SCALE

Structural CALculations Ensemble

SCALE is a Windows program and iPad app for producing neatly composed pages of structural engineers' calculations and component details for the design of a variety of components such as beams, columns and slabs, using steel, concrete, masonry and timber.

SCALE incorporates a library of over 600 proforma calculations, any of which may be selected for use when SCALE is run. The content of this library is continually under review, proformas being added or modified as codes of practice develop and change. The user can select between Eurocode design and previous British Standard design at the start of each proforma calculation.

SCALE also incorporates NL-STRESS, a software package for the elastic, plastic, and stability analysis of 2D/3D skeletal structures.

Your pdf viewer may show a list of bookmarks to the left of this document, click on the bookmarks to jump to the relevent section. Alternatively click on the highlighted links in the table of contents that follows, or click on any link inside the document to jump to the desired section.

i. Privacy

The SCALE iPad app offers In-App Purchases via the App Store to provide monthly and annual subscriptions. The app securely communicates with iTunes via the Fitzroy servers, but no user-identifiable information is passed to, or stored by, Fitzroy Systems or any third party.

Each existing SCALE user with a supported SCALE Sole Practitioner licence, can access the SCALE iPad app with their Fitzroy Systems subscription log-in details. Use of the log-in subscription details may be securely logged on the Fitzroy servers. Only the use of the log-in subscription details will be stored, no additional user-identifiable information is transmitted or stored via the servers.

When using the SCALE iPad app or Windows profgram, no user or usage data is passed to or collected by Fitzroy Systems or any third party.

No log-in is required to use the app.

No personal information is required to be entered to use the app.

The SCALE iPad app and Windows program does not use Location Services, nor does it use Contacts, Photos, or any other APIs to access user data.

Fitzroy Systems follows all requirements for complying with the European Union's General Data Protection Regulation ("GDPR").

For further information please email sales@fitzroy.com.

ii. Intellectual Property

SCALE utilises the following libraries and resources:

- Haru PDF 2.0.8 copyright 1999-2006 Takeshi Kanno
- icons8.com buttons
- LibPNG 1.2.16 copyright 2004,2006 Glenn Randers-Pehrson
- ZLib 1.2.3 copyright 1995-2005 Jean-loup Gailly and Mark Adler
- VFR Reader 2.8.6 copyright 2011-2015 Julius Oklamcak
- APNumberPad copyright 2017 Andrew Podkovyrin
- TLIndexPathTools copyright 2014 Tractable Labs
- OGRE 1.10.11 Graphics Engine copyright 2000-2017 Torus Knot Software
- FreeType 2.3.5 font library copyright 2000-2007 The FreeType Project
- FreeImage 3.11 image library copyright 2003-2008 FreeImage
- Liberation Mono fonts copyright 2012 Red Hat, Inc.
- JGActionSheet copyright 2014 Jonas Gessner
- SSZipArchive copyright 2010-2015 Sam Soffes.

1. Introduction to SCALE

2. SCALE User's Manual

3. LUCID User's Manual

4. SPADE User's Manual

5. NL-STRESS User's Manual

6. NL-VIEW User's Manual, a 3D viewer for NL-STRESS

7. NL-STRESS Reference Manual

8. TAPE User's Manual

9. Brief description of every proforma

10. PRAXIS Reference Manual

1. Introduction to SCALE - Structural CALculations Ensemble

SCALE is a computer system comprising several elements for producing:

- neatly composed pages of structural engineers' calculations which can be easily checked.

- component details and reinforced concrete details for the design of a variety of components such as beams, columns and slabs.

- elastic, plastic, and stability analyses of 2D and 3D skeletal structures.

The purpose of each is summarised in this section.

1.1 SCALE - Structural Calculations

SCALE incorporates a library of proforma calculations, any of which may be selected for use when SCALE is run. The content of this library is continually under review, proformas being added or modified as codes of practice develop and change.

A typical calculation produced by SCALE is shown in Figure 1.1.

The output from SCALE is a set of pages, neatly titled, dated and numbered, containing calculations and component details made to a standard suitable for submission to a checking authority.

After twenty five years of development of software-produced structural calculations, Fitzroy Systems remains committed to the fundamental concepts of SCALE:

- Interactive operation (question and answer), it is the easiest for the production of engineering calculations - see below.
- Practical calculations, for example you are not limited to just axial load on column bases, you may have bi-axial bending moments without having to create your own 'specials'.

Q&A is used where the data would not sensibly fit into a data box (see below). Many items of engineering data require help (background colour normally green) to explain what is required, to show a table, or to give general advice. The first prompt in SCALE option 070, which is a simple test example - and in the majority of other SCALE options - is 'Location'. It is expected that the engineer will type a description e.g. 'Beam Type M2 on grid lines A1-A3, B1-B3 and D1-D3', to locate the calculation to the project for which it has been prepared. SCALE uses Q&A for this prompt, displaying any previously given answer which may be edited.

A data box is used when the data is simple (e.g. giving sizes of a member or cross-section). Typically the left side of the screen shows a picture, the data box being on the right.

The third way by which data may be input is by extraction of the data from the stack file. The stack file may be created by running another option (e.g. LUCID reinforced concrete details create a stack file containing bar data which the bar scheduler uses), and provides a means of chaining together several routines to do a complex problem.

```
Knight (CMG) Ltd.                                         Page:  1
Unit 42, Castle Business Park, Marblewick.             Made by:  IFB
Job: New Civic Centre,                                    Date:  22 May
     Unit J, East Ancillary Building.                   Ref No:  JEAB7
______________________________________________________________________

Location: Revolve area under y=e^x about y axis from x=1 to 2

Integration of a general function using Simpson's rule
______________________________________________________

If y1,y2,y3 are the ordinates at a-h, a and a+h respectively,
Simpson's rule may be written:
 a+h
 ⌠
 |    f(x)dx  approx=  h/3 ( f(a-h) + 4f(a) + f(a+h) )
 ⌡
 a-h

Usually Simpson's rule is applied by dividing the area between the
ordinates y1 and y2n+1 into 2n strips.  This gives for 10 strips:
 b
 ⌠
 |  f(x)dx approx= h/3(y1 + y11 + 4(y2+y4+y6+y8+y10) + 2(y3+y5+y7+y9))
 ⌡
 a                         where h = (b-a)/10

Lower limit        a=1
Upper limit        b=2
Width of strip     h=(b-a)/10=0.1
Function of x      f(x)= 2*PI*x*e^x
Position 1         y(1)=2*PI*x*e^x=17.07946827
Position 2         y(2)=2*PI*x*e^x=20.76330477
Position 3         y(3)=2*PI*x*e^x=25.03309153
Position 4         y(4)=2*PI*x*e^x=29.97133177
Position 5         y(5)=2*PI*x*e^x=35.67140147
Position 6         y(6)=2*PI*x*e^x=42.23892371
Position 7         y(7)=2*PI*x*e^x=49.79331205
Position 8         y(8)=2*PI*x*e^x=58.46950286
Position 9         y(9)=2*PI*x*e^x=68.41989962
Position 10        y(10)=2*PI*x*e^x=79.81655449
Position 11        y(11)=2*PI*x*e^x=92.85361549
Integral
I=h/3*(y(1)+y(11)+4*(y(2)+y(4)+y(6)+y(8)+y(10))+2*(y(3)+y(5)+y(7)
  +y(9)))=46.42689878
```

Figure 1.1 Typical calculation produced by SCALE

1.2 LUCID - Reinforced concrete details

SCALE includes LUCID to produce reinforced concrete details for a variety of components such as beams, slabs and columns. LUCID works in the same way as SCALE.

The details and reinforcing steel schedules generally follow the recommendations of the old Cement & Concrete Association, the British Standards, and the Eurocodes.

LUCID generates drawings both as A4 pdf documents, and as hpgl text files which may be translated into dxf for inclusion in any other CAD system.

1.3 SPADE - Steelwork, timber and masonry details

SCALE includes SPADE to produce structural steelwork and other details for a variety of components such as beams, slabs and columns. SPADE works in the same way as SCALE.

SPADE generates drawings both as A4 pdf documents, and as hpgl text files which may be translated into dxf for inclusion in any other CAD system.

1.4 NL-STRESS - Structural analysis

The structural analysis program is called NL-STRESS which stands for Non-Linear STRESS. This is an advanced software package for the analysis of two and three-dimensional engineering structures. NL-STRESS has Department of Transport approval ref MOT/EBP/254C.

Options 800-999 give access to NL-STRESS via SCALE, these options permit the analysis of: arches, continuous beams, culverts, deep beams, beams curved on plan, suspended floor slabs, sub-frames, multi-storey frames, portal frames, raft foundations, a variety of roof trusses, shear walls, trestles and pipe racks.

NL-STRESS processes a text file containing the data for the structure to be analysed. Options 800 to 999 will prepare such a file from a question and answer dialogue with the user.

NL-STRESS data files can be amended or entered directly using the menu option "Run NL-STRESS directly". The structure can then be analysed and the results viewed on screen.

After analysis the results can be viewed interactively in 3D using the NL-VIEW option.

2. SCALE User's Manual

2.1 About SCALE

SCALE is an iPad app for producing neatly composed pages of structural engineers' calculations and component details for the design of a variety of components such as beams, columns and slabs.

SCALE incorporates a library of proforma calculations, any of which may be selected for use when SCALE is run. The content of this library is continually under review, proformas being added or modified as codes of practice develop and change.

SCALE's heritage started in the days of DOS, with the software being constantly upgraded over the years, keeping up with developments in Microsoft Windows, with this version for the iPad being released on the App Store in 2018.

Recent improvements include:

- now much easier to view the calculations mid-calculation, by simply scrolling the window,
- now much easier to go back and forwards when doing a calculation,
- can now switch between normal calcs, condensed calcs and summary calcs at any point by tapping on the Settings button.

2.2 User interface

All user interface elements in SCALE are native iPad controls and function like any other iPad app.

Items of data (such as the name of a file containing page headings) are entered in textfields. A textfield is a box in which the cursor can be placed. The cursor is a flashing vertical line.

To bring the cursor to the desired location:

- tap the screen, at the desired location in that field,
- move it to any other position using left or right arrow key.

Type the item to be entered in the field.

The first character typed appears at the current cursor position, the cursor stepping one place rightwards to make room for the next.

To delete the character after the cursor, tap the "del" key.

To delete the character to the left of the cursor (and drag the rest of the item one position leftwards), tap the Backspace key.

2.3 SCALE subscriptions

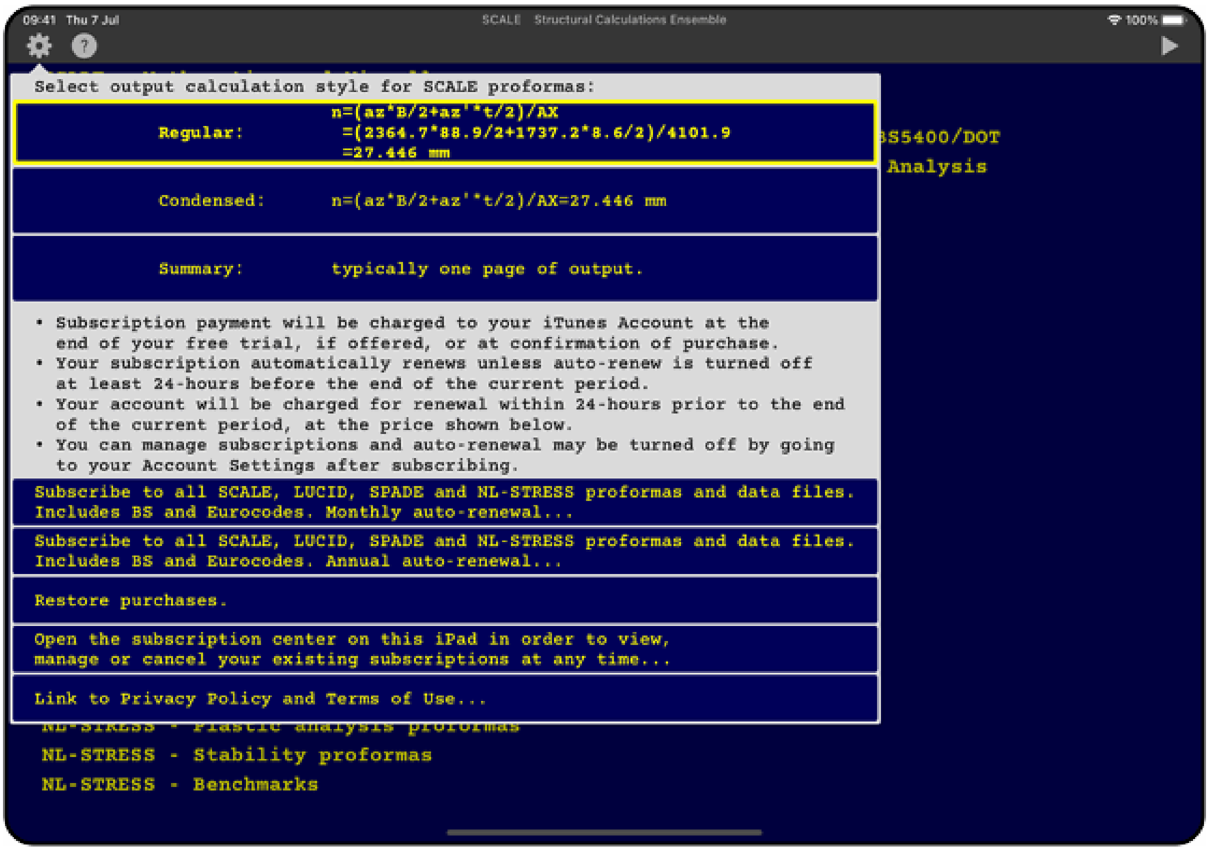


Figure 2.1: Settings: selecting subscriptions and calculation style

Tap on the "settings" button on the toolbar, as shown in Figure 2.1, to bring up a dialog with various subscription options. To start a monthly or annual auto-renewal subscription, tap on the appropriate button, and respond to the iTunes store popups as usual.

Tap on the "restore purchases" button if you already have a subscription on another iPad or have reinstalled SCALE on this iPad. SCALE will then communicate with the iTunes store, which may prompt for your details as usual.

If you would like to manage your existing subscriptions, tap on the "Open subscription center" button, this provides a shortcut to the subscription center from where you can manage or cancel existing subscriptions.

Existing SCALE Sole Practitioner users can run proforma 11 to enter their log-in details as part of their annual support and update package.

2.4 Proforma selection

Figure 2.2: Selecting from the list of proformas.

Tap on the SCALE icon to launch SCALE.

When SCALE launches you are presented with a list of group headings covering all the proformas, see Figure 2.2. Tap on a group heading to expand the list and show all of the proformas in that group. Tap the heading again to collapse the list if required. Scroll the screen up and down as required to find the desired proforma, you can do this by dragging up or down with one finger, or by flicking up or down. The group heading remains visible at the top of the screen even when the list of proformas in that group has scrolled off the top of the screen. To select a proforma to run simply tap on it.

You will then be presented with a short description of the proforma, as shown in Figure 2.3, simply tap on the "►" key, "►" button or press return to start this proforma.

Figure 2.3: Proforma description and option number.

If you require a different proforma tap on the "◄" back button on the top left of the toolbar, to go back the main proforma selection screen, which will remember where you last were.

2.5 Page headings file

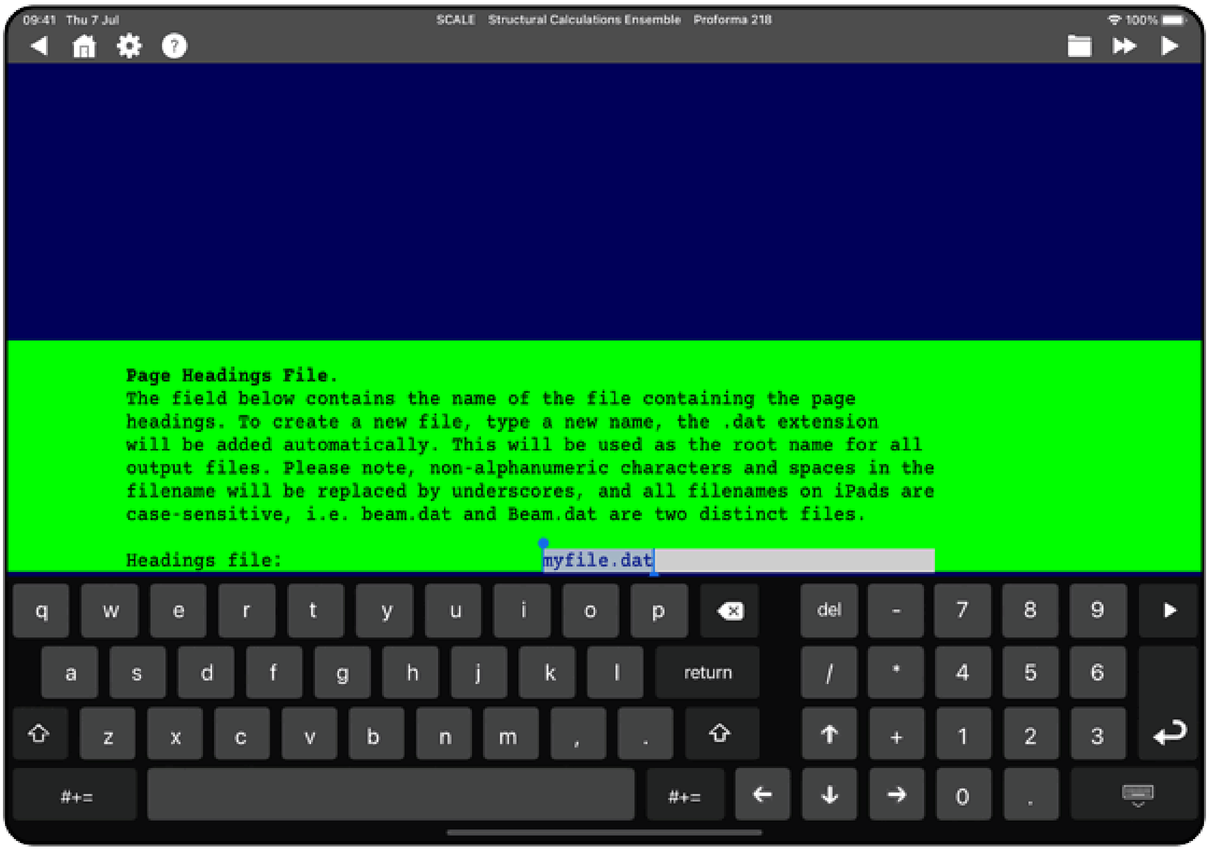


Figure 2.4: Entering page headings file.

On selecting a proforma, you move to a screen to enter the file name that contains the page headings, see Figure 2.4.

The page headings file allows you to re-use previous page headings, and its file name is used as the base file name for the creation of the final calculations, e.g. myfile.dat would lead to a calculations file called myfile.cal, with associated output files myfile.pdf and myfile.docx.

Type in a new file name if required and press return, or just press return to accept the last used file and its headings. You can use the on-screen keyboard for data entry, or an external bluetooth keyboard if you have one. The on-screen keyboard has three modes, the default - lower case letters, an upper case letters mode accessed by pressing either of the Shift keys (as seen in Figure 2.9), and a symbolic characters mode accessed by pressing the "#+=" key (as seen in Figure 2.11). The numeric keypad on the right side of the keyboard remains constant, and can be used for data entry at any time.

To go to the next screen press return, the "return" and "↵" keys on the keyboard are identical. You may also go to the next screen at any stage by tapping the "►" key on the on-screen keyboard, or the "►" button at the top right of the screen.

If you wish to choose a previously used file from a list, tap on the file browser button, to bring up a display of the list of headings files, as shown in Figure 2.5.

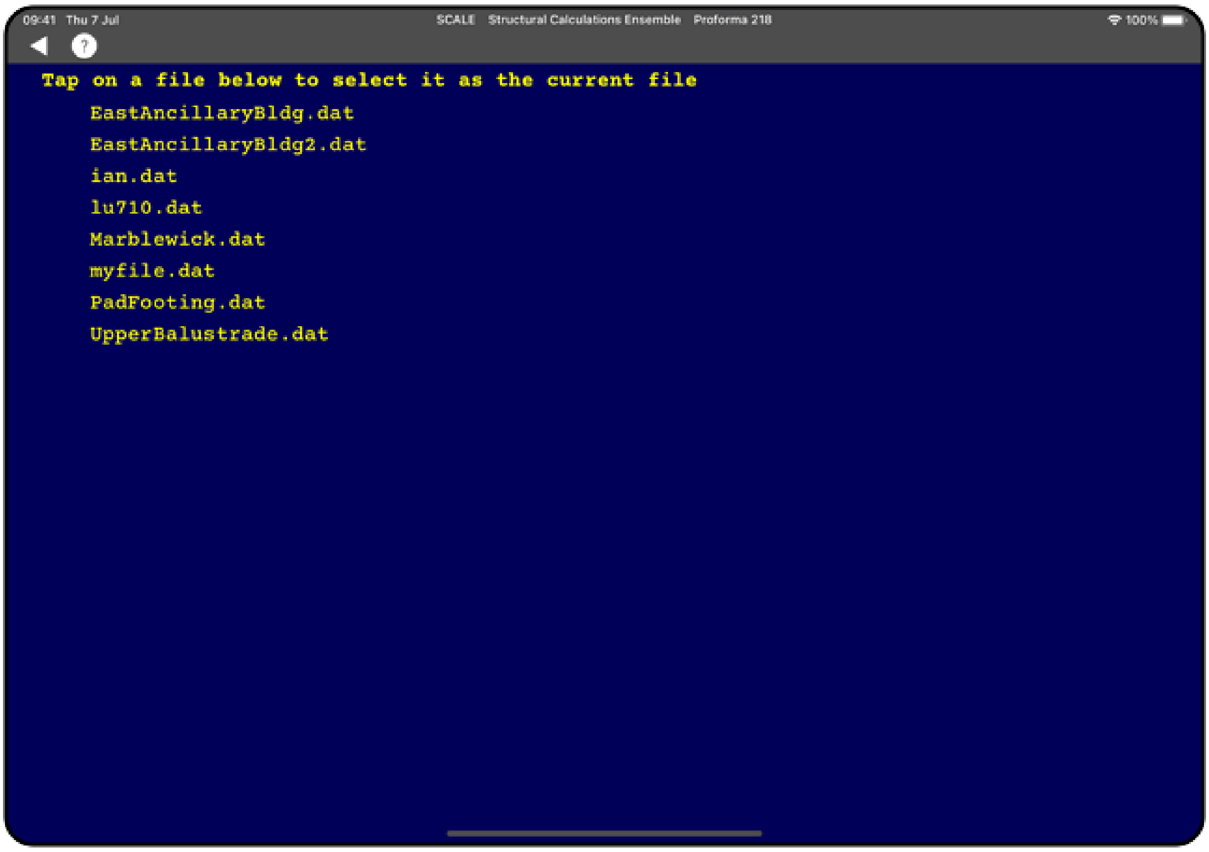


Figure 2.5: Select a previously used headings file.

To choose from the list, simply tap on the required file name, or tap on the "◀" back button to return to the Page Headings File screen without changing the file name.

2.6 Page headings

Figure 2.6: Editing the page headings.

On the Page Headings screen, see Figure 2.6, you can amend the details of the page headings if required. The data in each data field is automatically selected when the focus is moved to that data field, therefore you may simply start typing to replace the previous entry with new text. If you want to edit the existing text, move the cursor to the required starting point, either by touching the screen at the desired location, or by using the cursor keys on the keyboard. Move to the next data field by pressing return, or the down cursor key, or by tapping on the next data field on the screen. When touching and holding on the screen the standard Apple iPad magnifier will appear to make it easier to position the cursor at the chosen location.

If you wish to copy the contents of a previously used headings file, tap on the file browser button, then tap on the required file name from the displayed list of files, as illustrated in Figure 2.7.

Tapping on a file name, will copy the contents of the file selected from the list to the current file, and overwrite the contents of the current file.

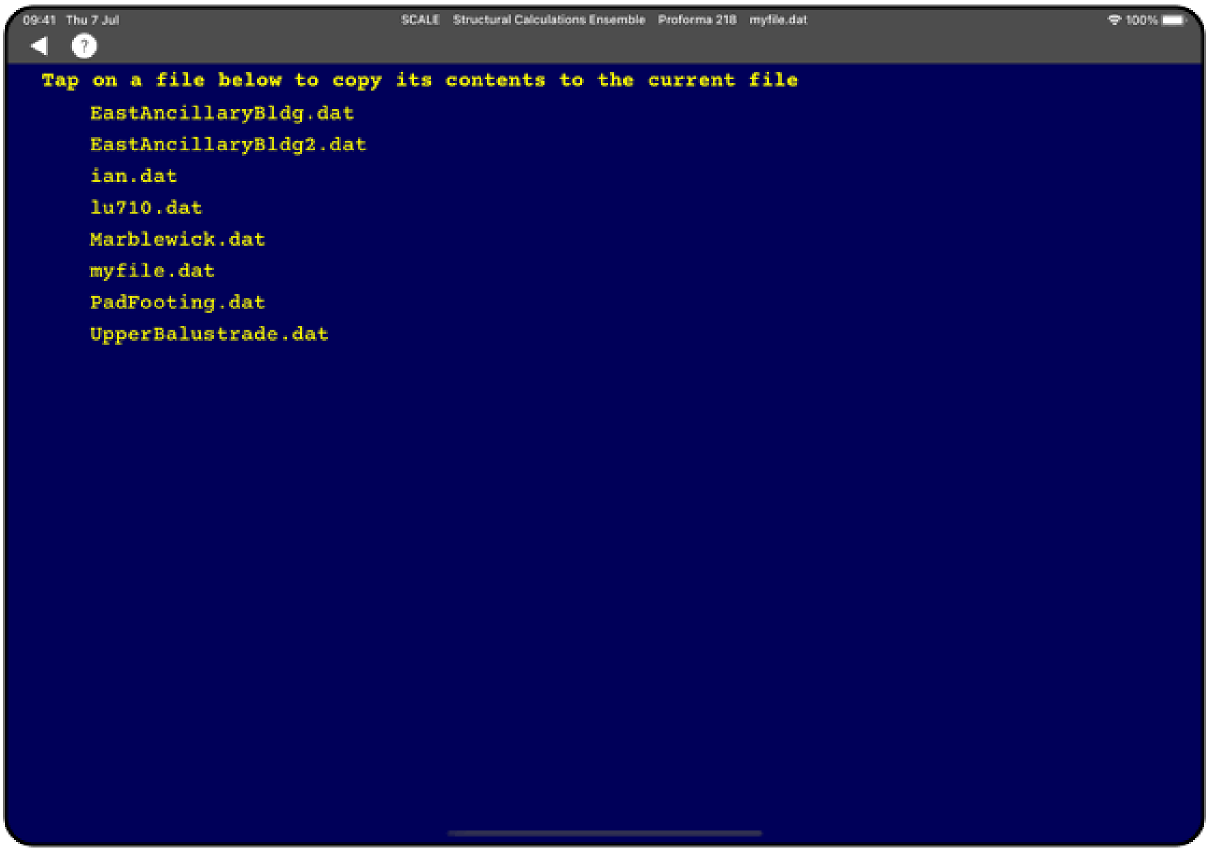


Figure 2.7: Copy the contents of an existing file.

To cancel, without overwriting the current file, tap on the "◄" back button to return to the Page Headings screen without changing the file contents.

2.7 Stack file

Figure 2.8: Selecting the stack file.

The next screen, see Figure 2.8, contains the name of the stack file where all the data entered for this current analysis is stored.

You can re-use an existing stack file, thus recalling previously entered data values, or you can provide a new file name. To reuse existing data values, type in the name of the stack file you used to store those values, and make sure to answer 0 for No to the later question of whether you want to use default values, otherwise the default values will be used instead of the previously stored values.

Tap on the file browser button to select from a list of previously used stack files (a screen similar to Figures 2.5 and 2.7 will appear) To edit the contents of the stack file, or headings file, open the file using Proforma 15 "Edit a file" see Section 2.21.

2.8 British Standards or Eurocodes

Figure 2.9: Choosing between BS and Eurocode calculations

Most proformas provide an option to choose between performing full calculations to the previous British Standards, or full calculations to the Eurocodes, see Figure 2.9. Answer 1 or 2 as appropriate to the prompt.

2.9 Default values

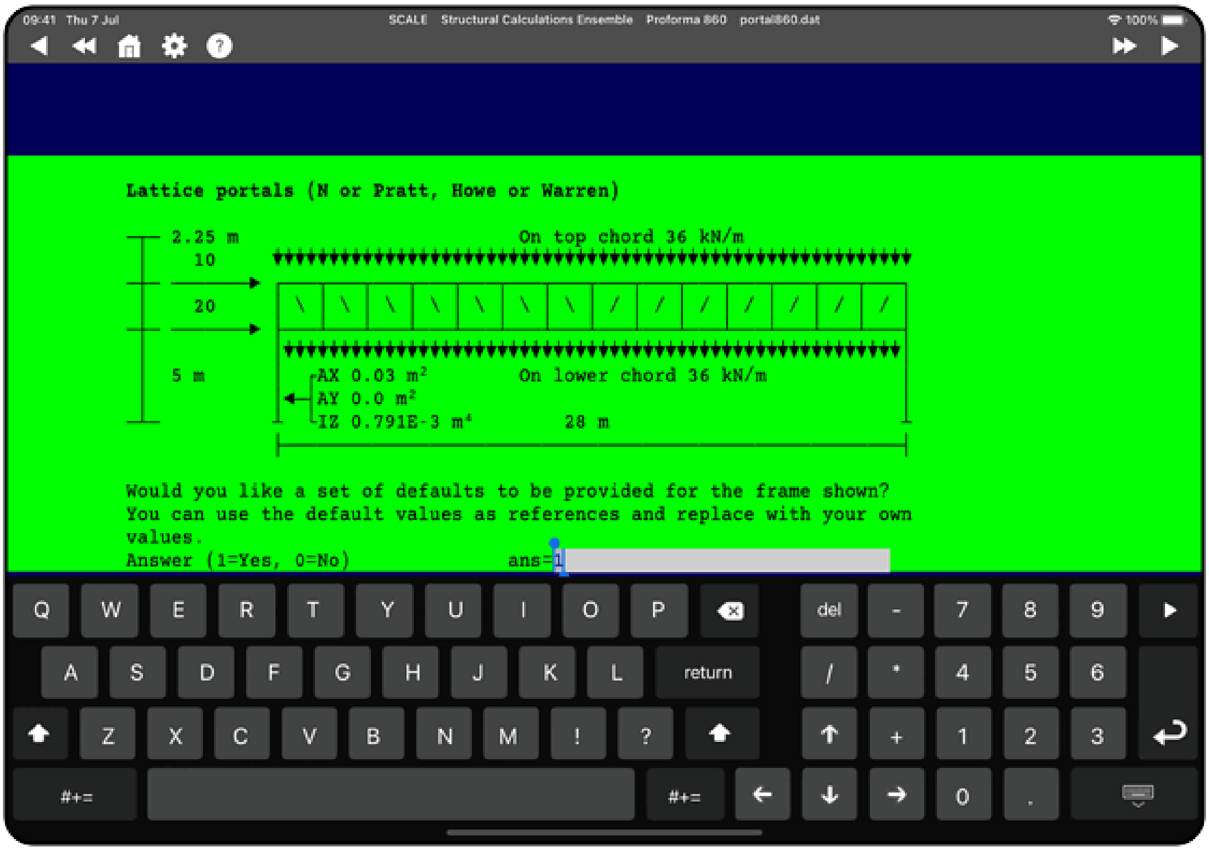


Figure 2.10: Choosing a set of default values

Most proformas offer a set of default values. Default values are values which are provided so that you can go through the calculation and see what information is needed without having to type in sensible values yourself. The default values are often taken from some published work. Select 1 for default values, 0 to re-use the previous values for this proforma stored in the stack file which you just selected.

You can run through the proforma to the end accepting the default values by pressing return as many times as necessary. If you wish to change one of the default values, edit or replace the value offered before pressing return.

2.10 Data input

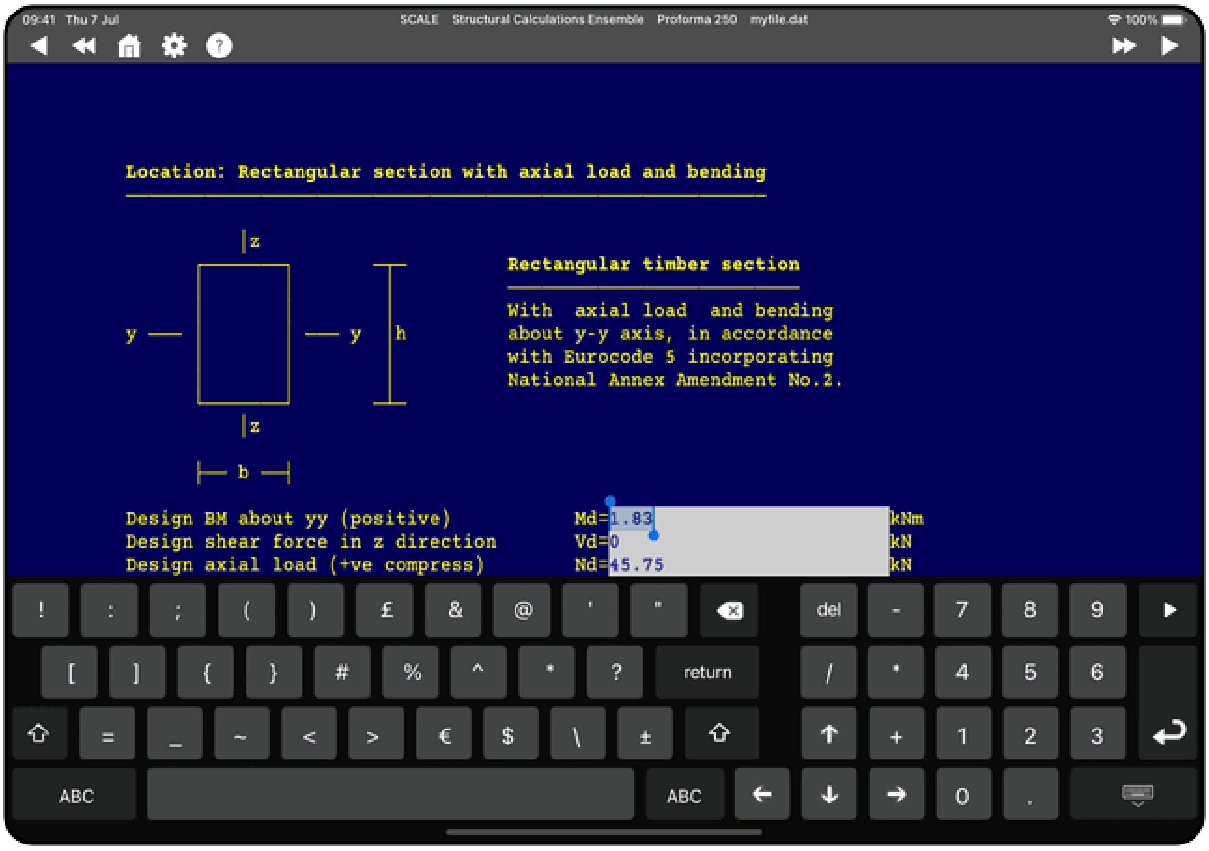


Figure 2.11: Entering data during the calculation

You then proceed to answer the questions as you run through the proforma, see Figure 2.11.

Most proformas start with 'Location' followed by a question prompt. The purpose of this prompt is to enable you to type in a response such as 'Beam on grid line B1-B3' and thereby locate the calculation. If you wish, you may press return alone to omit the location information and fill it in by hand at a later date.

Yellow text on a blue background denotes information sent to the calculations file; black text on a green background denotes help information only, which is not sent to the calculations file.

At any point you can drag or flick the screen down to view the calculation that has thus far been created. If you want to alter the calculation, tap on the "◀" button on the toolbar to go to the previous screen.

Because there can be thousands of calculations between each screen, when you go back a screen, in order to ensure that SCALE reaches the same internal state that was present for the previous screen, SCALE will re-calculate everything from the beginning, starting with the same original stack file and re-using all the responses thus far entered. This approach ensures consistency even when the previous screen is within a nested loop.

Many proformas have sections set up internally, such that tapping on the "◀◀" button on the toolbar will jump you back to the start of the current section.

If you are happy with the changes that have been made, and want to accept the default answers or the values stored in the stack file for all remaining questions, then tap on the "▶▶" fast forward button on the toolbar to jump to the end of the calculation. If the proforma is missing any data required to complete to the end then the proforma will drop out of fast forward mode and prompt you for the further data required.

2.11 Important note regarding moving back and forth

PLEASE NOTE: MOVING BACK AND FORTH
In many proformas, when you move forward through the calcs, SCALE will calculate values for you and present these on screen for you to alter as required. Going back and changing previously entered values, for example changing code=1 to code=2, will significantly change not only the order in which variables are input, but also in many cases the actual definition of what each variable represents. Because of this, when you go forward after going back, SCALE will present the new values it has calculated as if you reached this point in the calculation for the first time. To elaborate further: if you change a value, then go back and forward, the value you changed will not be stored and re-presented, the freshly calculated value will. Although this may appear frustrating at first, it is the most straightforward way to ensure the consistency of the data presented.

2.12 Normal, condensed, summary output

SCALE has three styles for the output calculations:

- Normal e.g.: n=(az*B/2+az'*t/2)/AX
  =(2364.7*88.9/2+1737.2*8.6/2)/4101.9
  =27.446

- Condensed e.g.: n=(az*B/2+az'*t/2)/AX=27.446

- Summary: typically one page of output.

To switch between these three styles, tap on the Settings button on the toolbar at the top of the screen, then tap on the desired style in the popup that appears, as shown in Figure 2.1. SCALE will then re-run through the data entered so far, and generate the required style of output.

Condensed and Summary styles are not available for the NL-STRESS proformas (chiefly 800 to 999), choosing an NL-STRESS proforma will automatically reset the style to Normal mode.

2.13 Viewing final calculations

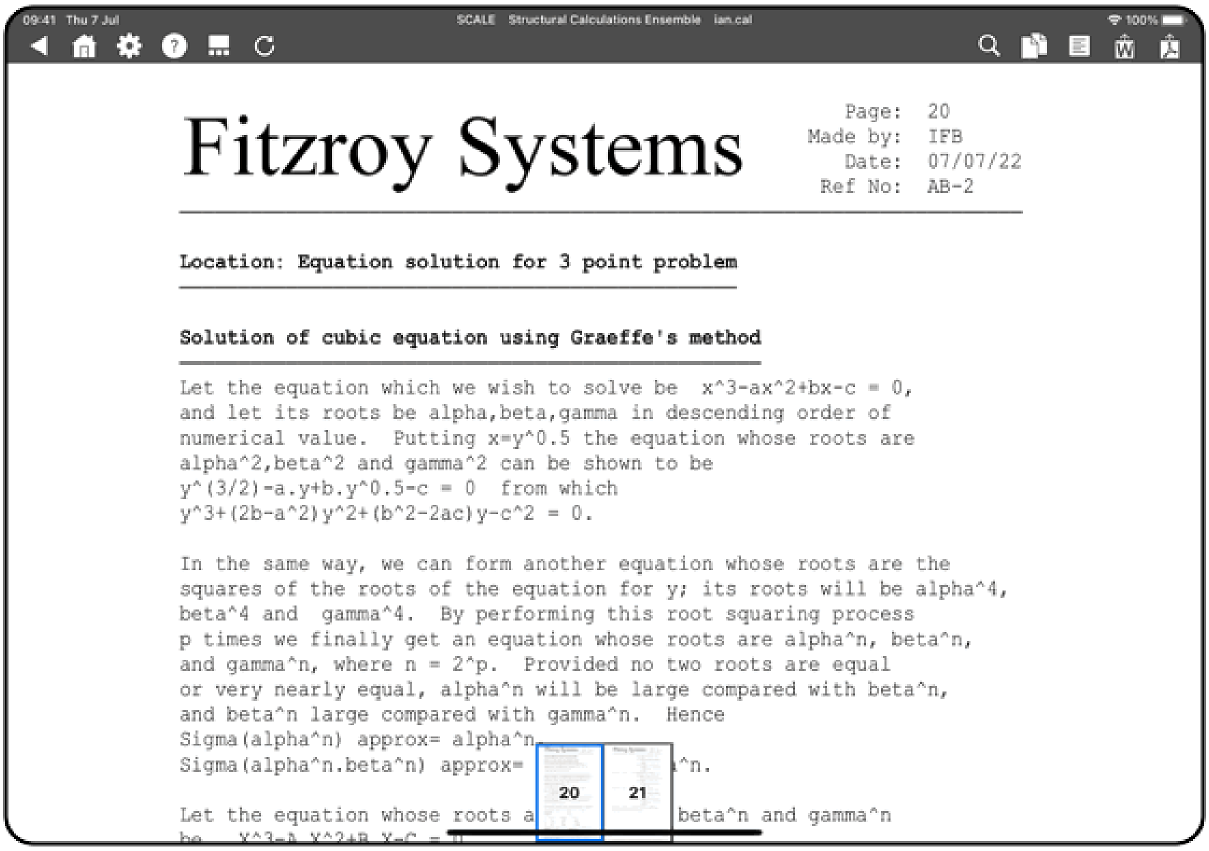


Figure 2.12: Viewing final calculations.
(Use proforma 18 to add a company logo to every page.)

At the end of the calculation you will go straight to viewing the pdf of the calculations file that has been created, this file includes any associated drawings and plots.

"Thumbnails" and "Rotate" buttons are added to the left hand side of the toolbar.

Tap on the "Thumbnails" button to toggle the display of the thumbnails navigation bar at the bottom of the screen. Tap on a thumbnail to jump directly to the corresponding page. The thumbnails prominently display the actual page numbers. Swipe the bar left and right to scroll the thumbnails to ends of the document.

Tap on the "Rotate" button to rotate the displayed pdf by 90 degrees. This is helpful for viewing NL-STRESS plots and LUCID and SPADE drawings which take up a full page in landscape format. Tap on the "Rotate" button again to switch back to portrait mode. The rotation is on screen only, and does not affect the actual pdf file.

The buttons at the right hand side of the toolbar now change to a selection of:

- Search,
- NL-VIEW,
- File,
- Edit,
- Upload DXF,
- Upload Word, and
- Upload pdf.

The exact buttons presented will vary depending on the calculations.

Tap on the "Search" button to search the calcs, see Section 2.22 for further details.

Tap on the "NL-VIEW" button to launch the NL-VIEW 3D viewer following an NL-STRESS analysis, see Chapter 6 for further details.

Tap on the "File" button to launch the File option to append sets of calculations together, see Section 2.20 for further details.

Tap on the "Edit" button to add comments to the calcs, see Section 2.14 for further details.

Tap on the "Upload DXF" button to save the LUCID or SPADE details as a DXF file, then copy that DXF elsewhere either via email, the cloud, printing, or in another app, see Section 2.15 for further details.

Tap on the "Upload Word" or "Upload pdf" buttons to copy the calculations elsewhere either via email, the cloud (e.g. iCloud/Dropbox), printing, or in another app, see Section 2.15 for further details.

2.14 Editing final calculations

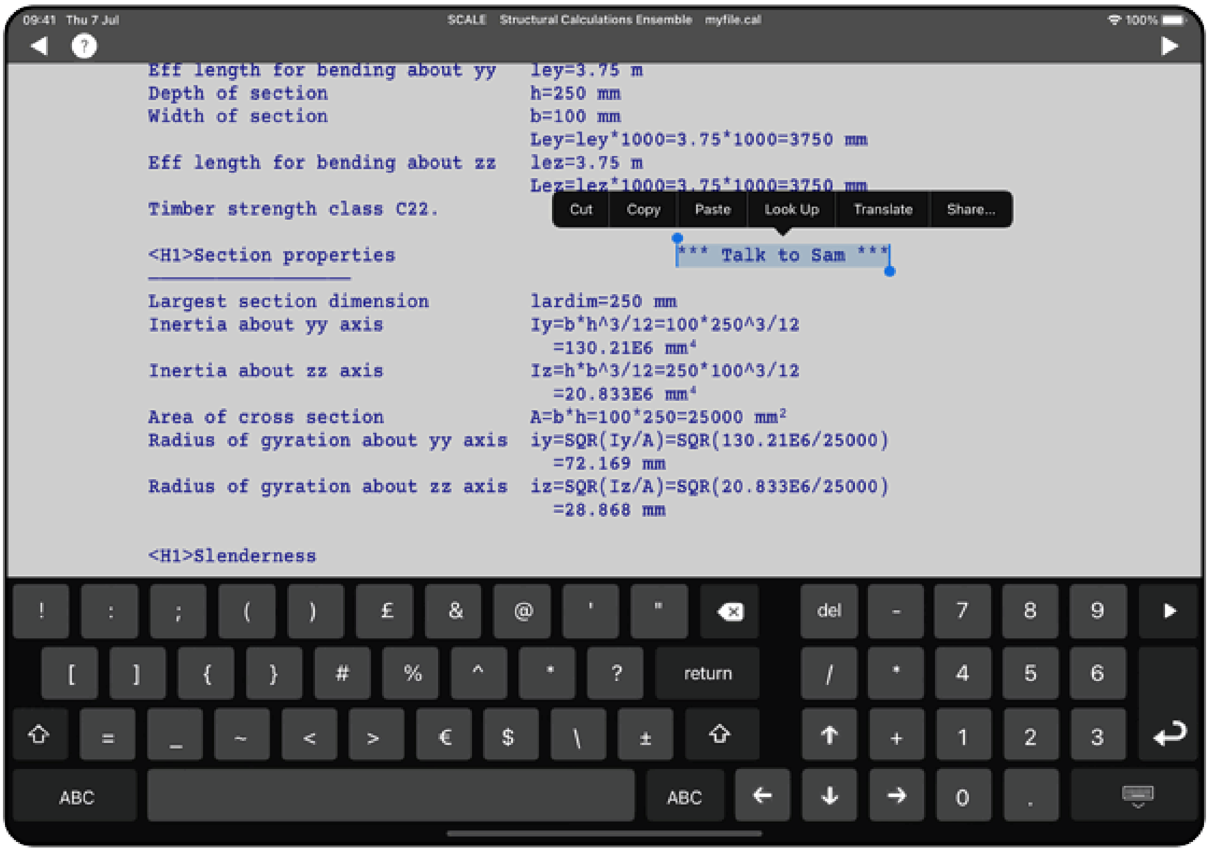


Figure 2.13: Adding comments to the final calculations.

On selecting the "edit" button you can now edit the calcs file to add any additional notes as required, see Figure 2.13. SCALE does not allow you to add extra lines to a calcs file as this could cause subsequent page headings to be misaligned.

When you have finished editing the calcs, tap on the next "►" button to save the changes and return to the viewing calcs screen.
If you wish to abandon your changes, tap on the back button "◄" at the top left of the screen to return to the viewing calcs screen.

Note: SCALE uses the four characters < H 1 > at the start of a line containing a heading to signify that the whole of that line is to be emboldened in the output.

2.15 Saving and uploading calculations

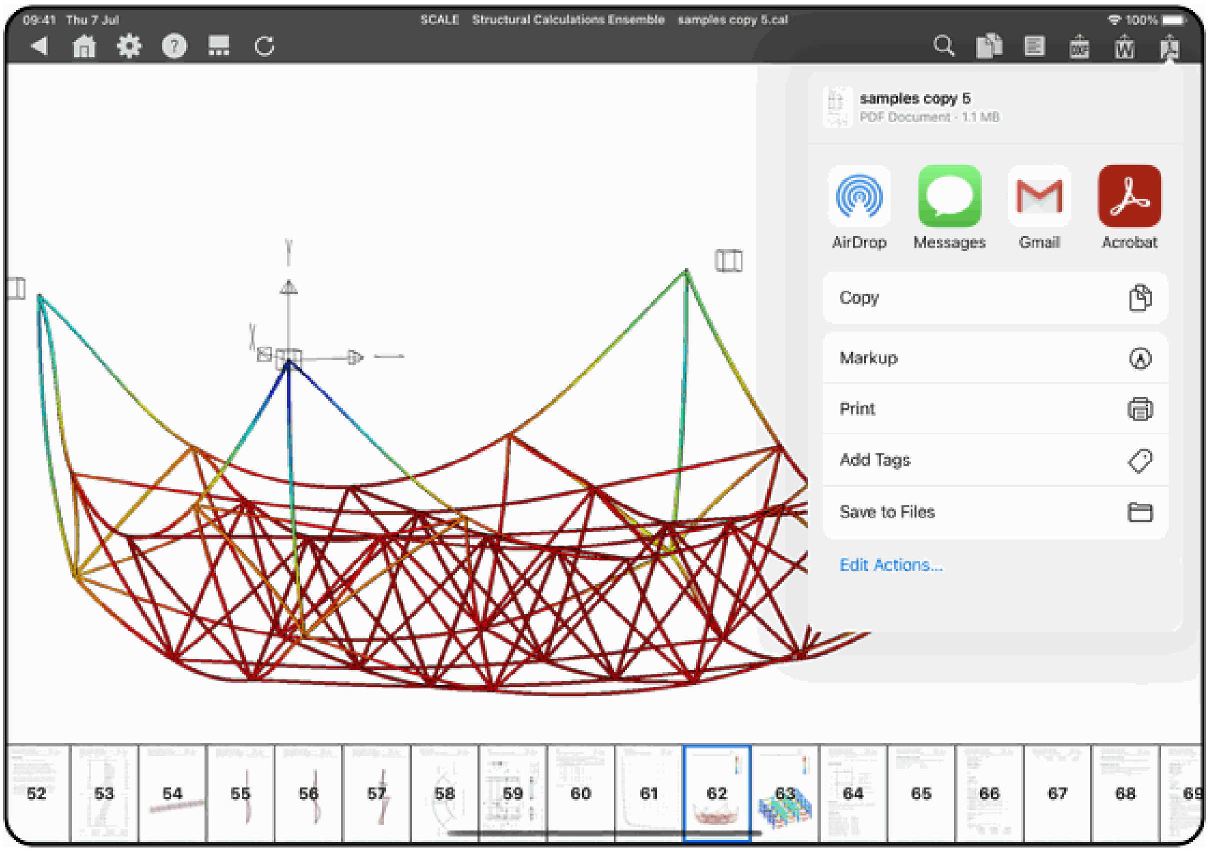


Figure 2.14: Uploading or printing the final calculations.

After editing, the calcs file can be uploaded or printed as a .pdf file by tapping on the Upload button, in the top right of the screen. This will launch a popup, e.g. as shown in Figure 2.14, which gives options to send to an AirPrint printer, send to email, send to cloud storage e.g. Dropbox, open in Adobe Acrobat etc, all depending on what other software is installed on your iPad.

The calcs file is saved into the current working directory, which is a sub-folder within the Documents folder of the SCALE app, see Section 2.19 for further details.

The contents of this folder can be viewed on the iPad using the iPad's Files app. The contents can also be viewed and files can be copied onto a computer by connecting the iPad to the computer and opening iTunes.

2.16 Viewing this help manual

Tap on the "help" button, a question mark in a circle, which is present at the top right of every screen, to display this Help Manual.

When finished viewing the Help Manual, tap on the back button "◄" at the top left of the screen to return to your previous screen.

The Help screen also displays an Upload button in the top right of the screen, tapping on this opens a popup, similar to the one described in Section 2.15, which allows you to print this file using an AirPrint printer, or export to another pdf viewer, or upload to a cloud service. But please note, this file covers the whole of SCALE and NL-STRESS, and so runs to nearly 500 pages!

Touch and drag, or touch and flick to move up and down between pages, pinch to zoom in and out as required. To aid navigation of the Help Manual, there are hyperlinked indexes at the start of the file, and over 200 hyperlinks within the text itself. Each page has a title with the chapter title and page number. Simply tap on any hyperlink to jump to that section.

Tap on the Search button above to search for a required word or phrase. (See Section 2.22 for further details.)

Tap on the Index button above (when viewing in the SCALE app) to display a list of bookmarks. Tap on a bookmark to jump to the relevant section. (See Section 2.23 for further details.)

2.17 Pull-through linking with NL-STRESS

Figure 2.15: Pull-through linking with NL-STRESS.

SCALE proformas which have a "#" symbol next to them in the list of proformas are able to pull-through axial loads, shear forces, bending moments, depths, breadths and lengths of any member of a structure which has been previously analysed using NL-STRESS with the same file name.

If the arrays file of a previous analysis is present (i.e. there is a .arr file present in the current working directory), then a proforma which has pull-through enabled will prompt you to select the required member number at the start of the proforma. If no pull-through of results is required then simply leave the prompt blank.

2.18 How to move the keyboard to the bottom of the screen

If you find the on screen keyboard positioned across the middle of the screen, and wish to move it, then you can drag it back to the bottom of the screen. To drag the keyboard, press and drag on the keyboard icon located at the bottom right of the on screen keyboard.

2.19 Use of files

In general, SCALE prompts for a headings file name which has the extension .dat and creates a new calculations file of the same name but with .pdf as its extension, or with .docx when output to Word is requested, or with .dxf when output to dxf is requested.

For example if SCALE were given the name george.dat the calculations would be found in a new files named george.pdf and george.docx.

All these files are saved into the current working directory, which is a sub-folder within the Documents folder of the SCALE app.
On first installation the working directory is called CALCS, but the working directory can be changed if required using Proforma 12 to set up separate directories for different projects. These files can be readily viewed using the iPad's built in Files app, by browsing to On My iPad->SCALE and opening the current working directory.
Many intermediate files are generated by SCALE in both text format and binary format, these intermediate files will also be found in the working directory.

The pages of results in the calculations file (.pdf) are given headings copied from the data file (.dat). The sole purpose of the data file is to provide such headings. (The page numbering, however, is not copied from the data file but given independently as later described.)

Filling out the page headings is simply a matter accepting or amending the name of the data file and the headings offered.

When SCALE reaches the end of the calculation, the responses you have typed in to replace the prompts are not lost; but are saved internally in a file of the same name as the proforma file but with the extension .stk (standing for stack of values). For example after running the proforma sc210.pro, the stack of values last used would be found in a new file named sc210.stk.

When SCALE option 210 is again requested, then providing you responded 0 to suppress the example defaults, those values previously given will be offered. The .stk file thereby saves you the need to retype loading and material properties which are standard for the job.

SCALE assumes that each .pdf calculations file will be printed out or sent elsewhere during the session, and to avoid cluttering up the storage, overwrites the last .pdf file. Thus if the .dat file (where the page headings were stored) was called c702.dat, then at the end of a SCALE session, the last calculation produced would be found in the file c702.pdf. It is expected that a single .dat file will suffice for many different sets of calculations e.g. columns in steel, concrete, timber or masonry. The .stk file (which contains the stack of input numbers), on the other hand, will only be appropriate to the proforma selected; obviously the values input for the design of a steel column will be different from those input for the design of a masonry column, and therefore the .stk file must be associated with its proforma. As an example, if SCALE option 410 was selected then on finishing this option, the stack of input numbers, would be found in sc410.stk. For most proformas, there are no more than a dozen numbers to be input, and it would be more hassle saving them to a named file, than typing them in again.

For more complex proformas the stack file can be saved at the end of the run by tapping on the File button, see Section 2.20.

In summary:

- on entering SCALE a file must be nominated, its name having .dat as its extension. This file should contain headings for the calculations to be produced by SCALE

- a new file is created by SCALE as a destination for calculations or error messages; this new file has the same name as that described above but with .pdf as its extension

- when SCALE is terminated normally, the responses typed in for loading and material properties etc. are saved internally in a .stk file and are available the next time SCALE is run.

2.20 The File option: collating results files, and saving stack files

Figure 2.16: The File option.

The File option provides the means of appending several sets of calculations to build a complete document comprising several hundred pages. Automatic page re-numbering based on the page number on the first page of calculations is done. After appending the current set of calculations to a document, the full document is brought into SCALE and displayed on screen.

Tap on the file browser button with a "c" on it to choose from a list of existing .cal files.

Tap on the file browser button with an "s" on it to choose from a list of existing .stk files.

2.21 Editing a file directly

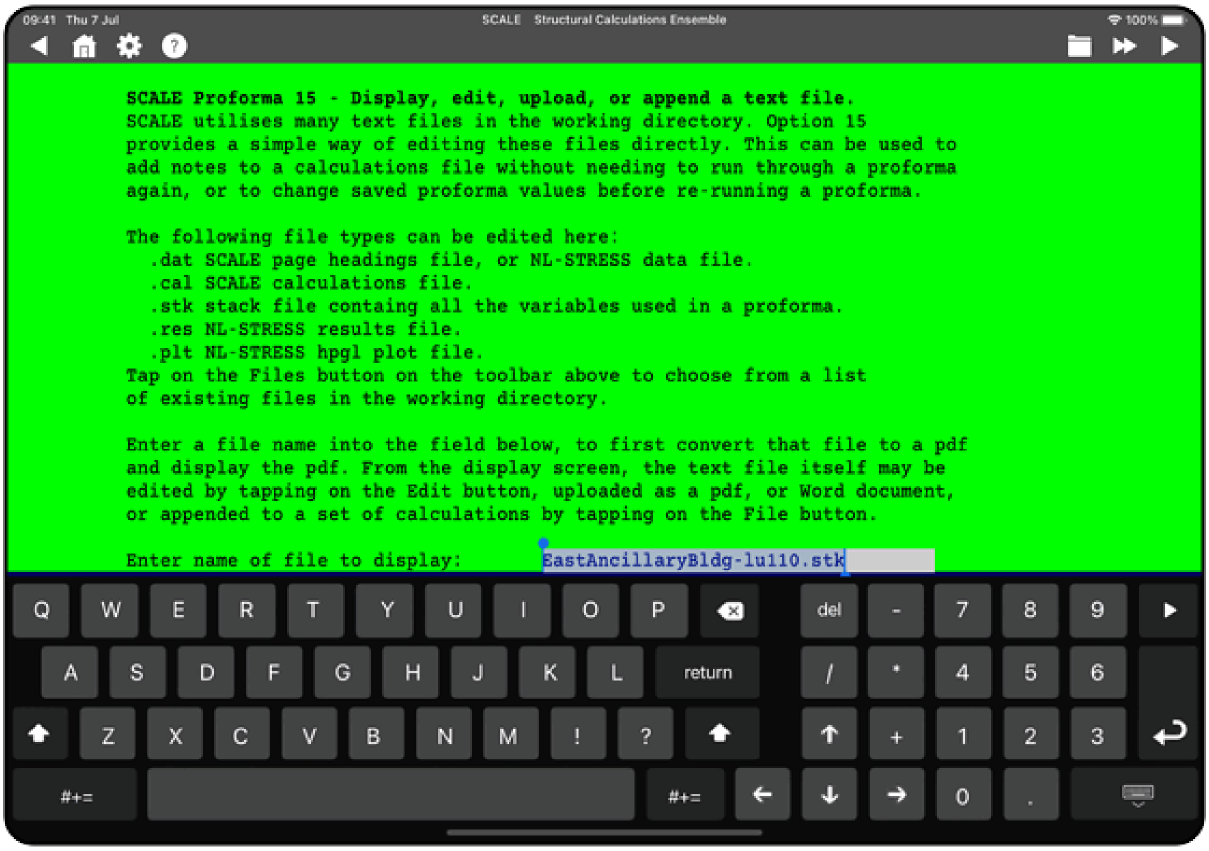


Figure 2.17: Proforma 15 - Edit a file

SCALE provides a way of editing the text files directly, as this can often be a quicker and easier way of making bulk changes than doing it within the question and answer interface of SCALE.

Select Proforma 15 from the main menu, see Figure 2.17, then enter the name of the file to edit.

To select from a list of files, tap on the file browser button at the top of the screen, this will then bring up a list of available files to edit, as shown in Figure 2.18. The available files are the .dat .cal .stk .res and .plt files in the current working directory inside the SCALE Documents folder on the iPad.

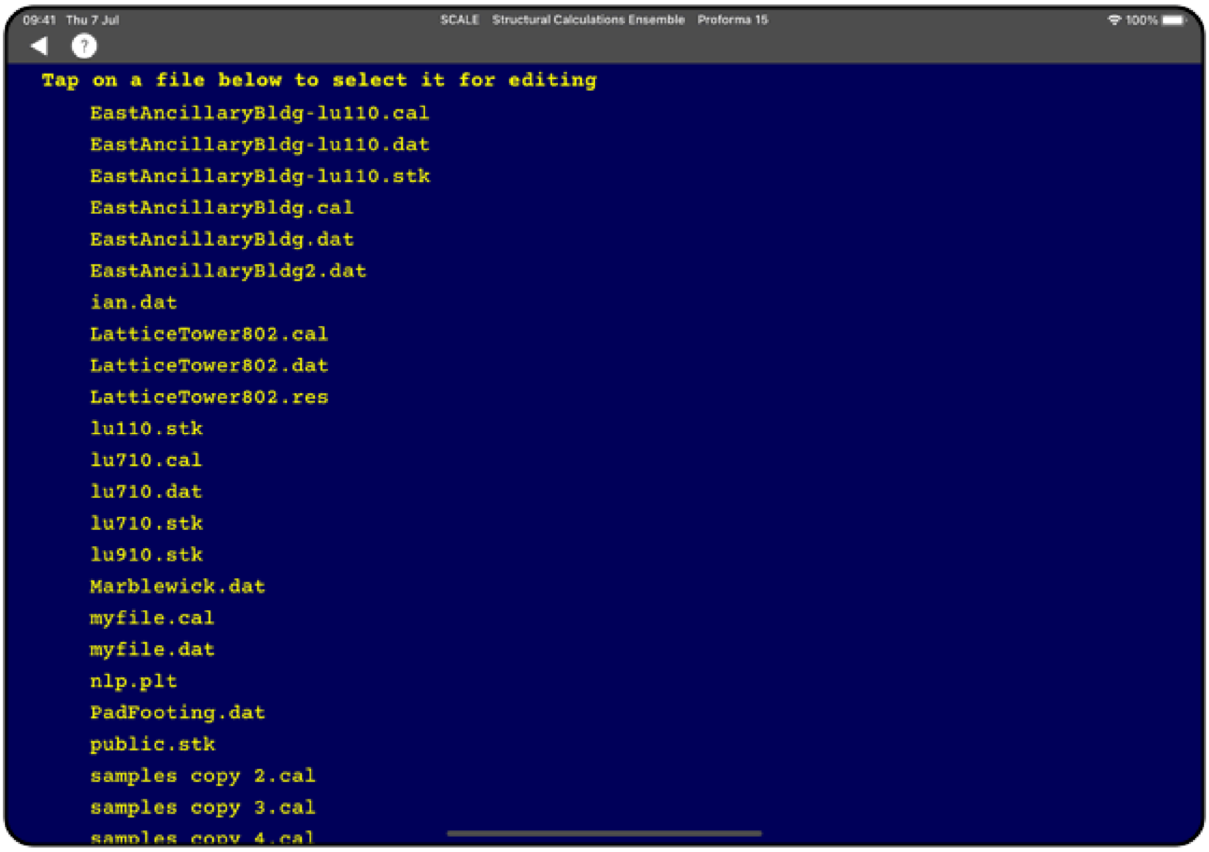
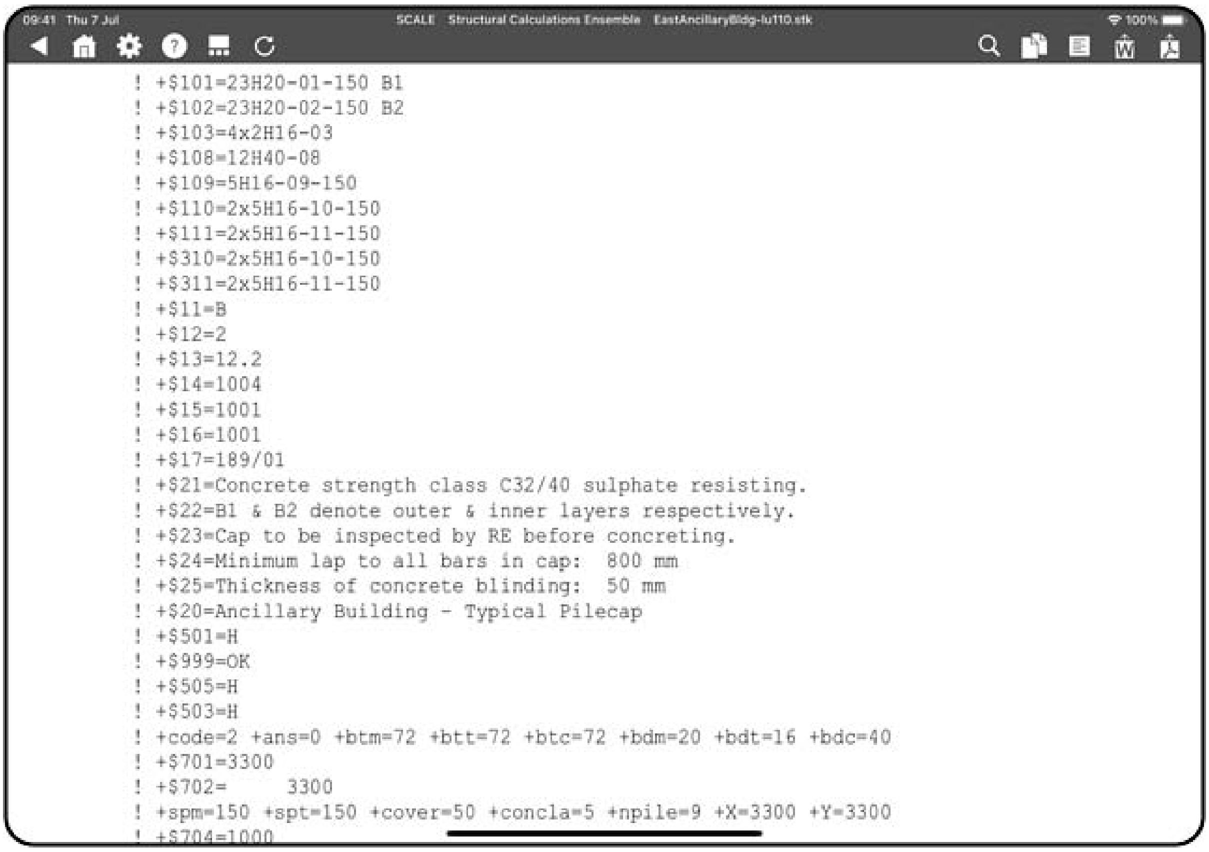


Figure 2.18: Select a file for editing.

Select a file from the list by tapping on it, or tap on the back button "◀" at the top left of the screen to return to the previous screen without choosing an alternative file.

If you wish to make more substantial changes, e.g. renaming, copying or deleting, then open the iPad's built in Files app, and navigate to On My iPad -> SCALE, then open the folder that corresponds to the current working directory and make the required changes to the files there.




```
! +$101=23H20-01-150 B1
! +$102=23H20-02-150 B2
! +$103=4x2H16-03
! +$108=12H40-08
! +$109=5H16-09-150
! +$110=2x5H16-10-150
! +$111=2x5H16-11-150
! +$310=2x5H16-10-150
! +$311=2x5H16-11-150
! +$11=B
! +$12=2
! +$13=12.2
! +$14=1004
! +$15=1001
! +$16=1001
! +$17=189/01
! +$21=Concrete strength class C32/40 sulphate resisting.
! +$22=B1 & B2 denote outer & inner layers respectively.
! +$23=Cap to be inspected by RE before concreting.
! +$24=Minimum lap to all bars in cap:  800 mm
! +$25=Thickness of concrete blinding:  50 mm
! +$20=Ancillary Building - Typical Pilecap
! +$501=H
! +$999=OK
! +$505=H
! +$503=H
! +code=2 +ans=0 +btm=72 +btt=72 +btc=72 +bdm=20 +bdt=16 +bdc=40
! +$701=3300
! +$702=     3300
! +spm=150 +spt=150 +cover=50 +concla=5 +npile=9 +X=3300 +Y=3300
! +$704=1000
```

Figure 2.19: Displaying the contents of a stack file.

To edit a file, you first have to display it with proforma 15.

Displaying the file first is necessary otherwise the toolbar buttons for appending using the File option, and for output as pdf and Word, would not work as expected. If a file was in the middle of being edited when a toolbar button was pressed, then it would be ambiguous as to which version of the file should be output. Only allowing output from the saved version removes any ambiguity.

When the file is displayed with proforma 15, as shown in Figure 2.19, tap on the Edit button to switch to editing mode, as shown in Figure 2.20.

Output files that are paginated with a set number of lines per page, for example, .cal and .res files, do not permit extra lines to be added: as this could mess up the pagination. For these files pressing 'return' will just move to the next line.

Input files, for example, .dat heading and data files and .stk stack files, allow extra lines breaks to be added: press 'return' in a line and that line will be split into two lines at the current cursor position. To remove a line break, position the cursor at the start on the line and press the backspace key. The current line will be moved onto the end of the line above, unless the combined line would be too long for SCALE in which case the lines will not be joined.

Figure 2.20: Editing a stack file directly.

Once the selected file to be edited has been chosen, make any changes required, then tap on the next "►" button to save the changes and go back to the screen displaying the file. If you wish to abandon your changes, tap on the back button "◄" at the top left of the screen.

This option provides a way of easily changes multiple values used by a proforma. If you are editing stack files, please note that the path through a proforma is dependent on the values entered. So, for example, making a beam longer in the stack file, may cause a proforma to ask for further inputs for slenderness. Values for these inputs might not already be present in the stack file, as they might not have been required for a previous run. SCALE will prompt for any such missing values when the proforma is run for the first time with the edited stack file. If the proforma is run in fast forward "►►" mode, then the proforma will automatically drop out of fast forward mode and prompt you when a missing value is required.

2.22 Searching the Help Manual and calculations file

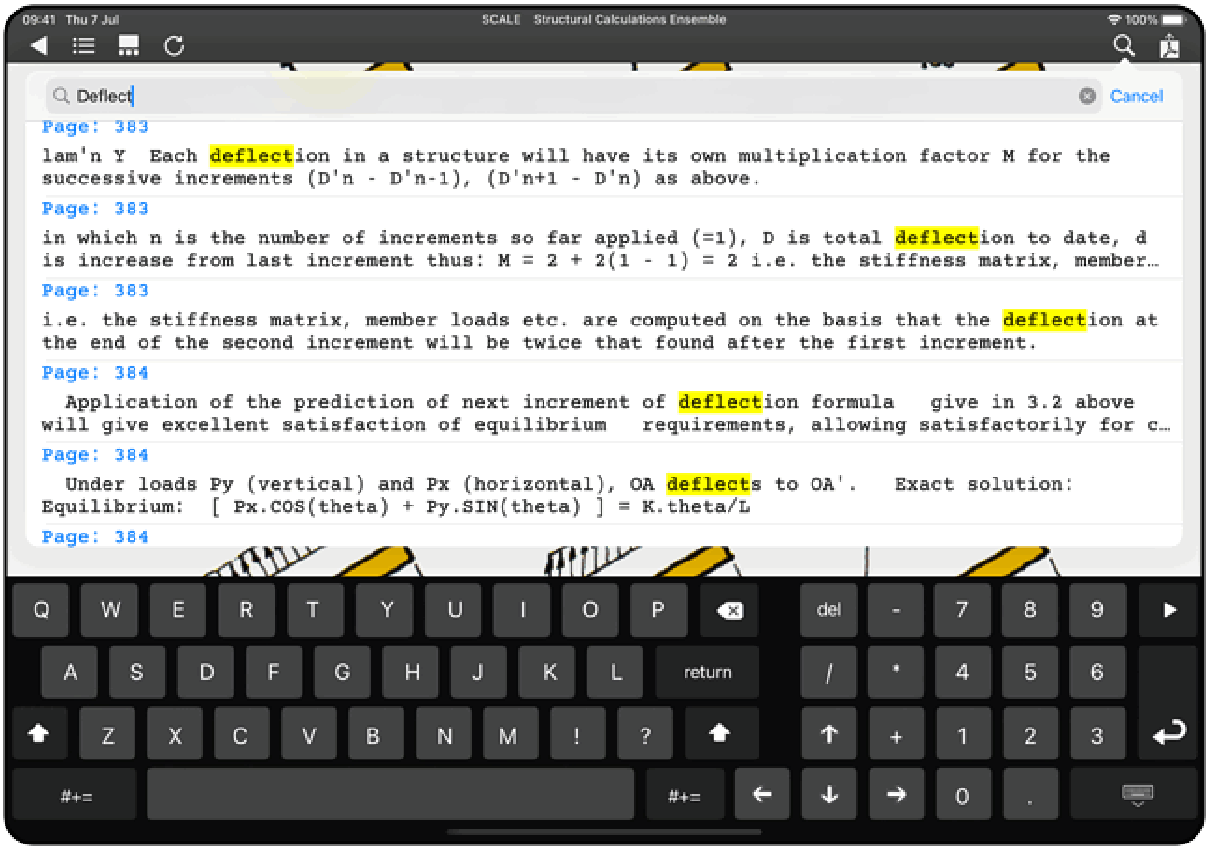


Figure 2.21: Searching for text in a document.

Tap on the Search button on the toolbar to launch the search bar. Type in the word or phrase that you need to look for, and the screen will be auto-filled with corresponding matches within the document. Swipe the list of matches up and down to scroll through as required, then tap on a match to jump to its location in the document, the searched word will then be present on the top line of the display.

Tap on the Cancel button to dismiss the search bar without moving to a different destination.

2.23 Using the Help Manual's index

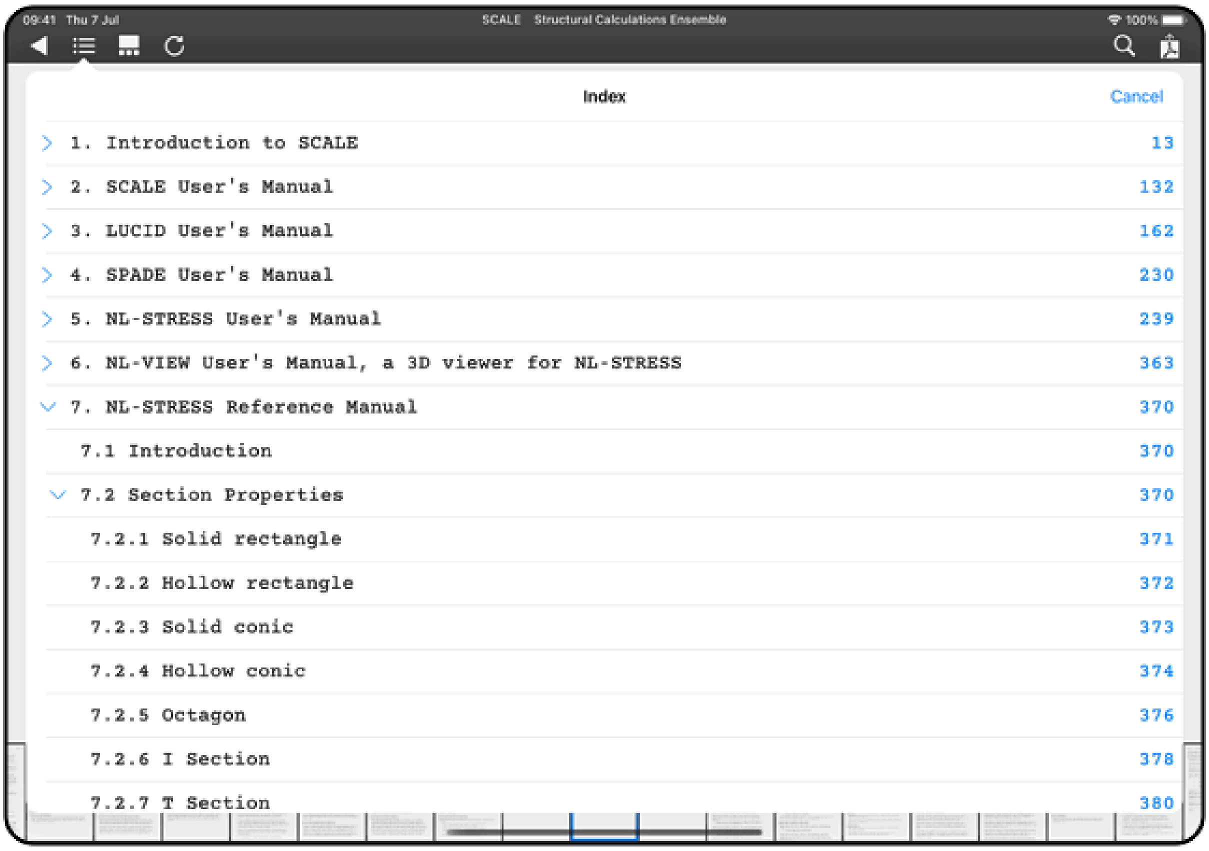
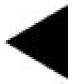


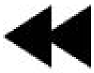
Figure 2.22: Using the Help Manual's index.

Tap on the Index button on the toolbar to bring up the table of contents for the Help Manual. The index includes hyperlinks to all the sections in the manual. Tap on a blue chevron on the left hand side to expand a section, tap again to collapse. Tap on a section heading to jump to that section in the Help Manual.

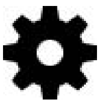
Tap on the Cancel button to dismiss the table of contents without moving to a different destination.

2.24 Buttons on the toolbar

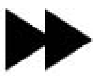

Go back to the previous screen.

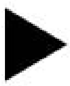

Go back to the start of the current section.


Go the initial proforma section screen.

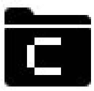

Open the Settings dialog.

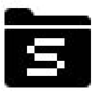

Open this help manual.


Fast forward to the end of the proforma.

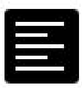

Go to the next input screen.


Open the file browser to select an existing file,
or copy the header information from an existing file.

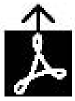

Open the file browser to select an existing
.cal file.


Open the file browser to select an existing
.stk file.


Launch the File option to append to an existing
file, and/or copy the stack file to a new file.

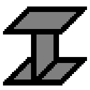

Edit the final calculations file.

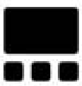

Rotate the pdf page between landscape and portrait,
the rotation is on screen only, and does not affect
the actual pdf file.

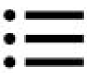

Upload/copy/print the final calculations as a pdf file.


Upload/copy the final calculations as a Word docx file.


Upload/copy the final calculations as a dxf file, this button is present for LUCID and SPADE calculations.


Launch the NL-VIEW 3D structural viewer, this button is present for the NL-STRESS proformas and after running NL-STRESS directly.


Toggle the display of the thumbnail bar on and off, when viewing the Help Manual or calculations file.


Launch the index to display the table of contents for this Help Manual.


Launch the search bar to quickly find text in the Help Manual or calculations file for desired text.

3. LUCID User's Manual

3.1 About LUCID

3.1.1 Scope

LUCID is a software package for producing reinforced concrete details for a variety of components such as beams, slabs and columns.

LUCID incorporates a library of proforma details, any of which may be selected for use when LUCID is run. The content of this library is continually under review, new details being added or modified as codes of practice develop and change.

The output from LUCID is an A4 drawing which may be sent directly to a printer, or saved as a pdf file. Prior to printing, the LUCID detail may be translated into a dxf file for inclusion in any CAD system.

3.1.2 Historical

Published papers:

- LUCID - a system for the production of detail drawings
  by RH Mayo and Professor LL Jones
  pub. Building Technology and Management, July 1973
- LUCID - an aid to structural detailing
  by Professor LL Jones
  pub. The Structural Engineer, January 1975
- LUCID - a cooperative venture in CAD
  by LL Jones, AJM Soane, RH Mayo and P Charlton
  CAD '82 Brighton

3.1.3 Computerised LUCID

LUCID scores over other reinforced concrete detailing systems in that the user does not have to draw the detail. LUCID is an 'expert system' which knows about detailing and builds the detail using the knowledge contained within the LUCID database. The engineer merely has to guide LUCID by giving answers to the various questions which LUCID asks.

3.1.4 Scales

LUCID scales the drawings wherever practical and shows the scale/s at the foot of the drawing.

A metric scale is chosen from the set:
1:10 1:15 1:20 1:25 1:30 1:50 1:75 1:100
which are generally available on triangular (Toblerone shaped) scale rules. The gap between 1:30 and 1:50 is significantly larger than any other and for this reason the scale of 1:40 is also adopted in preference to the not-to-scale alternative. For the same reason the scale of 1:60 is occasionally used for slabs, the philosophy being that it is better to use a non-standard scale than not to use any scale.

3.1.5 Dimensions

Reinforcement detailing is the art of choosing a suitable reinforcement pattern rather than great mathematical precision in positioning the bars. Traditionally dimensions are left off reinforcement details and this has been adopted in LUCID with the exception of the three main dimensions: length, breadth and depth of each component, are included to give the engineer something to put his/her scale against.

3.1.6 Curtailment

LUCID details adopt curtailment positions which are generally more conservative than code rules.

3.1.7 SI units

LUCID works in SI units (often referred to as metric units). Detailing is in accordance with BS8666:2005.

3.1.8 Calling up

The 'calling up' of reinforcement to British practice is as follows:

```
                           No.of   Dia(mm)     You may change No.of,
                             | Type | Mark     Type & Dia but please
                             |    | | |        do not change the bar
Examples of bar calling up:  └── 4H20-05       Mark if you wish to
                                 2X12-05       schedule the bars.
```

The engineer may depart from the above systems. If the engineer does depart it would be frustrating if the engineer's own method of 'calling up' kept being overwritten by the British system. Even if the engineer kept to say the British system, but changed the bar diameter from the 'calling up' automatically generated to save the engineer the trouble, then it would be frustrating if the bar diameter was changed back when the engineer was running through the proforma again. Accordingly it is necessary to keep track of any departure from the automatically generated data. This is done by keeping track of both the automatically generated calling up stored in e.g. string $310 and the engineer's amended calling up stored in e.g. string $110. A further complication arises when the engineer reruns the proforma changing e.g. a dimension. Changing just one dimension can affect several calling ups, so the system needs to update the $110 string from the $310 string if appropriate. This selectively updating is controlled by variables commencing 'new' e.g. 'newl' meaning new links/stirrups. If newl=0 then $310 does not get copied to $110, if newl=1 then $110 does get copied to $110.

3.1.9 Operation of the program

To check that the program is operating correctly, select LUCID option 410 at the start of SCALE, this is a single panel edge supported slab and (because it is simple) ideal for test purposes.

LUCID now reads the proforma detail for the slab. After the proforma detail has been read, accept Page 1 as the start page number and accept all the default values offered. When the detail has been completed the screen displays the detail on the screen.

3.2 Use of files

3.2.1 The data file

Page headings - comprising firm's name, address and job information - remain substantially unchanged for the duration of a project. This information is held in a data file - with name ending in the extension .DAT. An existing data file may be nominated, or a new one created. The information is stored on disk, a typical data file C702.DAT - as supplied - contains:

```
TITLE N G NEERS AND R K TECTS CO PARTNERSHIP
TITLE 101 HIGH STREET  PEVERILL  DORSET
TITLE JOB: NEW CIVIC CENTRE
TITLE      ANCILLARY BUILDING
MADEBY DWB
DATE 27.10.15
REFNO 95123
```

3.2.2 The proforma detail file

All LUCID proforma details have a file name starting with 'lu' followed by three digits. The three digits correspond to the option number used to select the proforma.

3.2.3 The stack file

When LUCID is terminated in a normal manner, the responses typed in by the user to replace the ???? prompts are not lost; but are saved in a file of the same name as the proforma file but with extension .stk (standing for stack of values). For example after running the proforma lu410, the stack of values last used would be found in a new file named lu410.stk.

When LUCID is restarted and proforma 410 is again requested, then providing the engineer refuses the example defaults, those values previously given will be offered. The .stk file thereby saves the user the need to retype data.

3.2.4 The finished detail file

In general, the LUCID program prompts for a file name which has the extension .dat and creates a new file of the same name but with .pdf as its extension which contains the finished detail written in HPGL.

The headings at the top of the finished detail are given headings copied from the data file (.dat). The sole purpose of the data file is to provide such headings. The page number, however, is not copied from the data file but given independently so that it may be changed easily for each detail.

The page number may have an upper case letter prefix e.g. FSL/3. Each time an option is selected the previous finished detail (held in the .pdf file) is overwritten. On exit from LUCID the .pdf file will contain the last finished detail produced for the job selected by the .dat file.

3.2.5 Bar schedules

LUCID options - which produce details - ask the user if bar schedule information is required. If the user responds positively, then the option being run writes a stack file (lu910.stk) at the same time as the detail is produced. This .stk file contains bar schedule information for use by the bar scheduler (option 910). When option 910 is run subsequently, and the user refuses the standard default values, then a bar schedule will be displayed for the detail. The user may then amend any or all lines in the schedule, if required.

3.3 The library of proformas

The library of proforma details is continually under review.
This section contains a brief description of all the LUCID proformas.

3.3.1 Beams

LUCID options 810 and 820 cover simply supported, continuous and cantilever beams. The notes in section 3.12 cover items such as: arrangements of bars, continuity through the column, pre-assigned bar marks, 'calling up', fixing dimensions, covers, concrete outlines, spacer bars, centre lines and grid lines, link shape codes, bottom span bars, links in section, right-hand support bars, scales, span top steel, details at external columns, cantilever beams.

3.3.2 Slabs

LUCID options 410, 420, 430, 440, 450 and lu460 cover one and two way spanning slabs supported along their edges, and flat slabs (with or without cantilevers) supported by columns. The notes in section 3.8 cover items such as: bar marks & 'calling up' fixing dimensions, covers, holes and openings, chairs for top reinforcement, centre lines and grid lines, bars shared by two panels, bar layers, scales, floors subdivided into smaller panels, isolated floor panels, bottom splice bars details, torsion steel, pre-assigned bar marks, orientation, drops and column heads, allocation of bar marks, column support shear reinforcement, edge reinforcement.

3.3.3 Columns

LUCID option 510 covers square, rectangular & circular columns, allowing for the 'column over' to have a variety of reductions in column shape. The notes in section 3.9 cover items such as: schematic elevation, section at mid height and near top of column, bar marks & 'calling up', L-bars at column head.

3.4 Using LUCID

Generally yellow text on a blue background denotes information used to build the detail; black text on a green background denotes help information only.

The proforma details usually offer a set of default values. Default values are values which are provided so that you can go through the calculation and see what information is needed without having to type in sensible values yourself. Accept the default values by pressing Return as many times as necessary to get you through the detail to the end. If you wish to change one of the default values, edit or replace the value offered before pressing Return.

See SCALE User's Manual for further information on program usage.

3.5 Foundations (lu110, lu120, lu130)

3.5.1 Introduction

These notes describes the use of the LUCID details available to enable the user to produce drawings both for isolated column foundations and for strip footings.

3.5.1.1 Isolated column foundations

The range of column foundations includes:

- pile caps, with 1, 2, 3... 9 piles in a group,
- square and rectangular reinforced concrete bases, and
- square and rectangular mass concrete bases.

The foundations are detailed both in plan and section, together with a detail of the connection to the column. This connection may take the form of starter bars, a pocket, or holding down bolts.

3.5.1.2 Strip footings

A range of fifteen reinforced concrete details are provided which cover independent footings, and both internal and external wall footings combined with a ground slab. The footings are detailed in section only and the extent of the flooring must be indicated on a general arrangement or key plan drawing.

3.5.1.3 Bar 'calling up' and scheduling

Bar 'calling up' follows the traditional method thus:

```
                              No.of    Dia(mm) Bar spacing(mm)
                                |  Type  |  Mark    |
                                |     |  |   |      |
Examples of bar calling up:     └── 14H20-05-200
                                     8H12-05-100 (Eurocode)
                                     8R12-05-100 (BS)
```

After printing the LUCID detail, use option 910 to produce a bar and weight schedule.

The bar schedule complies with the requirements of BS8666: 2000 using the shape code references, dimensioning and tolerancing given therein. The bar schedule is tabulated under the heading:

Member	Bar mark	Type and size	No. of mbr	No. of bar	Total no.	Lngth ofbar (mm)	Shape code	A (mm)	B (mm)	C (mm)	D (mm)	E/R (mm)	Rev ltr

Weights are given for each bar type (H, A, B, C, S or X) subdivided for bar diameters 16mm and under, and 20mm and over. (BS uses types H, R or X.)

3.5.2 Isolated column foundations

The basic linework for an isolated column foundation is built up from two details:

- the foundation detail; which consists of a plan and section of a pilecap, reinforced concrete base or mass concrete base, and
- the connection to the column detail; which shows starter bars, holding down bolts, a pocket, or a note that the connection is detailed elsewhere.

The foundation details consists of three quite separate subdivisions: pile caps, reinforced concrete bases and mass concrete bases.

3.5.2.1 Pile caps

The details for pile caps are summarised diagrammatically in Figure 3.5.1. The column always appears central on the pile cap and, as can be seen, for a given number of piles there is only one cap detail.

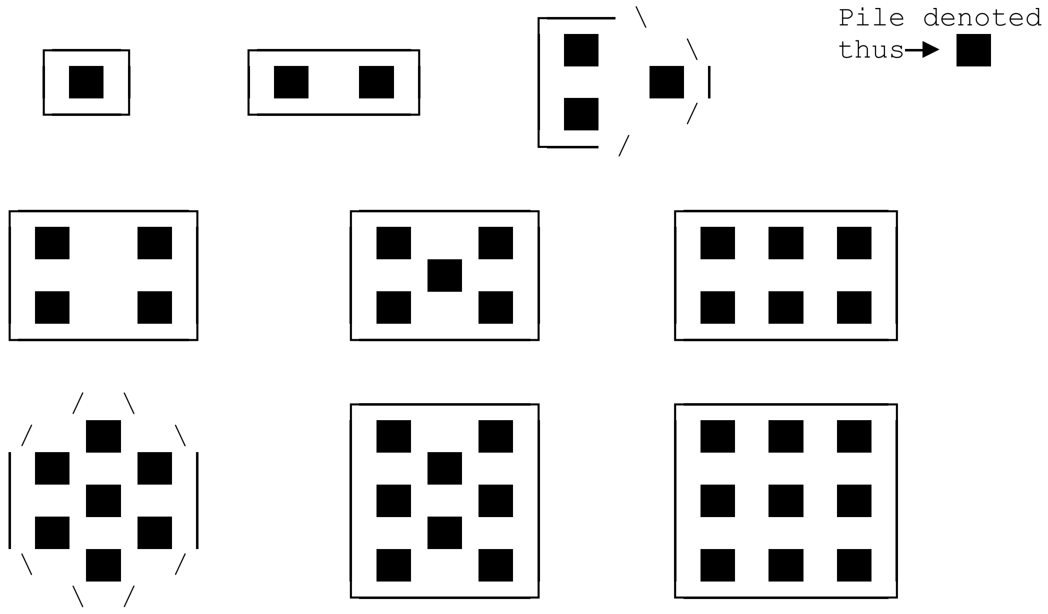


Figure 3.5.1 - SUMMARY OF DETAILS FOR PILE CAPS

The main bars on the pile cap details are drawn with a large radius bend, which is the detail adopted in many offices. However, these bars can be scheduled as having standard radius bends, providing such a detail satisfies the design requirements. Users may wish to add to their linework drawing a note on how reinforcement from the pile is to be exposed, bent and treated, if this has not been specified elsewhere.

Reinforcement calling up strings are generally completed in the standard manner but it should be noted that the 3 and 7-pile cap details have runs of bars which vary in length.

3.5.2.2 Reinforced concrete bases

The details for reinforced concrete bases are summarised diagrammatically in Figure 3.5.2. (For clarity, only bar runs spanning across the screen/paper are indicated.) Column position shown: ■

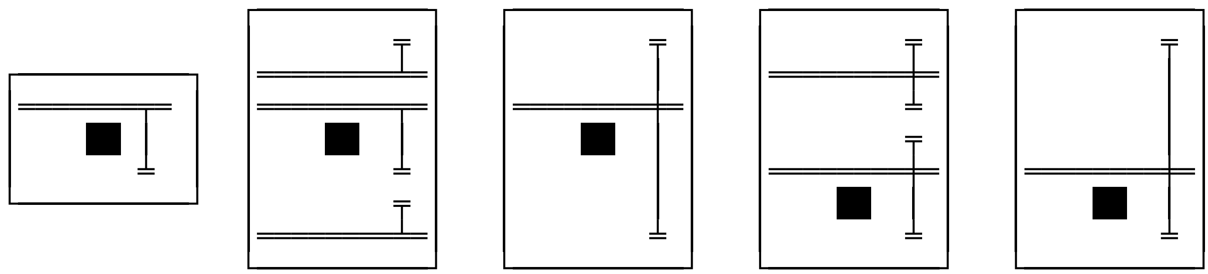


Figure 3.5.2 - SUMMARY OF DETAILS FOR R.C. BASES

The base is detailed in plan and section. Transverse bars may optionally be "bobbed" (bent up at their ends) in the section.

For square bases the bars are uniformly distributed but for rectangular bases with a central column the user has a choice of having the main steel uniformly distributed or grouped in three bands. For rectangular bases with a column eccentric along the major axis the user may have the main steel either uniformly distributed or in two bands. A third band has not been included since this is usually very short.

It should be noted that the column position always appears on the major axis and therefore if it is eccentric across the shorter dimension then the drawing must be amended.

3.5.2.3 Mass concrete bases

The details for mass concrete bases are summarised diagrammatically in Figure 3.5.3. Column position shown: ■

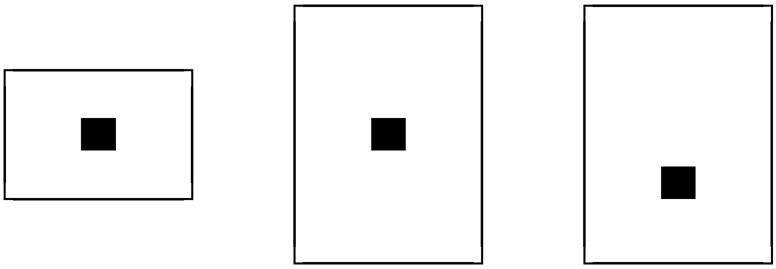


Figure 3.5.3 - SUMMARY OF DETAILS FOR MASS CONCRETE BASES

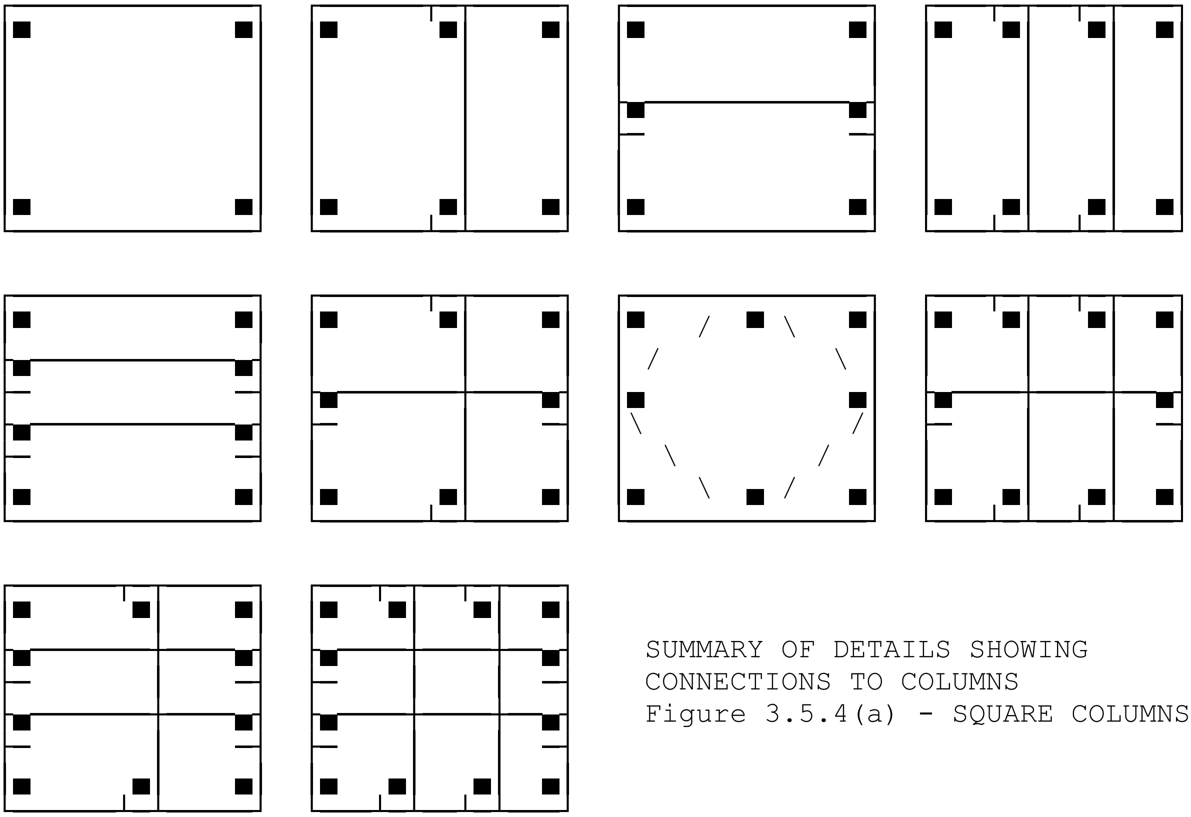
As with the previous foundation details, mass bases are shown in both plan and section. Three details give the user a choice of a square base with a central column, a rectangular base with a central column, or a rectangular base with a column eccentric along the longitudinal axis of the base.

3.5.3 Connection to column details

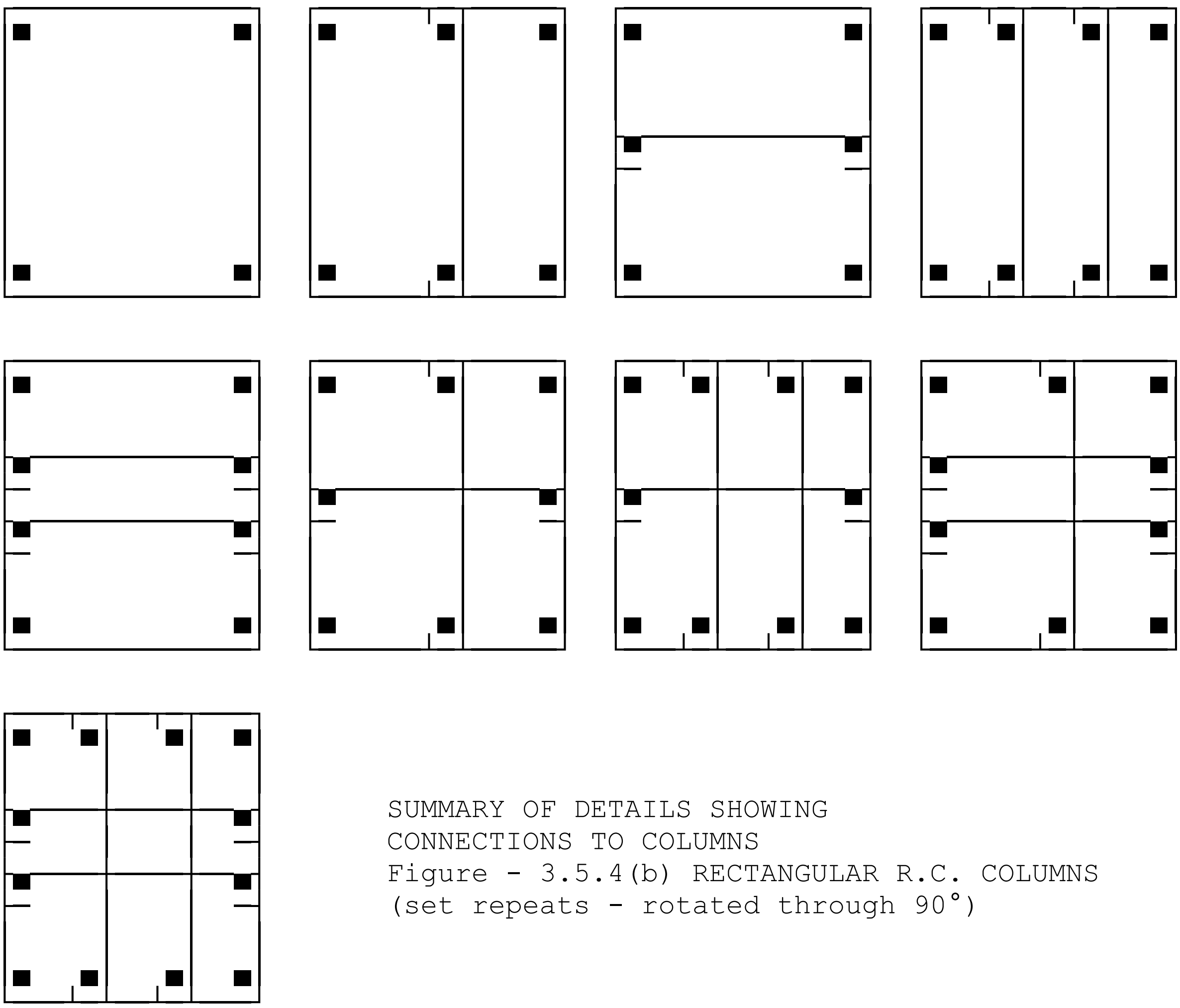
The following alternative connections to the column are provided:

- starter bars for a selection of differently reinforced square, rectangular, circular or very rectangular columns,
- holding down bolt details using 2 or 4 bolts,
- a square or rectangular pocket, and
- a choice of notes indicating that the starter bars, holding down bolts or other column connection is detailed elsewhere.

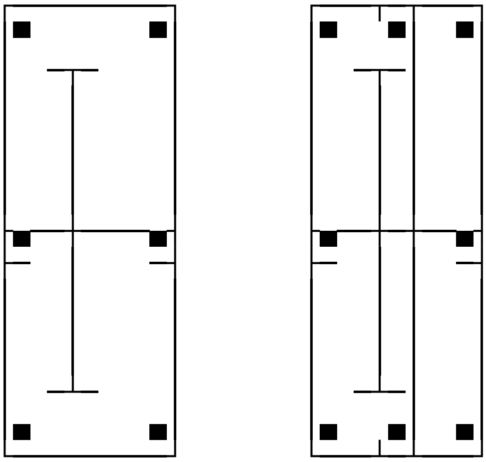
The details available are summarised diagrammatically in Figure 3.5.4 with the exceptions of: circular columns (which may have 6, 8, 10 or 12 bars) and the 'Detailed elsewhere' notes.



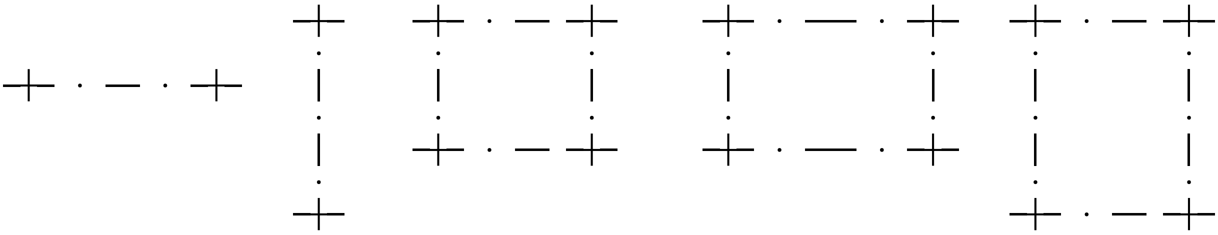
SUMMARY OF DETAILS SHOWING
CONNECTIONS TO COLUMNS
Figure 3.5.4(a) - SQUARE COLUMNS



SUMMARY OF DETAILS SHOWING
CONNECTIONS TO COLUMNS
Figure - 3.5.4(b) RECTANGULAR R.C. COLUMNS
(set repeats - rotated through 90°)



SUMMARY OF DETAILS SHOWING
CONNECTIONS TO COLUMNS
Figure 3.5.4(c) - VERY RECTANGULAR
R.C. COLUMNS
(set repeats - rotated through 90°)



SUMMARY OF DETAILS SHOWING CONNECTIONS TO COLUMNS
Figure 3.5.4(d) - HOLDING DOWN BOLTS FOR STEEL COLUMNS

SUMMARY OF DETAILS SHOWING CONNECTIONS TO COLUMNS
Figure 3.5.4(e) - POCKETS

3.5.3.1 Starter bars to columns

A detail showing starter bars gives a section of the column and its reinforcement, and it superimposes the column outline on the foundation plan and the starter bars on the foundation section. The details cover square, rectangular, circular and very rectangular columns all having various numbers or main bars.

Because the foundation detail always has the same orientation on the sheet, in the cases of asymmetrically reinforced square columns, and of all rectangular columns, two orientations of the column detail are provided so that the user can achieve the correct orientation of the starter bars relative to the foundation.

The user may wish to specify on the linework drawing the height of the kicker and the length the starters should project to ensure that an adequate lap length is obtained.

3.5.3.2 Holding down bolts

A detail showing holding down bolts superimposes the bolt holes on the foundation plan and the bolts on the foundation section; and includes relevant notes.

3.5.3.3 Pockets

A detail showing a pocket superimposes the plan view of the pocket on the foundation plan and the section of the pocket on the foundation section.

3.5.3.4 Detailed elsewhere

If users wish to detail the starter bars for columns somewhere other than on the foundation drawing, then their foundation drawing should show that the starter bars are detailed elsewhere. This will be necessary, for example, if the user wishes to use either the stub column facility or the starter-bars-only facility in the LUCID columns set of details.

3.5.4 Strip footings

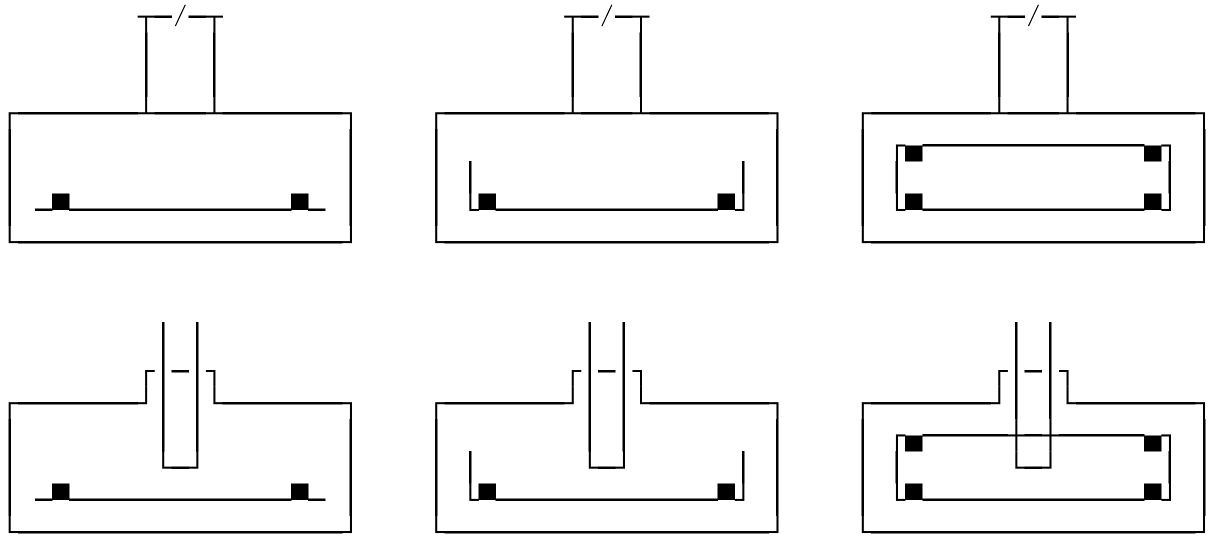
Strip footings are detailed in section only and are summarised diagrammatically in Figure 3.5.5, covering the following range:

- isolated strip footings,
- internal strip footings, and
- edge strip footings.

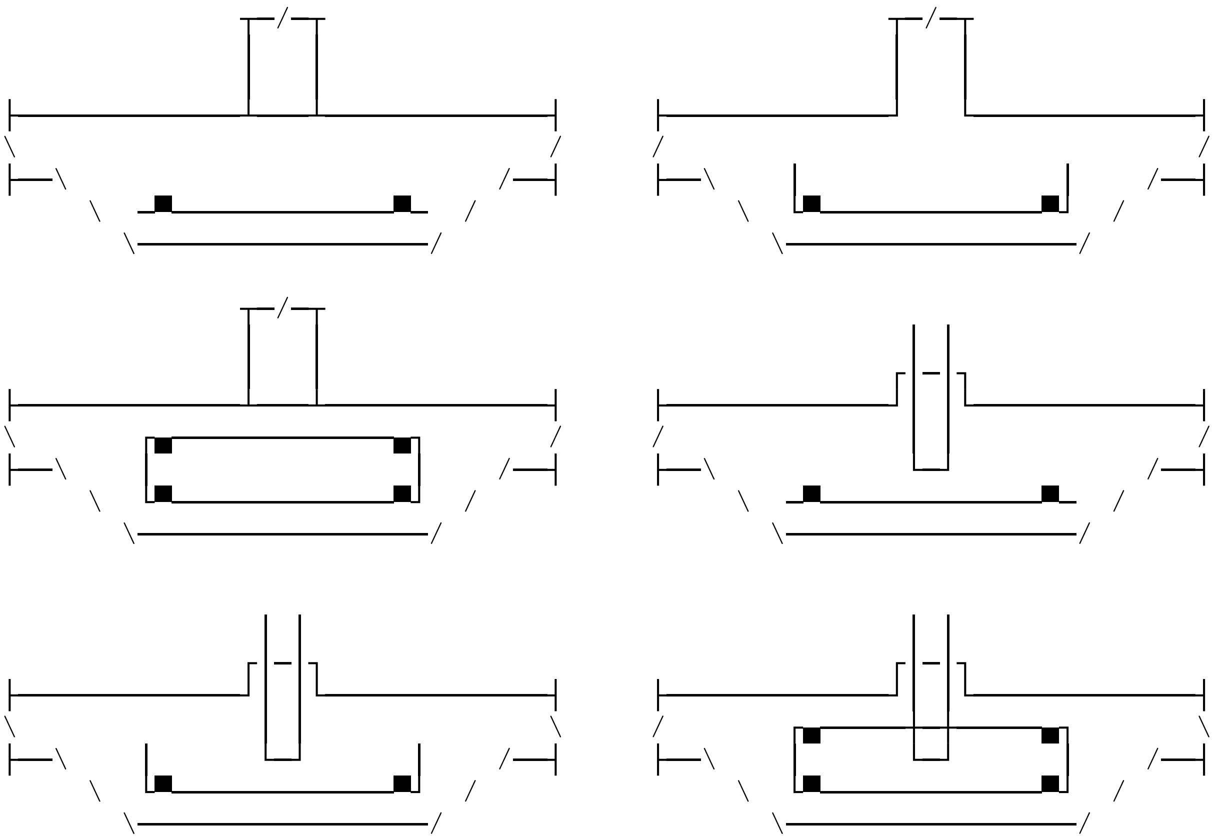
For the isolated and internal strip footings the transverse steel can be straight, bobbed or caged and the supported wall may have starter bars or not.

For edge strip footings three alternative steel or concrete outlines are offered.

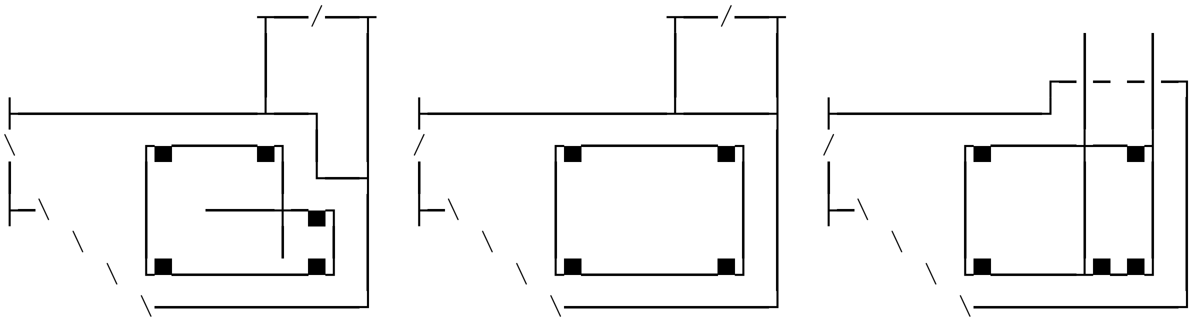
The extent of the footing to which the section applies should be indicated either by suitable addition on the LUCID drawing or on the relevant G.A. or key plan drawing. In some cases it may be necessary to indicate the form of wall above.



SUMMARY OF DETAILS FOR STRIP FOOTINGS
Figure 3.5.5(a) - ISOLATED STRIP FOOTINGS



SUMMARY OF DETAILS FOR STRIP FOOTINGS
Figure 3.5.5(b) - INTERNAL STRIP FOOTINGS



SUMMARY OF DETAILS FOR STRIP FOOTINGS
Figure 3.5.5(c) - EDGE STRIP FOOTINGS

3.6 Retaining walls (lu210, lu220)

3.6.1 General

These notes describes the use of the LUCID details available to enable the user to produce drawings both for free standing cantilever retaining walls and propped cantilever retaining walls.

3.6.1.1 Free standing cantilever retaining walls

The details in this set aid in the preparation of drawings for free-standing cantilever retaining walls whose stem height is less than about 30ft (9 metres) and their suitability, for use in higher walls, must be checked carefully.

3.6.1.2 Propped cantilever retaining walls

Such walls typically occur around the perimeter of basements, and are usually single storey height or a little more. For walls outside this range, the suitability of the details must be carefully checked.

Walls as a whole are detailed panel by panel, each panel being a straight section of wall in plan terminating in a plain end or right angle corner. Typical walls with the plan shapes shown in Figure 3.6.4 can be detailed using 1, 2 or 3 LUCID drawings, one for each straight wall panel. When propped walls form a 'T' junction in plan with a panel which does not carry horizontal loading, as shown in Figure 3.6.5, use may be made for the "unloaded" panel of a LUCID "Walls" drawing.

A drawing of an individual panel comprises an elevation together with a vertical and horizontal section. The elevation is always viewed from the side of the propping suspended slab, as shown by the arrows in Figure 3.6.4. Cross-hatching to indicate the retained material is shown to facilitate the reading of the drawing.

Details are provided which have a constant thickness up to the top slab, or panels with a step on the "earth face" (the face opposite to the propping slab). Both types have numerous variations where the panel joins the top slab, and a considerable variety of details for the left and right hand ends of the panels.

3.6.1.3 Bar 'calling up' and scheduling

Bar 'calling up' follows the traditional method thus:

```
                              No.of    Dia(mm) Bar spacing(mm)
                                |  Type | Mark |
                                |     | | |    |
Examples of bar calling up:     └── 14H20-05-200
                                     8H12-05-100 (Eurocode)
                                     8R12-05-100 (BS)
```

After printing the LUCID detail, use option 910 to produce a bar and weight schedule.

The bar schedule complies with the requirements of BS8666: 2000 using the shape code references, dimensioning and tolerancing given therein. The bar schedule is tabulated under the heading:

Member	Bar mark	Type and size	No. of mbr	No. of bar	Total no.	Lngth ofbar (mm)	Shape code	A (mm)	B (mm)	C (mm)	D (mm)	E/R (mm)	Rev ltr

Weights are given for each bar type (H, A, B, C, S or X) subdivided for bar diameters 16mm and under, and 20mm and over. (BS uses types H, R or X.)

3.6.2 Free standing cantilever retaining walls

The reinforcement is detailed panel by panel, using a full vertical section through the wall, a partial elevation of the wall and a horizontal section through the stem indicating the extent of the horizontal reinforcement at each end of the panel. The layout of the complete retaining wall and the length over which a particular reinforcement detail drawing applies must be shown elsewhere, such as on a general arrangement or key plan drawing.

The details allow for the concrete outline of the stem to be positioned towards the front, centre or back of the base, and a downstand key beneath the stem to be included if so required. Various reinforcement arrangements for the base and stem are included.

For low and medium walls the "front" of the retaining wall may be battered to reduce the thickness of the stem at the top to 80% of the thickness of the stem at the base. For high walls, either or both faces may be similarly battered. When both faces are battered the thickness of the stem at the top is 60% of the thickness at the base.

It should be noted that approximate ground levels are shown to make it clear which face of the wall is against the material being retained and that bars in similar positions in all walls have the same bar marks. The bar marks have generally been allocated in the probable order of fixing. Because of this bar marking system, some combinations of details result in gaps in the bar mark numbering sequence.

The details are divided into four subsets
a) the stem,
b) the base,
c) the panel end conditions, and
d) the base key

and these subsets are described later in detail.

3.6.2.1 The stem subset

Twenty stem details are provided, covering what will be loosely termed low, medium and high walls, as summarised diagrammatically in Figure 3.6.1. One reinforcement pattern for the vertical bars is provided for each of the low and medium walls, but two for the high walls (high wall 1 and high wall 2). Although the diagrams of the steel arrangements in Figure 3.6.1 show the steel vertical, on the details it is in fact parallel to the faces of the various concrete outlines.

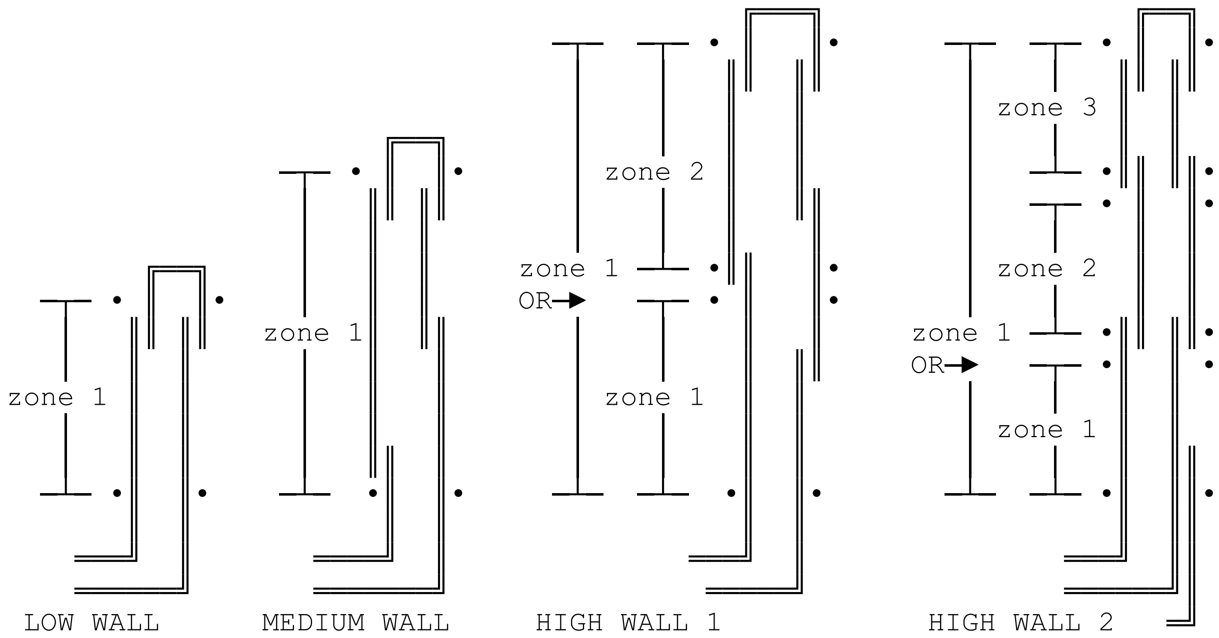


Figure 3.6.1 - SUMMARY OF STEM REINFORCEMENT PATTERNS

The horizontal steel is positioned outside the vertical steel, since research by the Cement and Concrete Association has shown that this is the better position to control cracking; this arrangement does of course reduce slightly the lever arm of the vertical steel.

The bars in the stem are called up on the elevation. The spacers are called up on the section.

The steel arrangements shown in Figure 3.6.1 are primarily intended to be used, as their description suggests, for low, medium or high walls and the ratio of stem height to base width changes for these categories so that normal proportions are maintained.

The vertical steel arrangements suggested for low walls, carry the main starter bars (mark 10) through to the top of the wall, while those for medium walls allow for one change in the vertical bars. For high walls, where considerable savings can be made by reducing the quantity of main steel up the wall, the detailer has the choice of making two or three changes in the steel quantity. In effect, therefore, either the main vertical steel can be all the same or there can be up to four different quantities.

On low and medium walls the horizontal distribution steel is assumed to be the same all the way up the wall but for high walls the detailer can have one, two or three different zones of distribution steel if he so chooses. It should be noted that the three zones offered with High wall type 2 can easily be reduced to two by using the same bars in two of the three zones.

The position of the starters for the vertical steel is fixed by the cover to the base, and vertical bars lapped onto these are positioned using lap length dimensions rather than by fixing dimensions from the base. The details have the advantage of allowing for a panel which has slightly differing stem heights between the ends of the panel, since the top lap can be a varying dimension with a minimum value.

The top two horizontal bars (mark 15) are shown inside the capping U-bar and are called up separately from the other horizontal bars. This allows the use of larger diameter bars so as to stiffen the top of the wall if required.

The L-shaped vertical starter bars mark 9, 10 and 11 are all drawn with a standard bend and two of the horizontal bars in the base (mark 6) are positioned at these bends to assist in distributing local stresses. The detailer's attention is drawn to the fact that these starter bars may require non-standard bends in some cases to satisfy the code regulations.

3.6.2.2 The base subset

The six base details cater for the stem to be central or towards the front or back of the base. They are summarised in Figure 3.6.2. Each stem position has two reinforcement arrangements: two lapping "trombone" bars (marks 3 & 4) together with longitudinal distribution steel; L-bars (mark 5) in addition to the trombone bars. Both the transverse and the longitudinal distribution bars are called up on the section. It should be noted that chairs are shown in the base and these are called up on the section.

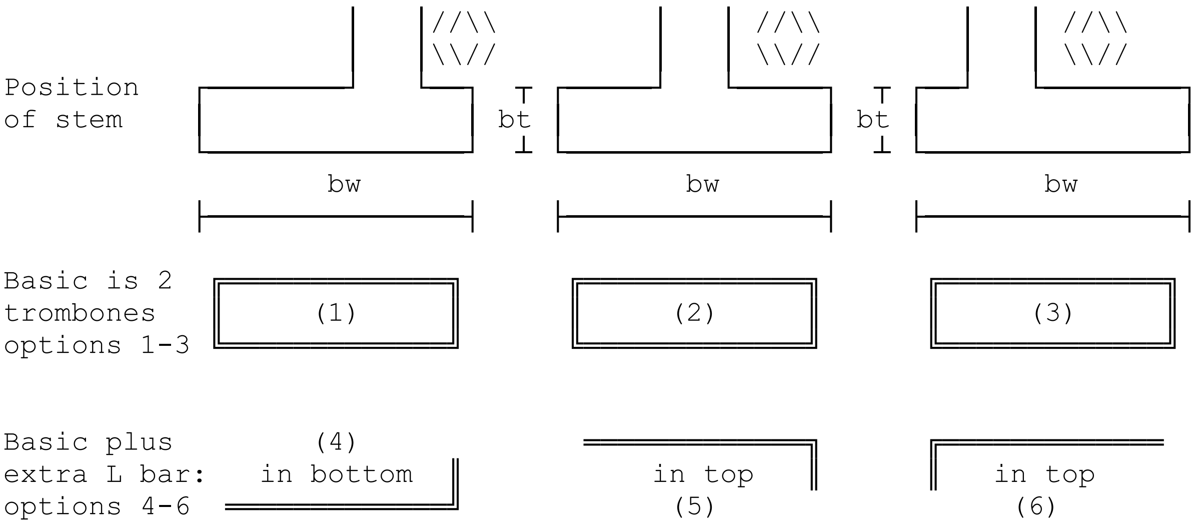


Figure 3.6.2 - SUMMARY OF BASE REINFORCEMENT PATTERNS

Any base detail may be used in conjunction with any stem detail but the detailer must ensure that the chosen base reinforcement pattern is satisfactory, particularly with regard to anchorage and local bond.

3.6.2.3 The panel end subset

The nine details show whether, at either end of the wall panel being detailed, the horizontal distribution bars
a) project into the adjacent panel,
b) project from the adjacent panel, or
c) terminate.

These details are summarised in Figures 3.6.3(a) & 3.6.3(b).

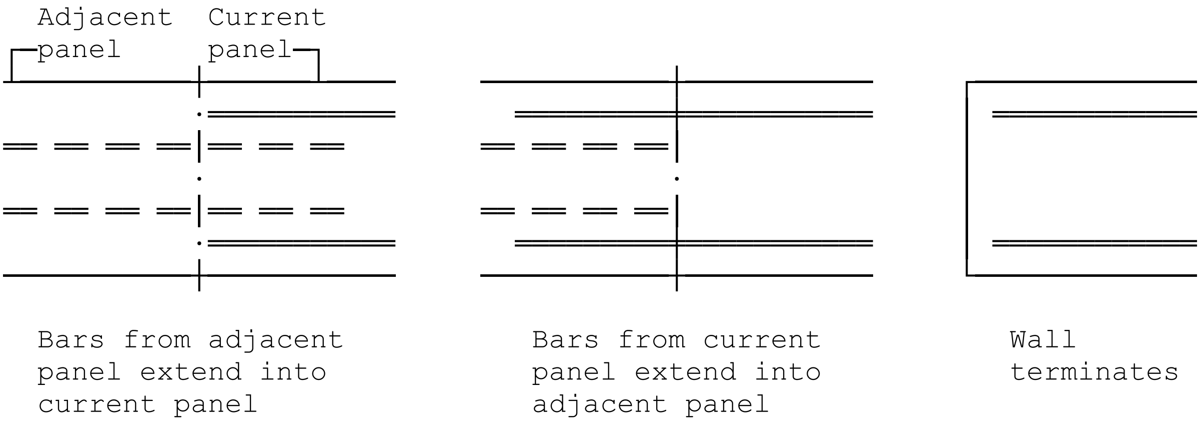


Figure 3.6.3(a) - PLAN ON LEFT HAND END OF WALL PANEL

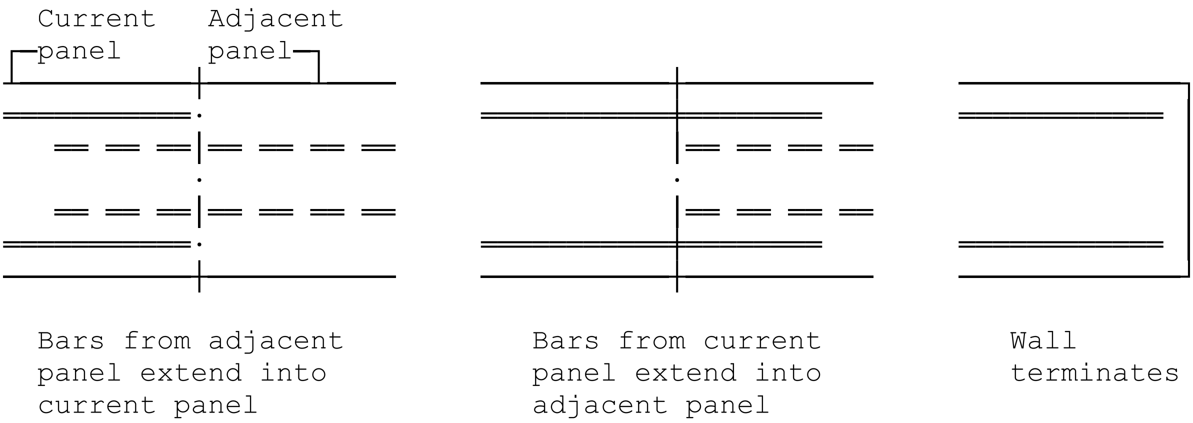


Figure 3.6.3(b) - PLAN ON RIGHT HAND END OF WALL PANEL

3.6.2.4 The key subset

An optional downstand key, directly below the wall stem, may be included.

3.6.3 Propped cantilever retaining walls

The details are divided into three subsets, each representing one of the choices to be made in selecting a suitable detail, namely:
a) the vertical section, including concrete outline, reinforcement pattern and connection to the propping slab;
b) the plan detail at the left hand end of the panel; and
c) the plan detail at the right hand end of the panel.

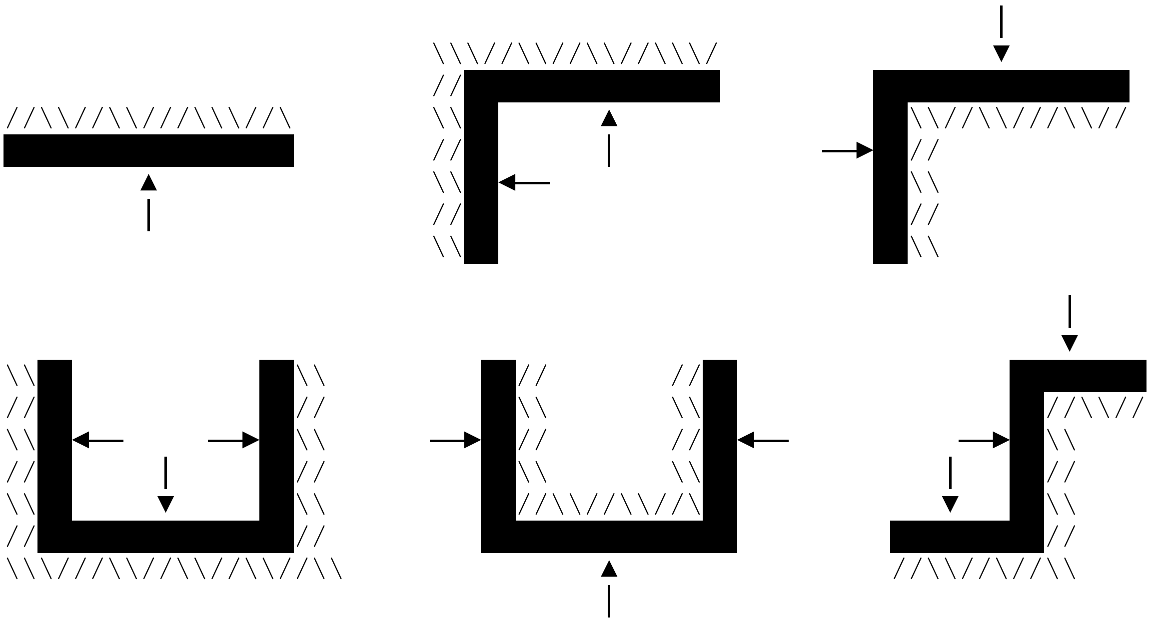


Figure 3.6.4 - TYPICAL PLANS ON PROPPED RETAINING WALLS
(arrow shows direction of elevation)

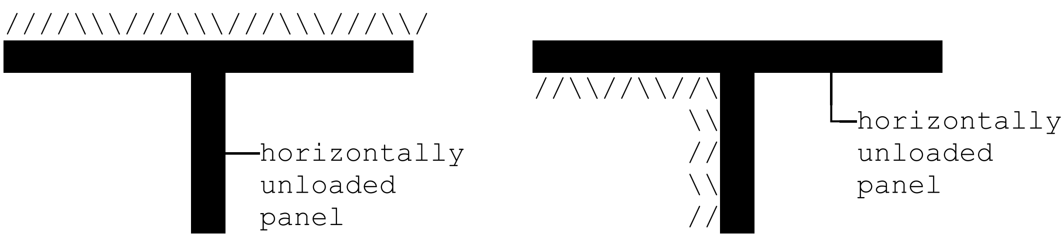


Figure 3.6.5 - TYPICAL PLANS ON HORIZONTALLY UNLOADED PANELS

3.6.3.1 The vertical section subset

The subset comprises two series of details, one for plain panels. They are summarised in Figure 3.6.6(a) for plain & 3.6.6(b) for stepped panels.

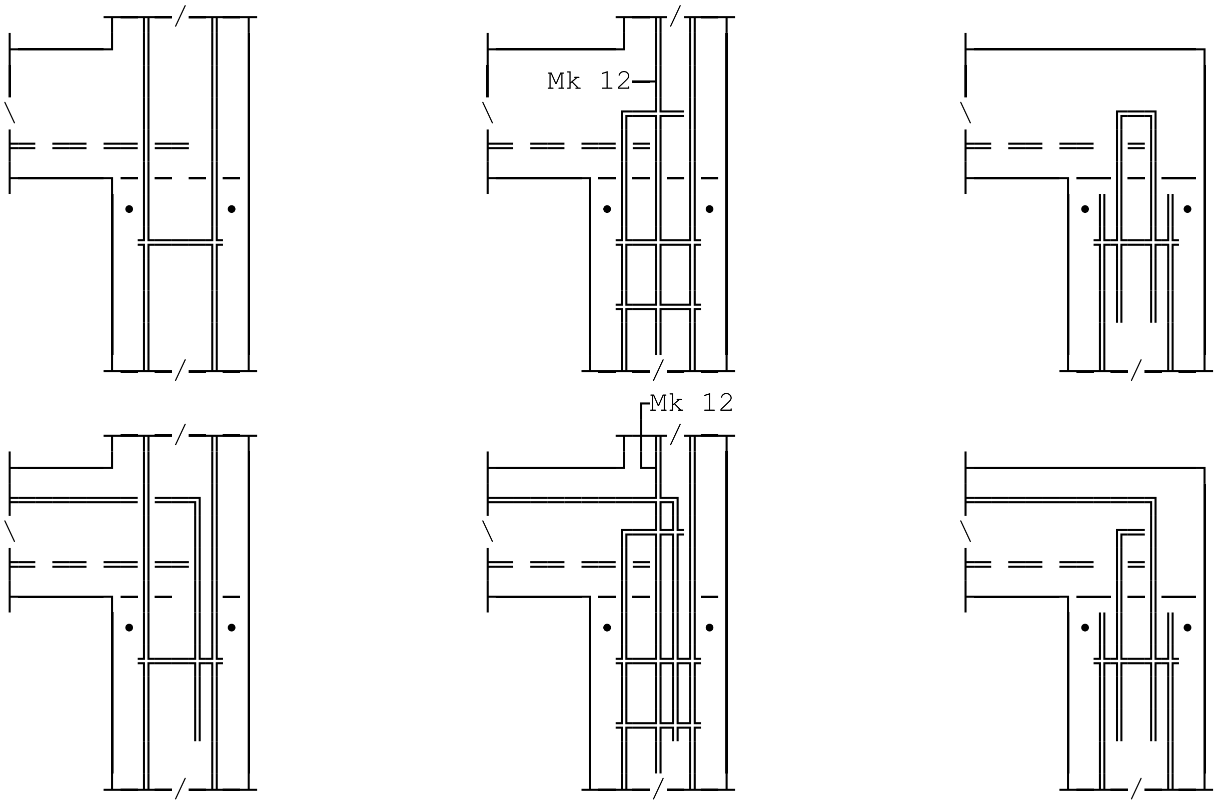


Figure 3.6.6(a) - PLAIN PANELS - SUMMARY OF VERTICAL SECTIONS
(N.B. the addition of extra horizontal bars to anchor the bottom ends of Mk 12 bars may be necessary)

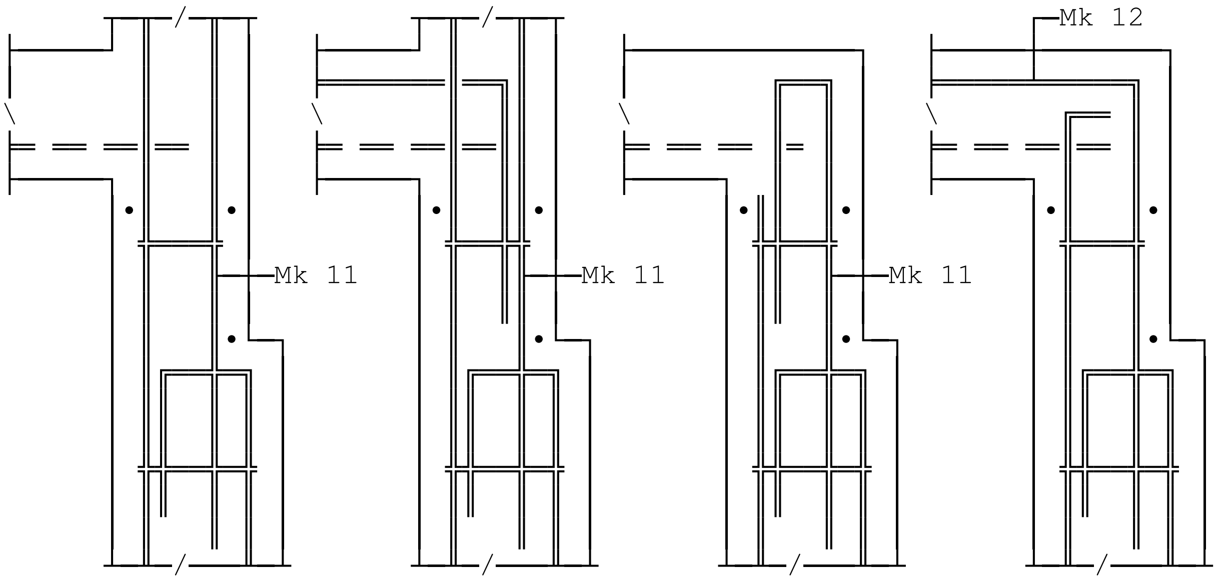


Figure 3.6.6(b) - STEPPED PANELS - SUMMARY OF VERTICAL SECTIONS
(N.B. the addition of extra horizontal bars to anchor the bottom ends of Mk 11 & Mk 12 bars may be necessary)

3.6.3.2 Vertical sections

The summarised details for plain panels details are provided for the same thickness above and below the top slab. Details also allow the wall above the slab to be thinner or to omit the wall above the slab.

The stepped panel details provide for the situation where a step is required on the earth face, below the propping slab level, as is common when an outer skin of brickwork is used.

It should be particularly noted in stepped panels that a horizontal bar is provided at the step where the two sets of vertical bars cross.

As well as giving the concrete vertical sections together with horizontal, vertical and spacer bars, these details also show some bars in the elevation.

3.6.3.3 Notes

These details permit two lines of notes, such as "wall not to be backfilled until permission to do so has been obtained", where the wall is not structurally adequate until the upper slab has been cast.

3.6.3.4 Starter bars from base slab

The starter bars are shown as detailed elsewhere (such as on a base slab drawing). If it is decided to detail them with the wall steel, suitable amendments should be made to the bars in the vertical section and the elevation.

3.6.3.5 Wall spacers

As part of the general LUCID policy, wall spacers are shown and called up on the drawings. The spacers are shown as U-bars with the legs horizontal in the same plane as the other horizontal steel.

3.6.3.6 The plan details

The two plan detail subsets basically both offer the same range of details and will therefore be dealt with in one section. They are summarised in Figure 3.6.7 for Left Hand End. The right hand end is similar but handed. The horizontal sections shown are taken through the lower part of the stepped wall or the corresponding portion of the plain wall. Consequently the plan details are virtually the same for plain and stepped walls. The elevation on each particular detail of course varies, the stepped walls showing additionally the step on the far face and the extra steel required above the step.

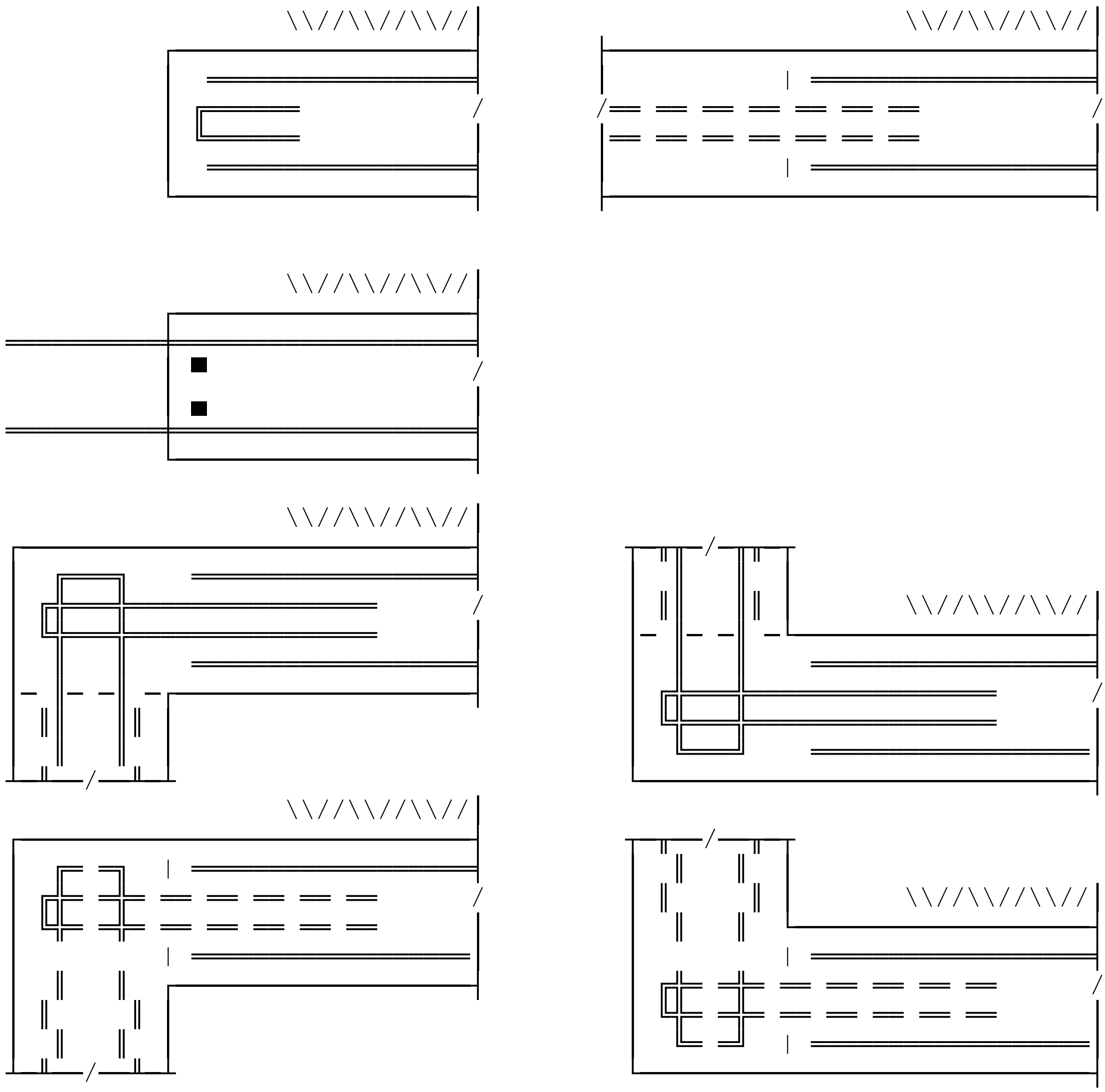


Figure 3.6.7 - SUMMARY OF PLAN DETAILS FOR LEFT HAND END
OF PLAIN & STEPPED PANELS
(summary of right hand end similar but handed)

Two basic types of end condition are provided. The horizontal steel can be stopped within the length of the panel, or it can be extended to form a lap length with the steel in the next panel. For corners, interlocking U-bars are used and the details allow both or neither of the sets of U-bars to be called up. Where the U-bars are called up they are shown solid and the vertical corner bars automatically included. If the U-bars have been called up on another drawing the corner with the U-bars dotted should be used. This omits the vertical bars, since they too will be on another drawing. The choice of detail is therefore influenced by the casting sequence of adjacent panels. This type of corner detail is adequate if the corner carries a zero or closing moment, but may not be suitable for an opening moment.

If use is made of a drawing from the "Walls" set; for "T" junctions, the junction details will be compatible. The Walls details always carry the detail of the interlocking U-bar cast within the wall panel, and reference has only to be made on the retaining wall drawings to steel detailed elsewhere. This reference must be added manually.

In common with other LUCID drawings, bar marks have been pre-assigned on the details to give maximum flexibility during detailing. Specifically, the U-bars in corners do not have duplicated bar marks, to allow them to be different if required. If a common bar mark is required, however, the drawing can be amended.

Many basement walls have buttresses or counterforts formed in them. No specific details are provided to cover these cases.

3.6.3.7 The elevation

There is no subset of details specifically covering the elevation, since these bars are on the vertical and plan section details.

The choice of vertical section dictates which vertical bars are present, and the length of the typical vertical bars on the elevation. On the other hand, the choice of the plan end details dictates how far these vertical bars extend. The details therefore show the vertical bars on the vertical section subset, extending as far as are required at the ends, these are shown on the plan subsets as separate bars, called up on their own and with their own bars mark(s). For the stepped walls, two sets of vertical bars are provided in any corner where they are required, one below and one above the step.

In elevation the details show the horizontal steel as single bars, running from end to end of the panel, and lapping with the corner U-bars if these are present. If the horizontal steel needs to be lapped in the middle portion of the panel, the drawing should be amended, together with the calling up notation.

3.6.4 Detailing procedure

If there is any variation in cross-section (except for slight increases in wall height) or reinforcement details along the length of the wall, it is suggested that it be subdivided into separate panels and these marked on a general arrangement drawing or key plan. Separate LUCID drawings should be prepared for each such panel and cross-referenced to the drawing which shows their length and layout.

3.7 Culverts and subways (lu310)

The details in this set aid in the preparation of drawings for single box culverts and subways. The reinforcement is detailed in cross-section, with one cross-section to each A4 drawing. The layout of the culvert or subway and the length over which a particular cross-section applies must be shown on a general arrangement drawing or on a simplified key plan drawing. The details permit the culvert cross-section to be square or rectangular.

Two further details show on plan the diagrammatic arrangement of reinforcement at curves and skew ends.

3.7.1 Bar 'calling up' and scheduling

Bar 'calling up' follows the traditional method thus:

```
                           No.of    Dia(mm) Bar spacing(mm)
                             |  Type  |  Mark  |
                             |    |   |   |    |
Examples of bar calling up:  └── 14H20-05-200
                                   8H12-05-100 (Eurocode)
                                   8R12-05-100 (BS)
```

After printing the LUCID detail, use option 910 to produce a bar and weight schedule.

The bar schedule complies with the requirements of BS8666: 2000 using the shape code references, dimensioning and tolerancing given therein. The bar schedule is tabulated under the heading:

Member	Bar mark	Type and size	No. of mbr	No. of bar	Total no.	Lngth ofbar (mm)	Shape code	A (mm)	B (mm)	C (mm)	D (mm)	E/R (mm)	Rev ltr

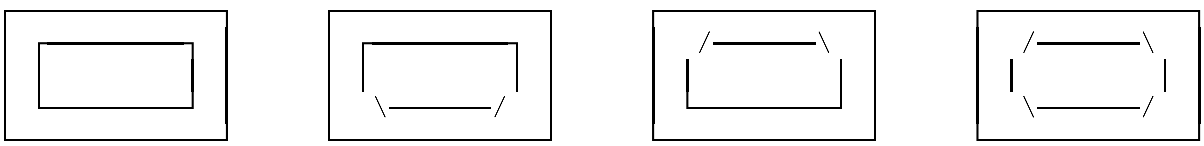
Weights are given for each bar type (H, A, B, C, S or X) subdivided for bar diameters 16mm and under, and 20mm and over. (BS uses types H, R or X.)

3.7.2 The detail subsets

The details contained in each subset are as follows:
a) the concrete outline
b) the reinforcement at the external face, and
c) the reinforcement at the internal face.

3.7.2.1 Concrete outlines

Four alternative concrete outlines are provided and these are shown in Figure 3.7.1. The variations cater for a choice of splayed or square internal corners.



Square corners — Splayed bottom corners — Splayed top corners — Splayed top & bottom corners

Figure 3.7.1 - CONCRETE OUTLINES

3.7.2.2 Reinforcement at external face

Nine alternative arrangements of the transverse reinforcement at the external face are provided and these are summarised in Figure 3.7.2. The reinforcement in both the external and internal faces is symmetrical about the centre-line.

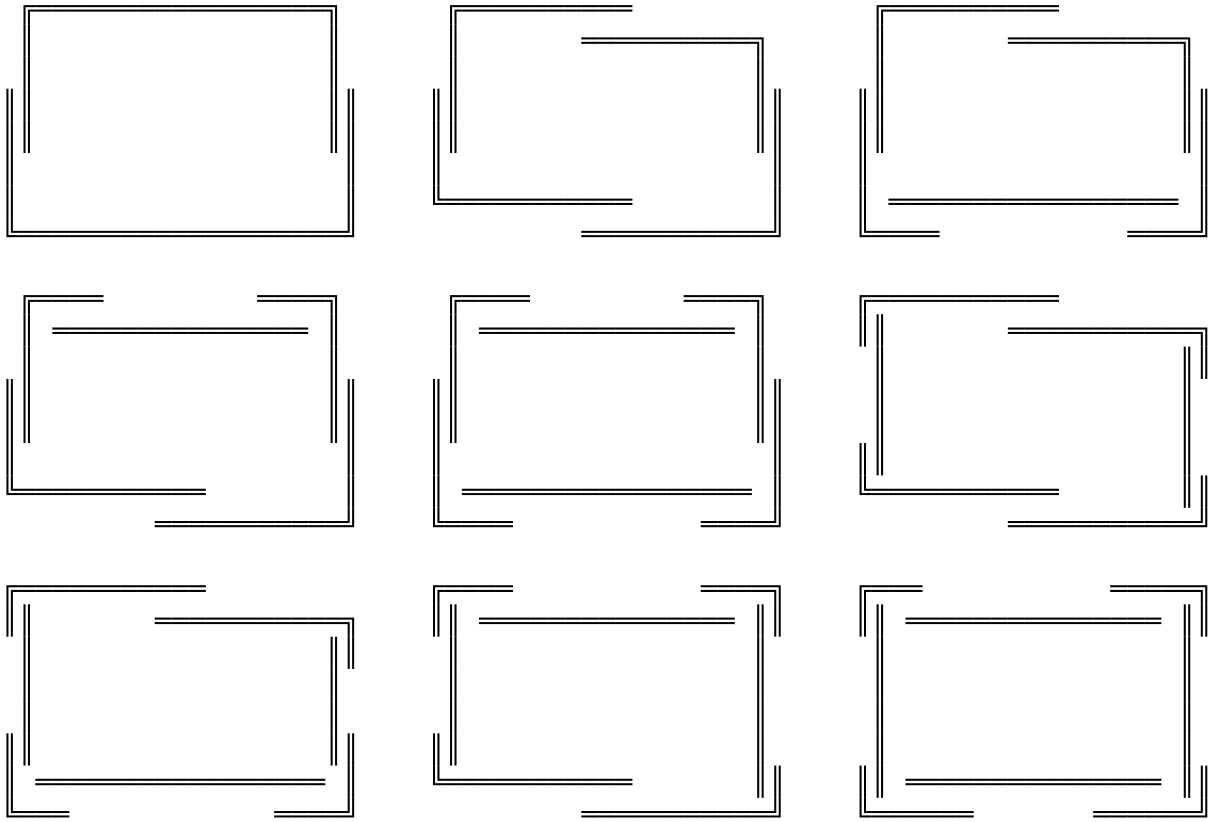


Figure 3.7.2 - EXTERNAL FACE REINFORCEMENT

3.7.2.3 Reinforcement at internal face

Six alternative arrangements of the transverse reinforcement at the internal face are provided and these are shown in Figure 3.7.3. These details have been designed for culverts and subways with tension on the outside face at the corner. If a culvert or subway can develop tension on the inside face at a corner, then users should note that the details, even if modified to give greater anchorage, are not considered to be structurally efficient.

Different bar marks are assigned for the longitudinal steel in the base, the walls, and the bottom and the top of the roof slab.

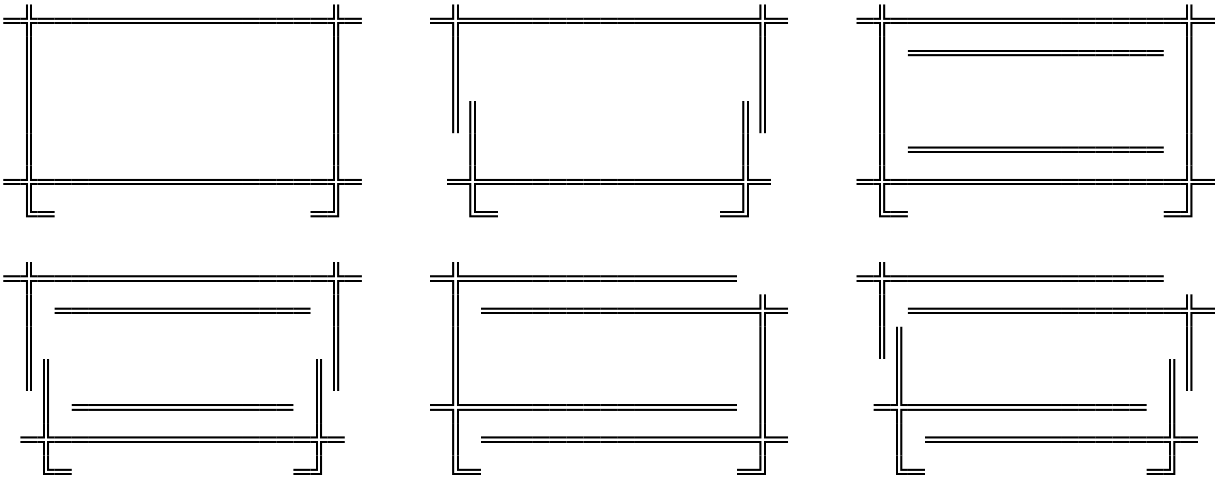


Figure 3.7.3 - INTERNAL FACE REINFORCEMENT

3.7.3 Diagrammatic plans

Two details are provided to show diagrammatically the steel arrangement on plan at curves and skew ends. These plans only show the arrangement of the bars and the full reinforcement details must be given on a separate cross-section drawing, to which a note should be added, referring to the diagrammatic plan. The overall length of the transverse bars in the base and the roof slabs varies for skew ends, and it may be convenient to use reinforcement details incorporating a lap and to vary the lap length. It is suggested that the skew end plan should be used only for small skews.

3.7.4 Detailing procedure

If there is any variation in cross-section or reinforcement details along the length of the culvert or subway, it is suggested that the culvert or subway be subdivided into separate lengths, each of constant cross-sectional detail, and these marked on a general arrangement drawing or key plan. A separate LUCID cross-section should be provided for each such length and cross-referenced to the drawing which shows their length and layout. Care should be taken to note clearly on the cross-section the lap length of the longitudinal bars and whether they stop at the end of a particular length or act as starters for the next length.

3.8 Slabs (lu410, lu420, lu430, lu440, lu450, lu460)

3.8.1 Introduction

These notes describes the use of the LUCID details available to assist in the detailing of solid reinforced concrete rectangular slab panels which span between line supports such as beams or walls; and flat slabs which are supported by columns.

There are four series of details which cover:

- simply supported single panels
- one-way spanning continuous panels
- two-way spanning continuous panels
- flat slabs.

The simply supported single panel series are for use on single, discrete, simply supported panels. The spanning series are for floors consisting of a number of connected panels such as that indicated in Figure 3.8.1. This Figure shows diagrammatically the plan of a typical floor.

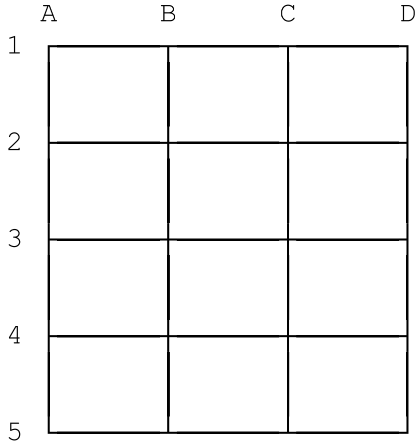


Figure 3.8.1

If there are only column supports at the intersection of grid lines then the flat slab series covers for this case including any cantilever edges.

If, however, the floor is supported around its whole perimeter and also along grid lines B and C and there are no supports, or only relatively trivial ones, along grid lines 2, 3 and 4, then the floor as a whole would normally be designed to span one-way between the line supports on grid lines A, B, C, and D. The one-way spanning series of details caters for this general type of connected slab. If, however, the line supports on grid lines 2, 3 and 4 are substantial then each panel may be designed as two-way spanning and the two-way spanning series caters for this type of slab.

3.8.2 General

A LUCID slab drawing gives the reinforcement for one panel only, and to detail a whole floor it must be subdivided into panels and one drawing produced for each panel. Each edge of a panel may be continuous or non-continuous and the range provided caters for all possible combinations of these edge conditions.

The use of the details is not restricted to regular rectangular floor slabs. For example, a floor with the plan view shown in Figure 3.8.2, could be detailed with the aid of the details available.

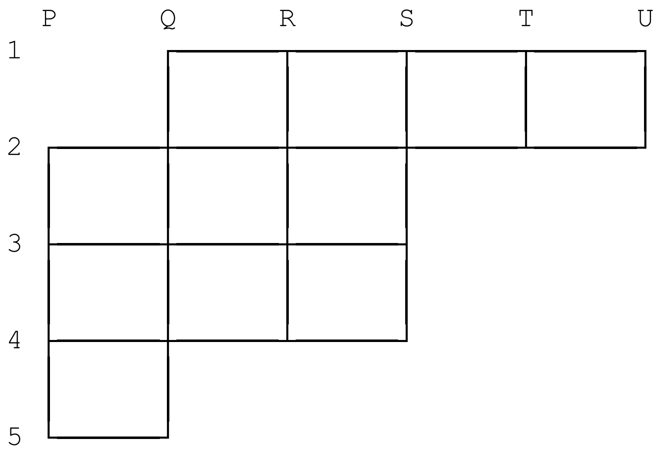


Figure 3.8.2

Since a LUCID drawing details only one panel, the way the panels fit together to form a whole floor should be shown on a separate General Arrangement or Key Plan drawing. The orientation of each panel should be carefully specified on its own drawing by inserting the appropriate grid line labels in the "balloons" provided.

3.8.2.1 Bar 'calling up' and scheduling

All the bars on the slab details have pre-assigned bar marks. From experience with LUCID on site it was found that steel fixers preferred to have such a system.

Bar 'calling up' follows the traditional method thus:

```
                              No.of    Dia(mm) Bar spacing(mm)
                                |  Type  |  Mark  |
                                |     |  |  |     |
Examples of bar calling up:     └── 14H20-05-200
                                     8H12-05-100 (Eurocode)
                                     8R12-05-100 (BS)
```

After printing the LUCID detail, use option 910 to produce a bar and weight schedule.

The bar schedule complies with the requirements of BS8666: 2000 using the shape code references, dimensioning and tolerancing given therein. The bar schedule is tabulated under the heading:

Member	Bar mark	Type and size	No. of mbr	No. of bar	Total no.	Lngth ofbar (mm)	Shape code	A (mm)	B (mm)	C (mm)	D (mm)	E/R (mm)	Rev ltr

Weights are given for each bar type (H, A, B, C, S or X) subdivided for bar diameters 16mm and under, and 20mm and over. (BS uses types H, R or X.)

Some bars, for example bars 12 and 18 on the one and two way spanning slabs are always in two bands and the call-up strings can be completed, for example, thus: 2+5H10-12-250 T2 where 2 is the number of bars in the first band and 5 is the number in the second band. To avoid any ambiguity, the two bands are always drawn as either the same, or markedly different, in width. Torsion bars sometimes occur in two bands, and then require calling-up as above.

3.8.2.2 Fixing dimensions

Detailing reinforcement is more the art of pattern selection rather than precision in bar location. As the bars are drawn to scale then the steelfixer may locate the bars by scaling the details. Where positions or lengths of reinforcing bars are critical the engineer should add the critical dimensions to the detail manually.

3.8.2.3 Covers

The general reinforcement covers for top, bottom and end should be given. Where it is required that a particular bar should have a special cover, the engineer should add it to the detail manually.

3.8.2.4 Holes and openings

Most of the LUCID slab details are too compact to allow the addition of complicated information concerning hole positions, sizes and trimmings. Such information is best presented on separate drawings, to which reference can be made in the space provided on each panel drawing.

The Concrete Society publication "Standard Reinforced Concrete Details" presents three methods of trimming holes in slabs. Detail A in that report covers the case where reinforcement which interferes with the hole may be moved to one side, and this case is usually best covered by a note on an appropriate drawing. Details B and C, however, cover more complex cases and details are provided to assist with the requisite trimmer bars for these two details.

Since the trimmer bars are shown in plan only, they are merely represented by single lines and these can represent any of several shape-codes specified by the engineer in his/her bar schedule.

These details each have a blank area in which the engineer may draw, on his/her linework drawing, a sketch of the positions and sizes of the holes, if this information is not already on an appropriate G.A. drawing. The space could also be used to provide a Table to identify which trimmer bars are used with which holes. Alternatively, one drawing can be produced for each hole.

Some engineers will prefer to schedule the appropriate trimmer bars on each panel's bar schedule and in this case it may be appropriate to add a note to the hole detail drawing to the effect that "trimmer bars are scheduled with the reinforcement for the relevant panel". Other engineers for example may wish to have a separate schedule for trimmer bars for the entire slab.

3.8.2.5 Chairs for top reinforcement

Most slabs require chairs to support the top reinforcement. Most engineers will want to schedule such chairs with the reinforcement for the panel in question, and details are available for two alternative ways of fixing a chair.

In the blank area provided on these chair supports drawings the engineer can, if required, draw a sketch of where chairs are required and how they are to be spaced; or this can be specified in words.

Each panel drawing has a space where the engineer can quote the drawing number for such a "Chair Supports Drg".

Some engineers may, of course, wish to schedule the chairs for an entire slab on one schedule.

3.8.2.6 Centre lines and grid lines

The drawings show a chain-dotted line as the centre line of each support. Grid line labels are used in the balloons to orientate the panels.

Where the centre line and grid line are not coincident, and the engineer wishes to avoid possible confusion of interpretation, sufficient clarifying amendments should be made manually.

3.8.2.7 Bars shared by two panels

Some bars, principally top steel across supports are common to two panels and hence should appear on two drawings. The convention adopted on the details is that bars will be detailed with that panel from which they protrude upwards or to the right of the panel as drawn on the page.

Conversely, where there are bars coming into a panel from the bottom or from the left their presence is indicated on the detail by means of a dotted bar line.

3.8.2.8 Bar layers

On all the details the steel in the bottom layer is drawn across the screen and each default bar 'calling up' terminates with the layer to which the bar belongs where: B denotes Bottom, T denotes Top, 1 denotes outside layer, 2 denotes inside layer. The layers used for the various types of slab panel are summarised in Figure 3.8.3.

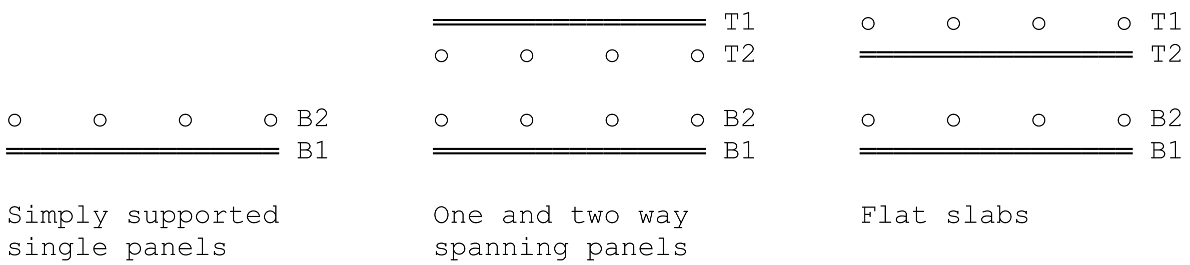


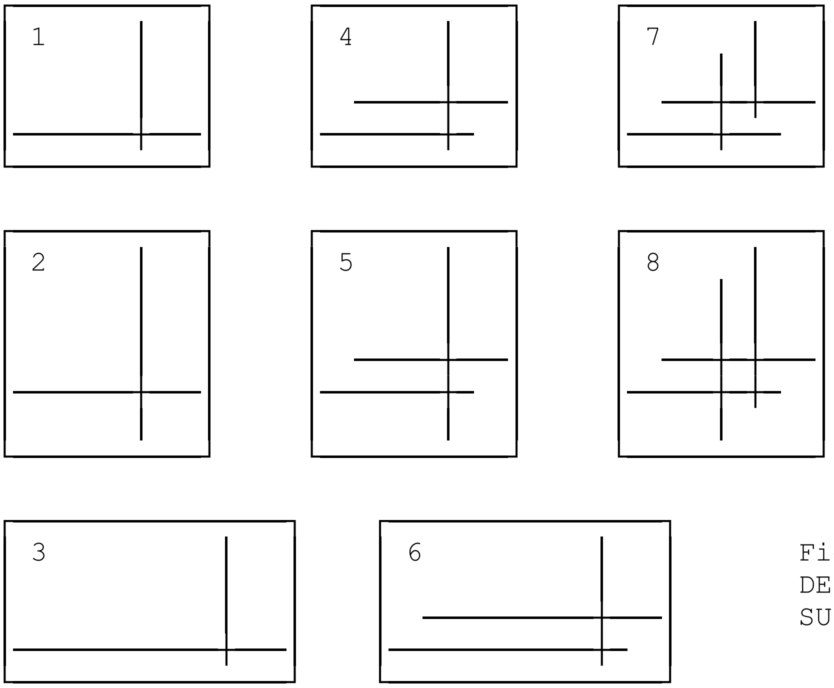
Figure 3.8.3 - BAR LAYERS

3.8.2.9 Scales

Two scales are used in the slab details; a general scale for slab length and width, and a special scale for slab depth. Where the slab aspect ratio (length/width) goes outside normal ratios, then the detail provided is not to scale.

3.8.3 Simply supported single panels

Each detail shows the concrete outline and the steel in both plan and section. These details are summarised in Figure 3.8.4, and give a choice of slabs which have either a square or rectangular shaped plan. The reinforcement is in the bottom face only and there is the choice of using staggered bars or not.



Notes:
No top steel provided.

Bars drawn across page are in (B1) layer.

Figure 3.8.4 - SUMMARY OF DETAILS FOR SIMPLY SUPPORTED SINGLE PANELS

Each detail shows a broken line, representing the inside of a support, on all four sides of the panel. However, the details may also be used for slabs supported on opposite sides, or on three sides, since if no support exists beneath an edge the relevant dotted line can be erased manually. Conversely, if required the form of support may be added e.g. a brick wall.

No top steel at all is shown in any of these details and they are therefore probably suitable only for slab panels which are not monolithic with their supports. If the engineer requires a drawing of a single panel with top steel around the edges, then the engineer should consider the use of a one or two-way spanning single panel.

In all eight details the reinforcement drawn horizontally is in the bottom layer, and the engineer may choose from the range whether or not to have the bars staggered. All non-staggered bars are drawn without "bobs" since they are likely to be lightly stressed at their ends, while all staggered bars are shown in section to be "bobbed". The engineer may easily amend these if required.

The slabs are provided in one orientation only. For example in Panel 2 the short bars are in the bottom (B1) layer while in Panel 3 it is the long bars that are in the B1 layer. Thus Panel 3 is not just Panel 2 turned through 90 degrees. Having chosen a particular detail, the grid lines must be labelled to give the proper orientation.

3.8.4 One-way spanning slabs

The sixteen details for one-way spanning slab panels are summarised diagrammatically in Figure 3.8.5. Each detail consists of a plan and two sections taken across the slab. If the panel has an external edge across the page in plan then a section, through that edge, is included.

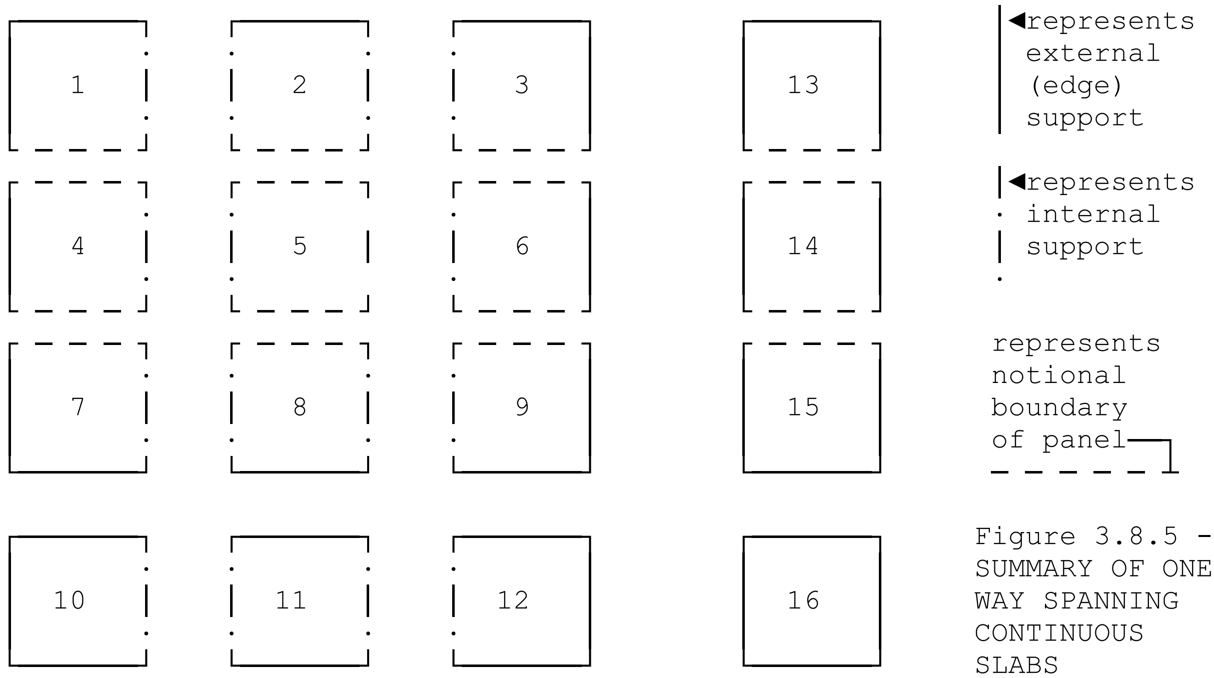


Figure 3.8.5 - SUMMARY OF ONE WAY SPANNING CONTINUOUS SLABS

The following general pattern of reinforcement is provided in the panels:

- staggered reinforcement in the bottom across the page, i.e. in the direction in which they are designed to span
- staggered top steel across the main internal supports
- top steel at any external edges
- appropriate distribution steel projecting into adjacent panels where necessary.

3.8.4.1 Uniformly reinforced floors

Details 10 to 12 are particularly suitable for floor areas in which:

- neither the top nor the bottom main steel changes within a panel
- there are no secondary line supports across the panel
- there is no lap in the distribution steel.

Where the distribution steel needs to be lapped with either random length or chosen length bars the panel may have a "break line" added to the plan and these distribution bars marked up.

3.8.4.2 Floors subdivided into smaller panels

Details are provided for use where the engineer wishes to subdivide the floor area spanning between main supports into a number of sub-areas. Such a subdivision would, for example, be appropriate in the case of a slab which although designed to span one way nonetheless has transverse beams. The engineer could then readily add these beams, and any top steel over them, to his/her drawings. The drawing of the adjacent panel would also have to be modified to indicate the presence of the beam and the top steel over it, and also to detail the requisite distribution top steel.

As an alternative to adding a break line to a drawing, it may sometimes be more convenient to detail a lap in the distribution bars by providing two drawings with their common boundary at the centre of the lap. This practice is particularly useful in slabs where a feature occurs which needs to be detailed as a special case manually, since it enables the engineer to keep to a minimum the area requiring special detailing, and thus probably permitting the feature to be catered for on an A4 size drawing, compatible with the main bulk of the drawings.

3.8.4.3 Isolated floor panel

Panel option 16 in the one way spanning slabs program is for an isolated one-way spanning panel, but it differs from the series of single panels in that top steel is provided at the edges. It also differs from the two way spanning single panel case which has staggered main bars in both directions, and also has "torsion" bars at its corners.

3.8.4.4 Bottom splice bars details

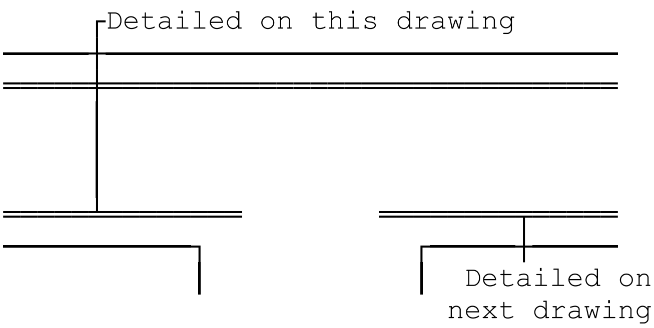


Figure 3.8.6(a)

The main bottom steel in LUCID provides the detail shown in Figure 3.8.6(a) at each internal support. If the engineer wishes the steel from the two panels to overlap, then the bars can be extended manually.

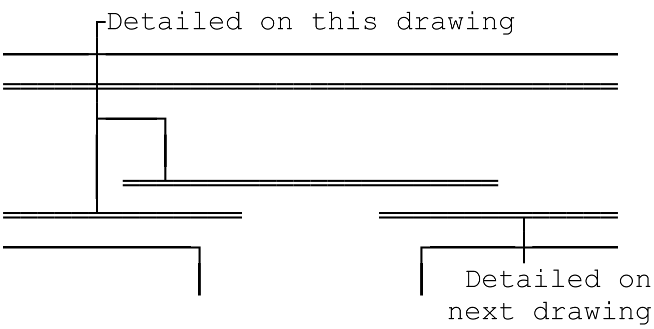


Figure 3.8.6(b)

Alternatively the engineer may wish to provide a 'splice bar' as shown in Figure 3.8.6(b), or indicate its presence if not detailed with the panel.

3.8.4.5 "Horizontal" external edges

All external edges are shown as being supported and in plan a dashed line represents the inside of the support. In cases where an external edge drawn horizontally on the page does not have a support the engineer should remember to erase this dashed line and to amend section B-B appropriately.

3.8.5 Two-way spanning slabs

The sixteen details for two-way spanning slabs are summarised in Figure 3.8.7.

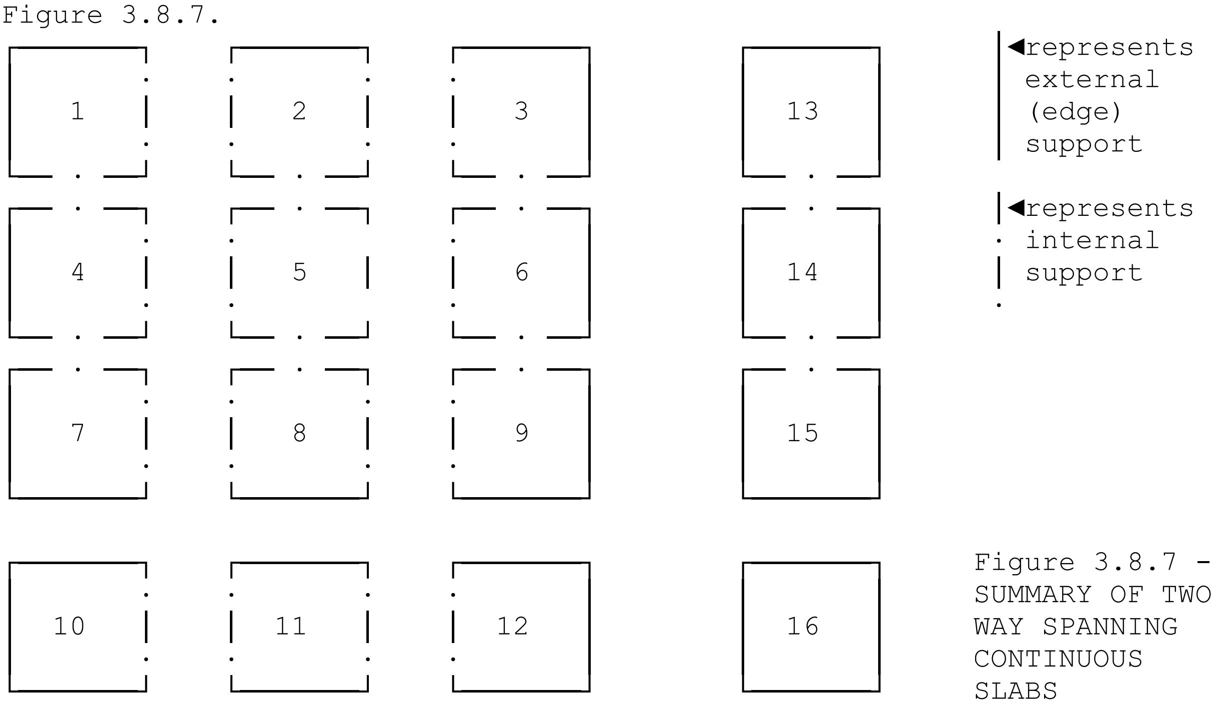


Figure 3.8.7 - SUMMARY OF TWO WAY SPANNING CONTINUOUS SLABS

Each two-way spanning slab detail consists of a plan and two sections taken across the slab, and those panels with an external "horizontal" edge have a third section through that edge.

The following general pattern of reinforcement is provided in the panels:

- staggered reinforcement in the bottom face in both directions
- staggered top steel over any internal supports
- top steel at any external edges
- extra top and bottom steel in two directions at all external corners to resist torsion
- appropriate distribution steel.

All edges are assumed to have line supports and choosing the appropriate detail is merely a matter of identifying the one with the correct boundary conditions.

3.8.5.1 Bottom splice bars details

The main bottom steel in LUCID provides the detail shown in Figure 3.8.6(a) at each internal support. If the engineer wishes the steel from the two panels to overlap, then the bars can be extended manually. Alternatively the engineer may wish to provide a 'splice bar' as shown in Figure 3.8.6(b), or indicate its presence if not detailed with the panel.

3.8.5.2 Torsion steel

Extra steel is provided at all external corners to resist torsion, but no additional bars have been included at other corners since the engineer can usually ensure that the other steel present is adequate to meet torsion requirements.

3.8.5.3 Pre-assigned bar marks

Within the one way and two way spanning slab details all bars in similar positions have the same bar mark. Thus, for example, the main bars across a right hand support in the outermost top (T1) layer are always numbered 17 if and when they occur. All the bar marks assigned to the various bars are identified in the composite Figure 3.8.8 so that, for example, the engineer can refer in his/her calculations to a specific bar mark, thus easing the process of communication. However this numbering system does mean that the bar marks in a drawing will not be sequential. The bar marks are arranged into an anticipated order of fixing.

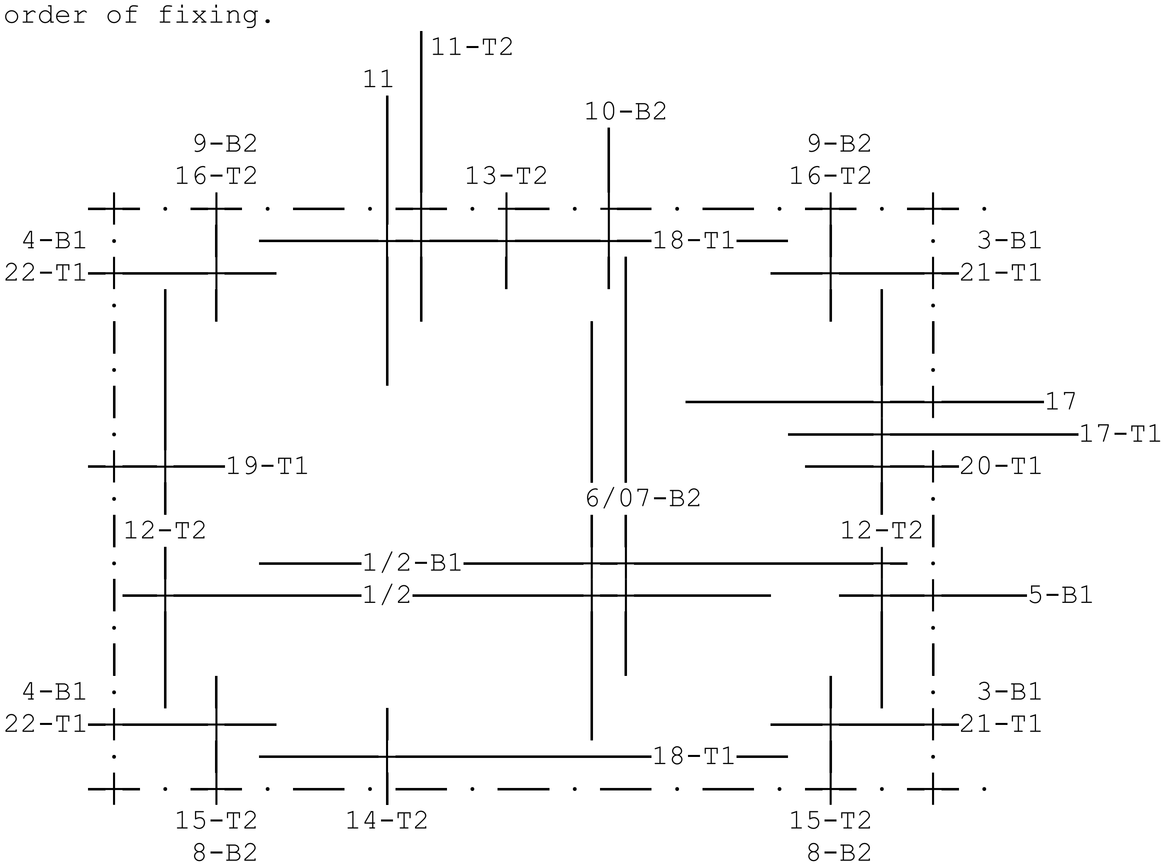


Figure 3.8.8 - ONE & TWO WAY SPANNING SLABS - COMPOSITE DRAWING SHOWING ALL POSSIBLE BAR MARKS (22 total)

3.8.5.4 Details at external edges

The detail at the edge of a slab is always shown on the details as in Figure 3.8.9(a). This can, however, be readily amended on the drawing to show a variety of other conditions, in respect of both concrete outline and reinforcement, and the engineer should make such changes as are required. For example, it may be required that the reinforcement take the form of a "trombone" bar, as shown in Figure 3.8.9(b).

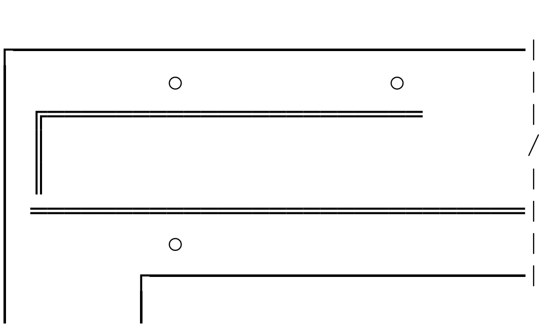


Figure 3.8.9(a)

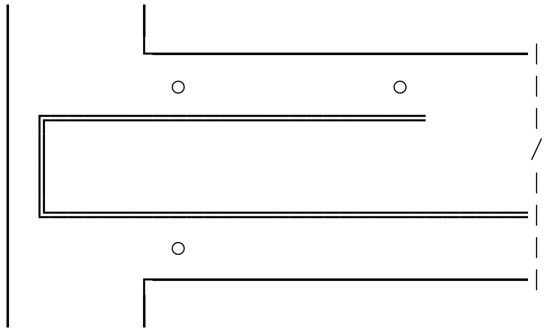


Figure 3.8.9(b)

3.8.6 Flat slabs

3.8.6.1 Panel types

On the assumption that flat slabs are at least two bays wide in each direction, the twenty-five panels summarised in Figure 3.8.10 cater for all possibilities including cantilever edges. The details comprise a plan and either two or three sections. Two sections are drawn across the page, and a third section is drawn through an outside edge.

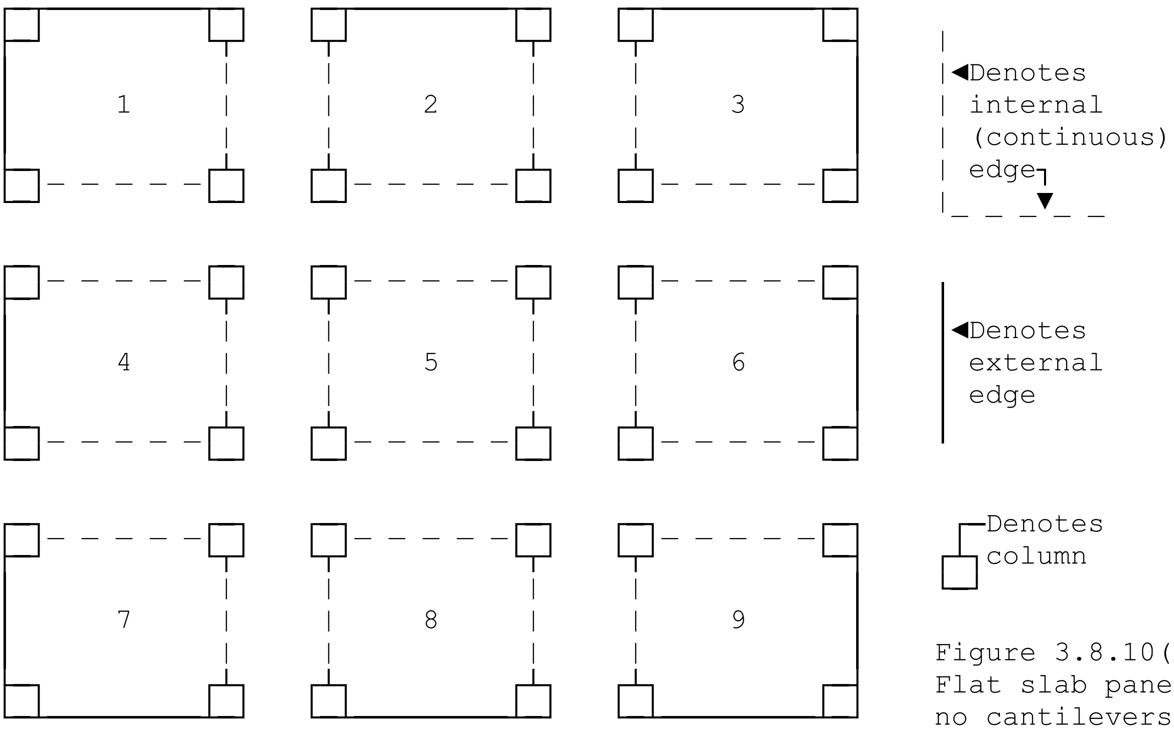


Figure 3.8.10(a) - Flat slab panels no cantilevers

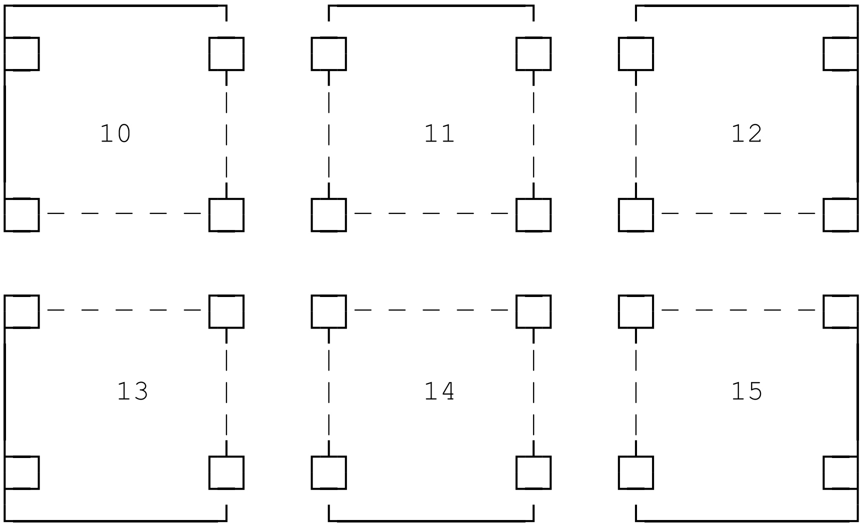


Figure 3.8.10(b) -
Flat slab panels
with one
cantilever edge

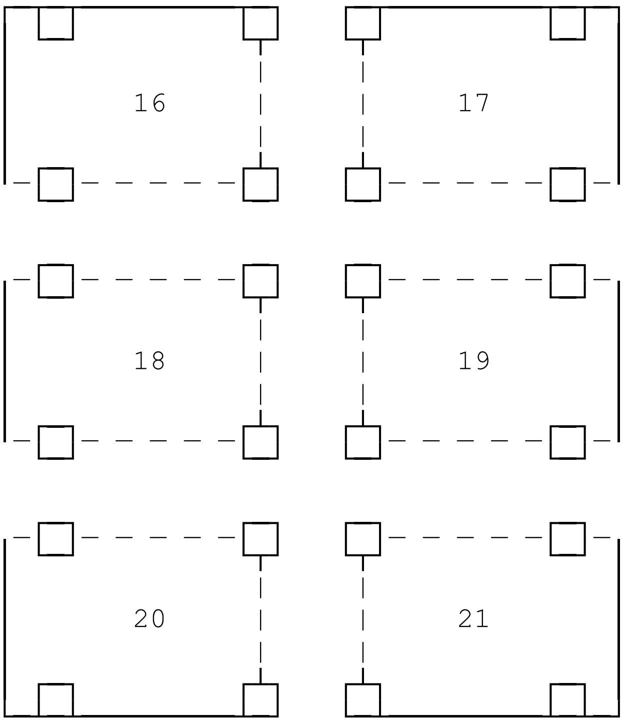


Figure 3.8.10(b) -
Flat slab panels
with one cantilever
edge (continued)

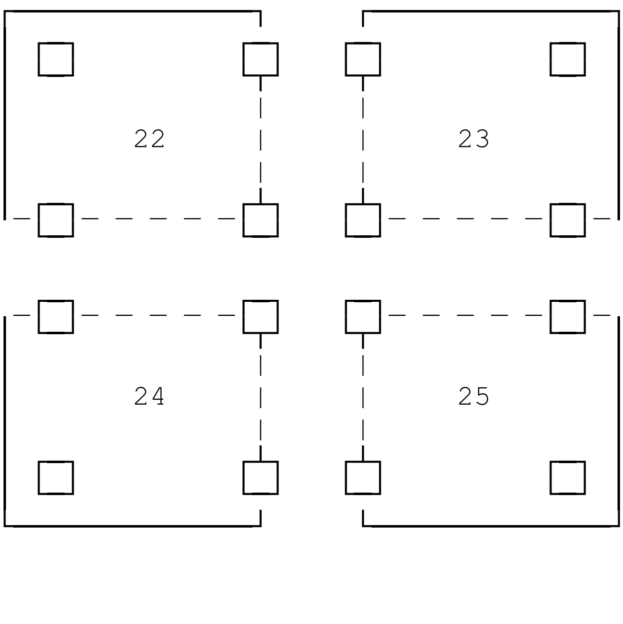


Figure 3.8.10(c) -
Flat slab panels
with two
cantilever edges

3.8.6.2 Orientation

Orientation of panels is by grid line reference, which may be dimensioned as offset from the column centre lines if required. In no case do the drawings need to be turned through more than 90 degrees in order to orientate them correctly with the General Arrangement or layout drawings.

The details show a symmetrical arrangement of columns; when columns are slightly off centre then it may be possible to use the details with minor amendments. For significant column eccentricities the details will not be applicable, and further detailing manually will be required.

3.8.6.3 Drops and column heads

The drop is not now generally used, a flat soffit to the slab being preferred. For this reason no drop details are provided.

Column head outlines should be detailed on the General Arrangement drawings, and any reinforcement required, detailed with the columns. The details are therefore independent of the column head.

3.8.6.4 Reinforcement details

Certain aspects of the details are common with those for edge supported slabs, in particular the detailing conventions and notation for the reinforcement layers, and the details for holes and support chairs. They may be summarised as follows:

- bars are detailed with panels from which they protrude upwards or to the right as drawn on the page
- the layers are identified B1, B2, T1 and T2
- separate details are issued for the reinforcement for holes and for chairs. The exact dimensions and position of a hole are preferably shown on the General Arrangement drawing with the slab reinforcement drawing giving only an indication of the hole.

3.8.6.5 Direction of reinforcement

For square panels it is preferable to have T1 and B1 bars at right angles so that if one direction benefits from increased lever arm at the support, the other direction benefits from increased lever arm in the span. As most flat slab panels are squarish, the LUCID details show the B1 and T1 bars at right angles to each other. This arrangement is also preferable for the column head shear reinforcement detail.

3.8.6.6 Reinforcement patterns

The reinforcement in the panels is in the following consistent pattern:

- bottom bars are provided over the full panel, including any cantilevers
- facilities are provided to have three different steel quantities in the central and two column strips in each direction
- staggered bars are used where no cantilever exists and alternate bars where there is a cantilever
- continuity into adjacent panels is provided by top bars across the column strip, and again facilities are provided for three different zones of staggered bars
- top bars at free edges consist of trombone bars which extend beyond the column strip
- top distribution bars are provided in the column strips where no other top steel exists.

3.8.6.7 Calling up strings and typical bars

Flat slab details are inevitably congested, due to the large number of different bar zones used in close proximity to each other. To reduce this congestion, bars in the same layer and the same strip have all been called up on the same string. Separate messages are used for each zone in the strip, but only one typical bar is shown.

In most cases, all bars called up by the one string will be the same length. If this is not the case, additional typical bars and fixing dimensions must be added manually. The bars are shown pictorially correct to the limits laid down in BS8110 for the empirical method of design.

To help identify which message applies to which zone, each message has been labelled a), b), or c), and a corresponding label included in a break in the calling up string within the applicable zone. The labels have been allocated such that zone a) is nearest to the relevant messages, and zone c) furthest away.

3.8.6.8 Allocation of bar marks

Bar marks have been allocated sequentially for each drawing without any gaps in the sequence, in the probable order of fixing. This is in the vertical order of layers (B1, B2, T2, T1), from the top and left hand side towards the bottom and right hand side of the drawing.

Each zone has a unique bar mark to allow for any variations of bar diameter (or length, if this is accompanied by alterations to the typical bar).

3.8.6.9 Column band at support

Some codes of practice require that two-thirds of the support reinforcement at columns be placed in a width equal to half the column strip. This would require up to 3 separate calling up messages in each direction over each column head. For simplicity it is recommended that this reinforcement be increased by 1/3 and provided at uniform spacing over the full width of the strip.

3.8.6.10 Bottom reinforcement

Although codes allow the bottom bars to be unlapped, and in fact stopped short of those of adjacent panels, current practice gives a nominal lap to these bars to distribute thermal cracks and the LUCID details show such a lap. If the engineer wishes to stop bottom steel short then this can be done by adjustment of the length of the typical bar and appropriate scheduling.

3.8.6.11 Column support shear reinforcement

Column support shear reinforcement is covered by separate drawings from the slab drawings. Four drawings are included to cover for:

- bent-up bars
- castellated bars
- sausage stirrups
- open stirrups.

It is necessary for the shear reinforcement to be mechanically anchored to the top and bottom mats and additional loose bars are detailed on the shear reinforcement drawings for this purpose. For castellated bars or sausage stirrups, both layers of tension reinforcement are tied as suggested by the Cement and Concrete Association.

3.8.6.12 Edge reinforcement

All outside edges of flat slabs have top reinforcement detailed as a trombone bar. This will allow the slab to be treated separately from any upstand or downstand edge beam which is subsequently added to the drawing.

At all discontinuous edges, both top and bottom steel should be bent through 90 degrees. With a sufficient lap to the bottom steel the trombone bar satisfies this, and also doubles as a chair for the top mat of reinforcement.

When an edge beam is required, then the slab reinforcement parallel and adjacent to that beam will be considerably reduced from that for the unsupported case. As each zone of bars has been called up separately with its own bar mark, the bars for the edge supported case may be reduced in diameter.

3.8.6.13 Stability ties

Stability ties may be provided by either top or bottom reinforcement. To provide a continuous tie in the bottom mat would require either a lap between bottom bars or the addition of splice bars.

The top reinforcement, in the column strips, is continuous in a normal flat slab in two directions at right angles, and by using this top reinforcement as the stability tie, it is thought that in general no additional reinforcement will be required.

3.9 Columns (lu510)

3.9.1 General

The details in this set aid the user in preparing reinforcement detail drawings for columns. Each A4 drawing shows a single lift of a column. The details provided cover columns of square, rectangular or circular cross-section and cater for cases where the size of the column over that being detailed has the same or reduced section.

The reinforcement is detailed on a schematic elevation together with a cross-section at mid-height. If desired a second cross-section near the top of the column may be provided when the column size reduces above the upper floor level.

The concrete outline is shown in section only. Full formwork requirements need to be shown elsewhere or added manually.

The majority of details provided, employ vertical bars which are straight or cranked at their lower ends and pass straight through the intersection at the upper floor level, and laps are positioned just above the floor zone. This arrangement allows the beam-column intersection detail recommended in the Concrete Society's "Standard Reinforced Concrete Details" to be employed.

When the column is reduced in section above the upper floor level facilities are provided to enable the starter bars for the upper column to be composed of dowelled splice bars, or of a combination of these and the main bars. Additional ties are provided in these cases to facilitate the correct location of the starters.

3.9.1.1 Bar 'calling up' and scheduling

Bar 'calling up' follows the traditional method thus:

```
                           No.of    Dia(mm) Bar spacing(mm)
                             |  Type  |  Mark  |
                             |     |  |   |    |
Examples of bar calling up:  └── 14H20-05-200
                                  8H12-05-100 (Eurocode)
                                  8R12-05-100 (BS)
```

After printing the LUCID detail, use option 910 to produce a bar and weight schedule.

The bar schedule complies with the requirements of BS8666: 2000 using the shape code references, dimensioning and tolerancing given therein. The bar schedule is tabulated under the heading:

Member	Bar mark	Type and size	No. of mbr	No. of bar	Total no.	Lngth ofbar (mm)	Shape code	A (mm)	B (mm)	C (mm)	D (mm)	E/R (mm)	Rev ltr

Weights are given for each bar type (H, A, B, C, S or X) subdivided for bar diameters 16mm and under, and 20mm and over. (BS uses types H, R or X.)

3.9.2 Details

3.9.2.1 Schematic elevations

The schematic elevation shows in elevation typical main column reinforcing bars, but not their correct position laterally in relation to each other which must be deduced from the section. These details also show the ties in elevation. A choice of 16 elevation details is available summarised diagrammatically in Figure 3.9.4.

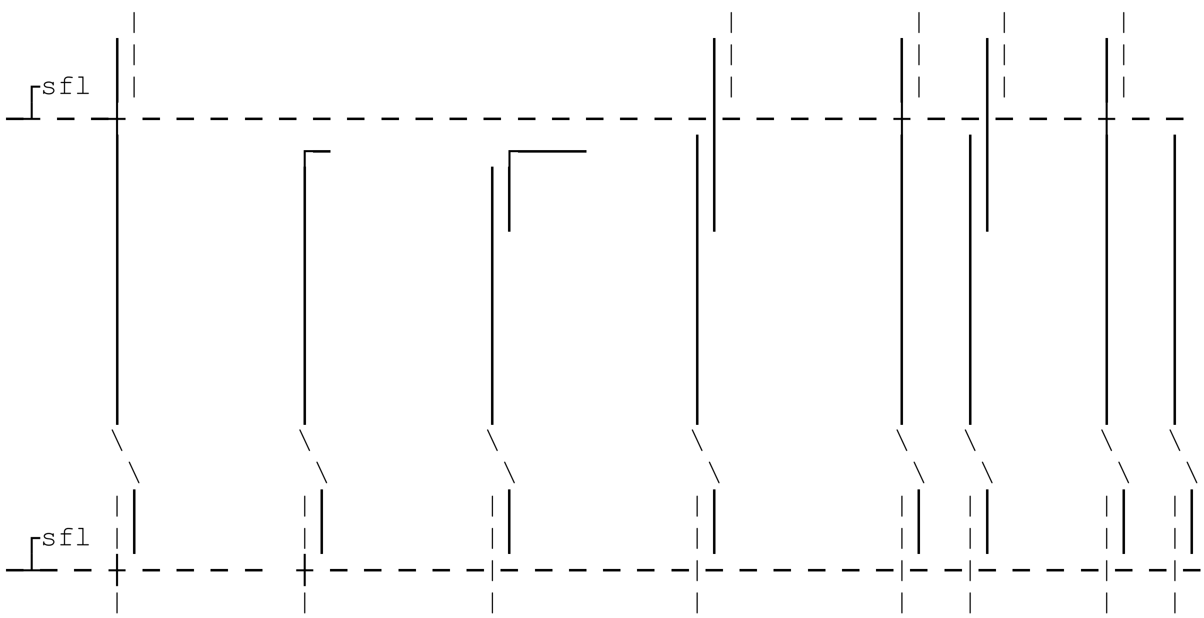


Figure 3.9.1(a) - VERTICAL BARS CRANKED AT LOWER END

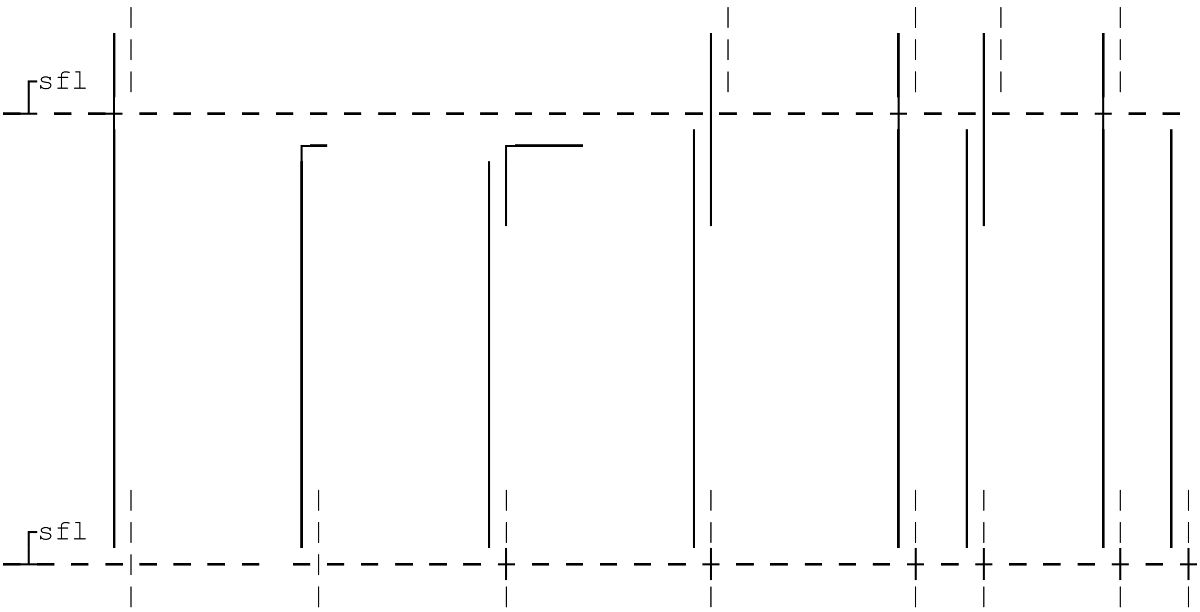


Figure 3.9.1(b) - VERTICAL BARS STRAIGHT AT LOWER END

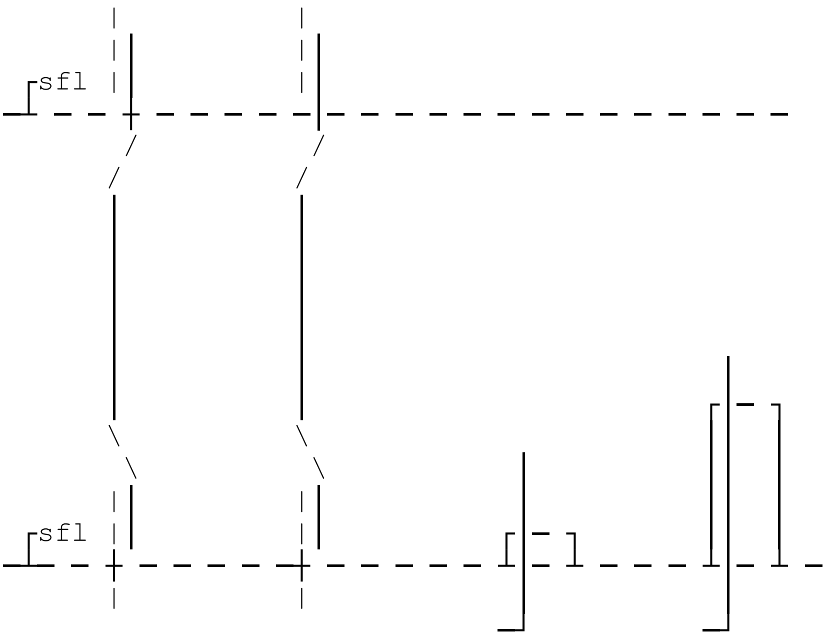


Figure 3.9.1(c) -

SUMMARY OF ELEVATIONS

VERTICAL BARS CRANKED AT UPPER & LOWER ENDS	STARTER BAR DRG	STUB COLUMN DRAWING

The vertical bars may be straight or cranked at their lower end. Cranked bars are compatible with the preferred beam-column intersection detail in that at their lower end they crank inside the bars from the column below which pass straight through the junction.

The "straight at lower end" details are intended primarily for two conditions:

- where the starter bars from a base, or from a column under, have increased side cover such that the vertical bars in the column are positioned outside the starters; or
- where circular columns are used, since the vertical bars can be positioned alongside the starter bars at the same cover.

Details are provided for the commonly occurring situation where all the vertical bars extend straight into the column over, and the same ties are detailed throughout the column length.

Details are provided for columns which stop at the upper floor level and also for use where there is only a nominal moment connection at the top of the column. If the anchorage of the main column bars becomes more critical, or if large moments have to be transferred to the column then L-bars at column heads are provided which may also be more appropriate for edge and corner columns.

Details are provided to cover the various possibilities which arise when the cross-section of a square or rectangular column is reduced in size above the upper floor level. The starter bars for the next lift, which are formed by dowelled splice bars and/or by continuing the main bars, are located by additional ties at the top of the column.

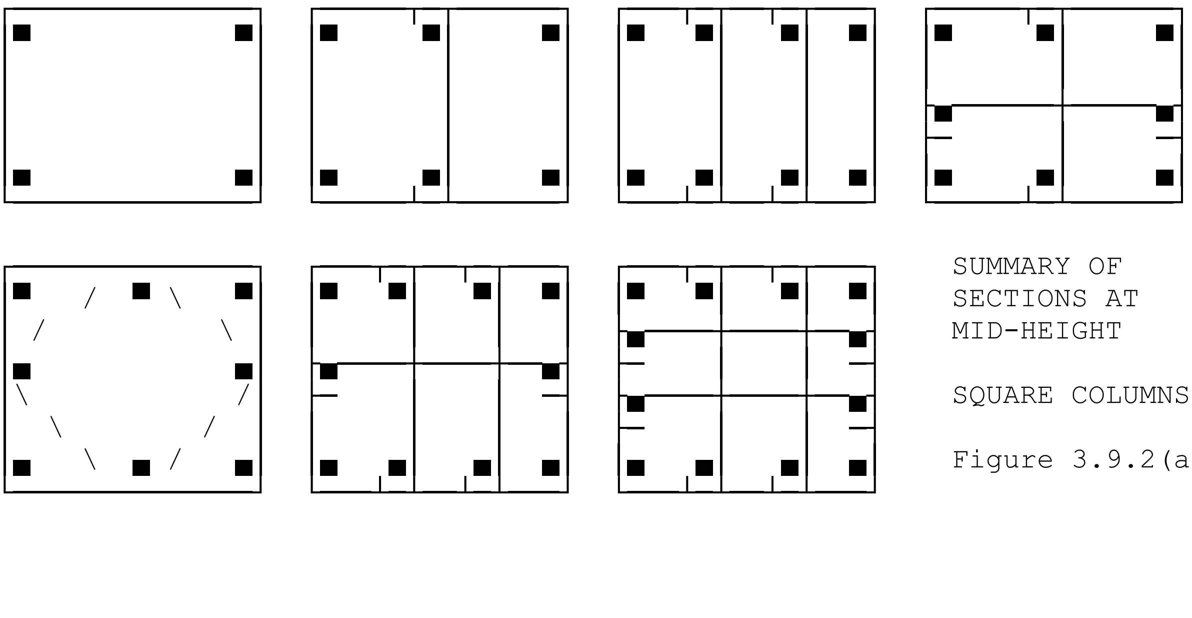
Details are provided to cover situations where there is only a small offset between column faces at reductions of column section, and the appropriate solution may be to crank the vertical bars through the depth of the beam or slab.

Some users may opt for the straight bottom/cranked top bar with the main bars for the next lift being positioned outside the starters. The upper crank however requires careful orientation on site and extra care must also be exercised to avoid a clashing of beam and column steel.

The final two schematic elevations offered cover starter bars and stub column drawings. Although it is generally recommended that column starter bars should be detailed and scheduled with the appropriate foundation, there may be cases where it is more convenient to be able to detail them separately, and a detail is provided for such cases. Similarly there may be cases where it is convenient to detail a stub column separately. For both these cases, the corresponding foundation drawings should, of course, indicate that the column starters are "detailed elsewhere".

3.9.2.2 Section at mid-height

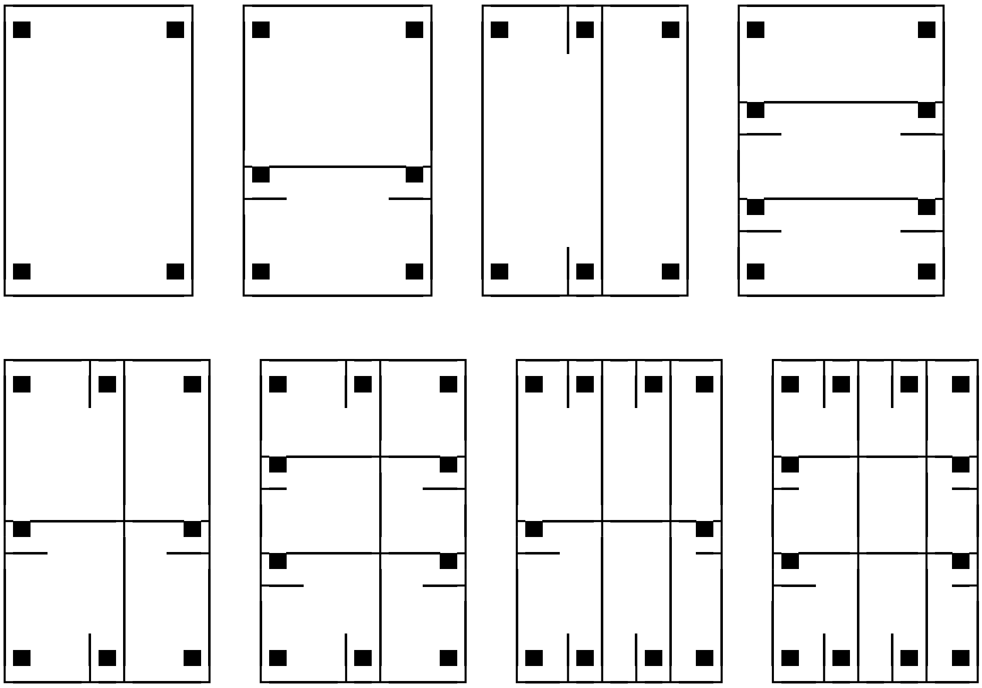
Twenty three details showing the mid-height section AA are available as summarised diagrammatically in Figure 3.9.2 (which omits circular columns containing 6, 8, 10 & 12 bars).



SUMMARY OF SECTIONS AT MID-HEIGHT

SQUARE COLUMNS

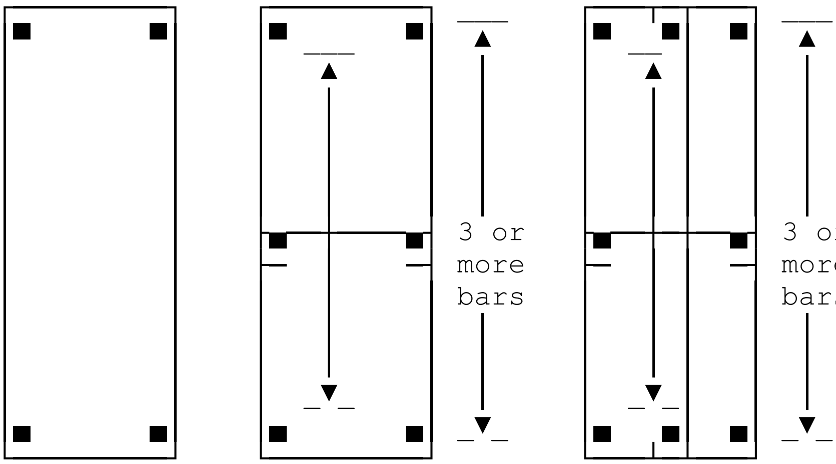
Figure 3.9.2(a)



SUMMARY OF SECTIONS AT MID-HEIGHT

RECTANGULAR COLUMNS

Figure 3.9.2(b)



SUMMARY OF SECTIONS AT MID-HEIGHT

VERY RECTANGULAR COLUMNS

Figure 3.9.2(c)

For rectangular columns, and for asymmetrically reinforced square columns, only one of the two possible section orientations is provided so the user must ensure the correct orientation of the reinforcement by labelling the grid line circles provided.

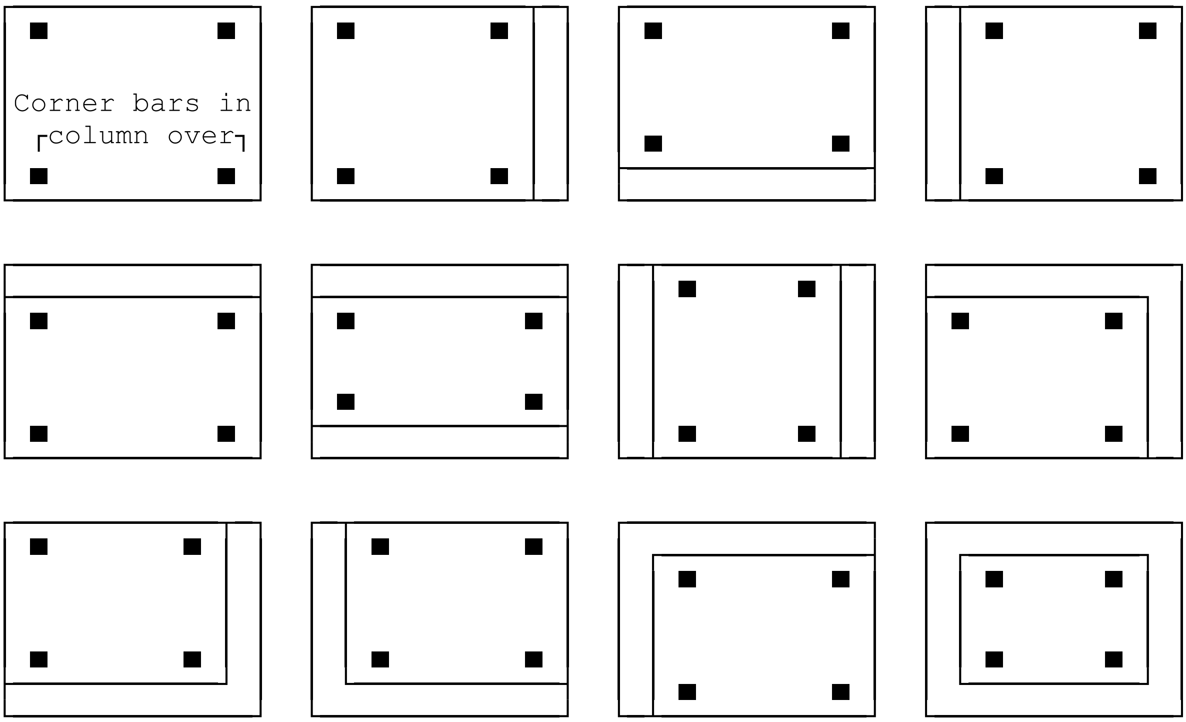
If opposite faces of a column are not reinforced in an identical manner the user must draw an extra grid line to one side of the column to ensure correct orientation.

Three options are provided to cover the case of columns which are very rectangular. When these very rectangular columns are used, the user must specify in the section the number of main bars in each long face and the number of transverse ties.

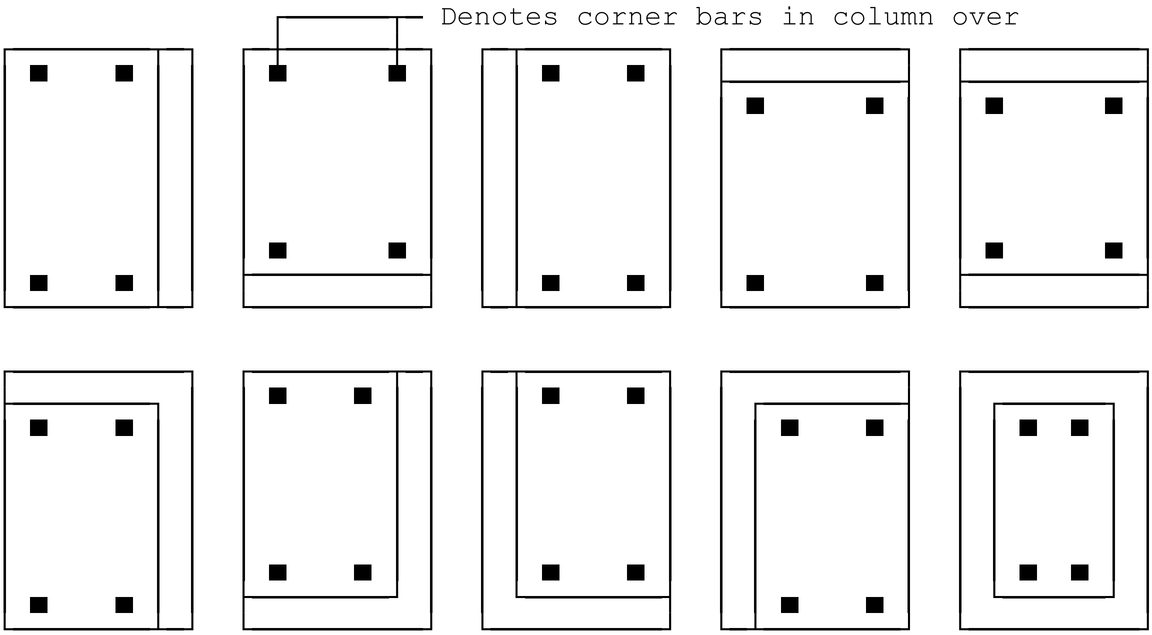
3.9.2.3 Section near the top of the column

When the column over that being detailed is reduced in size, it is necessary to provide a section at the top of the column to show any starters and additional ties.

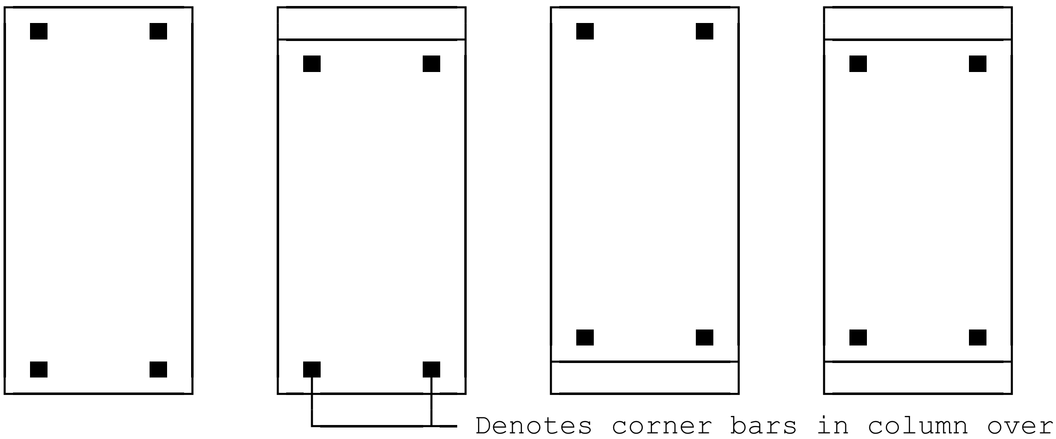
The details show the geometric relationship of the column over to the column under, the special tie(s) to locate the starters for the column over, and the starter bars. Any additional starter bars to the column over must be added to the linework drawing manually. The range of details available is summarised diagrammatically in Figure 3.9.3.



SUMMARY OF SECTIONS AT TOP OF THE COLUMNS
Figure 3.9.3(a) - SQUARE COLUMNS UNDER



SUMMARY OF SECTIONS AT TOP OF THE COLUMNS
Figure 3.9.3(b) - RECTANGULAR COLUMNS UNDER



SUMMARY OF SECTIONS AT TOP OF THE COLUMNS
Figure 3.9.3(c) - VERY RECTANGULAR COLUMNS UNDER

3.9.2.4 Extra ties detail

In the report "Standard Reinforced Concrete Details" it is suggested that, when employing cranked vertical bars, an adequate tie be provided to resist the outward horizontal component of the force in the inclined portion of the bar. If the ties provided throughout the column are of adequate size to resist this force the user may wish to indicate specifically on the drawing the tie positions at a crank.

If the ties provided are too small to resist this force, however, additional ties may be required. The most likely place for an extra tie is at the top of the crank.

3.9.3 Sequence of detail selection

If there is no change in column section at the upper floor level, the user will be asked to select section AA followed by the appropriate schematic elevation and, if desired, the extra tie detail.

If there is a change in section the user will first be asked to select section AA, then section BB appropriate to this change in geometry, then the appropriate schematic elevation, and if desired, the extra tie detail. However, if any of the corner bars in section BB are a continuation of non-corner bars from section AA, then these bars may not be in quite the same positions on the two sections on the linework drawing. The user may therefore wish to manually make slight amendments to ensure compatibility.

3.9.4 Completion of linework drawing

3.9.4.1 Pre-assigned bar marks

Generally all the bars on the elevation and all the ties in section AA and BB have pre-assigned bar marks.

3.9.4.2 Dimensions

Additional dimensions or covers may be added if thought necessary. The spread of main ties is located vertically by a dimension to the first tie. The drawing shows the size of the column cross-section.

3.9.4.3 L-bars at a column head

It is recommended that the user gives a positive indication on the drawing of the position of the horizontal legs of the L-bars when they are used. This is most easily achieved by adding to the drawing a plan of the L-bars, together with some section arrows on the elevation. It is further recommended that the L-bars be shown on the drawings of the elements into which they project as being "detailed elsewhere".

Although only one bar mark is given for the L-bars, there will be situations where two different bars are required. For example, at an edge column the L-bars in the external face may be different from those on the internal face. In such cases it is suggested that two calling-up messages be written against the L-bar on the schematic elevation, and that a plan on the L-bars should be added as outlined previously so as to completely specify the detail.

3.9.4.4 Heavy moment connections to beams

L-bars may sometimes be required to transmit a heavy moment from the beam to the top of the column. For column lifts other than those of the top storey, L-bars should be added manually to the column detail and shown as "detailed elsewhere" in the beam drawing.

3.9.4.5 Column sections outside the range

It is suggested that column sections not explicitly covered by the details may be achieved by selecting section AA and then manually adding further bars or ties to the detail to form the required section. Such a procedure should enable the user to cope with the vast majority of situations.

3.10 Walls (lu610)

The details in this set are an aid in the preparation of reinforcement detail drawings for walls comprising straight panels of constant height and thickness.

The walls shown in plan in Figure 3.10.1 are typical examples of what might be detailed. The details are primarily intended for panels whose length to thickness ratio is greater than 4:1 and the "Columns" details should generally be used for ratios less than this.

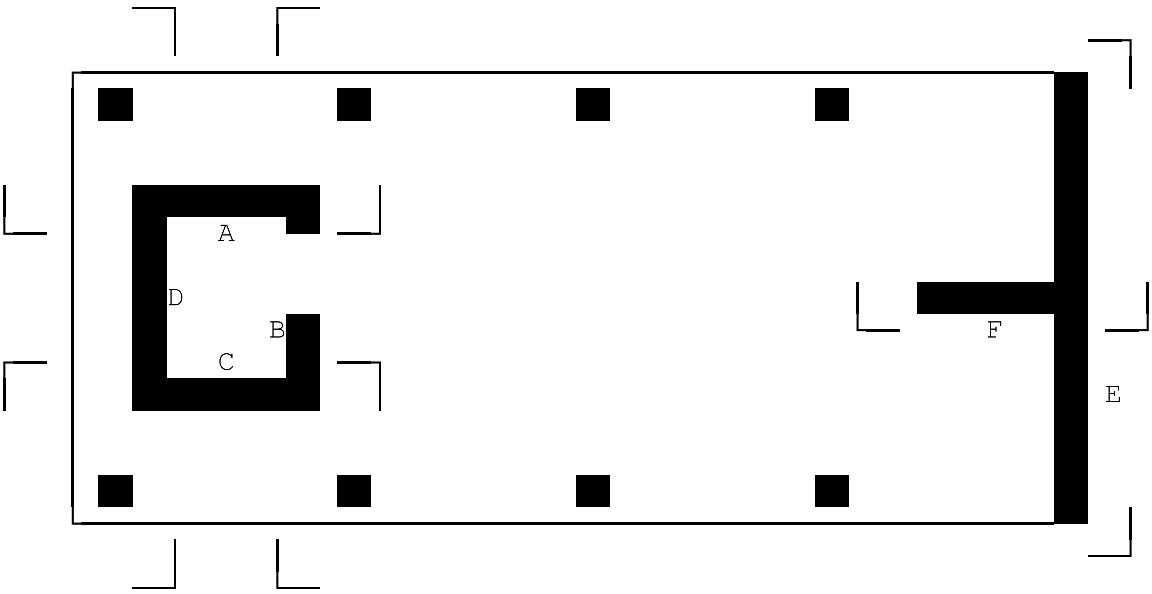


Figure 3.10.1 - TYPICAL FLOOR PLAN
(Wall panels marked A-F)

The details cover short, medium or long panels with uniformly spaced vertical steel, and medium and long panels with extra bars bunched at the ends. In all cases the vertical steel is either stopped off or carried on upwards as starters for the next lift. The medium range includes panels with openings for doors, which can be central or to the left or right of centre.

Each A4 drawing shows a single panel of wall detailed in elevation and with a horizontal section. For panels with a doorway a vertical section over the door is also shown.

When a wall is detailed using more than one panel the layout of the wall, showing the length over which each reinforcement drawing applies, should be shown elsewhere such as on a general arrangement or key plan drawing.

The bar marks have been allocated such that bars in similar positions in all panels have the same bar marks. Because of this bar marking system, for the simpler panels gaps occur in the numbering sequence.

3.10.1 Bar 'calling up' and scheduling

Bar 'calling up' follows the traditional method thus:

```
                               No.of    Dia(mm) Bar spacing(mm)
                                 |  Type |  Mark  |
                                 |     | |  |     |
Examples of bar calling up:      └─── 14H20-05-200
                                       8H12-05-100 (Eurocode)
                                       8R12-05-100 (BS)
```

After printing the LUCID detail, use option 910 to produce a bar and weight schedule.

The bar schedule complies with the requirements of BS8666: 2000 using the shape code references, dimensioning and tolerancing given therein. The bar schedule is tabulated under the heading:

Member	Bar mark	Type and size	No. of mbr	No. of bar	Total no.	Lngth ofbar (mm)	Shape code	A (mm)	B (mm)	C (mm)	D (mm)	E/R (mm)	Rev ltr

Weights are given for each bar type (H, A, B, C, S or X) subdivided for bar diameters 16mm and under, and 20mm and over. (BS uses types H, R or X.)

3.10.2 The details

The range of panels covered is summarised in Figure 3.10.2 and is subdivided into short, medium and long panels.

The panels are generally applicable to the following ratios:

short panels - between 4:1 and 10:1
medium panels - between 10:1 and 20:1
long panels - over 20:1

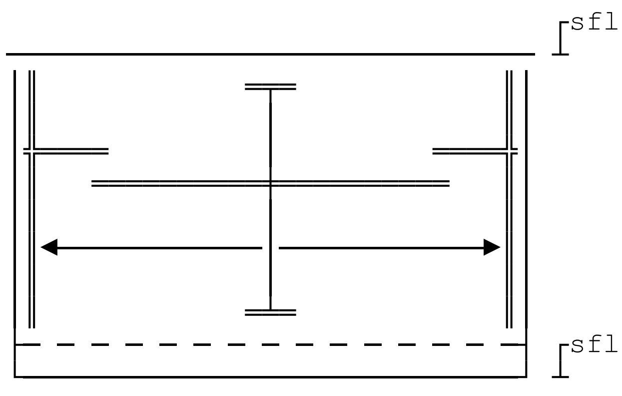
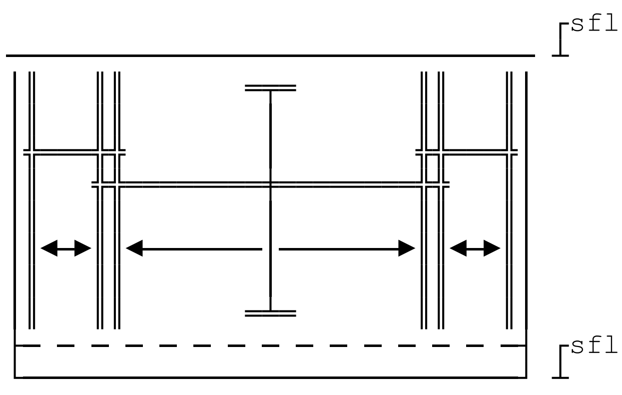
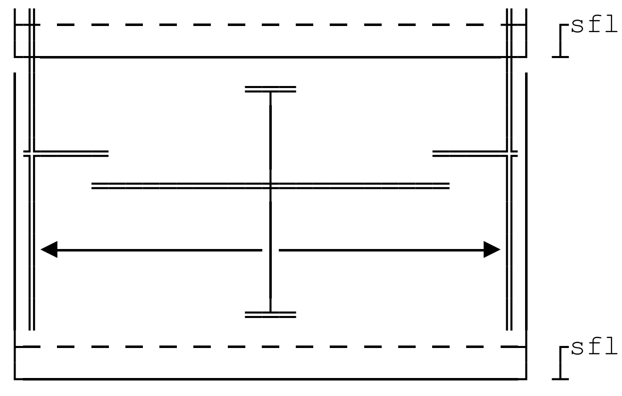
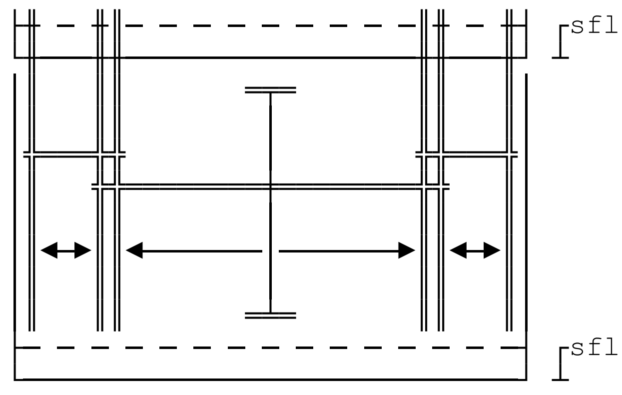
			SHORT PANEL	MEDIUM		LONG PANEL
				PLAIN	WITH DOOR	
PANEL STOPS	UNIFORM BARS		Yes	Yes	Yes	Yes
	BUNCHED ENDS		No	Yes	Yes	Yes
PANEL CONTINUES ABOVE	UNIFORM BARS		Yes	Yes	Yes	Yes
	BUNCHED ENDS		No	Yes	Yes	Yes

Figure 3.10.2(a) and (b) - SUMMARY OF DETAILS (Panel stops/continues above)

3.10.2.1 Short panels

Since short panels are unlikely to have doorways or bunched ends there are only two alternatives available. In both cases the bars are uniformly spaced and the choices allow the vertical bars to
a) stop off at the upper floor, or
b) continue up as starters for the next lift.

3.10.2.2 Medium panels

It is anticipated that the medium panels range will be used for the majority of the cases. The details offer the choice of:
a) uniformly spaced vertical bars; or
b) uniformly spaced vertical bars with bars bunched at each end of the panel.

For both categories a) and b) the vertical bars may either:
c) stop off at the upper floor; or
d) continue up as starters for the next lift.

Alternatives are provided for plain panels and for those with door openings which are to the right of centre, central or to the left of centre.

3.10.2.3 Long panels

The choices offered for long panels are those listed as a) to d) in section 3.10.2.2, but they are only provided for plain panels. If doorways are required they may either be added to a drawing from a long panel, or alternatively it is easy to choose the appropriate medium panel detail and add a breakline in a suitable position to infer a long panel.

3.10.3 Completion of linework drawing

3.10.3.1 Concrete outlines and dimensions

The details show only the minimum concrete outline and at possible points of intersection with other elements, such as the upper slab level and the ends of the panel, gaps have been left in the concrete outline. As required, the intersecting elements should be drawn in and the remaining gaps closed manually.

3.10.3.2 Reinforcement layout

The following general pattern of reinforcement has been adopted.
a) All horizontal bars are in the outermost layer at each face (i.e. the vertical steel is inside the horizontal).
b) Where the horizontal bars meet the end of the panel or a doorway opening they are always closed with a horizontal U-bar.
c) Above doorways the vertical bars are closed with U-bars at the bottom, and in addition short, straight bars at 45° in both faces are included at the upper corners of the doorway.
d) Spacers are shown on the drawing.

Under some circumstances the user may wish to fully tie the vertical steel either throughout the length of the panel of just in the end zones. In either case the ties should be added to the drawing manually.

3.10.3.3 Panel junctions

Walls which consist of more than one panel involve the detailing of the panel junction(s). There are many types of junction, the most common probably being L, T, or cruciform, and minor amendments are required to the linework drawings to cater for these.

The amendments are kept to a minimum if all walls are detailed separately.

In the case of two walls meeting at a corner, the outside corner bar will be duplicated and one bar should be moved to the inside corner.

3.10.3.4 Doorways

Details which show doorways and the start of the next storey above also show a doorway on this upper storey. If the doorways are not immediately above each other the upper doorway and starter bars must be amended manually

The details permitting an eccentric doorway assume the smaller area alongside the doorway to be not small. If the doorway is close to the return wall the horizontal bars mark 3 (or mark 4 for a right of centre door) may not be required, continuity of horizontal steel being obtained by lapping the U-bars mark 8 and 9. Typical bar mark 3 or 4 may be removed and typical bars mark 8 and 9 extended as required.

3.10.3.5 Holes and nibs

The only type of hole through a wall covered by the "Walls" details is a doorway opening. Any other holes, or nibs, corbels etc. should be added manually. The user is reminded that LUCID option 460 shows reinforcement patterns that may also be applicable to holes in walls. If these details are used for walls, the reinforcement cover notes should be amended as the terms "top" and "bottom" become incorrect.

3.10.3.6 Multiple use of drawings

Under some conditions a particular detail may apply to panels on more than one floor of a building and hence need cross referencing to more than one general arrangement drawing, key plan drawing or bar schedule. It is suggested that this can conveniently be done by including on the detail drawing a "Multiple Use Table", and adding the words "See Table" or similar in the relevant places of the detail manually.

3.10.4 Viewing convention

Since no universal system of viewing walls when preparing details is in general use, it is suggested that the user specify such a convention on his/her general arrangement and/or key plan drawings, such as in Figure 3.10.1, and that this convention is clearly cross referenced between these drawings and the reinforcement details.

It should be noted that the LUCID "Propped Retaining Walls" details show elevations viewed from inside the basement. If panels A-D of Figure 3.10.1 form a lift well that goes below ground slab level, care must be taken that all panels above one another are viewed consistently and in accordance with this convention.

3.11 In-situ Staircases (lu710)

The details in this set aid in the detailing of in-situ staircase flights and landings. Each staircase flight, spanning principally between line supports at its top and bottom, is detailed in section only. Since the section incorporates a break line, the detail can be used for flights with seven or more risers. A key plan can be provided on the drawings to assist in specifying the orientation and location of the staircase. The user has the choice of several arrangements of reinforcement and starter bars at each end of the staircase, and of whether or not finishes and the undercut are shown.

Staircase details are "not to scale" and thus differ from all other LUCID elements which are generally drawn "to scale".

3.11.1 Bar 'calling up' and scheduling

Bar 'calling up' follows the traditional method thus:

```
                           No.of    Dia(mm) Bar spacing(mm)
                             |   Type |  Mark  |
                             |      | |  |     |
Examples of bar calling up:  └── 14H20-05-200
                                  8H12-05-100 (Eurocode)
                                  8R12-05-100 (BS)
```

After printing the LUCID detail, use option 910 to produce a bar and weight schedule.

The bar schedule complies with the requirements of BS8666: 2000 using the shape code references, dimensioning and tolerancing given therein. The bar schedule is tabulated under the heading:

Member	Bar mark	Type and size	No. of mbr	No. of bar	Total no.	Lngth ofbar (mm)	Shape code	A (mm)	B (mm)	C (mm)	D (mm)	E/R (mm)	Rev ltr

Weights are given for each bar type (H, A, B, C, S or X) subdivided for bar diameters 16mm and under, and 20mm and over. (BS uses types H, R or X.)

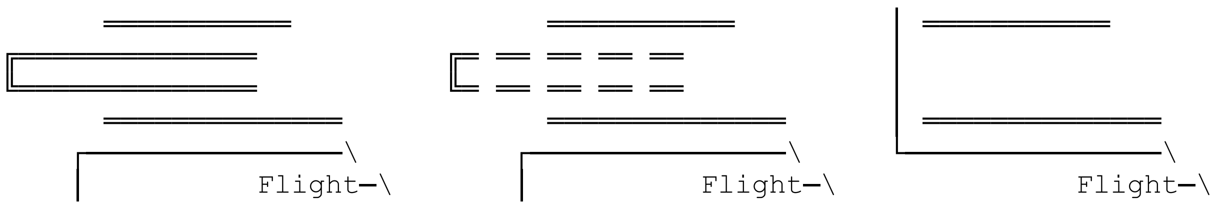
3.11.2 Staircase flight drawings

A staircase flight drawing is usually built up from four details, one from each of the following subsets:
a) top end;
b) tread profile (including dimensions, finishes and undercut options);
c) bottom end; and
d) key plan.

However, the user may omit the key plan and draw his/her own plan. Details for flight drawings are summarised in Figures 3.11.1 - 3.11.3.

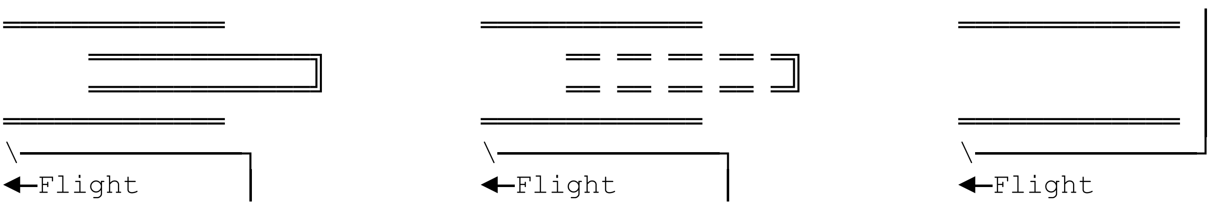
3.11.2.1 Top end details

The basic patterns of reinforcement are the same for the three top end details, and the only choice concerns the starter bars to the supporting structure. As shown diagrammatically in Figure 3.11.1 these may be detailed with the staircase, shown as detailed elsewhere or not detailed at all.



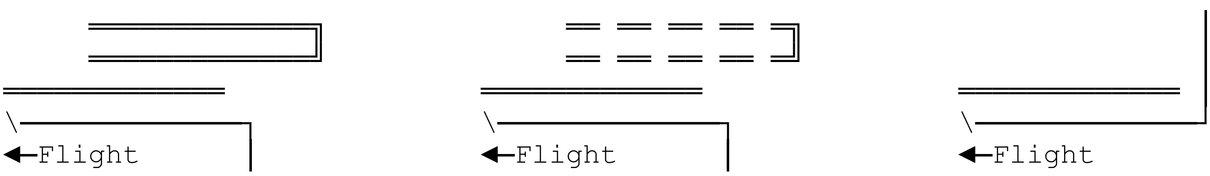
Starter bars detailed | Starter bars detailed elsewhere | No starter bars

Figure 3.11.1 - TOP END OF FLIGHT



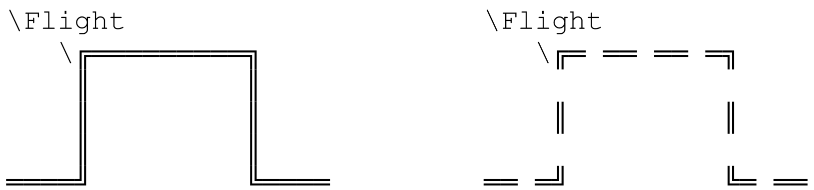
Starter bars detailed | Starter bars detailed elsewhere | No starter bars

Figure 3.11.2(a) - BOTTOM END - BARS IN TOP OF KNUCKLE



Starter bars detailed | Starter bars detailed elsewhere | No starter bars

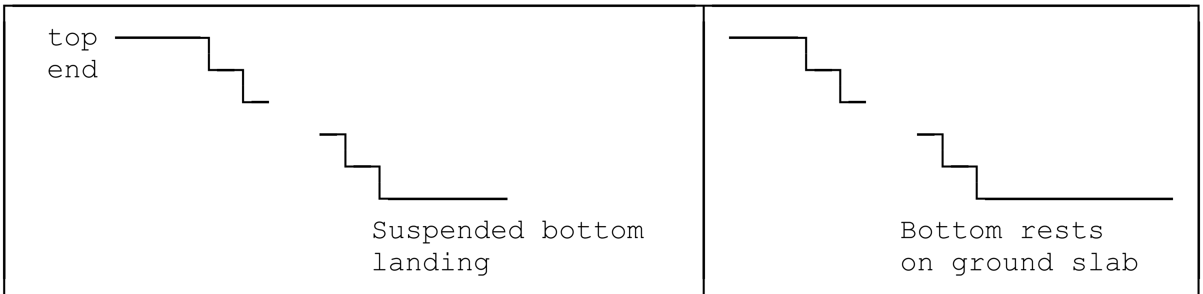
Figure 3.11.2(b) - BOTTOM END - NO BARS IN TOP OF KNUCKLE



Starter bars detailed

Starter bars detailed elsewhere

Figure 3.11.2(c) - BOTTOM END - GROUND SLAB



Tread profile	Undercut riser	Square cut riser	Undercut riser	Square cut riser
Separate finishes shown	1	2	5	6
Separate finishes not shown	3	4	7	8

Figure 3.11.3 - SUMMARY OF 'TOP END', 'BOTTOM END' and 'TREAD PROFILE' OPTIONS

3.11.2.2 Tread profiles

There are four tread profile details for staircases with suspended bottom landings, and a corresponding four for staircases whose bottom ends rest on ground slabs.

The options in these details provide square or cut back risers and the inclusion or absence of finishes. The eight details are summarised in Figure 3.11.3.

3.11.2.3 Bottom end details

Details provide six alternative arrangements of reinforcement at the bottom end of a staircase spanning onto a landing or similar slab. The basic reinforcement arrangement shows bars in the bottom face only but the user may also have bars in the top face of the bottom knuckle. As with the top end there are three choices of where starter bars are detailed.

The details which are summarised in Figures 3.11.2(a) to 3.11.2(c), provide two alternatives for cases where a staircase rests on a ground slab: the starter bars may be detailed with the staircase or merely indicated as being detailed elsewhere.

3.11.2.4 Key plan details

There are eight key plan details which are summarised in Figure 3.11.4. These details are also used in the preparation of drawings for landings.

The user can select any one of four orientations and show either a narrow or a wide stairwell.

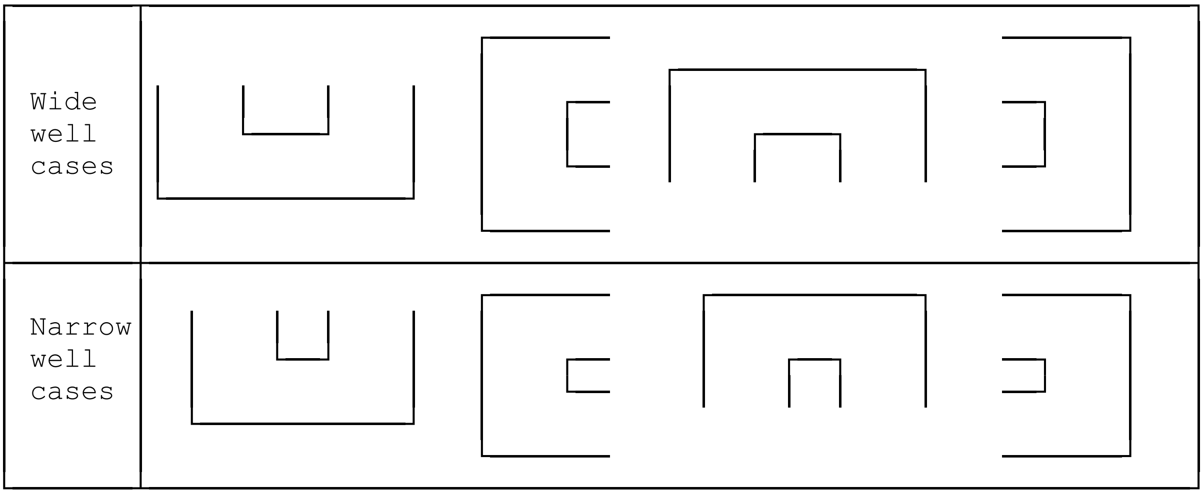


Figure 3.11.4 - SUMMARY OF KEY PLAN DETAILS
(for both flight drawings and landing drawings)

3.11.3 Completing a flight drawing

3.11.3.1 Landing bars

Linework drawings produced from the details show the landing bars as open circles, since it is presumed these will be detailed on a separate landing drawing. However if the transverse landing bars are detailed on the flight drawing, this drawing can be completed as shown below

Landing bars shown	5H12-11-300 T2
as open circles	5H12-11-300 B1

The landing bars are always shown in the first layer in the bottom face and second layer in the top face, but the user may readily amend these positions if necessary.

3.11.3.2 Transverse bars mark 10

In the bottom face of the flight, the transverse bars mark 10 have been shown at the rate of one per riser. In the top face, two bars mark 10 are shown associated with bars mark 08, and a further two with bars mark 03 (if the 03's are present).

If additional mark 10's are detailed, it is advisable to add appropriate dots to the drawing manually.

In the usual case where the staircase flight is not connected structurally to any adjacent wall, then the mark 10 bars will generally be straight or "bobbed" at each end. In these cases, no transverse section through the flight is normally required. If, however, there is a structural connection to a side wall, then a different arrangement of reinforcement may be required, in which case a transverse section may need to be drawn. It should be noted that in such a case the top and bottom transverse bars may not be the same and the necessary amendments must be made to the drawing.

3.11.3.3 End conditions

The three alternative end conditions provided may be amended in respect of reinforcement and/or concrete to give a large variety of alternatives.

3.11.3.4 Bar termination points

The details show bar stop off points based on average lap lengths and structural dimensions. The user should amend any stop off points which are not appropriate to his/her particular case. For example, if the distance from the bottom tread to the edge of the support is unusually short, or if the required lap lengths are unusually long, then the situation shown by the details may need amending.

3.11.3.5 The first riser fillet

Those details giving finishes show a fillet in the structural concrete at the bottom of the first riser. This is required to preserve the waist thickness if the finish on the bottom landing is thicker than that on a normal tread. If the finishes are the same thickness no fillet exists and the user may wish to erase it from the linework drawing.

3.11.4 Shapes and scheduling of flight bars

The bar marks on the flight drawings have been assigned so that each bar, if it occurs, always has the same bar mark. This means, for example, that if the bottom end starter bars are detailed elsewhere then bar mark 4 is not used. The bar marks are in a probable order of fixing.

The arrangement of reinforcement adopted is one that does not require any very critical cutting and bending, since almost every critical length incorporates a lap into which minor tolerance errors can be absorbed. However, since it is always the horizontal arm of a cranked bar which is the more critical for length, it is prudent to make this arm dimension 'A' in the bar schedule.

3.11.5 Landing drawings

The principal details which assist in the detailing of landings are summarised in Figure 3.11.5. They produce drawings on which the landing is detailed in section only. The engineer has the choice of including or omitting finishes.

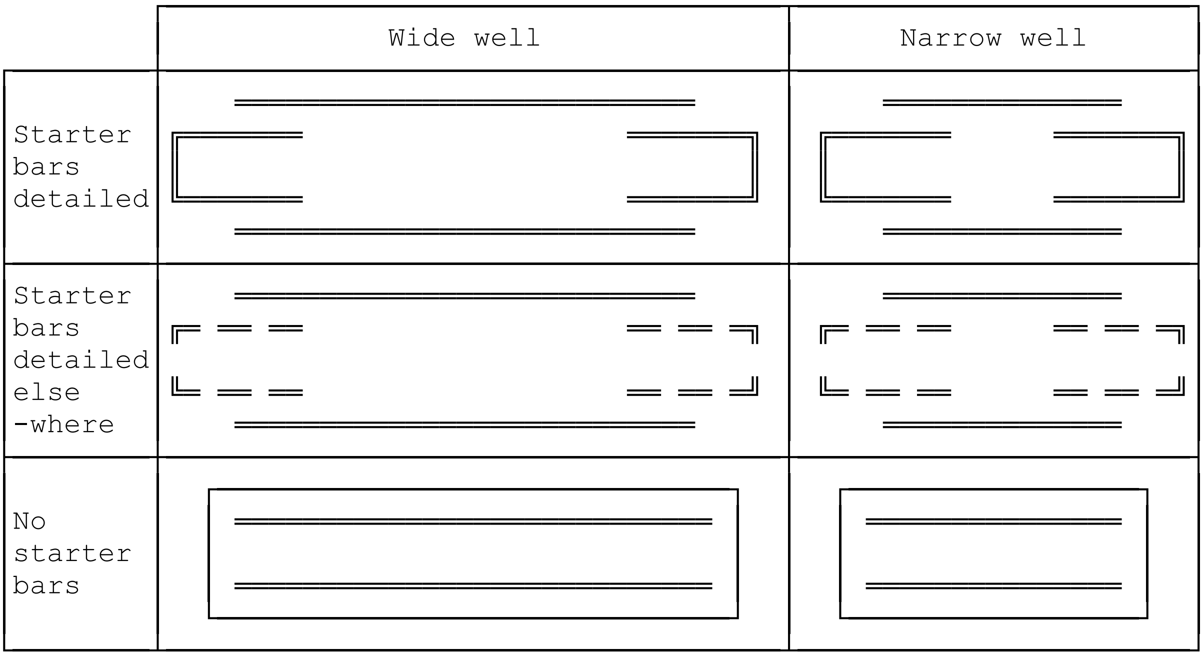


Figure 3.11.5 - SUMMARY OF DETAILS FOR LANDINGS

3.12 Beams (lu810, lu820)

3.12.1 Terminology

Although the terminology which is used is generally well known, to avoid any confusion it is defined in Figures 3.12.1(a) & 3.12.1(b).

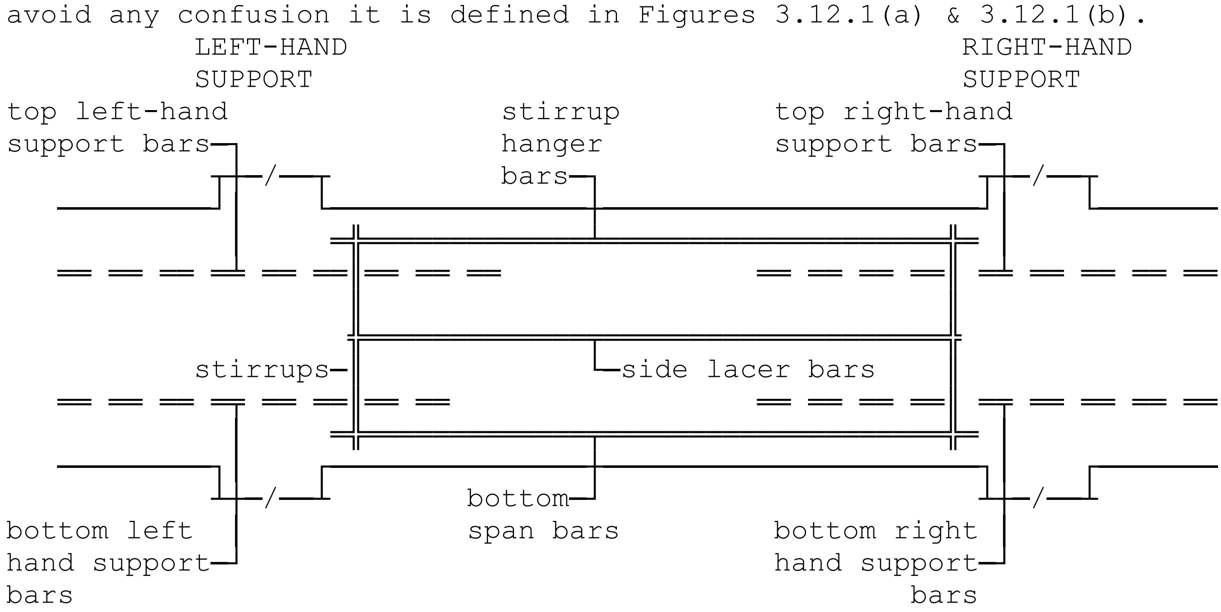


Figure 3.12.1(a) - EXPLODED VIEW OF BEAM

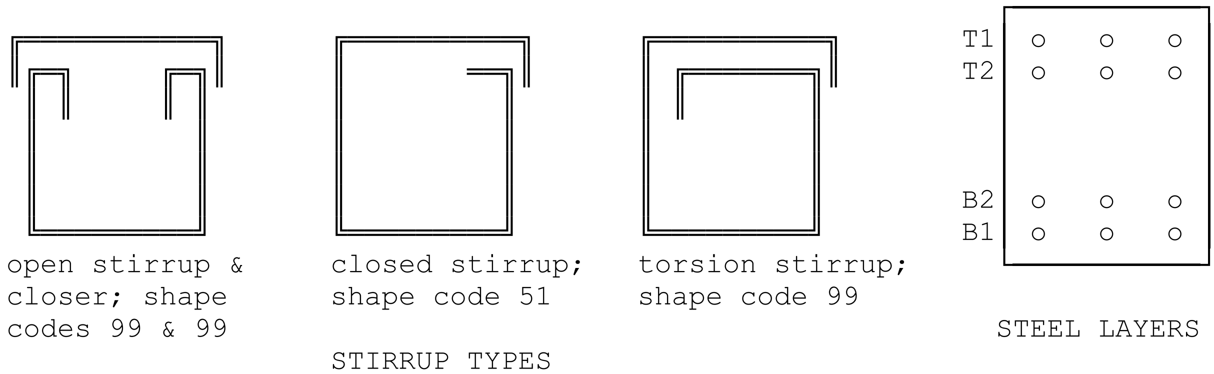


Figure 3.12.1(b) - DEFINITION OF TERMINOLOGY

The arrangements of bars used in the LUCID beam details follow the broad recommendations set out in the Concrete Society's Report "Standard Reinforced Concrete Details", namely:

- neither the bottom span bars nor the stirrup hanger bars extended into the column; and
- continuity through the column is provided by the top main support bars and by bottom support bars of appropriate size.

This arrangement of steel has two major advantages. First, the stirrups, bottom span bars and stirrup hanger bars can be completely prefabricated. Second, as the support bars do not have to be positioned in the corners of the stirrups, there is considerably more scope, without resorting to cranking, for them to be positioned so as to avoid column or intersecting beam reinforcement.

3.12.2 General

A LUCID beam gives the reinforcement for only one span and if a string of beams is to be detailed a drawing is required for each span. The span reinforcement in each beam is detailed by an elevation and two cross-sections and there are separate elevations and sections for the steel at each of the supports. The span and support elevations are separated both for clarity and to emphasise the prefabrication aspect of the span reinforcement.

The range of details covers beams with up to two layers of steel in both the bottom of the span and the top over a support. Each layer may contain from 2 to 8 bars. The shear reinforcement provided consists of vertical stirrups with from 2 to 6 legs at a section and these may be arranged in up to 3 different zones along the beam.

For a typical internal span of a continuous beam the convention that has been adopted is that the drawing details the span cage and right-hand support steel. The left-hand support steel is shown dotted, since this will have been detailed with the span to its left. Where the beam is the left-handed end span the left-hand support steel is included on the drawing. It is therefore recommended that, when detailing a string of beams, the user should start at the left-hand end. Grid lines are included on the drawing to help identify the span being detailed.

3.12.2.1 Bar 'calling up' and scheduling

All the bars on the beam details have pre-assigned bar marks. From experience with LUCID on site it was found that steel fixers preferred to have such a system in which - for example - all the stirrup hanger bars were Mark 5.

Bar 'calling up' follows the traditional method thus:

```
                            No.of    Dia(mm) Bar spacing(mm)
                              |    Type | Mark |
                              |       | | |    |
Examples of bar calling up:   └── 14H20-05-200
                                   8H12-05-100 (Eurocode)
                                   8R12-05-100 (BS)
```

After printing the LUCID detail, use option 910 to produce a bar and weight schedule.

The bar schedule complies with the requirements of BS8666: 2000 using the shape code references, dimensioning and tolerancing given therein. The bar schedule is tabulated under the heading:

Member	Bar mark	Type and size	No. of mbr	No. of bar	Total no.	Lngth ofbar (mm)	Shape code	A (mm)	B (mm)	C (mm)	D (mm)	E/R (mm)	Rev ltr

Weights are given for each bar type (H, A, B, C, S or X) subdivided for bar diameters 16mm and under, and 20mm and over. (BS uses types H, R or X.)

3.12.2.2 Fixing dimensions

The detailing of reinforcement is more the art of pattern selection rather than precision in bar location. As the bars are drawn to scale then the steelfixer may locate the bars by scaling the details. Where positions or lengths of reinforcing bars are critical the user should add the critical dimensions to the detail manually.

3.12.2.3 Covers

The covers for the main span and support reinforcement should be given. Left and right are as viewed in the sections, and some users may wish to add a comment on their G.A. or general notes drawing that this is so.

3.12.2.4 Concrete outlines

Some users may wish to add a concrete outline to the sections, as this may clarify the detail, particularly when there are nibs or upstands, etc. Four small dots are preprinted at the corners of sections AA and BB to act as a drafting aid.

3.12.2.5 Spacer bars

There are differing practices over whether spacers should be detailed and scheduled with a drawing or not. Where a detail with two layers of bars is selected, a spacer is shown in section, and a first and last spacer is shown on the elevation. Users who wish to call them up may do so manually.

3.12.2.6 Centre lines and grid lines

The drawings show a chain-dotted line as the centre line of each support. Grid line labels are used in the balloons to orientate the elevations.

Where the centre line and grid line are not coincident, and the user wishes to avoid possible confusion of interpretation, sufficient clarifying amendments should be made.

3.12.2.7 Stirrup shapes

Two basic types of stirrups are available: either open stirrups to shape code 99 with shape code 99 closers, or closed stirrups without "tags". (Tags, look like small bent up bar ends; they are drawn on bar elevations to show where a bar terminates.)

3.12.3 SIMPLY SUPPORTED, PROPPED CANTILEVER AND CONTINUOUS BEAMS

3.12.3.1 Bottom span bars

The sections which are offered have been chosen on the assumption that there will not be more bars in the second layer than the first, and that generally where the number of bars is in excess of three then there will be a minimum of two of each kind of bar. It is considered the choice offered covers the majority of cases which are likely to be required.

When making his/her choice of bottom span bars, the user must ensure that there will be sufficient bars in the bottom to act as corner bars for any stirrups. The number of bars which go through to the support, and therefore appear on section BB, can easily be seen from the elevation.

3.12.3.2 Stirrups in section

The user has the option of closed stirrups or open stirrups with closers along the whole length of the beam. The support section always has as many legs as, or more legs than, the midspan section.

The stirrups may be arranged in up to three zones. Section BB may be at the left-hand or right-hand support. However, as the number of longitudinal bars is always the same at each support, the bars at section BB are consistent with both section AA and the elevation irrespective of choice of position of section BB.

As span cross-sections showing stirrups are only taken in two places, if 3 stirrup zones are used the additional zone is not individually shown in section. Hence in this zone only those bars used in sections AA and BB may be used. However, as the bar sizes in sections AA and BB can be different, and the user can vary the spacing or have stirrups in pairs in the other zones, in practice this presents no restriction.

3.12.3.3 Right hand support bars

For continuous beams, both the top and the bottom steel may be in either one or two layers. It should be noted that although all the top bars are fully detailed the bottom support reinforcement and the end support reinforcement is shown as only two bars in section. Any more than this number must be added using manually.

3.12.3.4 Stirrup zones and left-hand support detail

Section arrows AA are shown at midspan and BB where the heaviest shear may be expected. For beams with symmetrical stirrup zones, section BB is drawn at the right-hand support but the user may easily change this if for some reason section BB is required to be at the left-hand support.

The range of left-hand support details allows the steel to be detailed elsewhere, which is the case when the previous left-hand span has been detailed, or gives a choice of three different bar arrangements at a non-continuous end. It should again be noted that only the two outside bars are shown in section and any other bars must be added manually.

3.12.3.5 Scales

Two scales are used in the beam details; a general scale for beam depth and width, and a special scale for beam spans. Where the beam width is less than 5/8 of beam depth or greater than 1.7 times the beam depth, then the detail provided is not to scale.

3.12.3.6 Span top steel

Although the stirrup hanger bars will generally be purely nominal there is no reason why they cannot be used for structural purposes such as compression steel or to resist hogging moments throughout the span. Extra bars may, of course, be added if they are required.

3.12.3.7 Details at external columns

The details available are not particularly suitable for cases where a considerable degree of fixity is required between the beam and the column, and the user must make any necessary amendments.

3.12.4 Cantilever beams

In the following, cantilever beams will be referred to as "cantilevers" and beams supported at both ends as "beams". As on the beams drawings, the reinforcement for cantilevers is shown on two elevations, one detailing the support steel and the other the span steel.

In all cases the support steel is detailed on the cantilever drawing. This means that the adjacent beam span must show the relevant support steel as "detailed elsewhere". Facilities are provided for this in the beams detail, for both right-hand and left-hand cantilevers.

3.12.4.1 Top support bars

The LUCID detail shows the top support bars in elevation and in section BB, which is taken through the support. The first layer top bars which run the full length of the cantilever are 'bobbed' at the free end of the cantilever.

The sections which are offered have been chosen on the assumption that there will not be more bars in the second layer than the first, and that generally where the number of bars is in excess of three then there will be a minimum of two of each kind of bar. It is considered that the choice offered covers the majority of cases which are likely to be required.

3.12.4.2 Bottom support bars

A choice of two details is available showing one layer of bars or two. In each case, however, only the outermost two bars in each layer are shown on section BB and any additional bars must be added. The number of bottom bars is not preprinted on the elevation, and this figure must be added manually to be compatible with the number of bars in section BB.

3.12.4.3 Stirrups and span bars in section

The user has the option of closed stirrups or open stirrups with the closers along the whole length of the cantilever.

The bar marks for the closers, nominal span bars and stirrups are preprinted on the section, and in addition the number of nominal span bars is preprinted for inclusion on the elevation.

As only one cross-section is taken through the span, if two stirrup zones are used, the additional zone is not individually shown in section. Hence in this zone only stirrup patterns adjacent to the support may be used. However, as the stirrup diameter and the spacing may be varied, in practice this presents no restriction.

3.13 Component detailing using LUCID

The Building Research Establishment's paper titled 'Working drawings in use' reports a study of working drawings which assesses to what extent they provide the technical information needed by site staff in order to build.

The LUCID 'component' details must be related to one another by a 'location' drawing. The BRE paper considers the referencing between any set of drawings and recommends:

- The set of drawings should have a systematic structure comprising separate groups of location, schedule, assembly and component drawings. Within each of these groups individual sheets are further classified as necessary on the basis of the elements of the building which they describe. This can be achieved by using the CI/SfB element categories.

- The size of drawing sheets from all sources should conform to international 'A' dimensions. The same sheet size should be used for all drawings which form a group which are to be stored together. (All LUCID details and schedules are A4 size.)

- Location drawings should be contained on sheets which are large enough to minimise fragmentation of overall plans and elevations. For most projects this means A1 sheets with 1:50 or 1:100 scale views. These views have two main purposes: firstly to provide basic overall dimensions and levels, and secondly to provide a basis for references to assembly or component drawings.

- The set should incorporate references which lead the drawing user directly to individual sheets in the majority of his/her searches. (The LUCID details show: Key (location) plan, Notes, GA & Schedule drawing numbers.)

- It is unrealistic to expect referencing to meet all search requirements. Therefore to aid search where no reference is provided, each sheet should have a title which is short and explicit. (Each LUCID detail shows a single component and thus titling can be short and explicit.)

- Detailed views should include information which fixes the position of each view. Grids and controlling lines representing key reference planes, for example finished floor level, are recommended for this purpose. (Each LUCID detail shows section indicators and where appropriate grid lines and structural levels.)

- For any set of drawings to be used effectively a brief guide to the arrangement of the set is important.

3.14 Reading reinforced concrete drawings

3.14.1 Introduction

The technical guidance notes here explain the way in which reinforced concrete drawings should be read and have been extracted from " The Structural Engineer", Volume 90, August 2012, Issue 8. In several cases reinforced concrete drawings are more diagrammatic than their general arrangement counterparts and carry with them their own unique set of rules. Note that the guidance provided here is based on European codes of practice.

The rules governing the detailing of reinforced concrete is a far more complex subject and is dealt with in the IStructE publication entitled "Standard Method of Detailing Structural Concrete" (3rd edition).

3.14.2 Drawing principles

The purpose of reinforced concrete drawings is to communicate to the installer the layout of bars within concrete elements of a structure. The only dimensions provided in them are those that relate to reinforcement whose placement cannot be fixed to a clear reference point. In some instances reinforced concrete drawings are not drawn to scale. However, with the dominance of drawings that are developed using CAD, this is rarely the case. A tabulated approach is also sometimes adopted for repeatable elements that have a modicum of variety to them in terms of one or two dimensions.

All reinforced concrete drawings should be read in conjunction with general arrangement drawings, as these provide the setting out dimensions for the concrete elements themselves, in exclusion to the reinforcement within them.

3.14.3 Reinforcement drawing terminology

All bars within reinforcement drawings have their dimensional information given on a separate document known as a "bar bending schedule". This schedule lists the quantity of the bar, its length, size and shape. In order to correlate the schedule against the bars located in the drawing, each bar is given a mark that can be cross referenced against the schedule. This "bar mark" is placed within a label that is attributed to each bar in the drawing alongside other information concerning the size, steel grade, frequency and elevation within the concrete element.

The steel designation/notation defines the grade of the reinforcement within a bar call-up and is typically designated with an "H". There are other grades: A, B & C that are steel reinforcing bars with varying degrees of ductility, with "C" having the highest ductility and are denoted with a corresponding letter. Grade A reinforcement bars are cold formed and are drawn from coils. They are commonly used for shear-links as they are easy to bend into shapes that feature tight bends. They cannot exceed 12mm in diameter however and this limitation does not apply to B & C grades.

It is important to note that a reinforcement drawing without a bar bending schedule is regarded as incomplete as one cannot be read without the other. Taking each column in turn:

- "Member" describes which element of the structure the bar is attributed to
- "Bar mark" is the unique identifier of each bar per drawing
- "Type and size" is the designation/notation and bar diameter
- "No. of members" is the number of elements this bar is located within
- "Total number" is the number of bars denoted with this bar mark occurs within the structure
- "Length of each bar" is the total length of the bar given in mm to the nearest 25mm
- "Shape code" is a code given to certain bent shapes of bars as defined in Table 3 of BS 8666:2005 Scheduling, dimensioning, bending and cutting of steel reinforcement for concrete - Specification
- "A to E" are dimensions stated to the nearest 5mm that need to be specified for shape codes in accordance with Table 3 of BS 8666:2005
- "Revision letter" is the revision of the bar bending schedule.

When determining the length of a bar, it must be carried out in accordance with the shape code as per Table 3 of BS 8666:2005.

When calculating the lengths of bars it is important to take into account tolerances and to allow for concrete cover. Clause 10.8.1 in the National Specification of Concrete Structures provides guidance on this. In summary: a tolerance of -10mm is allowed for when assessing distances between faces of concrete that are up to 150mm thick and -15mm for elements that are more than 400mm thick. In addition, it is necessary to assume a 10mm reduction in the specified concrete cover, which is the thickness of concrete to the surface of reinforcement.

3.14.4 Beam reinforcement drawing

Beam reinforcement drawings are amongst the simplest of the elements to create drawings for. They provide much of the required information diagrammatically and require little in the way of unique terminology within the bar mark call-ups. What does need to be shown clearly is the extent of the bars and how they lap with one another. This is done by showing marker arrows with a bar mark number showing how far one bar laps with another. Some bars need to be set out from a common point, typically the centreline of a support in order to locate them along the length of a beam. This is because such bars are installed to resist bending moments in the top section of a beam and therefore must be placed in such a way to cover the extent of the tension in the upper section of the beam.

3.14.5 Column reinforcement drawing

Column reinforcement can be more diagrammatic than other reinforcement drawings. They show the primary reinforcement together with dashed lines to indicate the extent of starter bars, which is the reinforcement that projects from the kicker and extent of containment links. A section is also taken through the column to indicate how the bars within it are laid out.

3.14.6 Floor slab reinforcement drawing

Reinforced concrete floor slab drawings tend to be the most complex of elements to draw. This is especially true of flat slabs due to the need to create concentrated sections of reinforcement that act as beams within the slab. With a minimum of four layers of reinforcement to plot and the curtailment of the bars needing to be carefully plotted out, it is common to find drawings for slabs becoming too cluttered to read. In some cases therefore it is preferable to create two separate plans for the upper and lower layers of reinforcement. Floor slab drawings also carry with them their own form of nomenclature for the level at which the bars are located within the slab.

"Extent indicators" are key to showing bars on a floor slab drawing. These indicators show the area that a bar occupies as well as the centres they are placed at. In instances where a bar shape is the same but has varying lengths, rather than having a bar mark for each bar, a variant marked with a letter is used. This prevents a mesh of bars being drawn on the slab.

3.14.7 Wall reinforcement drawing

There are similarities between floor slab and wall drawings in that they use a unique marker to indicate where the reinforcing bars are located within the concrete element.

In some instances the use of N1, N2 and F1, F2 is used in a similar fashion to floor slab reinforcement. Walls, like columns, have kickers within them and as such all reinforcing bars are set out with respect to the existence of the kicker. Other than the presence of the kicker, there is very little difference between a drawing for a reinforced concrete wall and a floor slab.

3.14.8 Applicable codes of practice

The applicable codes of practice for reinforced concrete detailing are as follows:

- BS EN 1992-1-1: Eurocode 2: Design of Concrete Structures. Part 1-1 General Rules and Rules for Buildings

- BS EN 1992-1-1: Eurocode 2: UK National Annex to Design of Concrete Structures. Part 1-1 General Rules and Rules for Buildings

- BS 8666:2005: Scheduling, dimensioning, bending and cutting of steel reinforcement for concrete - Specification.

3.14.9 Further reading

The Institution of Structural Engineers (2006) "Standard Method of Detailing Structural Concrete" 3rd edition.

Mineral Products Association (2010) "National Structural Concrete Specification" 4th edition.

4. SPADE User's Manual

4.1 About SPADE

4.1.1 Scope

SPADE is software for producing steelwork and other details for a variety of structural components.

SPADE incorporates a library of proforma details, any of which may be selected for use when SPADE is run. The content of this library is continually under review, new details being added or modified as codes of practice develop and change.

The output from SPADE is an A4 drawing which may be sent directly to a printer, or saved as a pdf file. Prior to printing, the SPADE detail may be translated into a dxf file for inclusion in any other CAD system.

4.1.2 Advantages of SPADE

SPADE scores over other steelwork detailing systems in that the user does not have to draw the detail. SPADE is an 'expert system' which knows about detailing and builds the detail using the knowledge contained within the SPADE database. The engineer merely has to guide SPADE by giving answers to the various questions which SPADE asks.

4.1.3 Operation of SPADE

To check that the software is operating correctly, select SPADE option 410 at the satrt of SCALE. This is for a flexible end plate connection, and (because it is simple) ideal for test purposes.

SPADE now reads the proforma detail for the connection. After the proforma detail has been read, accept Page 1 as the start page number and accept all the default values offered. When the detail has been completed the screen displays the detail on the screen.

4.2 Use of files

4.2.1 The data file

Page headings - comprising firm's name, address and job information - remain substantially unchanged for the duration of a project. This information is held in a data file - with name ending in the extension .DAT. An existing data file may be nominated, or a new one created. The information is stored on disk, a typical data file C702.DAT - as supplied - contains:

```
TITLE N G NEERS AND R K TECTS CO PARTNERSHIP
TITLE 101 HIGH STREET  PEVERILL  DORSET
TITLE JOB: NEW CIVIC CENTRE
TITLE      ANCILLARY BUILDING
MADEBY DWB
DATE 27.10.15
REFNO 95123
```

4.2.2 The proforma detail file

All SPADE proforma details have a file name starting with 'sp' followed by three digits. The three digits correspond to the option number used to select the proforma.

4.2.3 The stack file

When SPADE is terminated in a normal manner, the responses typed in by the user to replace the ???? prompts are not lost; but are saved in a file of the same name as the proforma file but with extension .stk (standing for stack of values). For example after running the proforma sp410, the stack of values last used would be found in a new file named sp410.stk.

When SPADE is restarted and proforma 410 is again requested, then providing the engineer refuses the example defaults, those values previously given will be offered. The .stk file thereby saves the user the need to retype data.

4.2.4 The finished detail file

In general, SPADE prompts for a file name which has the extension .dat and creates a new file of the same name but with .pdf as its extension which contains the finished detail written in HPGL.

The headings at the top of the finished detail are given headings copied from the data file (.dat). The sole purpose of the data file is to provide such headings. The page number, however, is not copied from the data file but given independently so that it may be changed easily for each detail.

The page number may have an upper case letter prefix e.g. FSL/3. Each time an option is selected the previous finished detail (held in the .pdf file) is overwritten. On exit from SPADE the .pdf file will contain the last finished detail produced for the job selected by the .dat file.

4.3 The library of proformas

The library of proforma details is continually under review.
This section contains a brief description of each SPADE proformas.

TIMBER

sp202 Timber connections. This option will detail beam to column connections of the following types:

- U-plate
- concealed fixing
- single plate
- T-plates on both sides.

sp250 Timber beam splice. This option will detail spliced joints of the following types:

- timber or steel splice on one side
- timber or steel splice on two sides.

sp252 Timber joists bearings on steel beams. This option will detail timber joists bearings on steel beams of the following types: continuous timber supported on top flange; timber on one side supported on top flange; timbers overlapping supported on top flange; timber on both bottom flanges; timber on one bottom flange.

sp265 Fan truss general arrangement. This option will detail fan trusses having four or six top bays. With each of the following options the basic truss setting out details will be shown on the elevation:

- full outline of truss members with member offset dimensions
- partial outline of truss members with member offset dimensions
- centre line of truss members only.

sp266 Fink truss general arrangement. This option will detail Fink trusses having 'equal bay joists' or 'four equal top panels'. With each of the following options the basic truss setting out details will be shown on the elevation:

- full outline of truss members with member offset dimensions
- partial outline of truss members with member offset dimensions
- centre line of truss members only.

sp267 Howe truss general arrangement. This option will detail Howe trusses having four top bays. With each of the following options the basic truss setting out details will be shown on the elevation:

- full outline of truss members with member offset dimensions
- partial outline of truss members with member offset dimensions
- centre line of truss members only.

sp268 Pratt truss general arrangement. This option will detail Pratt trusses having four equal top and bottom bays. With each of the following options the basic truss setting out details will be shown on the elevation:

- full outline of truss members with member offset dimensions
- partial outline of truss members with member offset dimensions
- centre line of truss members only.

sp269 Queen post truss general arrangement. This option will detail Queen post trusses. With each of the following options the basic truss setting out details will be shown on the elevation:

- full outline of truss members with member offset dimensions
- partial outline of truss members with member offset dimensions
- centre line of truss members only.

PORTAL CONNECTIONS

sp310 Portal eaves haunched connection - flush top. This option will detail portal eaves haunched connections which have a flush top.

sp320 Portal apex haunched connection. This option will detail a portal apex haunched connection. This option assumes that the haunch is cut from the same section as used for the rafter.

END PLATE CONNECTIONS

sp410 Flexible end plate connection - beam to beam. This option will detail flexible end plate beam to beam connections of the following types:

- one beam supported by another beam
- two beam supported by another beam.

sp412 Flexible end plate connection - beam to column. This option will detail flexible end plate beam to column connections of the following types:

- one beam connected to column flange
- one beam connected to column web
- two beams connected to column web.

sp416 Extended end plate connection - beam to column. This option will detail extended end plate beam to column connections.

ANGLE CLEAT & ANGLE SEAT CONNECTIONS

sp420 Double angle cleat - beam to beam connection. This option will detail double angle cleat beam to beam connections of the following types:

- one beam supported by another beam
- two beams supported by another beam.

sp422 Double angle cleat - beam to column connection. This option details one or two beams connected to the supporting column. The principal axes of all members lie on a common vertical plane. If two beams are to be connected to the column then the upper flange levels must be the same.

sp430 Angle seat - beam to column connection. This option will detail angle seat beam to column connections of the following types:

- one beam connected to column flange
- one beam connected to column web
- two beams connected to column web.

FIN PLATE CONNECTIONS

sp440 Fin plate connection - beam to beam. This option details one or two beams connected to a supporting beam by fin plates. If there are two beams to be connected, the principal axes must lie on a common vertical plane. The upper flange levels of all beams must be common.

sp442 Fin plate connection - beam to column. This option details one or two beams connected to a supporting column by fin plates. If there are two beams to be connected, the principal axes must lie on a common vertical plane. The upper flange levels of beams must be common.

MOMENT CONNECTIONS

sp450 Tongue plate connection - beam to column. This option will detail a beam to column connection using a tongue plate.

sp452 Direct welded connection - beam to column. This option details one or two beams directly welded to a supporting column. The principal axes of all members must lie on a common vertical plane. If two beams are to be connected then the upper flange levels must be common.

sp454 Tee connection - beam to column. This option will detail a beam to column connection using Tees at top and bottom of the beam.

sp458 Beam stub connection. This option will detail a beam stub connection.

MISCELLANEOUS CONNECTIONS

sp460 Beam over beam connection. This option will detail a beam over another beam connection.

SPLICES

sp490 Beam splice. This option will detail flange and web connecting plates for a beam splice.

sp492 Column splice. This option will detail flange and web connecting plates for a column splice.

COLUMN BASES

sp510 Column base plate. This option will detail a column base plate and holding down bolts.

sp512 Welded gusset column base plate. This option will detail a welded gusset column base plate and holding down bolts.

SHOP DETAILS

sp540 Shop details - non skew beam - drilled. This option will produce a shop detail for a non skew beam with groups of holes along its length and ends, and optionally top and bottom end notches.

sp550 Column shop details (drilled). The proforma will draw shop details for UB and UC sections. Columns could extend over 1 or 2 storeys provided even number of bolts at each level for flange bolt holes and web bolt holes are used.

FIRE CASINGS

sp580 Fire encasement (timber framing - up to 1 hour). This option will detail beam and column fire casings with timber framing for up to a 1 hour fire resistance.

sp582 Fire encasement (steel angle framing - up to 2 hours). This option will detail beam and column fire casings with steel angle framing for up to 2 hours fire resistance.

LOCATION DRAWINGS

sp590 Location drawing - single bay portal frame. This option will produce a location drawing for a double pitched portal frame, showing drawing reference numbers for eaves, apex and base plate connections.

sp592 Location drawing - multi-storey frame. This option will draw an elevation for a multi-storey frame, showing section sizes for beams and columns, together with other reference information.

MASONRY

sp605 Raft Foundation - wide toe. This option will detail a wide toe foundation with reference to Figure 5 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp608 Raft foundation - deep edge beam. This option will detail a deep edge beam to a raft foundation with reference to 'Structural design of masonry' by Andrew Orton Figure B2.11.

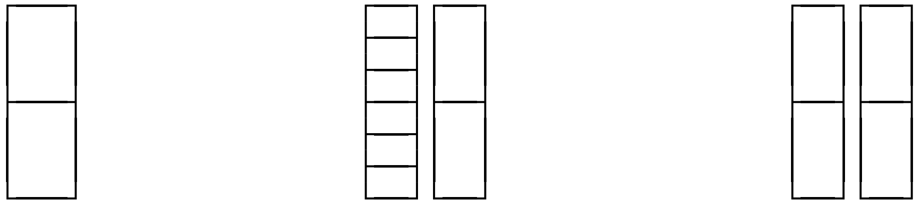
sp610 Raft foundation - plain edge detail. This option will detail a plain edge to a raft foundation with reference to Figure 6 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp612 Raft foundation - plain internal wall support detail. This option will detail a plain internal wall support with reference to Figure 6 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp613 Bored pile foundation to resist uplift. This option will detail a bored pile foundation to resist uplift in swelling clay conditions with reference to Figure 11 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp614 Bored pile foundation. This option will detail a bored pile foundation with reference to Figure 8 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp615 Masonry walls. This option will detail masonry walls of the following types:



solid block brick/block with cavity block with cavity

sp616 Trench fill foundation. This option will detail a trench fill foundation with reference to 'Structural design of masonry' by Andrew Orton Figure B2.7.

sp618 Traditional strip foundation with concrete floor. This option will detail a traditional strip foundation with reference to Figure 1(a) of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp619 Traditional strip foundation with suspended timber floor. This option will detail a traditional strip foundation with reference to Figure 1(b) of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp620 Wide strip foundation. This option will detail a wide strip foundation with reference to Figure 2 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp622 Pad and stem foundation for loose fill. This option will detail pad and stem foundations for houses sited on loose fill or soft compressible soils with reference to Figure 10 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp624 Precast driven segmental pile foundations for loose fill soft soils. This option will detail precast driven segmental pile foundations for houses sited on loose fill, soft compressible soils and open heavy clay with reference to Figure 9 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp630 Granular layer beneath slab venting through trench. This option will detail a granular layer beneath slab venting through trench foundation with reference to Figure 6(a) of 'Foundations for low-rise buildings' by RMC Driscoll, MS Crilly & AP Butcher, published in 'The Structural Engineer' June 1996.

sp632 Granular layer beneath slab venting through trench with riser. This option will detail a granular layer beneath slab venting through trench with riser foundation with reference to Figure 6(b) of 'Foundations for low-rise buildings' by RMC Driscoll, MS Crilly & AP Butcher, published in 'The Structural Engineer' June 1996.

sp634 Granular layer beneath slab venting through slotted pipe with riser. This option will detail a granular layer beneath slab venting through slotted pipe with riser foundation with reference to Figure 6(c) of 'Foundations for low-rise buildings' by RMC Driscoll, MS Crilly & AP Butcher, published in 'The Structural Engineer' June 1996.

GENERAL

sp801 Graph plot program. This option will accept X and Y coordinates defining a function and plot the shape of the function.

4.3.1 Default values

Each option starts by asking you if you want a set of default values. These are usually taken from a published design example. The purpose of the default values is to allow you to press Return at each and every prompt to see what data is required to produce the SPADE detail thereby making every option into its own User's Manual.

4.3.2 References

During the development of SPADE the following publications were consulted:

- Manual on connections for beam and column construction to BS449 by JW Pask; published BCSA.
- Manual on connections - volume 1 - joints in simple construction to BS5950 by JW Pask; published BCSA.
- Metric practice for structural steelwork; published BCSA.
- Plastic Design of Low-Rise Frames by MR Horne and LJ Morris; published by Constrado (now SCI).
- Steel Designers' Manual; published by BSP.
- Steelwork Design Guide to BS5950; published by SCI.
- Steel Detailers' Manual by Alan Hayward and Frank Weare; pub by BSP.
- Structural Steelwork Design to BS5950 by LJ Morris and DR Plum; published by Longman Scientific & Technical.
- Verification of design methods for finplate connections by DB Moore and GW Owens; published The Structural Engineer - Feb 1992.

If you feel that your own particular preferred detail should be included in SPADE please let Fitzroy have a copy of it.

4.4 Using SPADE

Generally yellow text on a blue background denotes information used to build the detail; black text on a green background denotes help information only.

The proforma details usually offer a set of default values. Default values are values which are provided so that you can go through the calculation and see what information is needed without having to type in sensible values yourself. Accept the default values by pressing Return as many times as necessary to get you through the detail to the end. If you wish to change one of the default values, edit or replace the value offered before pressing Return.

See SCALE User's Manual for further information on usage.

4.5 Component detailing using SPADE

The Building Research Establishment's paper titled 'Working drawings in use' reports a study of working drawings which assesses to what extent they provide the technical information needed by site staff in order to build.

The SPADE 'component' details must be related to one another by a 'location' drawing. The BRE paper considers the referencing between any set of drawings and recommends:

- The set of drawings should have a systematic structure comprising separate groups of location, schedule, assembly and component drawings. Within each of these groups individual sheets are further classified as necessary on the basis of the elements of the building which they describe. This can be achieved by using the CI/SfB element categories.

- The size of drawing sheets from all sources should conform to international 'A' dimensions. The same sheet size should be used for all drawings which form a group which are to be stored together. (All SPADE details and schedules are A4 size.)

- Location drawings should be contained on sheets which are large enough to minimise fragmentation of overall plans and elevations. For most projects this means A1 sheets with 1:50 or 1:100 scale views. These views have two main purposes: firstly to provide basic overall dimensions and levels, and secondly to provide a basis for references to assembly or component drawings.

- The set should incorporate references which lead the drawing user directly to individual sheets in the majority of his/her searches. (The SPADE details show: Key (location) plan, Notes, GA & Schedule drawing numbers.)

- It is unrealistic to expect referencing to meet all search requirements. Therefore to aid search where no reference is provided, each sheet should have a title which is short and explicit. (Each SPADE detail shows a single component and thus titling can be short and explicit.)

- Detailed views should include information which fixes the position of each view. Grids and controlling lines representing key reference planes, for example finished floor level, are recommended for this purpose. (Each SPADE detail shows section indicators and where appropriate grid lines and structural levels.)

- For any set of drawings to be used effectively a brief guide to the arrangement of the set is important.

5. NL-STRESS User's Manual

5.1 Introduction to NL-STRESS

NL-STRESS is a computer program for the elastic, plastic, and stability analysis of skeletal structures.

To select and run one of the supplied files, select the desired file from the opening screen.

For large structures having several hundred or several thousand joints, an ability to write the data directly, in the NL-STRESS language, becomes essential, and once mastered provides the simplest means of changing an existing file to suit a new problem.

Given a description of a plane or three-dimensional structure, and the loads applied to it, NL-STRESS can be made to analyse the structure by:

- conventional elastic analysis
- non-linear elastic analysis allowing for elastic instability
- elastic-plastic analysis under a given loading condition.

For years the individual members of structural frames have been designed on the basis of ultimate strength (the term 'design' is used here to mean deciding upon shapes and dimensions of cross sections). But structural frames themselves have continued to be analysed on assumptions of perfect elasticity and 'short' stable members. To consider ultimate behaviour of members in a frame which behaves elastically is an evident contradiction.

NL-STRESS has been written as a practical software package for non-linear analysis, its results may be based on elastic or ultimate-load theory according to the engineer's requirements. In particular there is no further need to accept the strange contradiction explained above.

The rest of this section discusses the points raised above in more detail.

5.1.1 Types of analysis

The assumptions inherent in the three possible types of analysis are now explained:

In elastic analysis the material from which the members are made is assumed to obey Hooke's law and not to yield. All members are assumed short enough to ignore the Euler effect of reduced stiffness under axial compression. Displacements induced in the frame as a whole are assumed small enough to justify ignoring changes in structural geometry (sway effect).

In non-linear elastic analysis the material from which members are made is assumed to obey Hooke's law and not to yield. The software applies the loading in small increments, correcting structural geometry at each increment. Thus the sway effect is catered for; in addition, the Euler effect in individual members may be catered for by dividing each member into a number of segments.

The plastic method of analysis incorporates the sway and Euler effects. Loading is applied in increments. After each increment the structural geometry is corrected and the members checked for the formation of plastic hinges. Further increments of loading are applied in this manner until the structure receives its full factored load.

There are three possible kinds of outcome:

- no plastic hinges: the results would be the same as for a non-linear elastic analysis
- total collapse: results are given for the loading just prior to collapse
- partial collapse: the results would locate those plastic hinges which would form under the specified loading condition.

5.1.2 Types of structure

Structures that may be analysed by NL-STRESS are skeletal in form, comprising straight uniform 'members' connected at their ends to form 'joints'. There are five possible 'types' of structure conforming to the above description:

- plane trusses, in which all joints are assumed to be frictionless hinges. Loads are confined to the plane of the structure
- plane frames, in which joints are assumed to be rigid unless hinges are specifically introduced at the ends of selected members. Loads are confined to the plane of the structure: moments act about axes normal to this plane
- plane grids, in which joints are assumed to be rigid unless hinges are specifically introduced at the ends of selected members. Loads act normal to the plane of the structure: moments act about axes lying in the structural plane
- space trusses, in which all joints are assumed to be

frictionless ball joints. Loads act in any direction

- space frames, in which joints are assumed to be rigid unless hinges are specifically introduced at the ends of selected members. Loads and moments act in any direction.

Typical structural problems which may be solved by modelling as one of the above structural types are:

- arches, balconies, bridge decks, continuous beams, culverts, deep beams
- dolphins: mooring, berthing and turning
- floor slabs, influence lines, multi-storey frames
- plates in bending and extension
- portal frames: single, multi-bay and irregular
- prestressing effects
- raft foundations and beam on elastic foundations
- shear walls and apportionment of load between frames and shear walls
- staircases: spiral, cantilever and dog-leg
- towers: radio masts, flare stacks, radar and pylons
- thermal effects
- trusses: couple, couple-close, collar-tie, Fink, Warren, king and queen post, valley and Mansard.

5.1.3 About this manual

This manual explains how to prepare data for analysis by NL-STRESS and how to interpret the results obtained.

Section 5.2 explains briefly the principles of linear and non-linear analysis, explains the principle of consistent units of measurement, explains the sign conventions to be used in the data and the interpretation of signs in the tables of results.

An introductory example is provided in section 5.3.

Section 5.4 defines the basic elements of a set of data and introduces terminology used throughout the rest of the manual.

Certain features - such as the ability to write an item of data as an arithmetical expression, or store a value for subsequent use - may be new to many readers. These features make preparation of data less tedious than was previously possible.

A notation for defining the composition of 'commands' and 'tables' from basic elements is explained in section 5.5. An understanding of this notation is useful when wondering whether such-and-such an arrangement of keywords and numbers would be permissible or not. Section 5.5 may be skipped on first reading but there is no reason to do so; the notation is explained in simple terms, and when understood, enables the full power of the software to be exploited.

Section 5.6 defines the order in which commands and tables should be assembled.

Section 5.7 - the biggest - defines and explains in detail every command and table that may be included in a set of data. A quick reference to section 5.7 is provided by section 5.8.

Section 5.9 lists error messages.

5.1.4 Operation of NL-STRESS

From the opening screen select an NL-STRESS file to open, make any changes required to the data file, then tap on the forward button to analyse the structure. When there are errors the results file is seen to contain nothing but error messages. The remedy is to run NL-STRESS for another try. When the analysis is successful NL-PLOT may be run to display or print: bending moments, shear forces or joint displacements.

Some structural analysis software claims to do an equilibrium check, but assumes that all deflections are negligible and consequently only check the accuracy of the arithmetic carried out by the software. A proper overall equilibrium check must take into account the displaced positions of all the applied loads and in general must satisfy: ΣX=0 ΣY=0 ΣZ=0 ΣMX=0 ΣMY=0 ΣMZ=0. NL-STRESS puts the applied loads on the displaced geometry and thus properly compares the applied loading with the computed reactions. It is important to inspect the EQUILIBRIUM CHECK; normally the check will show that the sum of forces balances the reactions to within one percent. If the check shows an imbalance greater than one percent, then non-linear analysis may be appropriate.

When NL-STRESS is run, it produces two new files, overwriting any existing files having the same name. Both files have the same name as the data file but each has a different extension viz:

- .arr (standing for arrays) holds the structure stiffness matrix, joint coordinates and member incidences and other arrays necessary for the production of bending moment diagrams etc. The .arr file is a binary file which cannot be viewed with a text editor, it is used internally by SCALE to facilitate the pull-through of results to many of the SCALE proformas (those identified with a # in the menu.

- .pdf results file, contains the joint deflections, member forces and support reactions.

5.1.5 NL-STRESS benchmarks

NL-STRESS Benchmarks give examples of NL-STRESS data files for a wide range of engineering structures. The purpose of providing the files is threefold:

- So that each of the benchmarks may be run and the results obtained compared to those embedded in the data, the embedded results were obtained by Fitzroy running their version of NL-STRESS.
- To allow the engineer to test their software against the standard Department of Transport benchmarks. NL-STRESS is Department of Transport approved program reference MOT/EBP/254C.
- Two give examples of how to write out NL-STRESS data for plastic analysis problems, dynamic problems etc.

5.1.6 NL-STRESS proforma data files

NL-STRESS may be used by selecting one of the NL-STRESS proforma data files, running through the usual SCALE question and answer to produce a data file for your structure. You may then open this data file in the NL-STRESS editor to add extra data, and duplicate load cases.

This manual describes the use of variables and expressions and 'looping' (REPEAT-UNTIL-ENDREPEAT) in an NL-STRESS data file; by such use it is simple using any text editor to set up data files in parametric form, to cover frequently analysed structures e.g. portals. For structures which are 'one off', the proforma nls.dat may be edited to produce a data file; proforma nls.dat contains only the commands and headings of the NL-STRESS language.

5.1.7 NL-STRESS parametric data files

For many of the more complicated structural types e.g. multi-storey frames, the NL-STRESS proforma will generate a parametric data file, so for example the number of bays and storeys become parameters of the structure. These parameters can be quickly edited in the NL-STRESS editor without the need to re-run the proforma.
The provision of parametric data has advantages:

- from the values supplied, the engineer recognises the 'units'
- from the values supplied, the engineer gets a 'feel' for the data
- the default data provided allows all parametric data files to be run as a batch and the ckecksum of all the runs in the batch used to find if any software modifications have affected the results.

There are several ways in which data may prepared for analysis by NL-STRESS. Parametric data generation is the most fundamental, and most powerful, in which the data is written in terms of parameters, and edited to suit the structural problem.

Parametric data allows the engineer to change the parameters e.g. the number of bays, height between chords, section properties etc. and thereby obtain a least weight design. To assist in the process of obtaining the least weight, a MEMBER SELF WEIGHTS loading will give the total weight in the equilibrium check. Theoretical approaches to the least weight problem do not take into account the engineering of the problem e.g. that a certain section depth cannot be increased, or that section sizes come in steps rather than a continuous function. The use of parametric data allows the engineer to vary those parameters which can vary, and hold constant those which cannot.

It is not necessary to understand the logic of a data file - in which the data is written parametrically - before it is used. There is no need to delete any of the lines which start with an exclamation mark which provide help, such lines are automatically removed from the results. Similarly, there is no need to remove any lines which follow the FINISH command.

When you have finished typing the data, tap the forward button to run NL-STRESS. If there are errors in the data, NL-STRESS will list the errors together with the line number and the line following the number. Go back and correct the data. After a successful analysis, continue to NL-PLOT, and display the bending moments on screen.

5.1.8 NL-STRESS verified models

Many of the NL-STRESS proformas combine parametric data with self checking. Between the SOLVE and FINISH commands of the .dat file generated by the proforma, comes a self check followed by a percentage comparison of results computed by NL-STRESS with those of the self check.

These "verified models" have the following aims:

- to avoid major disasters such as those described in the IStructE 'Guidelines for The Use of Computers for Engineering Calculations'
- to give assurance to the engineer that the numbers computed are OK
- to provide categorical results for any meetings which compare the results with those obtained from other modelling systems
- to provide an expert engineer in the form of specific advice given with each model
- to provide a system which is easy to maintain after the enthusiasm present at its development has waned
- to reconcile classical analysis with modern matrix methods for: linear-elastic, sway and within-member stability, rigid-plastic, non-linear elastic-plastic analysis, i.e. to provide bedrock between classical analysis and modern matrix methods
- to be useful to engineers with each model being immediately recognisable as a structural form
- to provide simple data as engineers are rightly suspicious of complexity
- to provide a common structure, so that any engineer who can use a text editor, can type: spans, section sizes, strengths etc. and run any model, thereby avoiding the time-waste and mistakes associated with starting with a blank sheet of paper.

5.1.9 Including diagrams in the data

One picture is worth a thousand words; it is helpful when an engineer finds an old data file, if that file includes a diagram of the structure.

Diagrams can be included in an NL-STRESS data file by using Graphics Characters. Their use has the advantage that 'scaling' to fit is avoided.

5.1.10 Including calculations and character strings in the data

NL-STRESS combines two languages:

- the NL-STRESS high level language which consists of MIT STRESS keywords such as: JOINT COORDINATES, MEMBER INCIDENCES, JOINT LOADS, MEMBER LOADS et seq. greatly extended.
- a programming language similar to PRAXIS, which provides 'looping' by the programming structure REPEAT-UNTIL-ENDREPEAT, and conditionals such as: IF... ENDIF; IF... THEN; IF... GOTO... etc.

This section 5.1.10, and section 5.11.2, describe the uses of the 'sense' switch, which is used to modify the behaviour of NL-STRESS.

Often values used in the data have been computed on a calculator, it is helpful if the data file shows how the values were computed.

NL-STRESS treats a line which start with an asterisk as a comment line which is to be included in the data file (normally printed at the start of the results) but does not normally contribute data for the structure. NL-STRESS does not ignore text after an asterisk; when it writes the results it substitutes numerical values for any expressions or assignments starting with a plus sign to make the data more meaningful for the engineer and checker.

If NL-STRESS finds an expression on a comment line e.g.

```
     * Deflection at joint 2  +q*l^4/(1024*e*iy)
```

it evaluates the expression and assuming the parameters have been set, prints the numerical result in the results file in place of the expression.

If NL-STRESS finds an assignment on a comment line e.g.

```
     * Deflection at joint 2  +d2=q*l^4/(1024*e*iy)
```

it deletes the plus sign, and copies the assignment to the results, followed by answer, replacing the assignment e.g.

```
     * Deflection at joint 2  d2=q*l^4/(1024*e*iy)=.3488
```

Of course the clever bit is that when the engineer changes the parameters, the values computed for any expressions or assignments will be amended automatically.

The engineer may include as many lines commencing with an asterisk as are necessary to explain the data, calculations and results for the benefit of the checker in the short term and the engineer in the long term. On occasions, it is desirable to store and retrieve text, especially when including post-processing between the SOLVE and FINISH commands. Any text string may be stored by assigning it using the $() pseudo-function e.g. a(1)=$(FORCE X) causes FORCE X to be hashed into a number and stored in a(1). To hash the string 'FORCE X' into a number, base 39 arithmetic is used with digits 0-9 valued as 1-10, upper-case letters A-Z as 11-36, lower case letters a-z as 11-36, +=37, .=38, remainder=39, a maximum of 9 characters may be hashed into a single variable. If the line:

```
* +a(1)
```

were included among the data or post-processing, then the hashed value would be printed; but if the line:

```
* $a(1)
```

were included among the data or post-processing, then

```
* FORCE X
```

would be printed; the prefix $ tells NL-STRESS to convert the hashed value back to text form. Of course, if we always wanted * FORCE X to be included in the data, we would not bother with storing it using $(). The purpose of $() is to allow us to store say:

```
a(1)=$(FORCE X) a(2)=$(FORCE Y) a(3)=$(MOMENT Z)
```

optionally shortened to: a1=$(FORCE X) a2=$(FORCE Y) a3=$(MOMENT Z) and then in a loop determining maximums, we can have a line such as

```
* Maximum $a(n) is +f(n)
```

for all values of n. The post-processing file 'wring.ndf' gives practical examples of the usage of $. When displaying the percentage difference between two values, it is tiresome to see a column of zeros, or a column of numbers with E exponent when there is an bad error. The logic below omits the number if it is zero, and if the percentage exceeds 99%, then prints BAD ERROR. The three lines below loop for percentages p=0 to 102, and prints them as a number (for a check) and then as the string $ok.

```
p=-1
:39
p=p+1
d1=INT(p/10) d2=p-d1*10
IF d1=0 THEN d1=-1
IF p<100 THEN ok=(d1+1)*63+d2+1
IF p>99 THEN ok=$(BAD ERROR)
IF p<1 THEN ok=0
* +p           $ok
IF p<102 GOTO 39
```

Either upper or lower case may be used in any string but not both. If a mixture of upper and lower case is given, the case is assumed as that of the first character within the brackets.

Substitution of values for variables preceded by a + as described above, generally uses the F format e.g. -12.46 for real numbers, E format for extreme numbers e.g. -1.0E+12, integer numbers being shown as integers. Occasionally, in a set of numbers it is tidier to output results of real numbers e.g. -23.6 in E format. To do this, include the assignment 'sense=1' to switch on E format, and 'sense=0' to switch it off. (Sense is used in memory of the 'sense switches' of yore.) As an example, if h=-23.6 & fsc=28 then the line:

```
* h= +h fsc= +fsc    will be shown in the results as:
* h= -23.6 fsc= 28   but if: sense=1  is set on a previous line as
* h= -0.23600E+02 fsc= 28
```

For convenience a list of 'sense=' follows:

```
sense=1  Keep to E format
sense=2  Ignore shear deformation for ISECTION
sense=3  Ignore Michael Horne's Q forces, sense=-3 to ignore
         Q forces but only if member is a cantilever
sense=4  ISIZF=1 & NCYC=MYC terminates analysis at current loading
         increment
sense=5  Dump of main stack
sense=6  Cause 'IF... THEN false' to be included in results.
```

5.1.11 General notes

NL-STRESS keywords such as COORDINATES should be in upper-case, symbolic names for variables must start with a lower case letter, the names can be of any length but only the first six letters are significant. Shorter names need less typing and are more easily used in assignments and remembered.

A line starting with an asterisk is a comment line which is included in the results if the keyword DATA follows the PRINT command. A line starting with an exclamation mark is a comment line which is ignored.

Characters which follow an isolated exclamation mark i.e. an exclamation mark which has a space before it and a space after it, are ignored save for including them (but not the exclamation mark) in the results if the keyword DATA follows the PRINT command. Generally, parameters precede ! and help follows.

An assignment e.g. nj=nm+1 (No. of joints = No. of members + one) is treated as an assignment only and does not contribute an item of data; thus in the statement: NUMBER OF JOINTS nj=nm+1 +nj the extra item nj is needed to contribute the number of joints. The plus sign in front of nj tells NL-STRESS to print the value of the expression which follows, assuming that the variable nm held the value 4, then the results would show:
NUMBER OF JOINTS nj=nm+1 5

Where it is desired that decimal points should line up in a table then include a double plus sign (++) in front of the variable. A fixed field of 12 characters is provided, the number being right adjusted in the field, then any trailing zeros removed, and if all figures after the decimal point are zero, the decimal point is also removed.

When there is insufficient space to contain all the data items on one line, subscripted variables may be written singly without their brackets, ax4 is identical to ax(4); but when used within a loop the brackets must be shown (obviously writing axi would be ambiguous, as it could be interpreted as ax(i) or a(xi)).

A # as the first character in a line causes the import of data from the named file which follows, e.g. #cc924.stk causes any data contained in the file cc924.stk to be imported, and thus re-assign any default values (i.e. those previously set for the parameters) to new values. This facility permits a thousand engineered sets of data to be analysed in a batch, thereby testing all sensible values of each parameter with all sensible values of every other parameter.

5.1.12 Introductory example

This section shows an introductory example.

- Some data
- Corresponding results

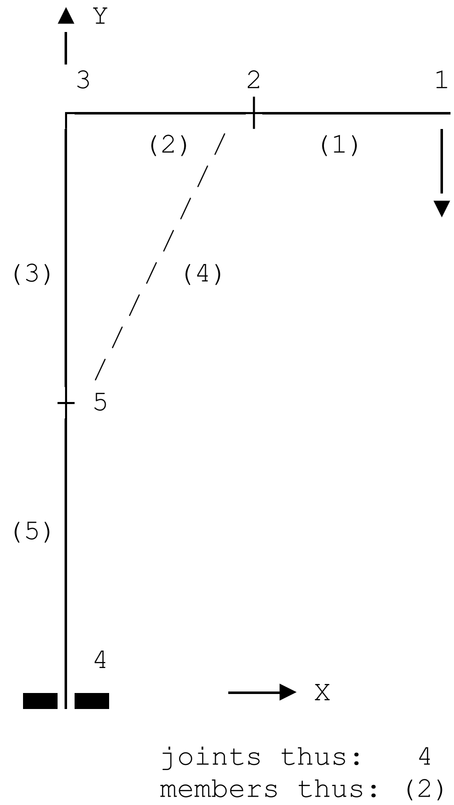
5.1.12.1 Some data

```
TITLE MANN WYFFE & PARTNERS
TITLE MEDIAEVAL JOUSTING FURNITURE
TITLE KNIGHT HOIST
MADEBY DWB
DATE NOV1145
REFNO KCMG-4ME
PRINT DATA RESULTS FROM 1
METHOD ELASTIC
TYPE PLANE FRAME

NUMBER OF JOINTS 5
NUMBER OF MEMBERS 5
NUMBER OF SUPPORTS 1
NUMBER OF LOADINGS 3

JOINT COORDINATES
1   1.8  3.2
2   0.9  3.2
3   0    3.2
4   0    0     SUPPORT
5   0    1.5
MEMBER INCIDENCES
1     2  1
2     3  2
3     5  3
4     5  2
5     4  5
CONSTANTS  E  6.6E6 ALL,  G  2.75E6 ALL
CONSTANTS  DENSITY  8.6 ALL
MEMBER PROPERTIES
1 THRU 3 RECTANGLE DY 0.35 DZ 0.2
5 AS 1
4 RECTANGLE DZ .2 DY .4

LOADING SELF WEIGHT OF HOIST
MEMBER SELF WEIGHT
1 THRU 5  1.0
LOADING LIVE LOAD
JOINT LOADS
1 FORCE Y -5.0
LOADING SELF + 150% LIVE
COMBINE 1 1.0,  2 1.5
SOLVE
FINISH
```



5.1.12.2 Corresponding results

```
MANN WYFFE & PARTNERS                                   Page:  1
MEDIAEVAL JOUSTING FURNITURE                         Made by:  DWB
KNIGHT HOIST                                            Date:  NOV1145
                                                      Ref No:  KCMG-4ME
____________________________________________________________________


TITLE MANN WYFFE & PARTNERS
TITLE MEDIAEVAL JOUSTING FURNITURE
TITLE KNIGHT HOIST
REFNO KCMG-4ME
DATE NOV1145
MADEBY DWB
PRINT DATA RESULTS FROM 1
METHOD ELASTIC
TYPE PLANE FRAME

NUMBER OF JOINTS 5
NUMBER OF MEMBERS 5
NUMBER OF SUPPORTS 1
NUMBER OF LOADINGS 3

JOINT COORDINATES
1   1.8  3.2
2   0.9  3.2
3   0    3.2
4   0    0     SUPPORT
5   0    1.5
MEMBER INCIDENCES
1     2  1
2     3  2
3     5  3
4     5  2
5     4  5
CONSTANTS  E  6.6E6 ALL,  G  2.75E6 ALL
CONSTANTS  DENSITY  8.6 ALL
MEMBER PROPERTIES
1 THRU 3 RECTANGLE DY 0.35 DZ 0.2
5 AS 1
4 RECTANGLE DZ .2 DY .4

LOADING SELF WEIGHT OF HOIST
MEMBER SELF WEIGHT
1 THRU 5  1.0
LOADING LIVE LOAD
JOINT LOADS
1 FORCE Y -5.0
LOADING SELF + 150% LIVE
COMBINE 1 1.0,  2 1.5
SOLVE
FINISH
```

MANN WYFFE & PARTNERS	Page: 2
MEDIAEVAL JOUSTING FURNITURE	Made by: DWB
KNIGHT HOIST	Date: NOV1145
	Ref No: KCMG-4ME

LOADING SELF WEIGHT OF HOIST
JOINT DISPLACEMENTS

JOINT	X DISPLACEMENT	Y DISPLACEMENT	Z ROTATION
1	.001342558	-.001077520	-.000607010
2	.001342558	-.000533181	-.000591501
3	.001341652	-.000013558	-.000584715
4	.000000000	.000000000	.000000000
5	.000374686	-.000012604	-.000499581

LOADING SELF WEIGHT OF HOIST
MEMBER FORCES

MEMBER	JOINT	AXIAL FORCE	SHEAR FORCE	BENDING MOMENT
1	2	.0000	.5418	.2438
	1	.0000	.0000	.0000
2	3	-.4652	-.2523	-.1592
	2	.4652	.7941	-.3116
3	5	.7711	.4652	.6316
	3	.2523	-.4652	.1592
4	5	2.5679	.8331	.9392
	2	-1.3983	-.2139	.0678
5	4	4.3334	.0000	1.5708
	5	-3.4304	.0000	-1.5708

LOADING SELF WEIGHT OF HOIST
SUPPORT REACTIONS

JOINT	X FORCE	Y FORCE	Z MOMENT
4	.0000	4.3334	1.5708

EQUILIBRIUM CHECK	SUM OF FORCES	REACTION
FORCES IN DIRECTION X	.0000	.0000
FORCES IN DIRECTION Y	-4.3334	4.3334
MOMENTS ABOUT AXIS Z	-1.5744	1.5708

MANN WYFFE & PARTNERS
MEDIAEVAL JOUSTING FURNITURE
KNIGHT HOIST

Page: 3
Made by: DWB
Date: NOV1145
Ref No: KCMG-4ME

LOADING LIVE LOAD
JOINT DISPLACEMENTS

JOINT	X DISPLACEMENT	Y DISPLACEMENT	Z ROTATION
1	.007804116	-.006600892	-.004050829
2	.007804116	-.003055904	-.003621462
3	.007797627	.000003965	-.003403809
4	.000000000	.000000000	.000000000
5	.002146833	-.000016234	-.002862444

LOADING LIVE LOAD
MEMBER FORCES

MEMBER	JOINT	AXIAL FORCE	SHEAR FORCE	BENDING MOMENT
1	2	.0000	5.0000	4.5000
	1	.0000	-5.0000	.0000
2	3	-3.3312	-5.4892	-1.3296
	2	3.3312	5.4892	-3.6107
3	5	-5.4892	3.3312	4.3334
	3	5.4892	-3.3312	1.3296
4	5	10.8289	1.9637	4.6666
	2	-10.8289	-1.9637	-.8893
5	4	5.0000	.0000	9.0000
	5	-5.0000	.0000	-9.0000

LOADING LIVE LOAD
SUPPORT REACTIONS

JOINT	X FORCE	Y FORCE	Z MOMENT
4	.0000	5.0000	9.0000

EQUILIBRIUM CHECK	SUM OF FORCES	REACTION
FORCES IN DIRECTION X	.0000	.0000
FORCES IN DIRECTION Y	-5.0000	5.0000
MOMENTS ABOUT AXIS Z	-9.0390	9.0000

MANN WYFFE & PARTNERS Page: 4
MEDIAEVAL JOUSTING FURNITURE Made by: DWB
KNIGHT HOIST Date: NOV1145
Ref No: KCMG-4ME

LOADING SELF + 150% LIVE
JOINT DISPLACEMENTS

JOINT	X DISPLACEMENT	Y DISPLACEMENT	Z ROTATION
1	.013048732	-.010978858	-.006683253
2	.013048732	-.005117037	-.006023695
3	.013038092	-.000007611	-.005690429
4	.000000000	.000000000	.000000000
5	.003594935	-.000036954	-.004793247

LOADING SELF + 150% LIVE
MEMBER FORCES

MEMBER	JOINT	AXIAL FORCE	SHEAR FORCE	BENDING MOMENT
1	2	.0000	8.0418	6.9938
	1	.0000	-7.5000	.0000
2	3	-5.4619	-8.4861	-2.1536
	2	5.4619	9.0279	-5.7277
3	5	-7.4627	5.4619	7.1317
	3	8.4861	-5.4619	2.1536
4	5	18.8112	3.7787	7.9391
	2	-17.6416	-3.1595	-1.2661
5	4	11.8334	.0000	15.0708
	5	-10.9304	.0000	-15.0708

LOADING SELF + 150% LIVE
SUPPORT REACTIONS

JOINT	X FORCE	Y FORCE	Z MOMENT
4	.0000	11.8334	15.0708

EQUILIBRIUM CHECK	SUM OF FORCES	REACTION
FORCES IN DIRECTION X	.0000	.0000
FORCES IN DIRECTION Y	-11.8334	11.8334
MOMENTS ABOUT AXIS Z	-15.2039	15.0708

5.2 Principles

Some of the principles embodied in NL-STRESS are familiar to engineers in particular guises. For example, elastic instability within a member is recognised as the problem of the slender column: the vertical load multiplied by the small lateral deflection at mid-height gives the value of induced bending moment which can lead to buckling. The sway effect (also called the p-delta effect) is also an old friend. Consider a portal frame with sideways wind force causing the eaves to move horizontally relative to the ground. The lines of action of vertical roof loads (transferred at eaves level) then shift horizontally too, inducing secondary bending moments in the legs of the portal (sway distance times vertical load).

The elastic-plastic effect is familiar to structural engineers who design industrial buildings. The simple geometry of portal frames made it feasible - even in the era of the slide rule - to calculate required sizes of rafter and stanchion by ultimate-load theory. These sizes are such as would cause the portal to become a mechanism if design loads were scaled up by their load factors.

All these non-linear effects may be incorporated in analysis by NL-STRESS or may be suppressed if desired. When incorporated, NL-STRESS derives a 'subloading' by dividing applied loads by a number of increments (typically 10 or 20). After each application of a subloading NL-STRESS adjusts the frame geometry and checks for the formation of plastic hinges. As each hinge forms it is incorporated into the structure ready for the addition of the next subloading.

A plastic hinge is assumed to form when the interaction (combined effect) of axial load, torsion and biaxial bending moments exceeds a critical value. The formulae used to establish whether a plastic hinge has formed are called 'interaction formulae'.

When an interaction formula establishes the formation of a plastic hinge, the hinge is modelled as a torsional hinge together with free hinges about the principal axes of the cross section. Pairs of constant equal-and-opposite moments are applied across the free hinges to model the plastic effect.

5.2.1 Sway effect

The sway effect is computed by applying the specified loading an increment at a time. The number of increments is specified in the data; the size of increment is found by dividing total load intensities by the number to give a 'subloading'.

After the application of the first subloading the displacements of the joints of the structure are used to modify the joint coordinates to give a new geometry. A frame with this new geometry is analysed under the previous loading plus the next subloading - and so on.

5.2.2 Stability within a member

A slender member under axial compression suffers a reduction in bending stiffness. The stiffness of a member hinged at both ends reduces to zero as compressive force reaches the Euler load, Pe, where:

$$Pe = \frac{\pi^2 EI}{L^2}$$

If there are no lateral constraints the second moment of area, I, is the minimum of Iy and Iz. In plane frames Iy is assumed to be infinite, preventing buckling normal to the structural plane. (In grids there are no axial loads to cause buckling). The assumption of Iy to be infinite is a convenient device - in reality it is being assumed that there is sufficient out of plane support to prevent buckling, not only buckling solely caused by axial force, but also lateral-torsional buckling.

Reduction of bending stiffness is complicated by end conditions other than simple hinges (also by loads applied laterally within the length of a member). An example of partial fixity is depicted in Figure 5.1. Let the bending stiffness, k, in the absence of axial load, P, be defined as M/a, where M is the applied moment and 'a' is the angular displacement. The figures show how bending stiffness, k, reduces to zero as axial load increases relative to Euler load.

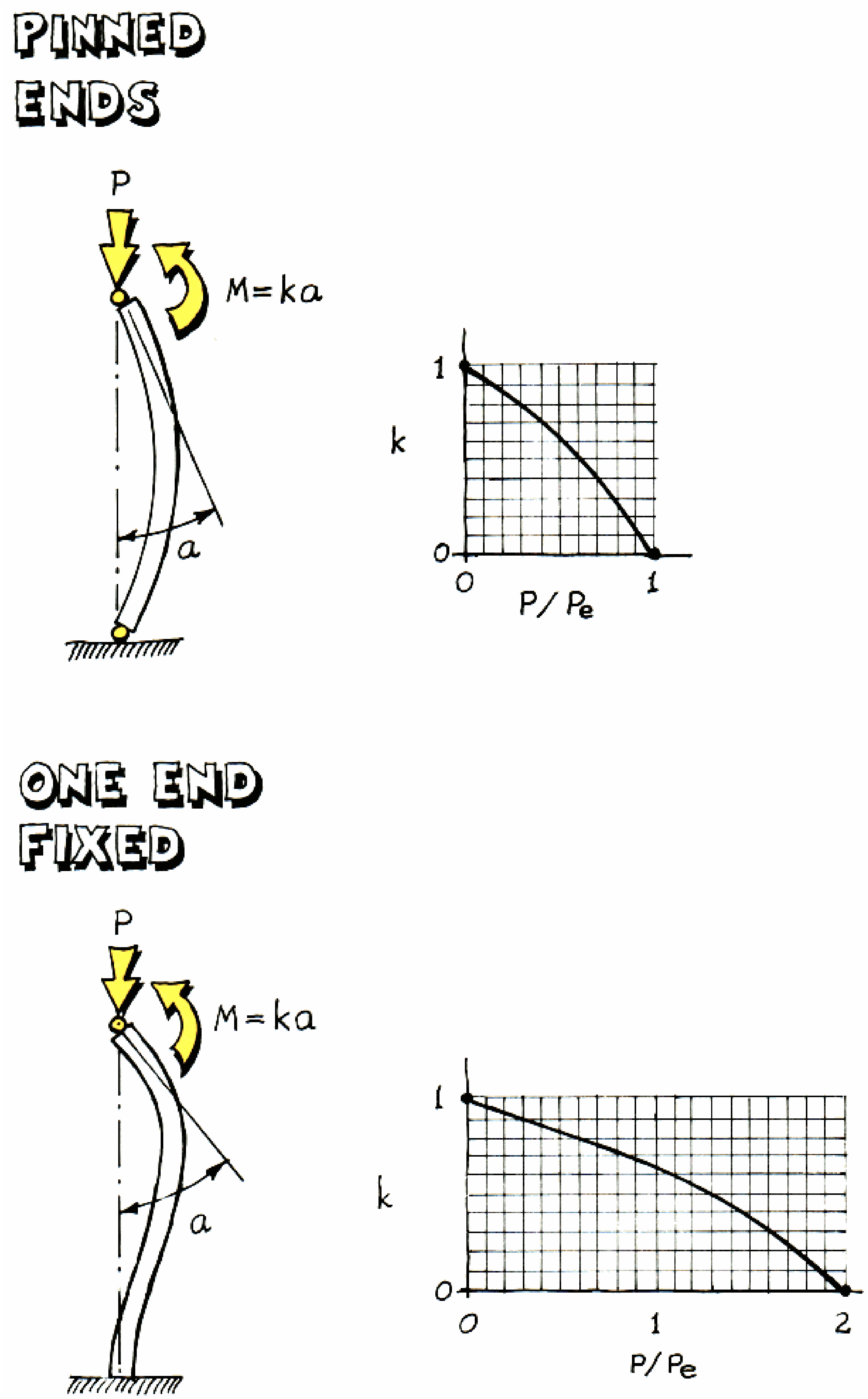


Figure 5.1: End effects on elastic instability.

5.2.3 Interaction formulae

For a plastic analysis a number of increments of loading must be specified. The intensities of all applied loads are divided by the given number of increments to give a 'subloading'. The subloading is applied to the structure; the members of the structure checked for the formation of plastic hinges; a further subloading is applied; and so on till all loading has been applied or the structure collapses.

A plastic hinge is assumed to form at an end of a segment when the interaction formula 'fails' at that point. NL-STRESS employs an interaction formula appropriate to the kind of cross section specified.

When the interaction formula 'fails', a plastic hinge is assumed to form. To model a plastic hinge NL-STRESS inserts three (in general) frictionless hinges at the end of the segment, then applies a pair of constant equal-and-opposite moments across each of the hinges. One hinge is axial, providing the plastic torque; the other two are hinges about the principal axes of the cross section. In plane frames there is no plastic torque; just a simple hinge about principal axis z. In grids there is a hinge with plastic torque about x, and a hinge about principal axis y.

Plastic hinges may develop only at ends of segments. It follows that for frames in which a hinge could develop part way along a member (say within the rafter of a portal frame) the more segments the more accurately will the hinge be located.

5.2.4 Plastic properties

For elastic-plastic analysis certain 'plastic' properties of cross sections are needed. These are, in general:

- squash load - the ultimate axial load for a short length of member
- plastic torque - the ultimate twisting moment on a cross section
- plastic moment - the ultimate bending moment of a cross section about a principal axis.

For cross sections specified in the data by shape and dimensions NL-STRESS is able to compute plastic properties using known geometry and yield stress.

5.2.5 Units of measurement

No unit conversions take place in NL-STRESS; the engineer should adopt:

	SI
• a unit of length	m
• a unit of force	kN
• a unit of temperature rise	°C

and stick to them. This may imply using unfamiliar units in some contexts. For example, assuming the above basic units, the following units would then become obligatory:

	SI
• coefficient of expansion	1/°C
• displacement	m
• point load	kN
• point-applied moment	kNm
• distributed load	kN/m
• cross-sectional area	m^2
• inertia (2nd moment of area)	m^4

Care is needed when abstracting section properties from handbooks where they are usually tabulated in cm or in units. An easy way to convert SI units is to use exponent form; a value of 2345 (units of cm^4) may be entered in the data as 2345E-8 (units of m^4) because the E says '...times ten to the power of'.

5.2.6 Sign conventions

A structure comprises joints connected by straight members. The location of every joint is specified by coordinates referred to a Cartesian system of axes called the global axes. The origin of global axes may be located anywhere convenient to the engineer. For example, in the case of a multi-storey building frame the most convenient place might be at ground level at the base of the left-most column or stanchion. On the other hand in a symmetrical portal frame the most convenient place might be at base level midway between the stanchions (then for every joint with a positive X coordinate there would be a corresponding joint in mirror image).

The global axes are denoted X,Y,Z. Movements in directions X,Y,Z are considered positive. Rotations about X,Y,Z are signed according to the right-hand rule. Think of the global axes as threaded bars; then turning X towards Y would cause linear movement in direction Z; turning Y towards Z would cause movement in direction X; turning Z towards X would cause movement in direction Y. The positive directions of all movements are depicted in Figure 5.2.

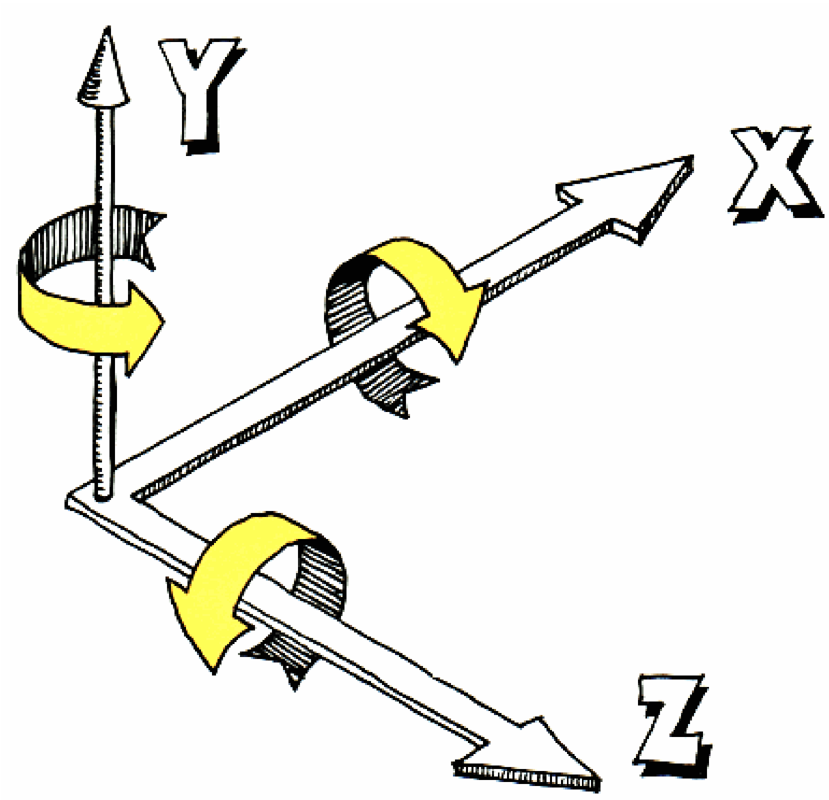


Figure 5.2: Positive directions of global axes.

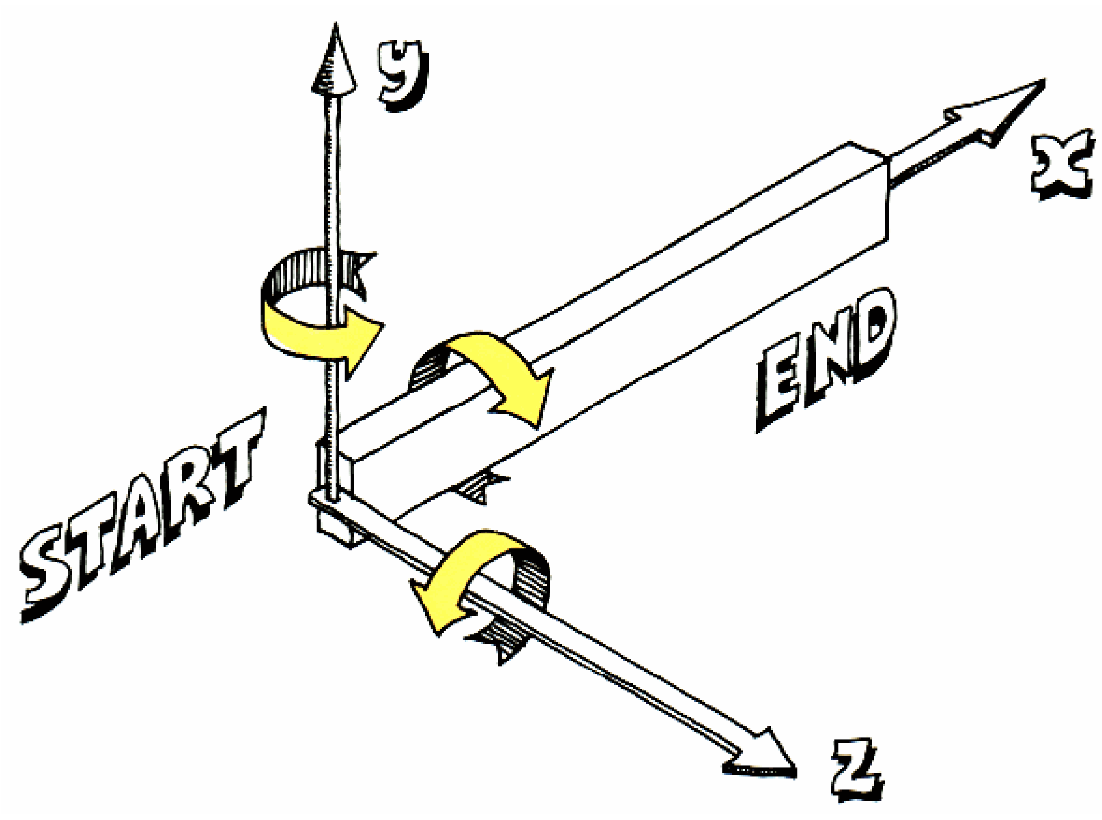


Figure 5.3: Positive directions of local axes.

Except in the special case of a continuous beam the members of a structure do not all run along the global axis X; they may be inclined to the global axis and each may run in a different direction. To fix the orientation of each member relative to axes X, Y and Z, every member is assumed to carry a local set of axes denoted x,y,z with axis x running from the START of the member (at the origin) to the END of the member. Movements along x,y,z - and about x,y,z - are considered positive as depicted in Figure 5.3.

In general, when dealing with the joints of a structure (their coordinates, conditions of support, settlements etc.) we refer to global axes X, Y and Z. When dealing with a member (its section properties, end releases etc.) we refer to its local axes x, y and z.

In plane frames, plane trusses and grids the structure is assumed to lie in the XY plane with Z pointing out of the paper. Accordingly the xy plane of every member lies in the global XY plane and the z axis of every member points out of the paper parallel to Z.

In space trusses and space frames the ZX plane is assumed to lie parallel to the ground; the Y axis being assumed to point vertically upwards. This is not arbitrary; the self weight of a member is taken to act in a direction opposite to that of Y. See Figure 5.4.

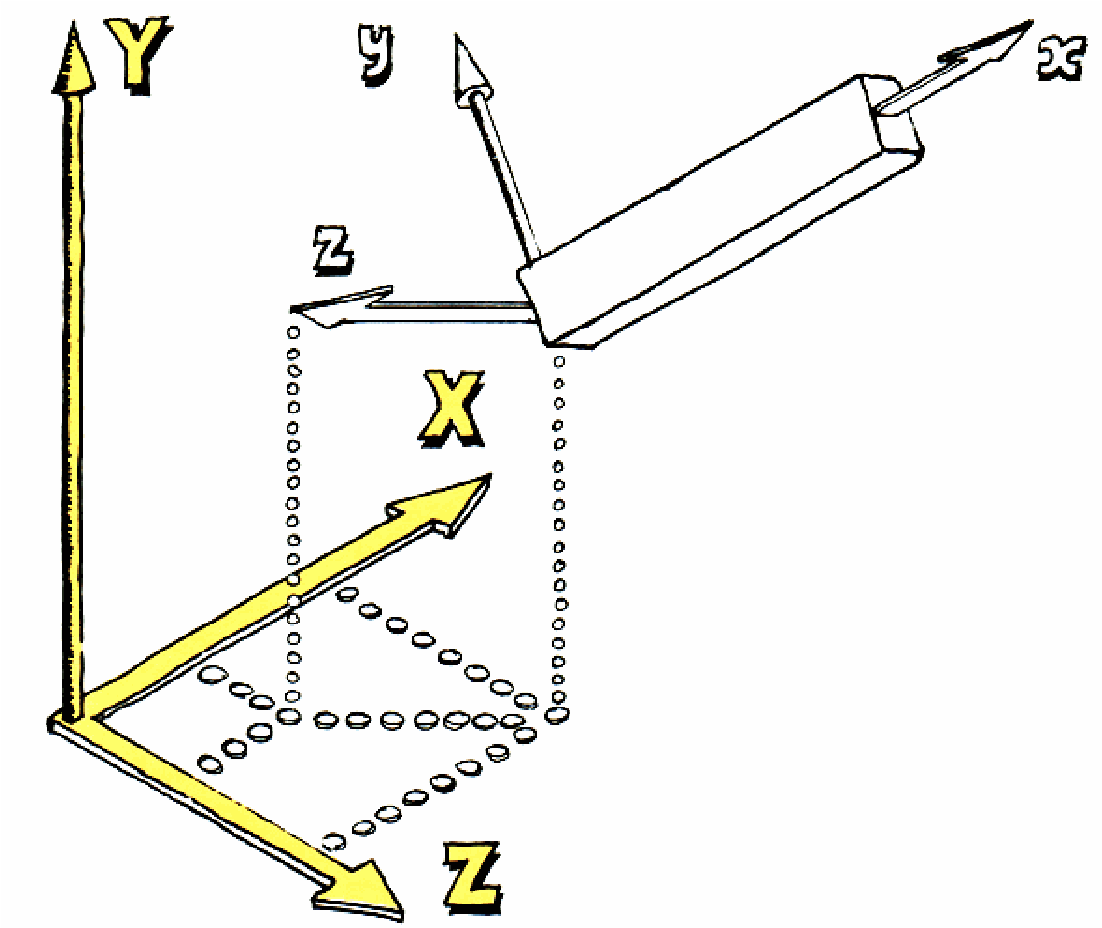


Figure 5.4: Relationship of local to global axes.

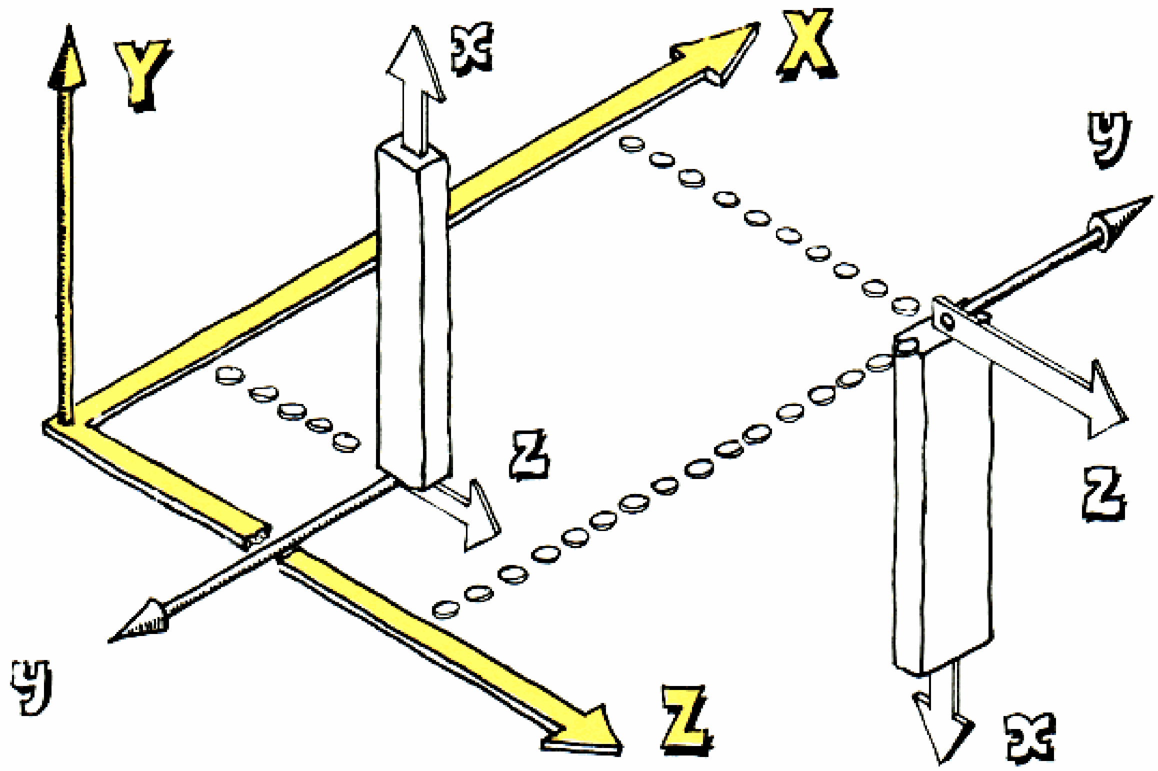


Figure 5.5: Vertical members.

Unless specified otherwise (see BETA below) the z axis of every member is assumed to remain parallel to the ground no matter what orientation the member has. The astute reader will notice that this is ambiguous; there being two positions for y when z is parallel to the ground. The one adopted makes the y axis lie entirely above the horizontal plane drawn through the START of the member (in other words y has a positive projection on Y).

The above definition is still incomplete; it does not cover vertical members for which the y axis lies neither above nor below, but IN the horizon through the START of the member. For vertical members it its assumed that local z then lies parallel to global Z and points in the same direction. See Figure 5.5.

The engineer may, however, specify an angle, BETA, for any member. The effect of this is to rotate the member about its x-axis to take up the required orientation. Positive BETA causes the z axis to dip below the horizon through the START end. See Figure 5.6.

The relationship of local to global axes in plane structures is depicted, for convenience, in Figure 5.7.

Take note that the relationship of local to global axes is treated differently among various versions of STRESS.

Few engineers have trouble with plane frames, space frames do cause difficulty, and to help in finding out what NL-STRESS is analysing, it is recommended that the two sets of axes shown in figs 2 & 3 are made out of paper; one for the global axes using upper case letters, the other for the local axes using lower case letters. Rolling up pieces of paper into tubes and using Sellotape and a stapler is a quick way of making the axes. Once made, position the global axes with the origin at the far left corner of your desk with the X axis pointing to the right, the Y axis pointing to the ceiling, and the Z axis pointing towards the front of the desk. Now imagine your structure sitting on your desk (it is normal to have all positive coordinates and therefore your structure will be in the positive quadrant). If you have a plan of your structure lie it on your desk correctly orientated to the X & Z axis. Now take the set of local axes and position them with the x axis going along one of the inclined (but not vertical) members, starting at the member start and pointing towards then end of the member. (The start of the member - it's local origin - is the first joint number in the MEMBER INCIDENCES table for the member. Keep the local z parallel to the top of the desk, with the local y axis pointing upwards, and you have now fixed the local axes in space. Unless you use one of the keywords GLOBAL or PROJECTED, loads listed in the MEMBER LOADS table are applied in the direction of the local axis. Tabulated member forces in the results file are always given in the direction of local axes.

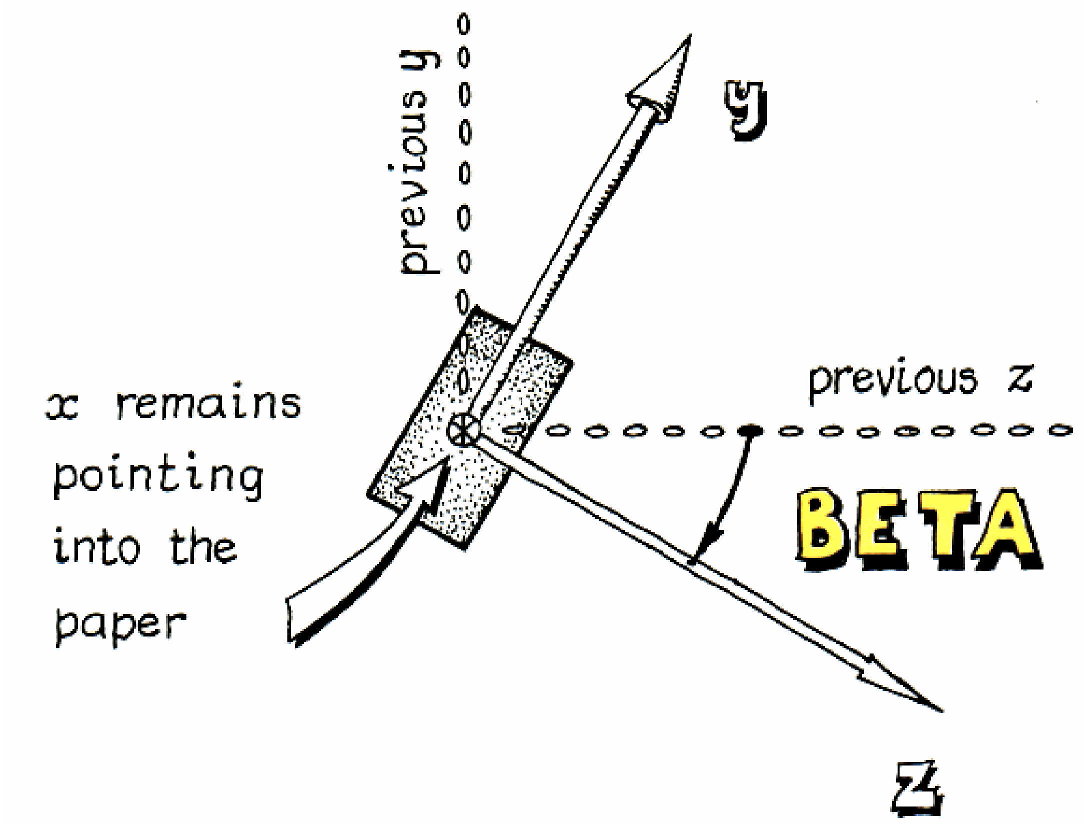


Figure 5.6: Angle BETA positive.

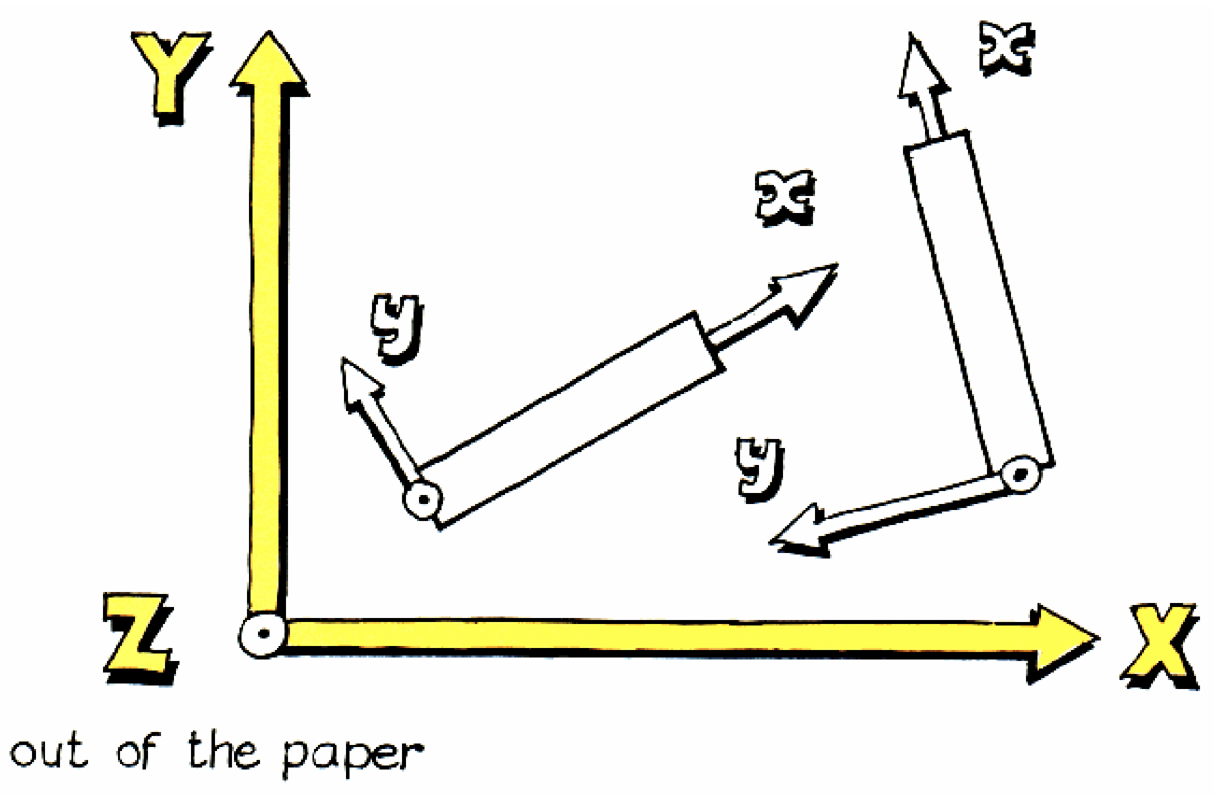
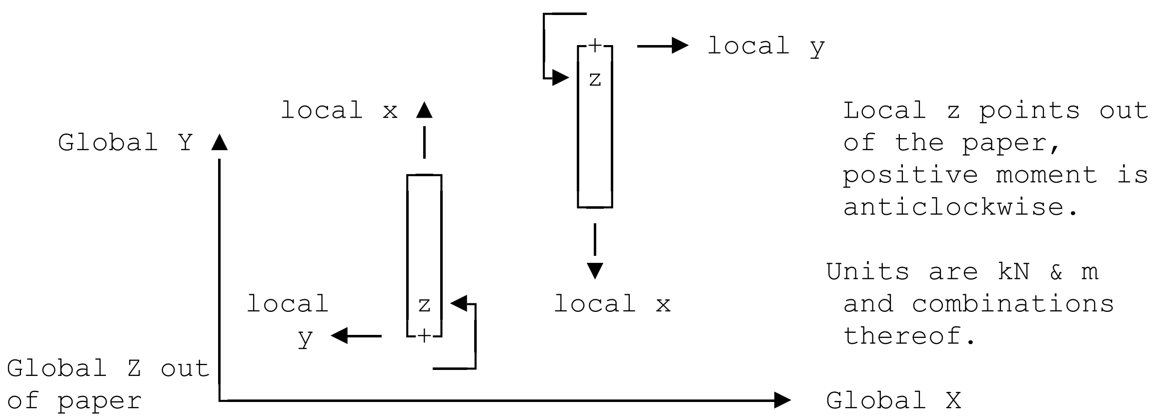


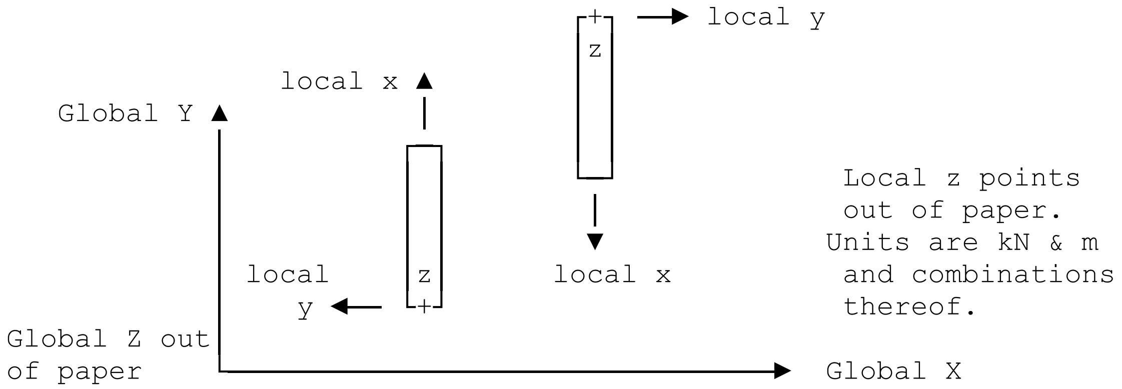
Figure 5.7: Axis relationship in plane structures.

5.2.7 Interpreting results

NL-STRESS - in common with all modern structural analysis software - has two sets of axes: a GLOBAL set and a local set. The GLOBAL set presents no difficulty as it conforms to that which we were taught at school: the origin in the bottom left corner with X pointing to the right and Y pointing upwards. For plane frames and space frames the Z axis points out of the paper, a positive moment is that which would cause a nut to travel in a positive direction along the axis; thus moments about the Z axis are anticlockwise when looking along the Z axis towards the origin. For plane frames and space frames all gravity loads are applied as negative in the Y direction.



ELEVATION ON ALL STRUCTURE TYPES EXCEPT PLANE GRIDS

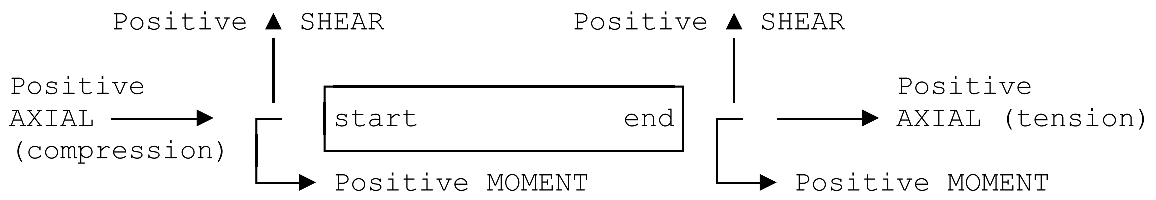


PLAN ON PLANE GRIDS

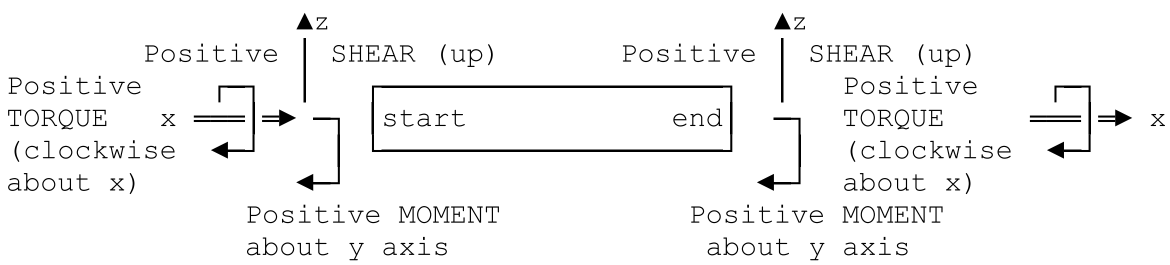
Generally, the members of a structure do not all run along the global axis X; they may be inclined to the global axis and each may run in a different direction. In the diagrams above, the left member has a positive projection on the Y axis, the right a negative. To fix the orientation of each member relative to axes X, Y and Z, every member is assumed to carry a local set of axes denoted x,y,z with axis x running from the START of the member (at the local origin) to the END of the member. The START of the member is the first joint given in the MEMBER INCIDENCES table, the END is the 2nd joint. We need this set of local axes for no engineer would want shear forces in a pitched rafter, referred to the GLOBAL axes.

Tabulated joint displacements give the displacements in the global directions and the rotations about the global axes in radians (1 radian = 57.3 degrees) for each joint.

Tabulated member forces give the forces imposed at the start and end of each member (or segment) by the joints. Think of the joints as being separate from the members and applying an axial load, shearing force and moment to the ends of the member in question. Positive forces are in the direction of the local axes, negative forces are opposite to the direction of the local axes. We cannot keep to the traditional convention that sagging is positive and hogging negative as sagging and hogging are undefined in the case of a vertical member. For a member in compression throughout its length there will be a positive force at the start end (local X goes along the member from the START end to the END end) and a negative force at the end, the joint at the end of the member pushing back along the member.



POSITIVE MOMENTS AND FORCES AT BOTH ENDS OF PLANE FRAME MEMBER



POSITIVE MOMENTS AND FORCES AT BOTH ENDS OF PLANE GRID MEMBER

When dealing with the joints of a structure (their coordinates, conditions of support, loads etc.) we refer to global axes X, Y and Z. When dealing with a member (its section properties, member loads

etc.) we refer to its local axes x, y and z. For quick reference the effects of positive results for forces acting on the ends of the members of a plane frame and grid are depicted above.
THINK OF THE JOINTS AS APPLYING FORCES TO THE MEMBER ENDS.

For a grid, all members lie in the X/Y plane; the depth of the grid members being measured in the Z direction. Results give moments about the local x & y axes and shears in the local z direction. Moments about the x axis (MX) are torsional, moments about the y axis (MY) are flexural.

For member forces we cannot keep to the traditional convention that sagging is positive and hogging negative as sagging and hogging are undefined in the case of a vertical member. For a member in compression throughout its length there will be a positive force at the start end (local X goes along the member from the START end to the END end) and a negative force at the end, the joint at the end of the member pushing back along the member. Figure 5.8 shows a summary of the directions of positive forces and moments at both ends of a member.

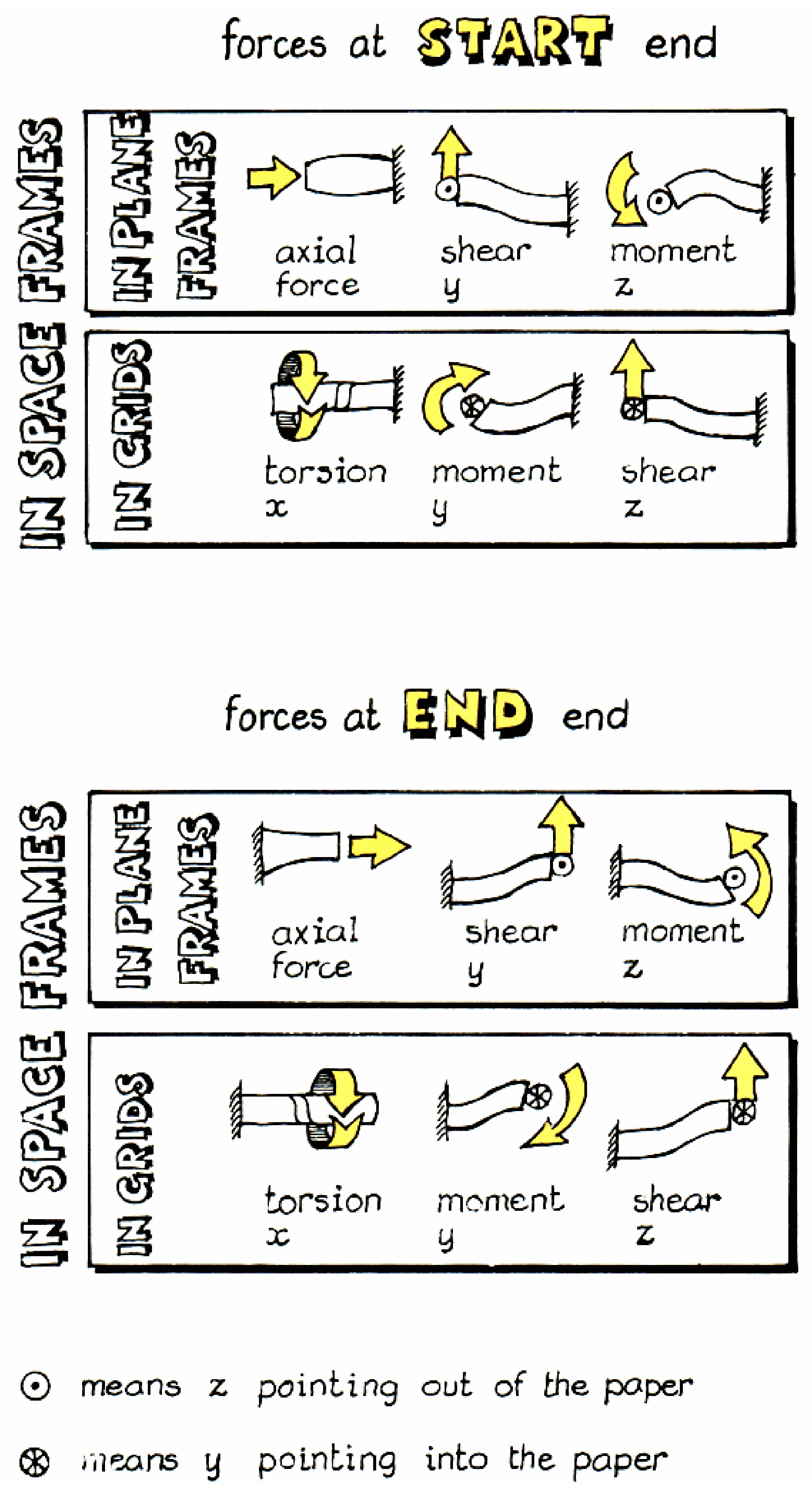


Figure 5.8: Directions of positive forces at ends of members.

Tabulated reactions - as with the joint displacements - are in the directions of GLOBAL axes. The reactions are followed by an EQUILIBRIUM CHECK. This is a true equilibrium check (not just a check on NL-STRESS's arithmetic) in that the sum of forces in each of the three directions, applied to the displaced structure, is compared with the sum of the reactions worked out by NL-STRESS from the loading applied to the structure in its undisplaced position.

A set of output comprises a summary of data and a selection of results of the analysis. Of these results the following are signed according to the conventions already described:

- deflections and rotations of joints
- forces and moments induced at ends of members
- reactions at supports - forces and moments acting on supported joints.

When positive, all these act in the positive directions already described. For quick reference the effects of positive results for forces acting on the ends of members are depicted in Figure 5.8. Think of the joints applying forces to the member ends.

Positive displacements at a joint move the joint in the directions depicted in Figure 5.2.

Positive reactions at a supported joint act on that joint in the directions depicted in Figure 5.2.

For simplicity, versions of NL-STRESS prior to 2.32 signed the stress according to the force or moment, thus a positive force or moment caused a positive stress and a negative force or moment caused a negative stress leaving the user to input a negative value for 'cy' if he/she wished to change the sign of the stress. For PLANE GRIDS a positive moment at the start of a member produces a compression in the fibres above the x axis and to combine the axial and bending stress at the start of the member we only need to add the stresses, if the result is positive then the fibres above the x axis are in compression, if the result is negative then the fibres above the x axis are in tension. Of course this simple treatment only works at the start of a plane grid member but if we adopt the convention that compressive stresses in the fibres above the x axis are always positive we can let NL-STRESS flip the signs for bending and axial stress as necessary for all structure types and both ends to accord with this convention. This flipping of signs applies only to stresses; forces and moment signs remain the same as they have been for the past 30 years (see Figure 5.2).

5.3 Introduction to the NL-STRESS GUI

The NL-STRESS GUI is an interface between you and NL-STRESS, which aims to simplify and speed up the process of preparing data for analysis. By using the interface you can focus on only a small section of the data set at any one time and can concentrate on the data, leaving the interface to assemble it correctly into the analysis file.

Direct editing of the data file is still possible, (and even required to go beyond the options available in the interface), through the "Identification->Edit the data file" menu option.

The current data set is presented in a main data window as a hierarchical list in which tables can be collapsed or expanded as necessary, reducing the apparent size of the data set and hiding data which the user may not be working with at a particular time.

Editing of data items is performed by double tapping (iPad) or double clicking with the left mouse button (Windows) anywhere on a text line or by pressing the Return key. This will open an edit dialog appropriate to that item or table. A tap/click on the dialog's Ok button will update the data set with the edited data. Selecting the dialog's Cancel "◄" button will ignore any changes made during the current visit to the dialog.

New data can be added by selecting from the menu bar at the top of the main window (iPad and Windows) or from a quick popup menu accessed by clicking the right mouse button (Windows only).

A detailed description of every aspect of data entry using the GUI is contained alongside the relevant description of the NL-STRESS Commands and Tables in sections 5.7.1 to 5.7.34

5.3.1 Launching the GUI

From the main SCALE menu, select the option "Run GUI data generator" from the section "Run NL-STRESS, the plot program and the GUI directly".

Enter your filename at the next screen, you can use the Files button to choose from the .dat files in your current directory.

SCALE will then silently run option 677 in the background to expand out any parametric variables, expressions and assignments, and replace them with their numerical values. These are removed because they would vastly complicate the process of adding extra data with the GUI. A copy of the .dat file is automatically saved with a .bak extension if you wish to revert to using the original file with expressions and assignments.

Once the GUI launches you will be presented with the main GUI window. This takes the form of a tree-view, as shown in Figure 5.9, where you can view all the file, and you can collapse and expand sections, by tapping on any heading line. You can tap/click on a heading with a minus icon to collapse that section, or tap/click on a heading with a plus icon to expand that section.

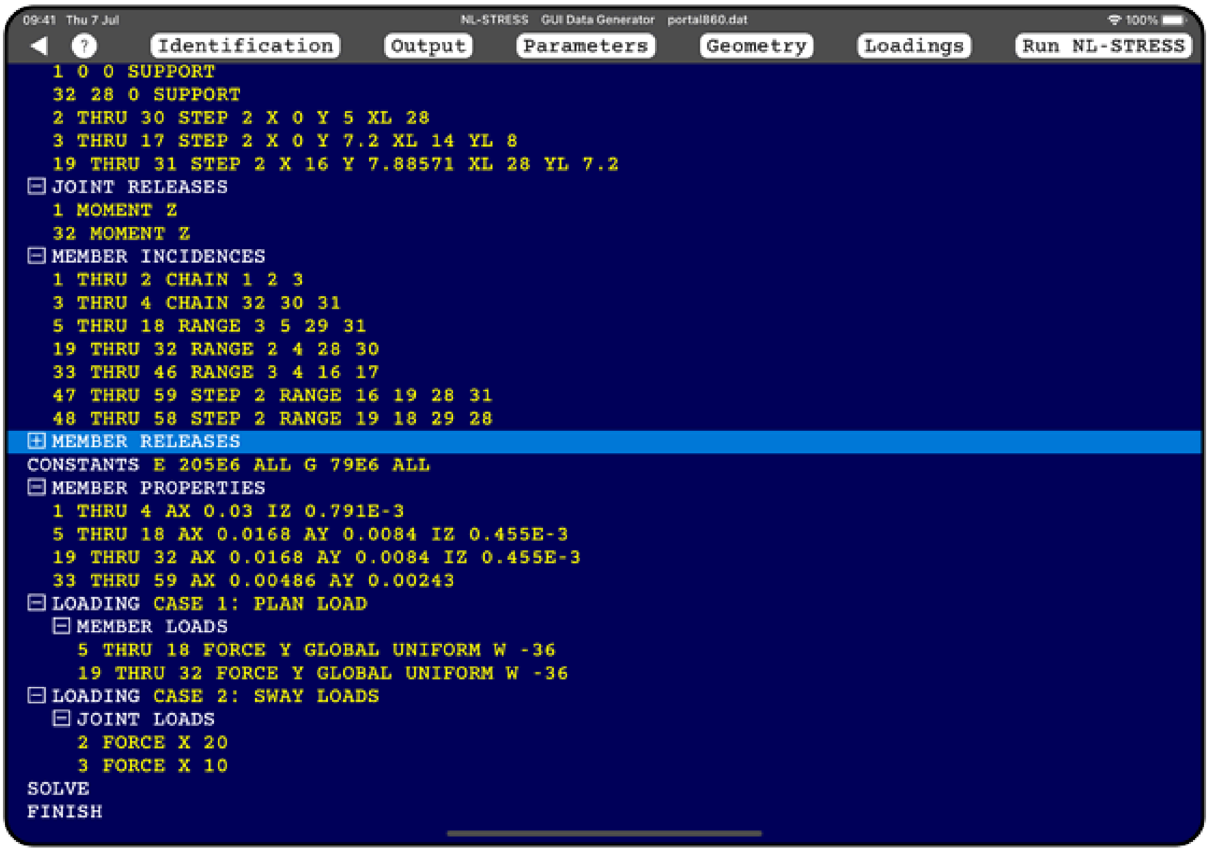


Figure 5.9: The main GUI window.

5.3.2 GUI - Draw Structure option

For 2D structures, the GUI's Draw Structure option may be used to graphically edit the location of joint and members. An outline may also be superimposed to help with the drawing, data for this outline is saved in the JOINT COORDINATES table for future editing if required.

To launch the Draw Structure option, select the "Draw structure" item from the Geometry menu option, or select the "Draw structure" button on the JOINT COORDINATES or MEMBER INCIDENCES windows.

See also JOINT COORDINATES,
MEMBER INCIDENCES.

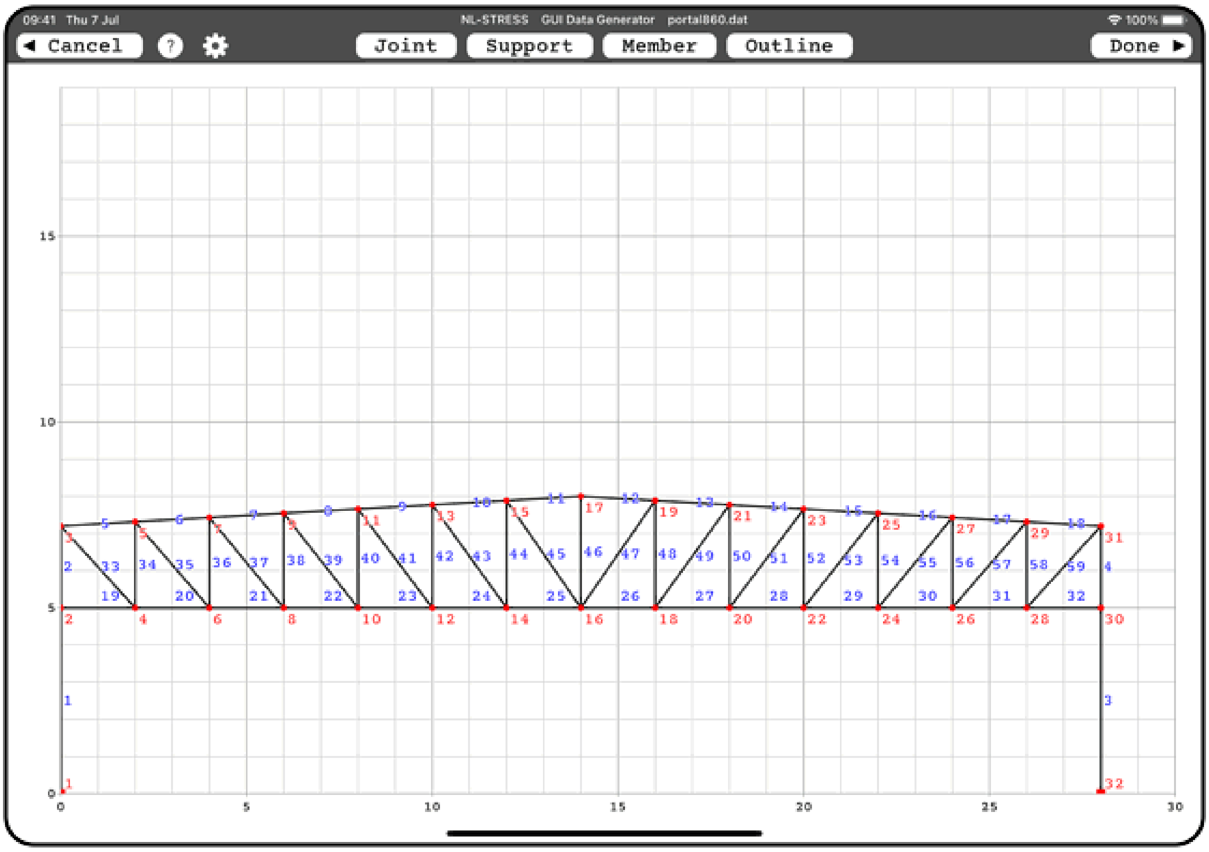


Figure 5.10: The GUI Draw Structure window.

When the Draw Structure window is first opened, as shown in Figure 5.10, axes scales are automatically calculated using the largest range on either axis. Re-sizing the window will result in a larger or smaller range on the relevant axis.

Only the first occurrence of a joint or member number will be recognised. Duplicates will be ignored.

Joint numbers are displayed in red, member numbers in blue. The members are displayed in black, any outline present is displayed in green.

Make sure that the maximum number of Joints, Supports, Members required for the structure has been entered into the Mandatory Parameters dialog. You will be prevented from exceeding these limits.

iPad:
To zoom in and out, use the usual pinch out and in gestures. With an external keyboard attached you can also use the plus and minus keys to zoom in and out.

To pan around, simply drag around with one finger. With an external keyboard attached you can also use the arrow keys to pan up, down, left and right.

If you edit joints and/or members in Draw Structure, then the JOINT COORDINATES and/or the MEMBER INCIDENCES tables will be expanded out to one joint or one member per line. i.e. all logic like 1 THRU 11 STEP 2 will be expanded. This is required to prevent duplication of data. Any outline data will be added to the start of the JOINT COORDINATES table, as show in Figure 5.11.

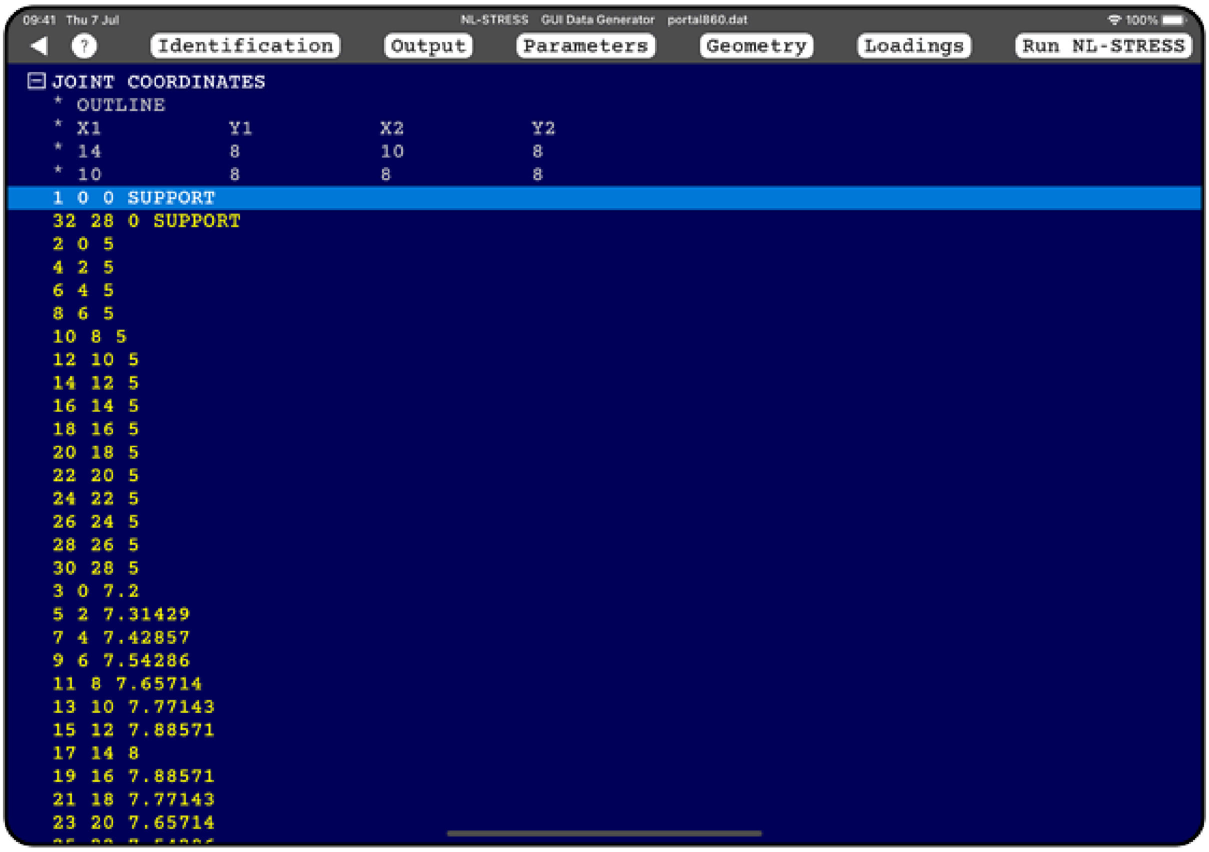


Figure 5.11: The GUI after returning from Draw Structure.

5.3.3 GUI - Draw Structure - Settings

Tap on the settings button to toggle Snap to Grid on and off, as shown in Figure 5.12. When Snap to Grid is on, then any joint or outline is restricted to lie on the intersection of grid-lines. The grid-line spacing is changed automatically as the model is zoomed in and out. To get a finer grid spacing, simply zoom in further.

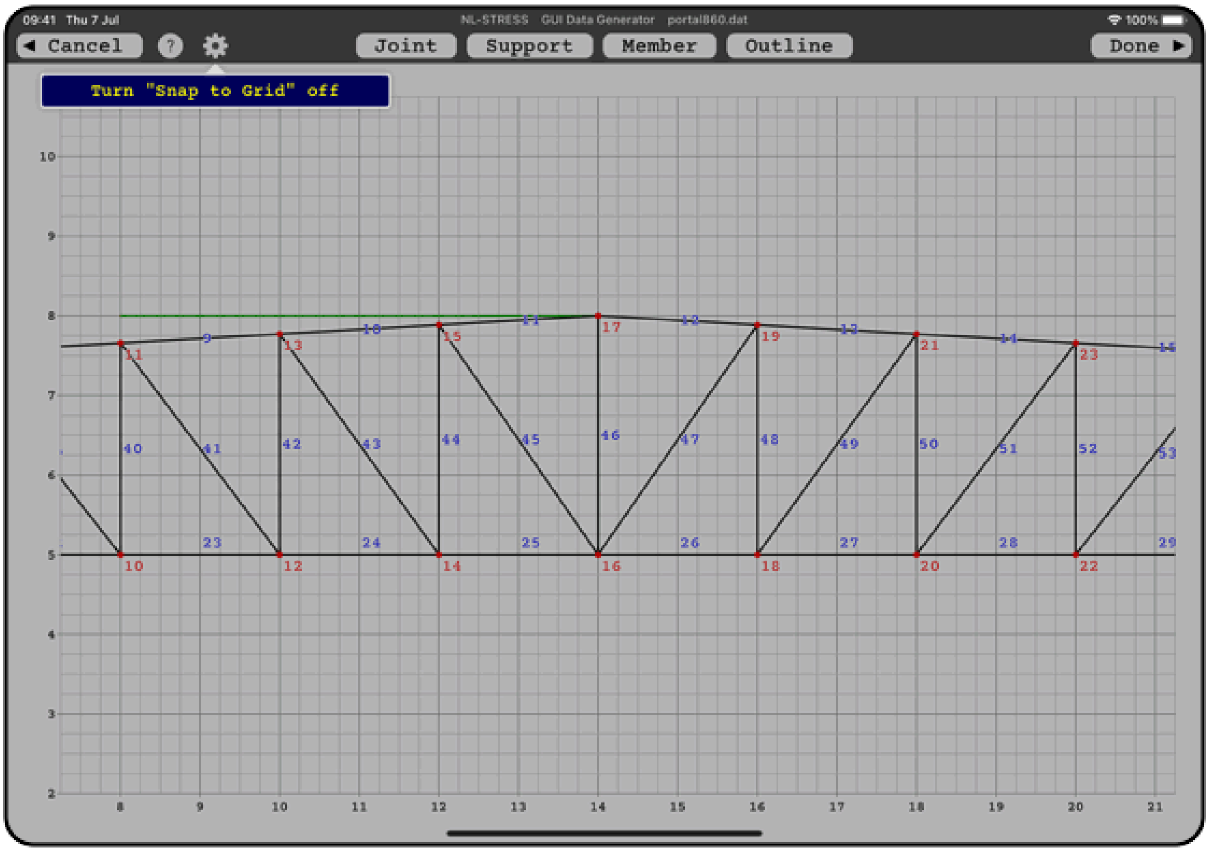


Figure 5.12: GUI Draw Structure settings menu.

5.3.4 GUI - Draw Structure - Joint

Tap on the "Joint" button to switch to joint mode, as shown in Figure 5.13.. Select the joint number required from the horizontal dial picker on the toolbar. Tap either side of the number to jump by one joint, or spin the dial to move quickly through the joints. Joints that are already present are coloured white in the picker, joints that are yet to be entered are highlighted in yellow.

Tap on the structure window to add/move the chosen joint to a new location, repeat tapping as required to keep moving the joint. For each tap, the required coordinates will be updated and the joint will be drawn. For an existing joint, any connected members will be redrawn.

Repeat for other joints as required.

You can zoom and pan as described in 5.3.2, or add an outline as described to 5.3.7 to help guide your joint placement.

Select the "Done" button when finished.

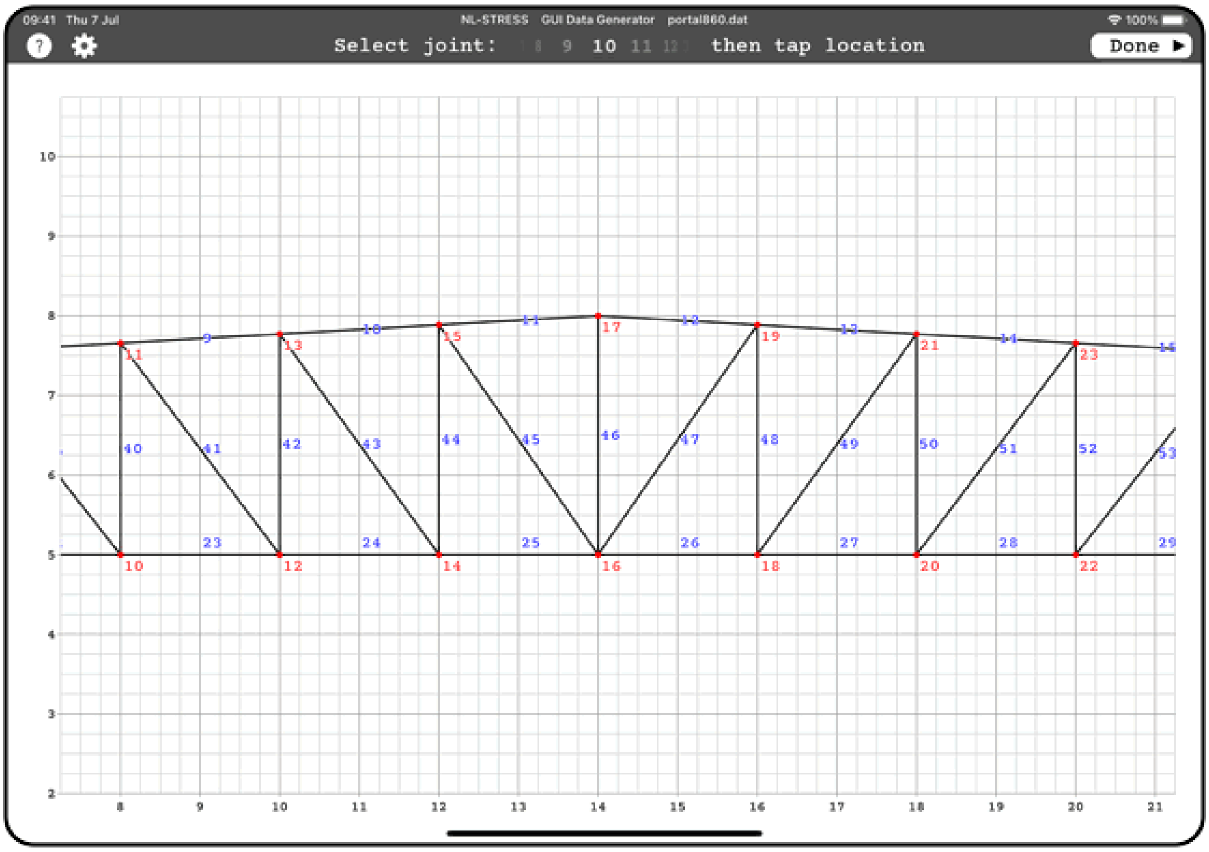


Figure 5.13: GUI Draw Structure adding a joint.

5.3.5 GUI - Draw Structure - Support

Select the Support button on the Draw Structure window to switch to Support mode. Tap on any joint to toggle it between supported and unsupported. Joints are indicated by a red dot, supports are indicated by a larger square.

Pan and zoom as required as described in 5.3.2

Select the "Done" button when finished.

5.3.6 GUI - Draw Structure - Member

Tap on the "Member" button to switch to member mode, as shown in Figure 5.14. Select the member number required from the horizontal dial picker on the toolbar. Tap either side of the number to jump by one member, or spin the dial to move quickly through the members. Members that are already present are coloured white in the picker, members that are yet to be entered are highlighted in yellow.

You can zoom and pan as described in 5.3.2.

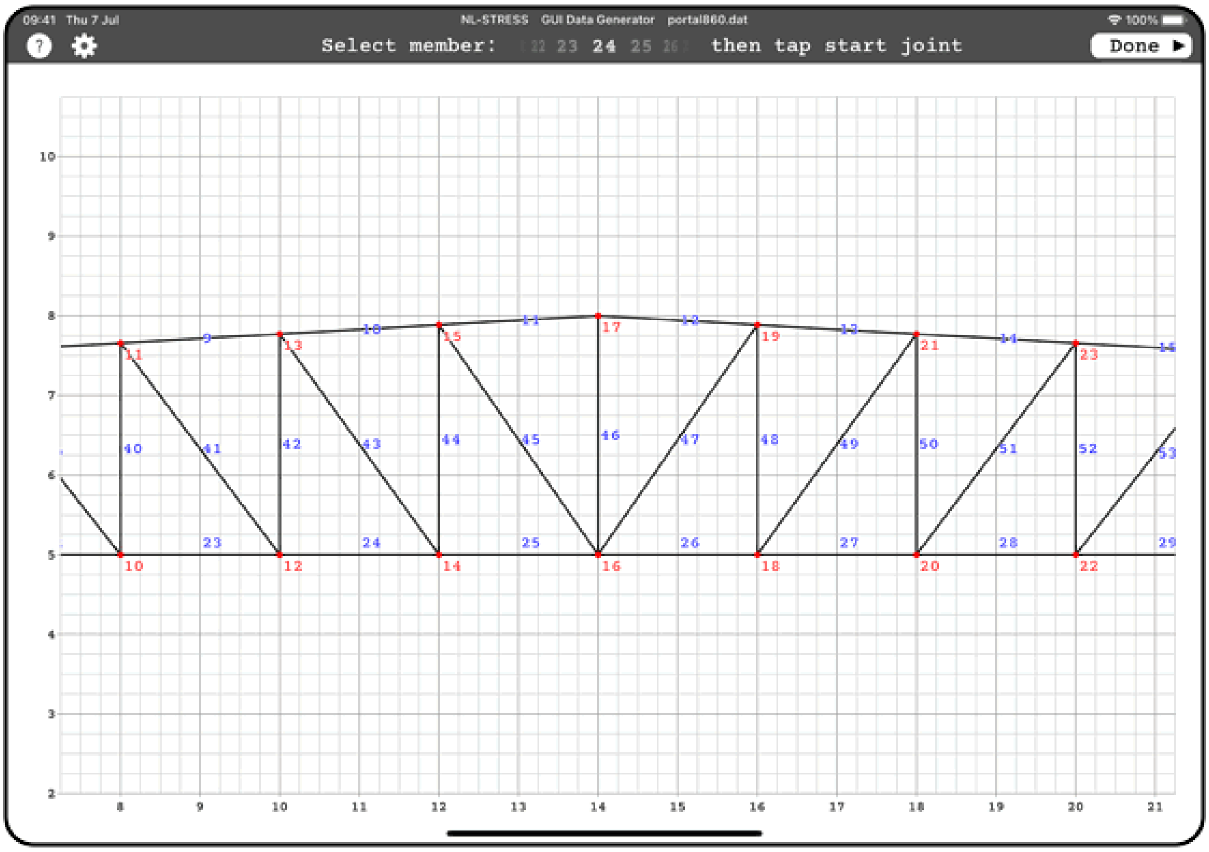


Figure 5.14: GUI Draw Structure adding a member, start joint.

Next, tap on the starting joint in the structure window. This starting joint will be temporarily highlighted with a target symbol, so you know which joint has been selected for the starting joint, as shown in Figure 5.15.

If you wish to abort adding this member, simply select the "Done" button when prompted for the member number, or after the starting joint has been selected.

Now, tap on the end joint for the chosen member.

When finished adding/changing members, select the "Done" button to go back to the main GUI window.

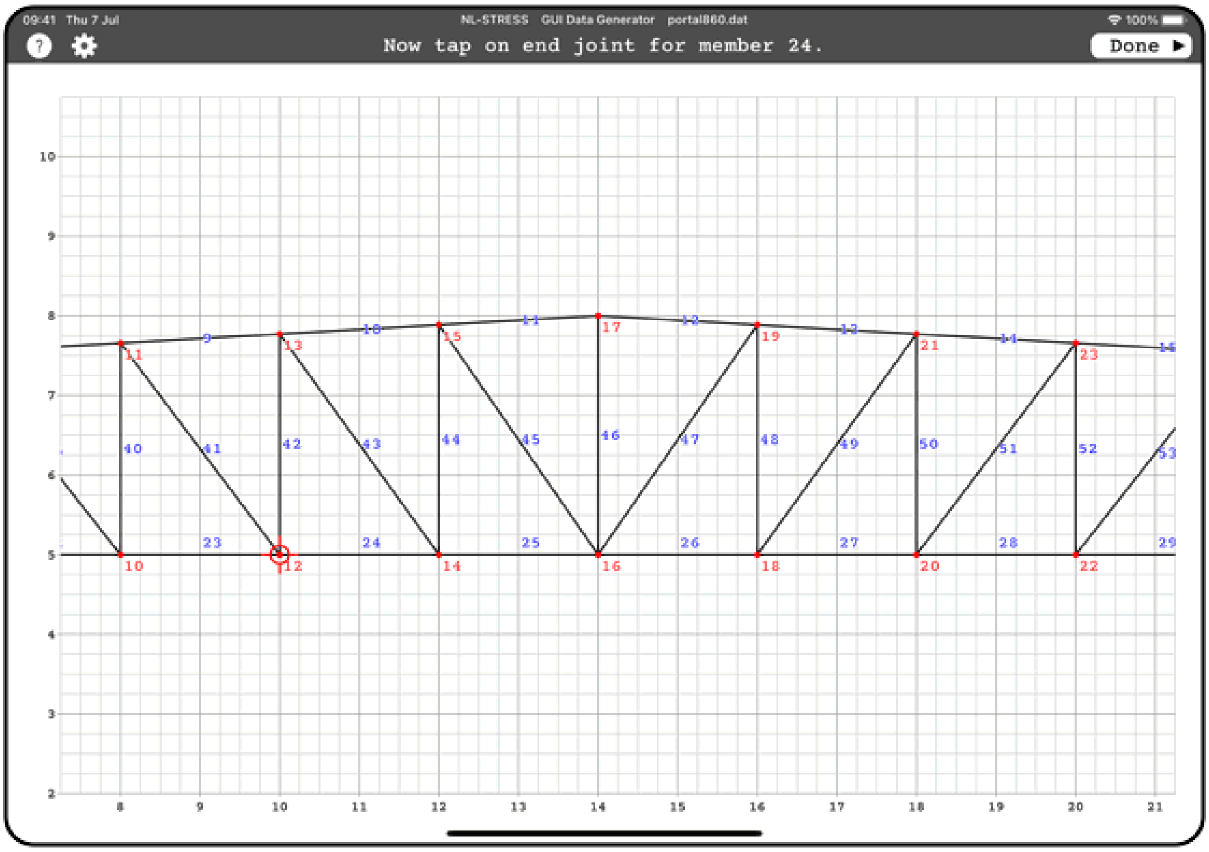


Figure 5.15: GUI Draw Structure adding a member, end joint.

5.3.7 GUI - Draw Structure - Outline

To help with established the construction of the model, an outline can be added to the screen. This outline supplies "sketch" construction lines without producing actual joint or member data.

Select the "Outline" button from the main Draw Structure screen, to enter the Outline mode as shown in Figure 5.16

Tap on the start position of the outline. This will then be marked with a target icon, so you see where you tapped, as shown in Figure 5.17.

If you're not happy with the start position, tap on "Done" and restart the process.

Next tap on the end position, and the outline will be added. Tap on "Done" when you have finished adding the outline.

Outline data appears at the start of the JOINT COORDINATES table as comments, as shown in Figure 5.11, and may be edited or deleted there by editing the data file. Outline data is saved with the .dat file.

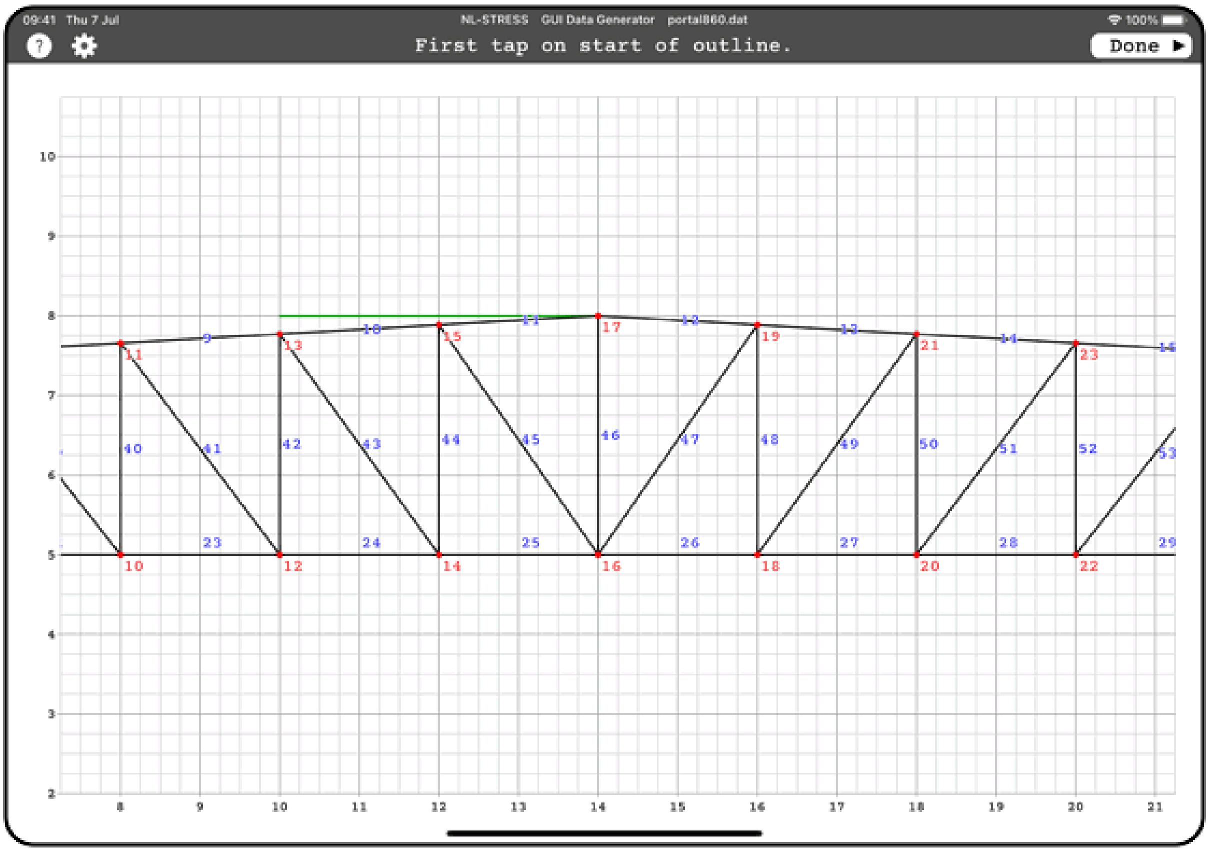


Figure 5.16: GUI Draw Structure adding an outline, starting point.

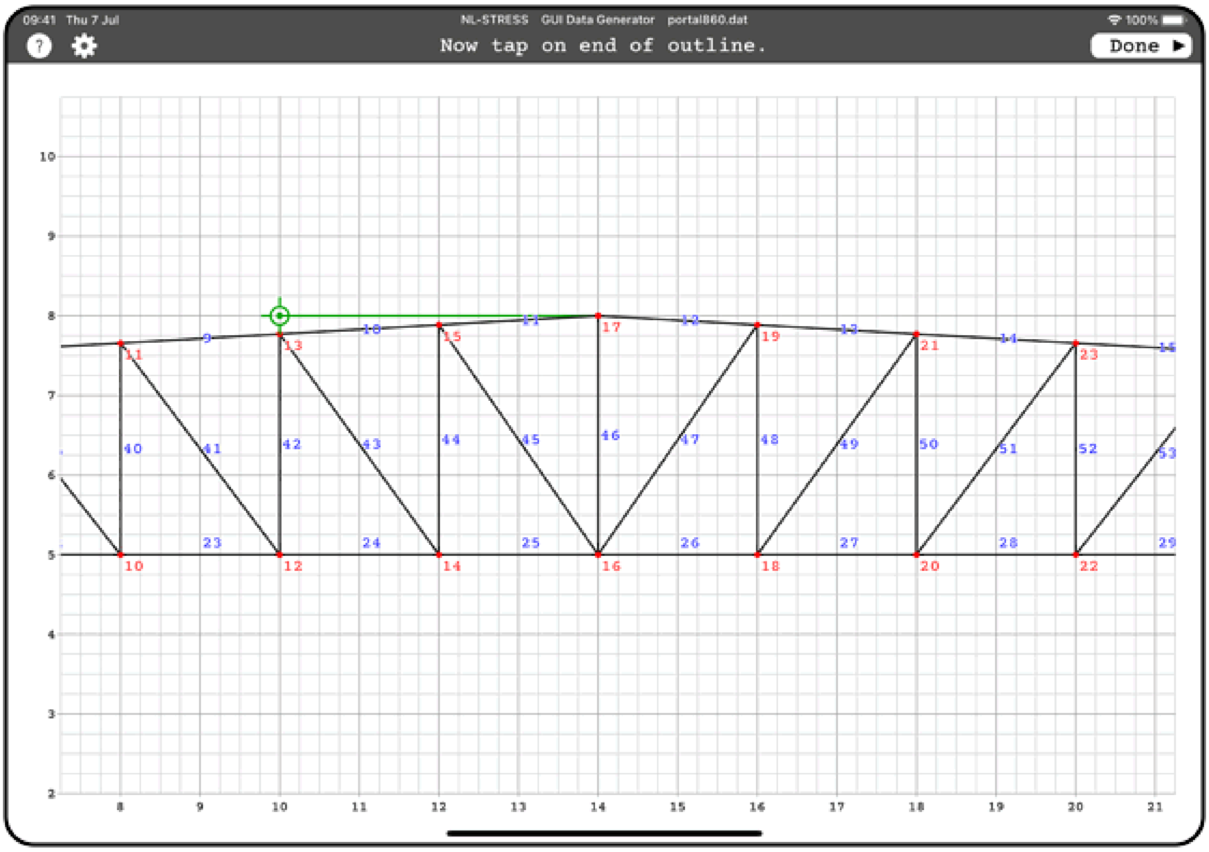


Figure 5.17: GUI Draw Structure adding an outline, part 2.

5.4 Basic data

This section defines the basic items - keywords, numbers, separators, expressions and so on - which, when properly arranged, constitute 'commands' and 'tables'. A correct arrangement of these, in turn, constitutes a set of data acceptable to NL-STRESS.

5.4.1 Keywords

Examples of keywords are MEMBER, THRU, AX, X. Keywords in the data must be typed in capital letters; each is recognised by NL-STRESS only if correctly typed and in correct context. Section 5.7 shows how keywords are used to build commands and tables.

Keywords may be abbreviated, but no further than their first four letters. MEMBER may be abbreviated to MEMBE or MEMB but not MEM. (In fact arbitrary letters may be typed after the fourth letter without causing an input error, e.g. MEMBAHS, but there is nothing to be gained from such practice.) Keywords with fewer than four letters (e.g. AX) may not be altered in any way.

Spaces in keywords are not allowed: MEM BER is two keywords neither of which is recognisable to NL-STRESS.

One keyword is unique in behaviour; it is PI. This is the name of a variable which is automatically made to contain 3.14159... at the start of an analysis. PI is intended for use in assignments such as:

```
area=PI*radius^2
```

Another unique keyword is LINE; it contains the line number of line in which the keyword appears. For an example of its use, see 4.11.

5.4.2 Values

A 'value' means an item of numerical data. A value may be typed at the keyboard in any of the following forms:

- as a number (e.g. -27.6)
- as a function (e.g. RAD(15.5))
- as a symbolic name for a variable, which has been previously assigned (e.g. psc)
- as an algebraic expression involving any or all of the above forms (e.g. -27.6*(RAD(15.5)+psc^2)).

Numbers, functions, variables and expressions are separately described below. All are stored and manipulated within memory using high precision arithmetic with 16 or more decimal digits of accuracy.

5.4.3 Numbers

Examples of numbers are 76, +76.0, -.25, 3.5E-6

In general, a number may be typed with or without a leading plus or minus sign; with or without a decimal point. A trailing or leading decimal point is permitted, as in 25. in place of 25.0 or .25 in place of 0.25

A number may be written in exponent form where the E says '...times ten to the power of'. Thus -3.5E3 (or -3.5E+3) is another way of writing -3500.0, and 3.5E-3 is another way of writing 0.0035 (shift the decimal point the number of places indicated by the exponent; left for minus). Exponent form is useful when converting units by multiplying by powers of ten.

5.4.4 Functions

Examples of functions are:

```
            INT(a+b)     SIN(2*PI+x)   EXP(x)
```

A function is a keyword followed immediately by an expression in brackets. The expression in brackets is called the 'argument' of the function.

There may be no spaces anywhere in a function.

When a function is encountered its argument is evaluated and transformed to return a single value in place of the function; for example INT(2*3.4) returns 6 which is the integral part of the argument 6.8. Because expressions may contain functions it is possible to have functions of functions; thus SIN(RAD(30)) returns 0.5 because RAD(30) returns 0.5236 - the number of radians in 30 degrees - then SIN(0.5236) returns 0.5.

The keywords of all available functions are listed below together with an explanation of what each function returns.

First the arithmetic functions:

ABS Absolute value. ABS(2.5) and ABS(-2.5) both return 2.5, ABS(0) returns 0

APR Approximate match to unity. APR(.99) returns 0.99, APR(.999999) returns 1. This function is for particular use in comparing two values say a & b thus: IF APR(a/b)=1 ...

INT Integral part by truncation of the absolute value. INT(2.9) returns 2, INT(-2.9) returns -2, INT(0) returns 0.
The INT function may be used to cycle for a special condition e.g. if it is required to set a value 'fac' =100 generally, but every sixth time in a loop, set to unity, proceed as follows:
set the base   b=6
set the value fac=100 arrange for the value 'a'
to cycle 1,2,3,4,5,6,1,2,3,4,5,6,1,2,3... and so on in a loop.
IF b=a-INT((a-1)/b)*b THEN fac=1 ENDIF
will set the value fac to: 100,100,100,100,100,1,100,100... etc.
For those who are familiar with the modulus programming function, see praxis help section 6.11.

DE0 DEcimal rounding to 0 decimal places. DE0(2.9) returns 3, DE0(2.3) returns 2, DE0(-2.9) returns -3, DE0(-2.3) returns -2.

DE1 DEcimal rounding to 1 decimal places. DE1(2.95) returns 3.0, DE1(2.35) returns 2.4, DE1(-2.35) returns -2.3.
DE2-DE3 similar to above for rounding to 2-3 decimal places.

DFR Decimal FRaction. DFR(3.235) returns 0.235, DFR(3) returns 0, DFR(-6.2) returns -0.2.

SGN Signum. Returns 1 if the argument is positive, -1 if negative, 0 if zero. SGN(0.01) returns 1, SGN(-270) returns -1. For switches (programming devices) using Signum see sc924.hlp.

LOG Natural (base e) logarithm. LOG(1.0) returns 0, LOG(2.718282) returns 1. LOG(0) or LOG(-1) provokes an error message. To convert between LOGe & LOG10 use: LOG10(e) =1/LOGe(10) =0.4342945
thus LOG10(2) =LOGe(2)*0.4342945 =0.69315*0.4342945 =0.30103

EXP Natural antilogarithm (e to the power of ...). EXP(0) returns 1, EXP(1) returns 2.718282, EXP(-1) returns 0.3678794 (i.e. 1/e)
To convert between EXP & ANTilog10 reverse above for LOG:
thus ALG10(0.30103) =EXP(0.30103/0.4342945) =EXP(0.69315) =2.

SQR Square root. SQR(16) returns 4, SQR(0) returns 0, SQR(-16) provokes an error message.

Next the trigonometric functions:

DEG The argument is an angle in radians; the function returns the value of the angle in degrees. DEG(PI) returns 180, DEG(-1) returns -57.29578, DEG(0) returns 0

RAD The argument is an angle in degrees; the function returns the value of the angle in radians. RAD(180) returns 3.141593, RAD(57.29578) returns 1

SIN The sine of an angle measured in radians. SIN(-PI/6) returns -0.5, SIN(0) returns 0

ASN Arcsine; "The angle whose sine is..." ASN(-0.5) returns -.5235988, ASN(0) returns 0

COS The cosine of an angle measured in radians, COS(-PI/6) returns 0.8660254, COS(0) returns 1, COS(PI) returns -1

ACS Arccosine; "The angle whose cosine is..." ACS(1) returns 0, ACS(-1) returns 3.141593

TAN The tangent of an angle measured in radians. TAN(0) returns 0, TAN(PI/4) returns 1

ATN Arctangent; "The angle whose tangent is..." ATN(0) returns 0, ATN(1E20) returns 1.5708 (very nearly PI/2), ATN(-1) returns -.7853982

Next the hyperbolic functions:

SNH Sinh; the hyperbolic sine of argument x, or (e^x-e^-x)/2

CSH Cosh; the hyperbolic cosine of argument x, or (e^x+e^-x)/2

TNH Tanh; the hyperbolic tangent of argument x, or SNH(x)/CSH(x)

Special functions:

MMI Millimetres to Inches conversion with rounding. If UNITS=2 then MMI(40) returns 1.5, else returns 40. The inches are in steps of 1/8" up to 1.5", then in steps of 1/4" up to 6", then in steps of 1/2" up to 12", thereafter in 1" steps.

ARR Used for accessing the NL-STRESS arrays file. See section 5.11.8.

VEC Used for multiple assignments. See section 5.4.8.

RAN Random number generator. RAN(seed) returns a random number in the range >=0 and <1.0; where 'seed' is any integer number in the range 1 to 32000.

5.4.5 Variables

A name invented for a variable should not be the same as any keyword. To avoid confusion, keywords in NL-STRESS are in capital letters; names of variables start with a lower case letter which may be followed by further lower or upper case letters, digits or apostrophes. Where it is necessary to start a variable with an upper case letter, then the variable should be prefixed by a plus.

Examples of names of variables are:

```
f'c  fy  d'   dia  fs2   hMIN  z   alpha1   +Psi   a(i+1)  bc(i,j)
```

Any number of characters may be used to compose the name of a variable but NL-STRESS ignores those after the sixth, epsilon6 and epsilon7 would both be treated as the same variable, epsilo.

Before use as an item of data or in an expression, the variable must be assigned e.g. fcu=30. Such an assignment may be placed in any line of data save in a title or comment line where it would be ignored.

A variable name may be subscripted as in the last two examples above. The penultimate example shows a singly subscripted variable, for which there is no need to declare size in a dimension statement. The subscript may be an integer or variable, or a single integer or variable with addition/subtraction of another integer or variable. A subscripted variable may have no more than three characters in its name (the part before the opening bracket) and there may be no spaces within or between the name or subscript. Each subscript must evaluate to an integer.

The last example shows a doubly subscripted variable bc(i,j), for which there is a need to declare its size. Doubly subscripted arrays must have the number of columns declared, i.e. one dimension needs to be set. The dimension is set by assigning it to the array name e.g. bc=3, before the first use of the array in doubly subscripted form. Functionality is important in programming; NL-STRESS allows subscripted variables to be used in: non-subscripted form e.g. bc6
singly subscripted form bc(a-7)
doubly subscripted form bc(i,j).
As stated above, it is necessary to declare the dimension of an array before its first use in doubly subscripted form. NL-STRESS stores its elements left to right, top to bottom, thus if bc=3 then the array bc(,) contains:

```
┌ bc(1,1) bc(1,2) bc(1,3) ┐
│ bc(2,1) bc(2,2) bc(2,3) │
│ bc(3,1) bc(3,2) bc(3,3) │
└ bc(4,1) bc(4,2) bc(4,3) ┘
```

It follows that bc6=bc(6)=bc(2,3). One use of such functionality is that a doubly subscripted array may be assigned on a single line e.g. ac=3 ac1=VEC(1,0,0,0,1,0,0,0,1) which would set up a unity matrix ac(,)= $\begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$ without the need for doubly nested loops.

As just a single dimension (for the number of columns) is set for doubly subscripted variables, the number of rows in the array may be less than, equal to, or greater than the number of columns. The array bc() above has 3 columns, and four rows.
The width (number of columns) may be re-dimensioned as required, e.g. if bc=6 is assigned then the array bc(,) is referenced:

```
[bc(1,1) bc(1,2) bc(1,3) bc(1,4) bc(1,5) bc(1,6)]
[bc(2,1) bc(2,2) bc(2,3) bc(2,4) bc(2,5) bc(2,6)]
```

i.e. two rows when previously dimensioned =3, have now been put on a single row; the order - as stated before - is always left to right, top to bottom, and has not been changed, only the referencing.

Internally, NL-STRESS has two stacks for variables: VSTAK() for general variables of all types; VAR() for variables va(1:8000), vb(1:8000), vc(1:8000), vd(1:8000). Read/write access to these special arrays is quicker than access to the general stack. These special arrays may be used as singly or doubly subscripted variables, as described above.

5.4.6 Expressions

Examples of expressions are (1.5+7)/3, 1+SIN(RAD(45)), 1+SIN(x). An expression comprises terms bound together by operators and nested within brackets - much as expressions in algebra. A 'term' may be:

- a number (e.g. 1.5)
- a function (e.g. RAD(45))
- a variable (e.g. x).

An 'operator' may be:

^ to raise to a power. This operator has highest precedence; in other words in the absence of brackets it is applied before any other operator

* and / to multiply or divide respectively. These operators have equal precedence beneath that of ^

+ and - to add or subtract respectively, or as a prefix (e.g. -3). These operators have equal precedence beneath that of * and /.

Brackets may be used to change the precedence of operators from the pattern described. Thus 4-2^2 reduces to 4-4=0 whereas (4-2)^2 reduces to 2^2=4.

Brackets may be 'nested'. For example: (x^(2*(3+5)))/6

Spaces are not permitted in an expression: 6+3^2 may not be typed 6 +3^2 because that would signify two values: 6 and 9.

Although NL-STRESS is not intended to be used for the production of general calculations (SCALE is designed for this); nevertheless NL-STRESS does permit the engineer - when writing NL-STRESS data in parametric form - to replace expressions (& assignments, see 4.8) by their final numerical value. If it is required that an expression such as 3*(12.4+a) should be shown as 52.2, preface the expression with a plus sign. Thus the expression

```
+3*(12.4+a) kN/m
```

will be shown in the data (at the beginning of the results) as

```
52.2000 kN/m
```

assuming the variable 'a' held the value 5 when the results were being written. Please note that:

- the printed field has a maximum of 12 characters with four decimal places shown, left adjusted at the '+'
- to avoid causing confusion when using this facility do not re-assign the variable 'a' in the data file, rather use a new variable e.g. 'b'
- the text which follows the expression i.e. kN/m, has the single space between preserved; had there been more than a single space between, the text would not have been shifted to the left.

Assuming assignments: +AX'cd=3 a=3 +B=a*RAD(AX'cd) +C=a*10+.5+B
Examples of EXPRESSIONS and how they are SHOWN in the results:

```
 a                          a
+a                          3
+AX'cd                      3
 a*10+.5+B                  a*10+.5+B
+a*10+.5*B                  30.6571
```

N.B. AX'cd on its own will be faulted.

5.4.7 Separators

Keywords and values in a line of data must, in general, be separated from one another by spaces or commas or both. For example the keyword AX, qualified by a value of 0.623, may be typed in any of the following ways:

```
AX   0.623

AX,0.623

AX,      0.623
```

but not as AX0.623 which would be treated as an unrecognisable keyword.

No separator is allowed between the function keywords SIN, INT etc. and the subsequent value or expression enclosed in brackets. SIN (alpha) is an error.

No separator is allowed on either side of the equals sign in an assignment. No separator is allowed inside an expression.

In a title, commas and spaces are simply part of the title; they do not 'separate' anything, but there must be a separator between the title and its introductory keyword.

If errors are found, then the line reference in the error message is that corresponding to the line reference by any text editor. Examples of several NL-STRESS statements follow on the next 13 lines.

```
NUMBER OF JOINTS 2
NUMBER OF SUPPORTS 2
NUMBER OF MEMBERS 1
```

```
NUMBER OF LOADINGS 3
JOINT COORDINATES
*/5
1 0 0 SUPPORT
2 a 0 SUPPORT
JOINT RELEASES
1 MOMENT Z
2 FORCE X MOMENT Z
MEMBER INCIDENCES
1 1 2
```

5.4.8 Assignments

Before use as an item of data, or in an expression, a variable must be assigned. For clarity it is best not to mix assignments with other kinds of data on the same line. An example of a line containing assignments is:

```
f'=0.7*0.98,  ang=f'*0.89^2/12
```

As an alternative, it is permissible to include an assignment in any line of data save a title or comment line. For example in a line of joint coordinates:

```
12   x=2700+2*2300   x   x*COS(theta)
```

It is necessary to include the x on its own for the assignment serves only to assign the value of 7300 to x and does not associate x with any keyword. To avoid confusion it is suggested that the assignment is placed at the start of a line of data or preferably on a separate line.

The rules for spacing in the assignment line are the same as those in an ordinary line of data. Assignments should be separated from one another and from other items of data by separators. An assignment, such as f=0.7*0.89, may have no space on either side of the equals sign or in the expression itself.

Examples of ASSIGNMENTS and how they are SHOWN in the results:

```
+AX'cd=3                      AX'cd=3
+a=3                          a=3
 a=3                          a=3
+B=a*RAD(AX'cd)               B=a*RAD(AX'cd)=.1571
                              N.B. zero in front of decimal
                              point omitted to save space.
+C=a*10+.5+B                  C=a*10+.5+B=30.6571
```

WARNING - AVOID REASSIGNING VALUES TO THE SAME SYMBOLIC NAME/VARIABLE

```
b=2
NUMBER OF JOINTS +b
JOINT COORDINATES b=3
1 0 b
```

In the previous four lines of data, which commence and end with b, the variable b is firstly assigned the value 2 for use as the number of joints, then re-assigned with the value 3 for use as the Y ordinate for joint 1 in a plane frame. Such re-assignments have the potential to introduce bugs into the data. NL-STRESS, as with other software languages, carries out more than one pass through the data. The first pass reads the data, the second pass re-reads the data and writes it to the results. During the second pass, assignments are not carried out, the data is copied directly to the results only substituting the latest numerical values from the stack in any expressions which commence with a plus sign. Thus the results will contain:

```
b=2
NUMBER OF JOINTS 3
JOINT COORDINATES b=3
1 0 b
```

The above example will not cause the wrong results to be produced, but will cause confusion for the engineer when checking the number of joints. The remedy is to avoid re-assigning values to the same symbolic name, colloquially referred to as a 'variable'. As a further example suppose we require the smaller of two variables 'a' and 'b' to be stored in 'a'

```
IF b<a THEN * +a=b +c=27
SOLVE
...
c=54
```

If a=5 & b=3 then the result of the conditional IF would be to assign a=3 during the first pass through the data; during a second pass as both 'b' and 'a' hold the same value the conditional 'IF b<a' will not be carried out. If 'c=54' has been assigned after the SOLVE and before the FINISH then the engineer may expect that 'c=27' during the second pass through the data before the SOLVE but 'c' will contain 54.

On occasion when using parametric data it is necessary to assign a sequence of subscripted variables e.g. a(12)=3.2 a(13)=b a(14)=-5.7

As an alternative to the above NL-STRESS has a VEC function, VEC is short for VECtor, e.g. a12=VEC(3.2,b,-5.7). In a strict mathematical sense a set of value is not necessarily a vector, but in a programming sense the term vector is used to describe a one dimensional array. For the above example, a(12) is assigned the first value =3.2, a(13) the second =b, a(14) the third =-5.7. Each data item within the VEC() function, must be a single non-subscripted variable, or a single number prefixed with an optional minus sign. Negative decimal numbers less than 1, should have a leading zero before the decimal point e.g. b1=VEC(127,-0.45,tot).
The maximum permissible number of data items within the brackets is 25. The data items must be separated by commas and there must be no spaces between the brackets. For regularly repeating values it is permissible to add a multiplier after the closing bracket e.g. a12=VEC(3.2,b,-5.7)*2 which causes the assignments to be continued for a second time thus a(15)=3.2, a(16)=b, a(17)=-5.7. As a further example, assuming the variable b=200 then a(1)=VEC(24345)*b would assign a(1) thru a(200) with the value 24345.

VEC may be used for regularly repeating values which are incremented
each time around. VEC(v1,v2,...vl)/n says repeat the values v1,v2...
'n' times incrementing the values by vl each time around, e.g.
a1=VEC(1,1)/5 will assign a1=1 a2=2 a3=3 a4=4 a5=5.
As a further example: asta=4 ainc=3.5 anum=11 followed
by a1=VEC(asta,ainc)/anum will generate: a1=4 a2=7.5 a3=11 a4=14.5
a5=18 a6=21.5 a7=25 a8=28.5 a9=32 a10=35.5 a11=39. The VEC function
7 is for assigning 2 or more variables, thus a1=VEC(3) will be faulted.
The VEC function is cosmetically changed when it is included in the
results e.g. pro(1)=VEC(1.6,-1E3,27.32) is printed in the results as:
pro(1)...=(1.6,-1E3,27.32) to be more meaningful to the checker.

Another example: a set of 37 bending moments may be established by 3 parameters e.g. the maximum bending moment mbm=240 kNm, the number of lines nol=37, the bending moment increment bmi=-mbm/(nol-1), followed by BM1=VEC(mbm,bmi)/nol would generate bending moments:

```
240,233.33,226.67,220,213.33,206.67,200,193.33,186.67,180,173.33,
166.67,160,153.33,146.67,140,133.33,126.67,120,113.33,106.67,100,
93.333,86.667,80,73.333,66.667,60,53.333,46.667,40,33.333,26.667,
```

20,13.333,6.6667,0 on a falling scale
or the maximum bending moment mbm=240 kNm, the number of lines nol=37, the bending moment increment +bmi=mbm/(nol-1), followed by BM1=VEC(0,bmi)/nol would generate bending moments:
0,6.6667,13.333,20,26.667,33.333,40,46.667,53.333,60,66.667,
73.333,80,86.667,93.333,100,106.67,113.33,120,126.67,133.33,
140,146.67,153.33,160,166.67,173.33,180,186.67,193.33,200,
206.67,213.33,220,226.67,233.33,240 on a rising scale.

Although NL-STRESS is not intended to be used for the production of general calculations (SCALE is designed for this); nevertheless NL-STRESS does permit the engineer - when writing NL-STRESS data in parametric form - to replace assignments (& expressions, see 4.6) by the final assigned value. If it is required that an assignment such as c=3*(12.4+a) should be shown as 52.2, preface the assignment with a plus sign. Thus the assignment
+c=3*(12.4+a)
will be shown in the data (at the beginning of the results) as
52.2000
assuming the variable 'a' held the value 5 when the results were being written. Please note that:

- the printed field has a maximum of 12 characters with four decimal places shown, left adjusted at the '+'
- to avoid causing confusion when using this facility do not re-assign the variables 'a' or 'c' in the data file, rather use new variables e.g. 'd' and 'e'
- the text which follows the assignment i.e. 'N/m', is not shifted to the left as there is more than one space between it and the end of the assignment; had there been just a single space between, the text would have been shifted to the left
- assignments on lines which are not comment lines are made at the start of the analysis when the data is read
- assignments on lines which are comment lines (those starting with an asterisk) are made when the results are being written.

NL-STRESS benchmark SW21.BMK gives examples of assignments on lines which start with an asterisk and those which do not.

Occasionally there is a requirement to write conditionally a variable to the main stack so that it may be tested. To add a variable to the stack so that its value can be tested, without altering its value if it already has a value, assign to it the pseudo value 1E39 e.g. d=1E39.

If 'd' was already on the stack, then this assignment would be ignored, and the value held by 'd' would be that held before the assignment. If 'd' was not already on the stack, then the value held by 'd' would be 1E39. This numerical device has use in proforma sc461.pro which writes a data file for portal frame analysis and then invokes NL-STRESS. The device is similar to that used in SCALE and described in praxis.hlp.

PI (see 4.1) is a variable which is put on the stack containing 3.14159... so that it may be available for use in other assignments. One special assignment is 'gen=0', which is put on the stack to permit values to be passed to NL-STRESS when it is being run in batch mode. For example proforma sc964 generalises a verified model which is invoked by NL-STRESS which in turn is invoked by SCALE which passes the pagelength =-1 to NL-STRESS via cc924.stk to tell NL-STRESS to omit page headings from the results as the results are required to be included in the results file of SCALE.

5.4.9 Comment lines

A 'comment line' is a line beginning with an asterisk. For example:

```
* Allow 8% for connections.
```

The purpose of a comment line is to allow the engineer to add remarks in the data to help subsequent checking. NL-STRESS permits the inclusion of expressions and assignments in comment lined in the results if the PRINT DATA command has been used. For example:

```
* EN 1993-1-1:2005 (E) Clause 6.2.1. is a conservative interaction
* formula +uns=ABS(Ned)/Nrd+ABS(Myed)/Myrd+ABS(Mzed)/Mzrd
```

An asterisk followed by a slash is used for page control. When a figure/table is required in the NL-STRESS results; it would be annoying if half the figure were to be shown on one page with the remainder on another. To avoid this, a '/' following an asterisk tells NL-STRESS to go to start of next page before printing anything which follows. If an integer number follows the '/' as in the example below (in the range 1 to the number of lines available on a page) then it is interpreted as: if there are not 16 lines available on the current page, go to the start of next page before printing anything which follows. If an integer number does not follow the '/' then it is interpreted as go to the start of the next page regardless of how many lines are printed on the current page. In either case the line starting with the asterisk is printed as a blank line.

```
*/16
```

5.4.10 Exclamation mark

A line starting with an exclamation mark is a comment line which is ignored, any characters following the exclamation mark including the mark itself are ignored and are not copied to the results file.

Characters which follow an isolated exclamation mark i.e. an exclamation mark which has a space before it and a space after it, are ignored save for including them (but not the exclamation mark) in the results, if the keyword DATA follows the PRINT command. Generally, parameters precede ! and help follows.

The purpose of the exclamation mark is to allow the engineer to include private comments in the data e.g.

```
!Revised by architect on 7.7.97.
```

because a space does not follow the exclamation mark, the mark itself and all characters which follow are not copied to the results file.

The purpose of the isolated exclamation mark is to allow the engineer to include public comments in the data e.g.

```
IF nbay>2 THEN nstory=nbay+1  ! When > 2 bays, set storeys = nbay+1.
```

which would be copied to the results file as:

```
IF nbay>2 THEN nstory=nbay+1    When > 2 bays, set storeys = nbay+1.
```

5.4.11 Control and repetition

The REPEAT facility makes it unnecessary to duplicate sets of lines in the calculation file when the only difference lies in the values of variables. The structure is introduced by the line:

```
REPEAT
```

On a line somewhere below REPEAT must be the line:

```
ENDREPEAT
```

to match the REPEAT. Between REPEAT and ENDREPEAT must be the control word UNTIL followed by a single condition e.g. UNTIL a>b or a compound condition e.g. UNTIL a>b OR i=5

On meeting REPEAT, NL-STRESS takes note of the number of the line which follows REPEAT. This is the line to which NL-STRESS must return on meeting ENDREPEAT. NL-STRESS would "loop" indefinitely unless offered an escape by the condition after UNTIL. On meeting UNTIL, NL-STRESS evaluates the associated condition. If the condition proves to be true NL-STRESS leaves the loop and deals with the line following ENDREPEAT.

The REPEAT-UNTIL-ENDREPEAT has special usage when it encloses a loading condition. NL-STRESS recognises that the loading condition is within the REPEAT-UNTIL-ENDREPEAT programming structure and automatically generates a loading case number at the end of the LOADING command when it tabulates the results, thus avoiding the problem of having identical titles for all enclosed loadings.

Any number of REPEAT-UNTIL-ENDREPEATs may occur in an NL-STRESS data file, each must be closed before the next opened. (Although 'nesting' of REPEAT-UNTIL-ENDREPEATS is supported in SCALE, it is not supported in NL-STRESS.) Nesting in NL-STRESS, to any level, is provided by the more intuitive 'conditional GOTO'. The following example of a doubly nested loop needs no explanation:

```
i=0 j=0
:100
i=i+1
...
:200
j=j+1
...
IF j<20 GOTO 200
IF i<10 GOTO 100
```

WARNING in any looping, whether it be as above, or as a REPEAT-UNTIL -ENDREPEAT, or when using the BLOCK command, do not increment the counter on a line which commences with an asterisk, thus

```
* +i=i+1
```

on the line following the label :100, would give unreliable results.

When data is written parametrically, sometimes the parameters need to be changed, for example if one of the parameters is storey height then it will generally be constant, except the ground to first will likely be different. Two 'IF' programming structures are available: IF-ENDIF, and IF-THEN. Example of each form follow:

```
height=3.0
IF n=1 THEN height=3.3...

height=3.0
IF n=1
height=3.3
....
ENDIF
```

Both of the 'IF' examples shown to the left do the same thing. The first has the advantage that only one line of data is needed. The second has the advantage that within the IF-ENDIF programming structure, many lines of data can be included.

THRU,STEP,BUMP keywords of NL-STRESS permits many joints or members to be referenced in a single line of data e.g. 6 THRU 15 STEP 3 refers to the joints or members: 6, 9, 12 & 15. NL-STRESS keywords REPEAT-UNTIL-ENDREPEAT combined with the use of 'variables' in an NL-STRESS data file (symbolic names m and j below) extend the repetition, as shown in the finite element example:

```
m=-36 j=-1
REPEAT
m=m+36 j=j+1
m+1 THRU m+6  ELEMENT j+9,j+8,j+1,j+2
m+7 THRU m+12 ELEMENT j+16,j+15,j+8,j+9
m+13 THRU m+18 ELEMENT j+23,j+22,j+15,j+16
m+19 THRU m+24 ELEMENT j+30,j+29,j+22,j+23
m+25 THRU m+30 ELEMENT j+37,j+36,j+29,j+30
m+31 THRU m+36 ELEMENT j+44,j+43,j+36,j+37
UNTIL j=5
ENDREPEAT
```

The above may be condensed further to:

```
m=-36 j=-1
REPEAT
m=m+36 j=j+1
m+1 THRU m+36 ELEMENT j+9,j+8,j+1,j+2 BUMP 7
UNTIL j=5
ENDREPEAT
```

As mentioned above, one REPEAT-UNTIL-ENDREPEAT may not be nested within another, though many REPEAT-UNTIL-ENDREPEAT's may be contained within a single data file, or may be repeated with a different set of parameters, using the GOTO command. In the data below, 'ELEMENT T 0' makes a hole in a plate or wall. The first time through the loop, the positions for hole xs(nh),xe(nh)... refer to the first hole (nh=1). After the ENDREPEAT, the hole-number is incremented to 2, and the GOTO 7 causes the block of data, contained within, to be repeated to make the second hole and so on until all 'nh' holes have been made. The label (7 in this case) must be in the range 1 to 1000.

```
:6
hn=1
:7 <-----------------------------------------+
n=ys(hn)                                     |
REPEAT                                       |
ms=6*(nx*n+xs(hn)) n=n+1                     |
ms+1 THRU ms+6*(xe(hn)-xs(hn)) ELEMENT T 0   |
UNTIL n=ye(hn)                               |
ENDREPEAT                                    |
hn=hn+1                                      |
IF hn<=nh GOTO 7 ----------------------------+
```

Engineer beware, because the GOTO is a non-structured programming device, it is very easy to cause a never ending loop, e.g. if GOTO 7 is changed to GOTO 6, the REPEAT-UNTIL-ENDREPEAT is forever repeated for hole 1.

Nested IF-ENDIF are faulted in NL-STRESS data, so rather than write

```
┌ IF a=b                                       ┌ IF a<>b GOTO 100
│ IF c=d                ┌ IF a=b AND c=d       │ IF c=d
│ ...           write   │ ...              or  │ ...
│ ENDIF                 └ ENDIF                │ ENDIF
└ ENDIF                                        └ :100
```

Frequently NL-STRESS loading data takes the form of many load cases which contain blocks of very similar data e.g.:

```
LOADING CASE 2: Live pattern  1 1 0 1 1
MEMBER LOADS
1 FORCE Y UNIFORM W -(dlf*d1+llf*l1)
2 FORCE Y UNIFORM W -(dlf*d2+llf*l2)
3 FORCE Y UNIFORM W -(dlf*d3+0.0*l3)
4 FORCE Y UNIFORM W -(dlf*d4+llf*l4)
5 FORCE Y UNIFORM W -(dlf*d5+llf*l5)
```

To avoid repeating similar blocks of data to the above, for load cases 3, 4, 5 & 6, we can use the VEC() function (section 5.4.8) to make the MEMBER LOAD DATA for each block identical and then by using the BLOCK command (see CASE 3), tell NL-STRESS to use the same block again and again.

```
LOADING CASE 2: Live pattern  1 1 0 1 1
p1=VEC(1,1,0,1,1) line=LINE
MEMBER LOADS                                ! Line 63
1 FORCE Y UNIFORM W -(dlf*d1+p1*llf*l1)
2 FORCE Y UNIFORM W -(dlf*d2+p2*llf*l2)
3 FORCE Y UNIFORM W -(dlf*d3+p3*llf*l3)
4 FORCE Y UNIFORM W -(dlf*d4+p4*llf*l4)
5 FORCE Y UNIFORM W -(dlf*d5+pf*llf*l5)      ! Line 68

LOADING CASE 3: Live pattern  0 1 1 0 1
p1=VEC(1,1,0,1,1)
BLOCK line+1 line+6
```

It would have been permissible to give the above block command as say: BLOCK 63 68
but if the engineer added additional data at the start of the file then the BLOCK line numbers would need to be changed for each and every subsequent reference. LINE is a special variable which holds the current line number where the keyword LINE occurs, by inspection of the CASE 2 data it will be seen that MEMBER LOADS starts at line+1 and finishes at line+6; thus the command 'BLOCK line+1 to line+6' will point-to the correct block even if the engineer adds or deletes lines before CASE 2. It is advisable (though not mandatory) to keep the keyword LINE out of the BLOCK to be copied; this is done by the assignment line=LINE, the variable 'line' could have any name, but once assigned will keep its assigned value until it is re-assigned, cf. LINE which varies in value dependent on the line in which it occurs.

The BLOCK command and usage must not straddle the SOLVE command. When used before the SOLVE command, the defined block is not duplicated as the purpose of the BLOCK command is to minimise the data. When used after the SOLVE command the defined block is duplicated, so that it is consistent with REPEAT-UNTIL-ENDREPEAT looping used after the SOLVE command.

The BLOCK command is not supported by SCALE option 676/677 which replaces expressions with their numerical values and removes loops;

the BLOCK command can be flawed by editing the data file but omitting to update the line numbers; it is recommended that the following structure be used rather than the BLOCK command.

```
MEMBER INCIDENCES
a=9 b=2 c=b-1 d=a-1 e=-1 m=-5 fin=3
iret=10
:5
e=e+1 m=m+6 m THRU m+5 ELEMENT a+e,b+e,c+e,d+e
IF e<fin GOTO 5
GOTO iret
:10
a=17 b=7 c=b-1 d=a-1 e=-1 fin=7      iret=15
GOTO 5
:15
a=28 b=16 c=b-1 d=a-1 e=-1 fin=9     iret=20
GOTO 5
:20
a=41 b=27 c=b-1 d=a-1 e=-1 fin=11    iret=25
GOTO 5
:25
...
```

In the above, the 'block' is contained between a=9 and the first GOTO 5, and for this example sets element connectivity using the data: a=9 b=2 c=b-1 d=a-1 e=-1 m=-5 fin=3 iret=10.
On completion of the block, control is returned to label :10 by the 'GOTO iret'. The next set of data 'a=17 b=7 c=b-1 d=a-1 e=-1 fin=7 iret=15' is read and control goes back to label :5 and the cycle repeats, and so on.

It is permissible to write 'GOTO iret', but not permissible to have a label ':iret'. The colon must always be followed by an integer number in the range 1 to 32000.

| ASCII(124) as the first character on a line followed by numbers, variables, or expressions will cause the integer/real values to be displayed on the screen as a trace to keep a track of looping.
The first number following the bar should be the time in seconds for the display of each subsequent value, or a zero which WAITS FOR A MOUSE CLICK before displaying the next number.

All the programming structures of PRAXIS including IF-ELSE-ENDIF, DEFINE-ENDDEFINE, REPEAT-UNTIL-ENDREPEAT with high levels of nesting may be used in a SCALE proforma for the production of an NL-STRESS data file. SCALE proformas 560 to 600 give examples of proformas written to produce NL-STRESS data.

5.5 Notation for describing data

The arrangement of items in a line of data is defined by a special notation defined below. An example of a definition of a command in this notation is:

```
PRINT [ DATA|RESULTS ]  (FROM <page number>)
```

This illustrates three notational devices: vertical bars which say 'or'; square brackets which indicate that more than one item may be chosen; round brackets which enclose optional data.

Examples of lines of data written according to the above definition are:

```
PRINT DATA
PRINT RESULTS DATA
PRINT RESULTS, FROM 8
```

These notational devices are described in detail in this section.

5.5.1 Capital letters

Capital letters indicate keywords. When using a definition each keyword should be copied from the definition in full, or be abbreviated as far as its first four letters.

5.5.2 Pointed brackets < >

Words in pointed brackets describe the kind of data required; for example <page number> in the definition reproduced above.

Certain words are standard:

<title> indicates a title comprising any visible characters and spaces; for example: COMBINATION OF 1.1*DEAD LOAD + 1.5*LIVE is a title.

Words, numbers and symbols in titles have no intrinsic significance; they are simply characters in a title.

Titles are limited in length according to context, but may never extend beyond the end of the line of data in which they are typed.

<value> indicates a number, function, variable or expression

<members> denotes a sequence of one or more member numbers.

This term may be expressed in five ways; <value>
or <value> <value> BOTH
or <value> THRU <value>
or <value> THRU <value> STEP <value>
or [<value>] INCLUSIVE. For example:

```
27   (signifies member 27 only)
27 31 BOTH (members 27,31)
27 THRU 31 (members 27,28,29,30,31)
27 THRU 31 STEP 2  (members 27,29,31)
```

    27 5 6 32 33 INCLUSIVE (members 27,5,6,32,33)

The last form of this command is not available in the data generators but may be given when using any editor, including the one built-in to NL-STRESS, to prepare NL-STRESS data.

The THRU sequence should not miss the terminal value: 27 THRU 30 STEP 2 is an error. INCL is shown in the fourth example above; all NL-STRESS keywords longer than 4 characters may be shortened to just 4 characters.

<joints> denotes a sequence of joint numbers by the same conventions as <members>

In general the significance of words in pointed brackets is explained underneath the definition in which the words appear.

5.5.3 Vertical Bars |

A vertical bar says 'or'. Thus DATA|RESULTS offers a choice of precisely one of two keywords: DATA or RESULTS.

5.5.4 Spacing in definitions

Spacing in the definitions is significant. X 3.5, Y 3.6, Z 3.7 are all correct interpretations of X|Y|Z <value>. The close spacing of X|Y|Z specifies a single keyword as the first item. The space after the first item shows that <value> is the second item.

5.5.5 Square brackets [ ]

Square brackets indicate that the pattern of data defined inside the brackets may be given more than once. For example [DATA|RESULTS] permits any of the following interpretations:

```
DATA
RESULTS
DATA RESULTS
RESULTS DATA
```

The third example shows that items do not have to be in the same order as that in which they are listed in the definition.

Square brackets may be nested. For example:

```
[ FORCE|MOMENT [ X|Y|Z <value> ] ]
```

which may be interpreted as:

```
FORCE Y 27.5 X 36.2 MOMENT Z -9.7
```

5.5.6 Round brackets ( )

Round brackets signify optional data. For example, (GLOBAL) indicates that the keyword GLOBAL may be included in the line of data or omitted. Implications of omission are individually explained for each command or table.

5.6 Order of keywords in data

A complete set of data for NL-STRESS is shown in the Introductory Example. Analysis of this example shows commands and tables arranged in distinct groups:

- identification - TITLE, MADEBY, DATE etc.
- output wanted - PRINT etc.
- parameters - TYPE, NUMBER OF JOINTS etc.
- geometry - JOINT COORDINATES, MEMBER INCIDENCES, CONSTANTS etc.
- basic loadings - MEMBER LOADS, SELF WEIGHTS etc.
- combined loadings - COMBINE
- termination - SOLVE, FINISH.

This analysis demonstrates the usual ordering of a set of data. The remainder of this section defines the allowable order of data more formally.

5.6.1 Identification

The following keywords begin commands for ensuring that results are properly identified by titles, date etc.

TITLE,
MADEBY,
DATE,
REFNO

All these commands are optional. These commands, in any order among themselves, are usually placed first in a set of data. However, it is allowable to intersperse them among those listed in 'Output' and 'Parameters' below.

A set of results is shown in facsimile in the Introductory Example.

5.6.2 Output

The following keywords begin commands for specifying what output is wanted:

TABULATE,
PRINT

These are optional commands. They are usually placed after the Identification commands. However, they may be interspersed among the Parameters if desired.

The effect of TABULATE is 'global' in the sense that it applies to all subsequent loading conditions for which no contradictory 'local' TABULATE is given.

The TABULATE command may also be placed among data which specify a loading condition or combination. In such cases the 'local' TABULATE command supersedes any contradictory 'global' one, but only for that particular loading condition.

5.6.3 Parameters

The following keywords begin commands which declare global parameters. These specify the type of structure, number of joints etc. so that the software may set certain 'switches' and allocate storage space. The keywords are:

TYPE,
METHOD,
NUMBER OF JOINTS,
NUMBER OF MEMBERS,
NUMBER OF SUPPORTS,
NUMBER OF LOADINGS,
NUMBER OF INCREMENTS,
NUMBER OF SEGMENTS

The METHOD, NUMBER OF INCREMENTS and SEGMENTS commands are optional; all others mandatory.

These commands usually follow the Identification and Output commands, and can be in any order among themselves. However it is allowable to intersperse them anywhere among the Identification and Output commands.

5.6.4 Geometry

The following keywords are used as headings to tables of numerical data describing the geometry of the structure and properties of materials to be used in building the structure.

JOINT COORDINATES	mandatory: precisely one table
JOINT RELEASES	optional: no more than one table
MEMBER INCIDENCES	mandatory: precisely one table
MEMBER PROPERTIES	mandatory: precisely one table
MEMBER RELEASES	optional: no more than one table
CONSTANTS	mandatory: at least one command, more allowed

The JOINT COORDINATES table must be the first table of geometrical data; the other tables may follow in any order among themselves. The Identification, Output and Parameter commands must precede the JOINT COORDINATES table.

5.6.5 Basic loadings

A basic loading condition comprises a set of loads. The term 'basic' is used to distinguish such a condition from a 'combination' of basic loading conditions.

After all tables describing geometry and properties of materials comes the LOADING command to introduce a set of tables describing loads of the first loading condition. The end of this loading condition is marked by another LOADING command to introduce the next loading condition and so on. The final loading condition (basic loading or combination loading) is terminated by the commands SOLVE and FINISH.

The following keywords are used as headings to tables of numerical data describing a basic loading condition:

JOINT LOADS,
JOINT DISPLACEMENTS,
MEMBER LOADS,
MEMBER DISTORTIONS,
MEMBER TEMPERATURE CHANGES,
MEMBER SELF WEIGHTS,
MEMBER LENGTH COEFFICIENTS.

Each of these tables is optional; any may be used more than once in a single loading condition. The tables may be arranged in any order among themselves.

The TABULATE command may be interspersed among the above tables and would then apply locally to that loading condition.

5.6.6 Combinations

For METHOD ELASTIC only, after all basic loading conditions may come one or more combinations of basic loading conditions. Each combination is heralded by a LOADING command in the same manner as a basic loading condition. In the data for a combination the only commands permitted after LOADING are:

TABULATE (employed as described above) and:

COMBINE or
MAXOF or
MINOF or
ABSOF

One of these keywords is mandatory and must occur precisely once when defining a combination.

5.6.7 Termination

To terminate when data is incorrect, include a line such as:
IF ng<2 OR ns<1 THEN Number of girders/stiffeners too low.

The following commands come last in a set of data:

SOLVE,
FINISH

Every set of data must end with FINISH, and FINISH may be used nowhere else but at the end. The SOLVE command is optional; if omitted NL-STRESS checks the set of data, reports any errors found, then stops without attempting a solution.

5.7 Commands and tables

In this section every command and table is defined and its usage explained in detail.

The order in which commands and tables are presented in this section is the same as the usual order of arrangement of a set of data. The previous section defines this order and allowable departures from it.

A quick reference to the commands and tables defined in this section is provided in the next section.

5.7.1 The TITLE command

Syntax TITLE <title>

Purpose Records the title at the top of every page of results to identify the structure, contract, firm etc. responsible for the analysis.

Usage This command is optional. As many as four TITLE commands are permitted corresponding to the four lines at the top of each page of results. There should be no more than fifty characters in <title>.

Examples TITLE JOHN BROWN - ESTABLISHED 1867
TITLE SADDLER, HARNESS MAKER and HORSE CLOTHING
TITLE 309 ST VINCENT STREET GLASGOW and
TITLE CORN EXCHANGE BUILDINGS KILMARNOCK

5.7.1.1 GUI - Page headings entry

Select any of the menu options "Identification->TITLE, MADEBY, DATE, REFNO" or double click/tap on any of those commands displayed in the main GUI window to switch to the page headings entry screen, as shown in Figure 5.18. Amend the headings as required, then select "►" to accept changes, or "◄" to abandon changes.

Figure 5.18: GUI page headings.

5.7.2 The MADEBY command

Syntax MADEBY <title>

Purpose Records at the top of every page of output the initials of the engineer and checker.

Usage This command is optional. It may be used no more than once. There should be no more than eight characters in <title>.

Examples MADEBY DGA/DWB
MADEBY IF BROWN

5.7.3 The DATE command

Syntax DATE <title>

Purpose Records the date at the top of each page of output.

Usage This command is optional. It may be used no more than once. There should be no more than eight characters in <title>.

Examples DATE 30/6/15
DATE Mar.2015
DATE 07.07.15

5.7.4 The REFNO command

Syntax REFNO <title>

Purpose Records a reference number for a contract or job at the top of each page of output.

Usage This command is optional. It may be used no more than once. There should be no more than eight characters in <title>.

Examples REFNO JOB 534
REFNO HECB3302

5.7.5 The TABULATE command

Syntax TABULATE ([FORCES|REACTIONS|DISPLACEMENTS|STRESSES] |ALL)

Purpose Specify which of the available sets of results is to be printed.

Usage This command is optional. When used, its effect depends on where it is placed among the data:

- preceding JOINT COORDINATES, a TABULATE command applies to all loading conditions except those for which a local TABULATE command is given
- following a LOADING command, a TABULATE command applies only to that particular loading condition (or combination).

The order of keywords given after TABULATE establishes the order in which corresponding sets of results are printed. Omission of this command before JOINT COORDINATES implies TABULATE DISPLACEMENTS, FORCES, REACTIONS by default. It excludes STRESSES. TABULATE ALL implies TABULATE DISPLACEMENTS, FORCES, STRESSES, REACTIONS.

TABULATE, on its own, means tabulate nothing. This is a useful form of the command when results of a basic loading condition are of no interest on their own but only in combination with others.

Following the TABULATE command with the keyword STRESSES, causes a table of stresses to be produced for each member. Obviously bending stresses cannot be computed unless the data includes the distance from neutral axis to the extreme fibre. If the section properties are specified by the shape of sections which have symmetry about two axes, i.e. RECTANGLE, ISECTION, HSECTION, CONIC, then NL-STRESS can work out the distance to the extreme fibre by halving the overall depth of the section.

If the section has symmetry about one axis e.g. TSECTION, then NL-STRESS needs the distance CY i.e. the distance from the neutral axis to the required stress position. This may be the furthest distance to an extreme fibre - which will be the distance from the neutral axis to the end of the web which does not have the flange - thus computing and tabulating the higher stress. When CY is omitted then the bending stress is omitted from the results.

For sections which do not have an axis of symmetry e.g. angles when used in bending about axes x-x or y-y, then the NL-STRESS standards file (\SAND\NLS.STA) deliberately omits providing the distance to the extreme fibre. This omission is to stop angles being used in bending for - unless they are continuously restrained - they suffer from instability. Forty years ago structural hollow sections were expensive in comparison to angles, not so today, structural hollow sections are now a good choice for the internal members of lattices.

Examples
```
TABULATE ALL
TABULATE
TABULATE STRESSES, REACTIONS
```

5.7.5.1 GUI - TABULATE command entry

Either select the menu options "Output->TABULATE" or "Loading->TABULATE", or double click/tap on any TABULATE command displayed in the main GUI window to switch to the TABULATE entry screen, as shown in Figure 5.19.

Windows: the TABULATE keyword may also be added using the right mouse button popup menu.

If the current line is a loading title, TABULATE will be local to that loading.

You may click on Displacements, Forces, Stresses, Reactions in any order. The last button selected will place that item at the end of the line.

Clear clears the field. If you select "►" after this, the TABULATE line will be removed from the data window.

Click on none for TABULATE on its own.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.19: GUI TABULATE command entry.

5.7.6 The PRINT command

Syntax PRINT [DATA|SUMMARY|RESULTS|COLLECTION|DIAGRAMS|TRACE] (FROM <page> (LENGTH <length>))

Purpose Specify what kind of information NL-STRESS is to print.

Usage This command is optional; omission implies PRINT RESULTS FROM 1 by default.

The keyword DATA makes NL-STRESS print the set of data as received. The data are then printed on pages headed with name of firm, job number, date etc. as illustrated in the introductory example.

The keyword SUMMARY makes NL-STRESS print a summary of the joint, member and loading data. The summary is useful for checking purposes when the NL-STRESS data contains: variables, section properties specified by shape, REPEAT-UNTIL-ENDREPEAT, SYMMETRY and other shortened forms of the data especially MEMBER PROPERTIES such as: 3 THRU 4 AS 2 THRU 1 when the properties of members 3 and 4 vary.

Displacements, member forces, member stresses and support reactions are normally tabulated separately for each loadcase. Conversely, including the word COLLECTION in the print command causes the displacements for all loadcases for each joint to be collected and printed as a group. Similarly for member forces, member stresses and support reactions.

The keyword DIAGRAMS also collects the results of each loadcase together for each member, but presents the information as a diagram showing the bending moment and shear force envelopes. In addition to the values plotted, values of maximum and minimum deflection, dimensions and properties of the member, and maximum and minimum axial loads are tabulated.

In space frames two plots are made per member so as to show results relative to the local z axis as well as to the local y axis.

The diagrams define envelopes; a worst positive and worst negative result at each point along the member. The lines of the diagram are seen to be composed of digits. Digit 1 signifies that this value arises from loading case 1, digit 2 that the value arises from loading case 2 etc. (for load cases 10-35 the letters A-Z are used and for load cases 36 onwards the letters a-z). All load cases are shown in the diagrams (even if the load case has been suppressed by the TABULATE command) as one purpose of this facility is to show the build-up of the envelopes.

The keyword TRACE causes the appropriate arrays used in The analysis to be printed in the results; the array values may then be checked to trace that the various stages in the analysis have been carried out correctly, such a check requires a knowledge of the various arrays, see 5.11.8 for a summary.

The optional FROM clause (FROM 1 by default) specifies the number to be recorded as the page number of the first page of output. This facility enables the user to present results from NL-STRESS as a properly-numbered sequence of pages within a larger report. There are a number of features associated with the <page> given. If a starting page number (e.g. 132) is given, it will be printed following the word Page in the top right corner of the first page of calculations, subsequent pages being numbered 133, 134 etc.

If a starting page number such as C3/12 is given, it will be printed following the word Page in the top right corner of the first page of calculations, subsequent pages being numbered C3/13, C3/14 etc. NL-STRESS picks up the string of characters looking backwards from the end to extract an integer (12 in the example) and increments this on subsequent pages keeping the character string C3/ unchanged. An example of the first page is:

```
MANN WYFFE & PARTNERS                        Page:  C3/12
MEDIAEVAL JOUSTING FURNITURE              Made by:  DWB
KNIGHT HOIST                                 Date:  NOV1145
                                           Ref No:  KCMG-4ME
-------------------------------------------------------------
                                           Office:  5234
```

The number following 'Office' beneath the page heading is the licence number of the office in which SAND is in use.

If a starting page number is given as 0, this will be treated as an instruction to omit all page numbering following the word Page on all pages of the results. To facilitate page numbering, a number starting from 1 for each particular run will be printed at the end of the line immediately below the heading as in the following example:

```
MANN WYFFE & PARTNERS                        Page:
MEDIAEVAL JOUSTING FURNITURE              Made by:  DWB
KNIGHT HOIST                                 Date:  NOV1145
                                           Ref No:  KCMG-4ME
-------------------------------------------------------------
                                           Office:  5234   1
```

If a starting page number without an integer suffix is given (for example CJA alone) then this will be printed following the word Page in the top right corner of the first page of the calculations and on subsequent pages. A number starting from 1 will be printed at the end of the line immediately below the heading to facilitate subsequent page numbering.

For those who wish to use NL-STRESS with their own printed stationery, enter the 'Start page number' as #. The # causes the top four lines of each page to be left empty.

The optional LENGTH clause (LENGTH 66 by default) specifies the number of lines in the page. Alternatively the page length may be set by altering SCALE.STA (NL-STRESS looks for this file and if found uses the page length set therein). Note from the syntax that if the LENGTH clause is specified, it is also necessary to specify the FROM clause. LENGTH -1 is interpreted as 'omit page headings', also LENGTH gen (where 'gen' is a variable holding -1).

Examples PRINT DATA

```
PRINT RESULTS FROM 11              (number pages 11, 12,...)
PRINT RESULTS FROM DB/3 LENGTH 70  (number pages DB/3, DB/4,
                                    ... using A4 continuous stationery)
```

5.7.6.1 GUI - PRINT command entry

Either select the menu option "Output->PRINT" or double click/tap on the PRINT command displayed in the main GUI window to switch to the PRINT command entry screen, as shown in Figure 5.20.

Windows: the PRINT keyword may also be added using the right mouse button popup menu.

You may select DATA, SUMMARY, RESULTS, COLLECTIONS, DIAGRAMS in any order. The last button selected will place that item at the end of the line.

Select the FROM field and enter the required data as required, the data in the FROM field is duplicated in the Page field on the GUI Page Headings screen.

Clear clears the field. If you select "►" after this, the PRINT line will be removed from the data window.

Finally select "►" to accept changes, or "◄" to abandon changes.

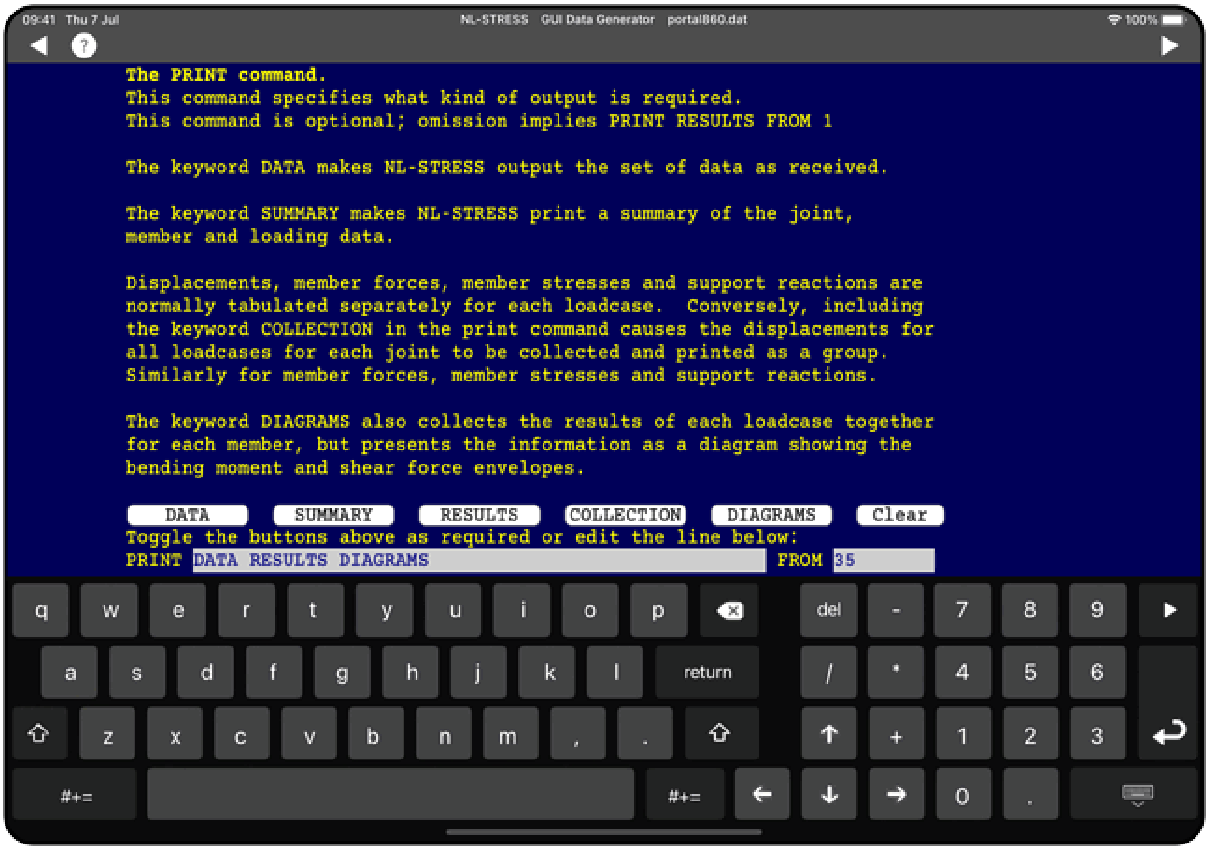


Figure 5.20: GUI PRINT command entry.

5.7.7 The TYPE command

Syntax TYPE PLANE|SPACE TRUSS|FRAME|GRID

Purpose Specify the type of structure: PLANE TRUSS, SPACE FRAME etc. This information is fundamental to all subsequent work on the problem.

Usage This command is mandatory. There are five possible forms of the command; all are given in the following examples.

Examples
```
TYPE PLANE TRUSS
TYPE PLANE FRAME
TYPE PLANE GRID
TYPE SPACE TRUSS
TYPE SPACE FRAME
```

5.7.7.1 GUI - TYPE command entry

Either select the menu option "Parameters->TYPE" or double click/tap on the TYPE command displayed in the main GUI window to switch to the TYPE command entry screen, as shown in Figure 5.21.

Windows: the TYPE keyword may also be added using the right mouse button popup menu.

Select the button representing the type of structure required.

Finally select "►" to accept changes, or "◄" to abandon changes.

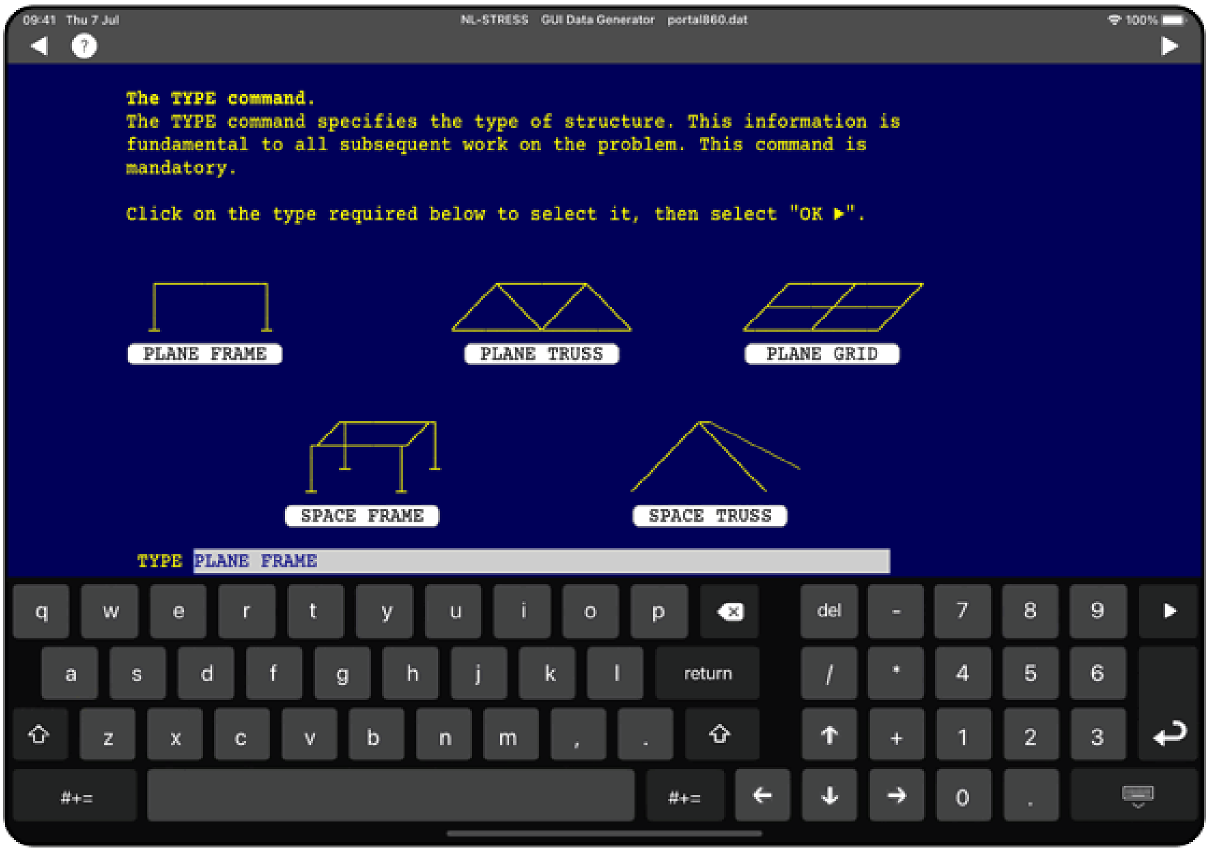


Figure 5.21: GUI TYPE command entry.

5.7.8 The METHOD command

Syntax METHOD ELASTIC|SWAY|PLASTIC (<percent>) (JOINTS|NODES)

Purpose Specify the method of analysis required. Also, the method by which NL-STRESS is to allocate 'node' numbers for setting up stiffness method equations.

Usage This command is optional; if omitted NL-STRESS assumes METHOD ELASTIC by default.

The keyword ELASTIC specifies elastic analysis with sway effect and Euler effect ignored. (The command NUMBER OF INCREMENTS should not follow the command METHOD ELASTIC).

The keyword SWAY specifies elastic analysis incorporating sway effect and Euler effect if desired. A NUMBER OF INCREMENTS command ought to follow METHOD SWAY. If within-member stability effects are required a NUMBER OF SEGMENTS command should also follow METHOD SWAY.

The keyword PLASTIC specifies elastic-plastic analysis (with sway effects incorporated) by increments of loading. A NUMBER OF INCREMENTS command should follow METHOD PLASTIC, and if plastic hinges are expected within any member, a NUMBER OF SEGMENTS command should also follow.

After each increment of loading the ends of all members and segments are examined for plastic behaviour and hinges inserted or modified according to the interaction formula employed.

NL-STRESS permits the engineer to model the hinge stiffness remaining after a plastic hinge has formed by specifying a percentage of the plastic moment following the METHOD command e.g. METHOD PLASTIC 5 which would specify that 5% of the plastic moment be used as the hinge stiffness. If the percentage is omitted NL-STRESS assumes a percentage of 100/(number of loading increments) i.e. 1% for a loading applied in 100 increments.

Omission of the keyword JOINTS as well as the keyword NODES makes NL-STRESS allocate 'node' numbers to joints in the order in which joint numbers are presented in the data.

For example, if the order of joint numbers in the JOINT COORDINATES table reads 2, 4, 3, 1,... then joint 2 becomes node 1, joint 4 becomes node 2, joint 3 becomes node 3, joint 1 becomes node 4,...

The keyword JOINTS signifies that joint numbers are to be treated as node numbers.

The keyword NODES tells NL-STRESS to derive a correspondence between joint numbers and node numbers such as to reduce the 'band width' to a suitably small value. The 'band width' may be found by looking at every member and finding the difference between the node numbers at its ends. The biggest difference establishes the 'band width'. The smaller the band width, the more efficiently NL-STRESS can analyse the frame.

Wherever a member is divided into segments NL-STRESS includes invisible nodes at the junctions. For any structure having invisible nodes, NL-STRESS acts as though the keyword NODES had been given - overriding the actual METHOD command if contradictory.

Any joint renumbering to minimise bandwidth, takes place after the member incidences have been read. Before joint renumbering the node and joint numbers are the same. When fixities (-1) are applied in the JOINT RELEASES table to the additional joints added for the segmenting, it is recommended that the MEMBER INCIDENCES table comes before the JOINT RELEASES, so that renumbering will have already taken place and in consequence the node and joint numbers for the additional joints at the ends of each segment have been established.

For compatibility with an earlier version of STRESS, the command METHOD STIFFNESS JOINTS may be given in place of METHOD ELASTIC JOINTS; similarly METHOD STIFFNESS NODES in place of METHOD ELASTIC NODES.

Examples METHOD ELASTIC
METHOD SWAY NODES
METHOD PLASTIC JOINTS

5.7.8.1 GUI - METHOD command entry

Either select the menu option "Parameters->METHOD" or double click/tap on the METHOD command displayed in the main GUI window to switch to the METHOD command entry screen, as shown in Figure 5.22.

Windows: the METHOD keyword may also be added using the right mouse button popup menu.

Select the button representing the method required and optionally select NODES or JOINTS.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.22: GUI METHOD command entry.

5.7.9 The NUMBER OF JOINTS command

Syntax NUMBER OF JOINTS <number> (LISTING <extent>)

where <extent> may take any of the following four forms:

<joint> <joint> BOTH

[<joint>] INCLUSIVE

<joint> THRU <joint> (STEP <increment>)

#filename

Purpose Declare the number of joints to enable NL-STRESS to allocate storage space for the analysis, and optionally list those joints required to be tabulated.

Usage This command is mandatory. Numbers given as an expression must reduce to a positive integer: 2*7 is acceptable as 14 but 1.9999*7 would be an error. In the second example below, joint displacements would be reported for joints 1 2 and 12. In the third example below, joint displacements would be reported for joints 1 3 5 and 7. In the fourth example below, joint displacements would be reported for any joint numbers listed in the text file 'myfile'. The joint numbers in 'myfile' must be listed singly (the programming structure THRU-STEP is not supported). As with all NL-STRESS data, a maximum line length of 80 characters is assumed for 'myfile'.

Examples
```
NUMBER OF JOINTS 123
NUMBER OF JOINTS 12 LISTING 1 2 12 INCLUSIVE
NUMBER OF JOINTS 24 LISTING 1 THRU 7 STEP 2
```

NUMBER OF JOINTS 1246 LISTING #myfile

5.7.9.1 GUI - NUMBER OF JOINTS, MEMBERS, SUPPORTS, LOADINGS entry

Select any of the menu options "Parameters->NUMBER OF JOINTS, MEMBERS, SUPPORTS, LOADINGS" or double click/tap on the NUMBER OF JOINTS, MEMBERS, SUPPORTS, LOADINGS commands displayed in the main GUI window to switch to the NUMBER OF JOINTS, MEMBERS, SUPPORTS, LOADINGS command entry screen, as shown in Figure 5.23.

Windows: each of the above keywords may also be added using the right mouse button popup menu.

Enter the required numbers in the fields provided.

Finally select "►" to accept changes, or "◄" to abandon changes.

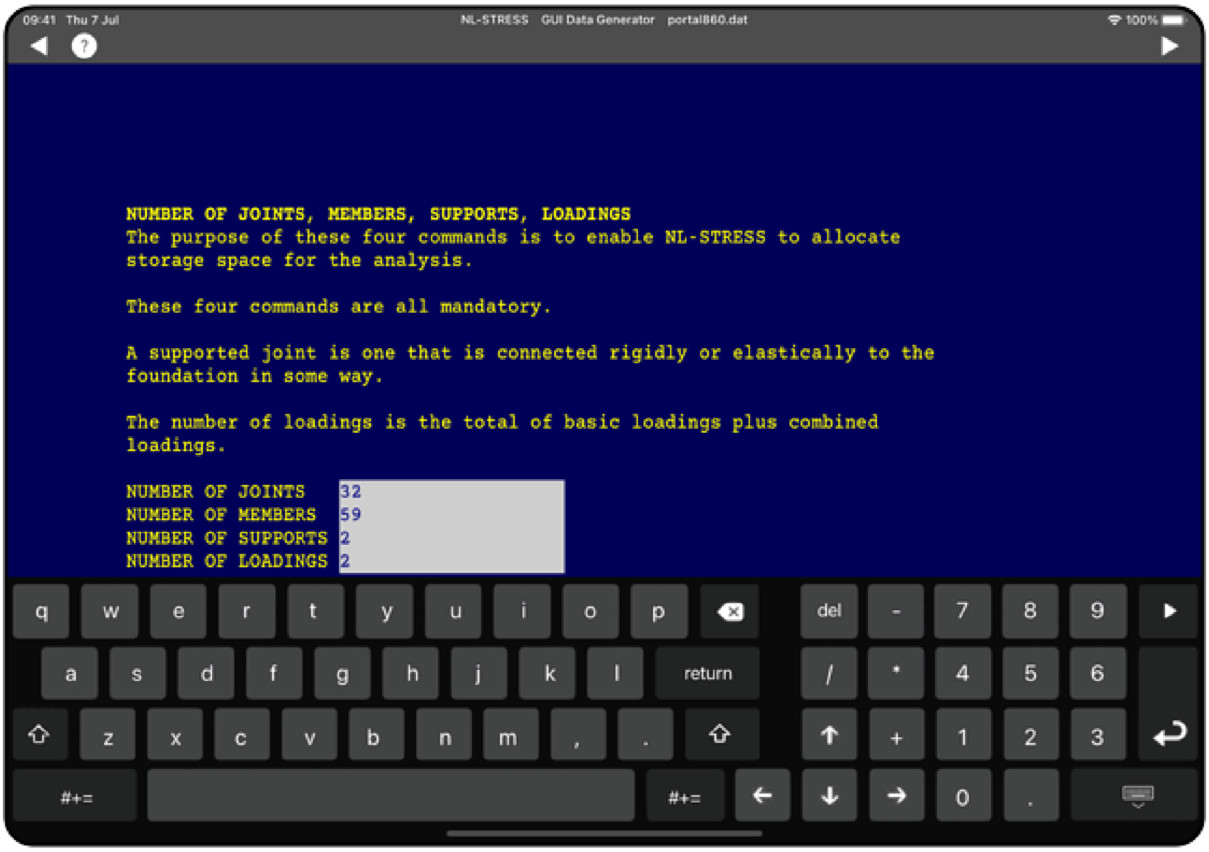


Figure 5.23: GUI NUMBER OF JOINTS, MEMBERS, SUPPORTS, LOADINGS.

5.7.10 The NUMBER OF MEMBERS command

Syntax NUMBER OF MEMBERS <number> (LISTING <extent>)

where <extent> may take any of the following four forms:

<member> <member> BOTH

[<member>] INCLUSIVE

<member> THRU <member> (STEP <increment>)

#filename

Purpose Declare the number of members to enable NL-STRESS to allocate storage space for the analysis.

Usage This command is mandatory. In the second example below, member forces & stresses would be reported for members 1 thru 12. In the third example below, member forces and stresses would be reported for any member numbers listed in the text file 'myfile'. The member numbers in 'myfile' must be listed singly (the programming structure THRU-STEP is not supported). As with all NL-STRESS data, a maximum line length of 80 characters is assumed for 'myfile'.

Examples
```
NUMBER OF MEMBERS 123
NUMBER OF MEMBERS 12 LISTING 1 2 12 INCLUSIVE
NUMBER OF MEMBERS 24 LISTING 1 THRU 12
NUMBER OF MEMBERS 1246 LISTING #myfile
```

5.7.11 The NUMBER OF SUPPORTS command

Syntax NUMBER OF SUPPORTS <number>

Purpose Declare the number of supports to enable NL-STRESS to allocate storage space for the analysis.

Usage This command is mandatory.

A supported joint is one that is connected rigidly or elastically to the foundation in some way.

MIT STRESS, STRESS-3, SuperSTRESS and NL-STRESS expect the engineer to tag 'S' or 'SUPPORT' in the JOINT COORDINATES TABLE to those joints which are supported. An error will be reported when the number of tagged joints found does not match the number of supports declared by the above command. Joints which have been tagged with 'S' or 'SUPPORT' but which are released in one or more directions (e.g. a pinned foot of a portal frame) have their releases or springs listed in the JOINT RELEASES table.

NL-STRESS recognises a spring stiffness of -1 entered in the JOINT RELEASES table (see 7.16) as full fixity in the declared direction for any joint, whether tagged or not tagged with 'S' or 'SUPPORT' in the JOINT COORDINATES table. Thus it is allowable to give the command 'NUMBER OF SUPPORTS 0' and list all the joint fixities as -1 in the JOINT RELEASES table. Although this practice may cause confusion to the checker, it can save considerable time in designating supports when a large finite element mesh has been generated by using i-THRU-j inside a REPEAT-UNTIL-ENDREPEAT. One word of warning, when using this feature to fix 'additional joints' (i.e. those that have been added to segment the members), it is essential that the MEMBER INCIDENCES' table comes before the JOINT RELEASES, for it is only after the MEMBER INCIDENCES have been read that the additional joints are defined.

Example NUMBER OF SUPPORTS 8

5.7.12 The NUMBER OF LOADINGS command

Syntax NUMBER OF LOADINGS <number>

Purpose Declare the number of load cases to enable NL-STRESS to allocate storage space for the analysis.

Usage This command is mandatory.

<number> is the total of basic loadings plus combined loadings. The maximum number of loadcases is 200; but note that other size limitations (e.g. insufficient disk space) may prevent an analysis being carried out.

Example NUMBER OF LOADINGS 12

5.7.13 The NUMBER OF INCREMENTS command

Syntax NUMBER OF INCREMENTS <number> (<accuracy> <cycles>) (TRACE)

Purpose When studying non-linear effects, to specify the number of equal increments over which a single loading is to be progressively applied.

Usage This command is optional. If omitted then INCREMENTS 1 is implied. <accuracy> is used as the percentage accuracy for convergence before the next increment of load is applied. If <accuracy> is omitted, a default value of 0.1% is used. <cycles> is used as a limit for the number of cycles that are carried out to satisfy compatibility and equilibrium, before the next loading increment is applied. If <cycles> is omitted, a default maximum value of 500 cycles is used. If <cycles> is given, then <accuracy> must also be given. <cycles> has particular importance in plastic analysis when the structure approaches instability due to the formation of plastic hinges.

The second example below, would cause the loading to be applied in 20 increments and make NL-STRESS ensure that all deflections had converged to within 0.5% of their predicted values before the next loading increment is applied; but would terminate the analysis if the deflections had not converged after 100 cycles at constant load.

TRACE, specifies that a set of results is to be printed after the application of each increment of load. From such results it is possible to trace the history of degradation from linear elastic behaviour through to collapse.

The number of increments should be in the range 1 to 500. The higher the number of increments the less the chance that a lower collapse mode is missed. In non-linear analysis, non-linear effects can cause local failure of a member which could be missed if the loading was increased by too high an increment.

The required contents of each set of results is specified by the TABULATE command currently in force.

Examples NUMBER OF INCREMENTS 10 TRACE
NUMBER OF INCREMENTS 20 0.5 100

5.7.13.1 GUI - NUMBER OF INCREMENTS command entry

Either select the menu option "Parameters->NUMBER OF INCREMENTS" or double click/tap on the NUMBER OF INCREMENTS command displayed in the main GUI window to switch to the NUMBER OF INCREMENTS command entry screen, as shown in Figure 5.24.

Windows: the NUMBER OF INCREMENTS keyword may also be added using the right mouse but right mouse button popup menu.

Enter the required data in the field provided.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.24: GUI NUMBER OF INCREMENTS command entry.

5.7.14 The NUMBER OF SEGMENTS command

Syntax NUMBER OF SEGMENTS <number> (<percent>) (TRACE)

Purpose Divide a member into segments of equal length. Additional nodes are then intrinsically defined between such segments.

Usage This command is optional. If omitted then NUMBER OF SEGMENTS 1 is implied. The number of segments should be in the range 1 to 100. <percent> is used to specify the percentage of the member length to be used to 'bow' each member for stability analyses. In the second example below, the 10 segments of each member are arranged in a bow such that the maximum displacement from the chord is 0.5% of the length of each member. Whereas MEMBER DISTORTIONS are used for studying 'lack of fit' problems (the member is distorted in the directions specified, then 'clamped' into the structure and let go); the bow specified in the NUMBER OF SEGMENTS command only tells NL-STRESS that each member has a parabolic bow (which does not give rise to stresses due to 'lack of fit'). The last item, TRACE, specifies that the set of results is to include the additional nodes in any displacements table; also the forces at the end of each segment in any table of member forces or stresses. When a bow percentage is specified, then NL-STRESS assumes that the additional nodes and members added to form the bow should be reported in the results and sets TRACE to 'on', thus in the second example, the keyword TRACE is redundant. Bow percentages less than 0.001% are ignored, i.e. ignored.

The bow is primarily intended for buckling analyses of plane frames and assumes a hogging bow. Bows in space frames are complicated by the BETA angle of rotation of the member; to keep things simple for space frames, the same treatment as that used for plane frames is used; but when the member to be bowed lies in the X-Y plane, the bow is also applied in the Z direction. To see a table of the coordinates for the bow, add the keyword SUMMARY to the PRINT command.

Examples NUMBER OF SEGMENTS 5
NUMBER OF SEGMENTS 10 0.5 TRACE

5.7.14.1 GUI - NUMBER OF SEGMENTS command entry

Either select the menu option Parameters->NUMBER OF SEGMENTS or double click/tap on the NUMBER OF SEGMENTS command displayed in the main GUI window to switch to the NUMBER OF SEGMENTS command entry screen, as shown in Figure 5.25.

Windows: the NUMBER OF SEGMENTS keyword may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.25: GUI NUMBER OF SEGMENTS command entry.

5.7.15 The JOINT COORDINATES table

Syntax JOINT COORDINATES (SYMMETRY X|Y|Z (<distance>))

<joint> <X-coord> <Y-coord> (<Z-coord>) (S|SUPPORT)

<joints> SYMMETRY <other joints> (S|SUPPORT)

<joints> [X|Y|Z|XL|YL|ZL <coord>] (S|SUPPORT)

<joints> AS <other joints> [X|Y|Z <bump>] (S|SUPPORT)

Purpose Define the coordinates of every joint in the structure relative to a convenient origin in global axes.

Usage This table is mandatory and there may be no more than one. It must be the first table describing the geometry of the structure. Rows of this table may take any of the three forms defined above. Every joint number - from 1 to the total number of joints - must be represented in the left hand column, but not necessarily in numerical order (see METHOD).

Coordinates are measured from any convenient origin. There should be either two or three coordinates per joint depending upon the TYPE of structure (PLANE or SPACE) declared.

SYMMETRY may be specified in the body of the table only if SYMMETRY is specified in the heading, in which case the specified joint is assumed to take coordinates which are the mirror image of those of some other joint. The coordinates of the 'other' joint/s must be specified earlier in the table; forward reference is not permitted.

The 'mirror' lies normal to axis X or Y or Z as specified in the heading. The mirror is located a given distance along that axis or at the origin by default.

In the second example below, joint 5 is located at a point with coordinates 2.5, 1.0. Notice that the SYMMETRY facility copies most coordinates of the joint to be reflected, recomputing only the coordinate along the axis on which the mirror is specified. The first two examples below illustrate alternative ways of presenting the same data.

A sequence of joints may be specified in the leftmost column of the table; for example 2 THRU 6. A corresponding sequence of coordinates must then be specified after appropriate keywords: X, Y, Z denote coordinates of the first joint in the sequence; XL, YL, ZL the coordinates of the last joint in the sequence. Omission of XL, YL, ZL and its value implies that the last coordinate is the same as the first. The final six examples below show different ways of presenting the same data.

Joints supported by the foundation must be tagged with the word SUPPORT or S. The number of such tags must add up to the number declared after NUMBER OF SUPPORTS. (Unsupported joints may be tagged FREE or F if desired; this only adds confusion so these keywords have been omitted from the definition of syntax above.)

Notice in the second example that joint 5 is tagged as a SUPPORT; the mirror is concerned only with coordinates, not support conditions.

When supports are applied to joints involved in implied sequences, they refer to all joints in the sequence.

In the last form of the syntax i.e.
<joints> AS <other joints> [X|Y|Z <bump>] (S|SUPPORT),
the AS says that <joints> are to have the same coordinates as the other joints incremented by the <bump> in the X, Y (and Z for space structures). <bump> may be positive or negative. The coordinates of the 'other' joints must be specified earlier in the table; forward reference is not permitted.

The last two examples below show identical use of the AS; in both examples the X & Y values are correctly shown as positive for in both examples the <joints> are to the right of the <other joints> and therefore have increased X coordinate. The AS command saves much time when say, a hundred untidy joints for the first floor has to be repeated for the second and subsequent floors with just the Y coordinated 'bumped'.

Examples

```
JOINT   COORDINATES
7       1.0,  1.0
5       2.5,  1.0   SUPPORT

JOINT   COORDINATES  SYMMETRY  X  1.75
7       1.0,  1.0
5       SYMMETRY 7,  SUPPORT

JOINT  COORDINATES
```

```
2       3.5   0
4       4.5   0
6       5.5   0
8       6.5   0

JOINT   COORDINATES
2  THRU  8  STEP  2, X 3.5, Y 0, XL 6.5, YL 0

JOINT   COORDINATES
2 THRU 8 STEP 2, X 3.5, XL 6.5, Y 0

JOINT   COORDINATES SYMMETRY X 5
2       3.5   0
6       5.5   0
4 THRU 8 STEP 4 SYMMETRY 6 THRU 2 STEP 4

JOINT   COORDINATES
2 THRU 4 STEP 2 X 3.5 Y 0 XL 4.5
6 THRU 8 STEP 2 AS 2 THRU 4 STEP 2 X 2.0 Y 0

JOINT   COORDINATES
4 THRU 2 STEP 2 X 4.5 Y 0 XL 3.5
8 THRU 6 STEP 2 AS 4 THRU 2 STEP 2 X 2.0 Y 0
```

5.7.15.1 GUI - JOINT COORDINATES table

Either select the menu option Geometry->JOINT COORDINATES table or double click/tap on any line in the JOINT COORDINATES table displayed in the main GUI window to switch to the JOINT COORDINATES table entry screen, as shown in Figure 5.26. The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

Use the edit box to the right of the JOINT COORDINATES title to enter any SYMMETRY data if required.

Select the "Draw Structure" button to enter the coordinates for 2D structures graphically with Draw Structure.
Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.26: GUI JOINT COORDINATES table.

5.7.16 The JOINT RELEASES table

Syntax JOINT RELEASES

<joints> [FORCE|MOMENT [X|Y|Z (<spring const>)]]

Purpose Simulate hinged joints, hinge-and-roller joints, elastic supports, and other special details at supported joints.

Usage This table is optional; if omitted then all supported joints (those tagged SUPPORT or S in the JOINT COORDINATES table) are assumed to be built in to the foundation. There may be no more than one such table.

Directions are specified with reference to global axes. Allowable directions of release depend on the type of structure declared:

Plane truss: FORCE X, FORCE Y
Plane frame: FORCE X, FORCE Y, MOMENT Z
Plane grid: FORCE Z, MOMENT X, MOMENT Y
Place truss: FORCE X, FORCE Y, FORCE Z
Space frame: FORCE X, Y, Z, MOMENT X, Y, Z

Linear spring constants are measured in force units/length unit (e.g. kN/m, k/ft). Angular springs are measured in 'moment units' (e.g. kNm, k-ft) per radian turned through. Radians are dimensionless, so angular spring constants are measured in 'moment units'. To guard against springs being associated with wrong directions, when a spring constant is applied in one of the given directions then spring constants must be applied in all given directions. Thus FORCE X 123.4 Y is a mistake; there should be a 0 after Y.

A special spring stiffness having the value -1 is recognised by NL-STRESS as full fixity. This spring stiffness may be applied to both supported and unsupported joints. One word of warning, when using this feature to fix 'additional joints' (i.e. those that have been added to segment the members), it is essential that the MEMBER INCIDENCES' table comes before the JOINT RELEASES, for it is only after the MEMBER INCIDENCES have been read that the additional joints are defined.

When post processing, to simplify extracting values from the stiffness matrix, all joints need some flexibility, the following treatment when combined with METHOD ELASTIC JOINTS ensures that every joint has some flexibility in all directions and therefore the row and column numbering are in joint order. If 'NUMBER OF SUPPORTS 0' has been given then the joints must be fixed (by -1) and then set as a high spring stiffness, the following gives an example:

```
JOINT RELEASES
nj=49 ny=6 i=0 fix=-1
:15
i=i+1
* Fix fully for first loop, then apply springs.
1 THRU 1+ny FORCE Z fix MOMENT X fix MOMENT Y fix
nj-ny THRU nj FORCE Z fix MOMENT X fix MOMENT Y fix
fix=1E12
IF i<2 GOTO 15
```

The first example below allocates a vertical fixity to joints 1 to 1000 of a structure of type PLANE GRID. The joint releases in the second example below are depicted in Figure 5.27.

Examples

```
JOINT RELEASES
1 THRU 1000 FORCE Z -1

JOINT RELEASES
27    FORCE X, MOMENT Z
28    FORCE X  12.3
29    FORCE X  10,  Y 10,  MOMENT Z 34.5
```

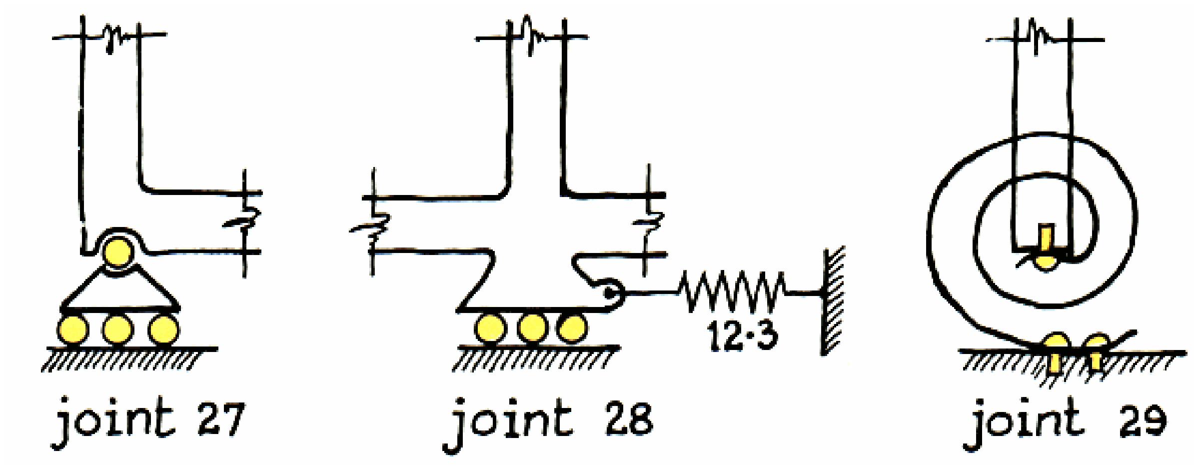


Figure 5.27: Examples of releases in joints.

5.7.16.1 GUI - JOINT RELEASES dialog

There are two ways of entering JOINT RELEASES in the GUI, either by entering one release at a time using the JOINT RELEASES dialog, or by editing the JOINT RELEASES table.

To enter data using the JOINT RELEASES dialog, select the menu option Geometry->JOINT RELEASES or select the button "Switch to JOINT RELEASES GUI" on the JOINT RELEASES table screen.

Examples of the JOINT RELEASES dialog are shown in Figures 5.28 and 5.29. The releases available for your current structure type are displayed on the dialog. The available directions for each structure type are listed in the previous section.

Enter the joint number data in the field provided and select the direction of joint release required. If no spring constant is required, leave that field blank.

Select "►" to add the joint release, or "◄" to return to the main GUI window without adding the release.

Select the "Switch to JOINT RELEASES table" button to switch to entering data using the GUI JOINT RELEASES table option.

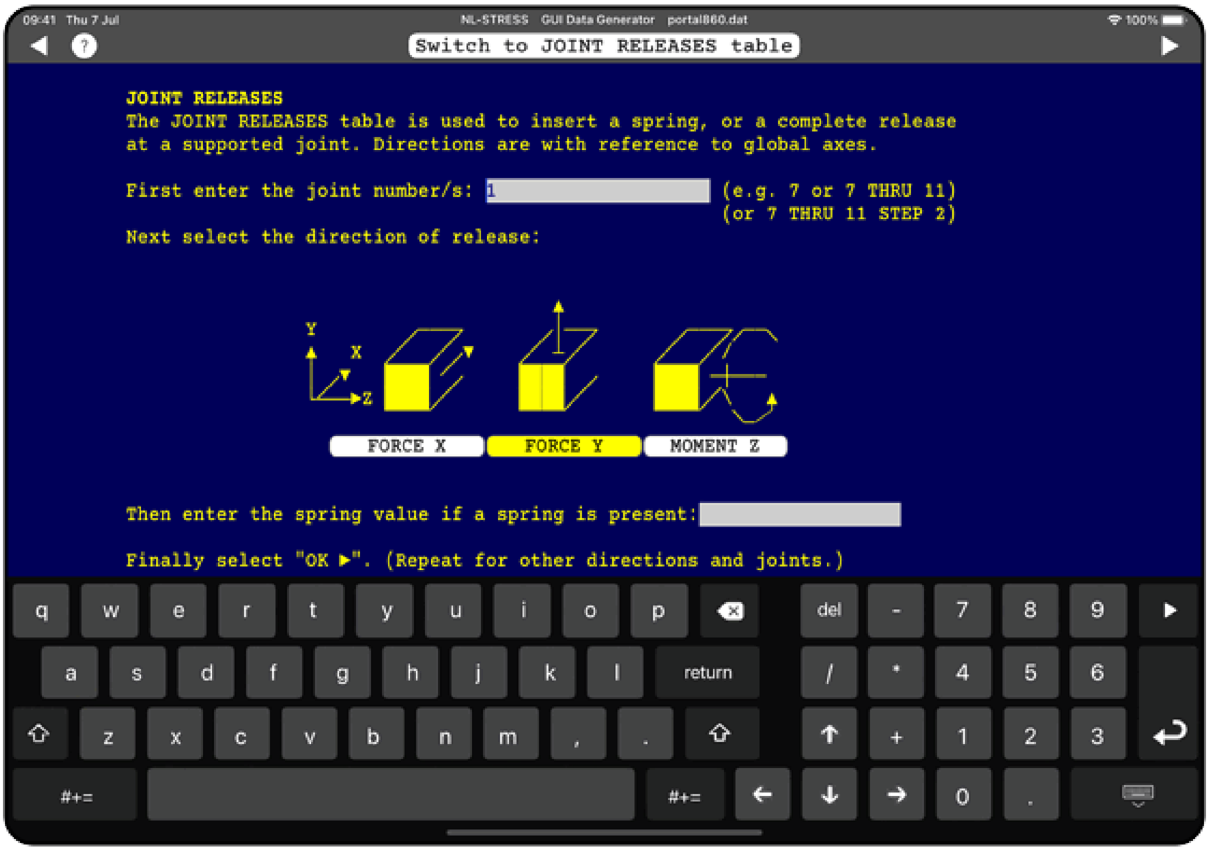


Figure 5.28: GUI JOINT RELEASES entry for PLANE FRAME.

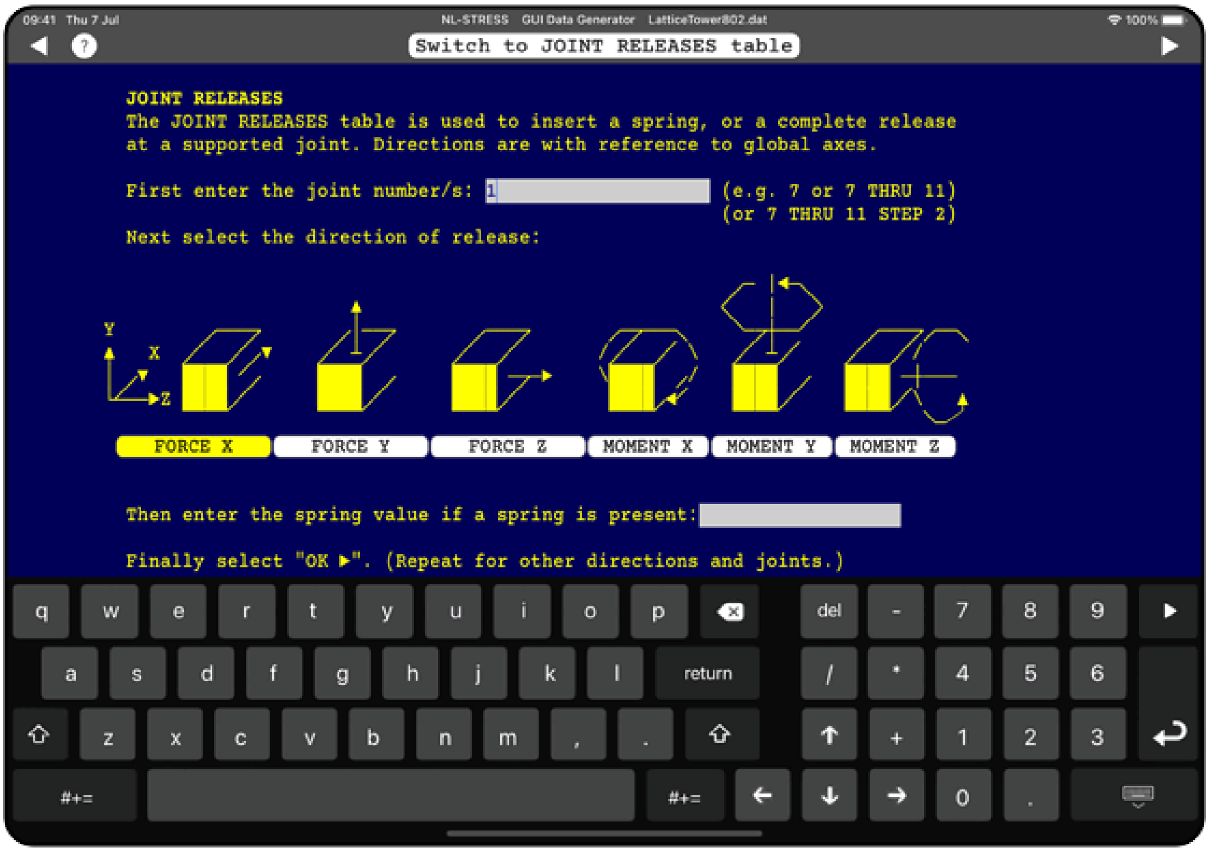


Figure 5.29: GUI JOINT RELEASES entry for SPACE FRAME.

5.7.16.2 GUI - JOINT RELEASES table

Either select the menu option Geometry->JOINT RELEASES table, double click/tap on any line in the JOINT RELEASES table displayed in the main GUI window, or select the "Switch to JOINT RELEASES table" on the JOINT RELEASES dialog, to switch to the JOINT COORDINATES table entry screen, as shown in Figure 5.30.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

You may select the "Switch to JOINT RELEASES GUI" button to switch to the JOINT RELEASES dialog entry screen.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.30: GUI JOINT RELEASES table.

5.7.17 The MEMBER INCIDENCES table

Syntax MEMBER INCIDENCES

<member> <i> <j>

<members> CHAIN [<j>]

<members> RANGE <fi> <fj> <lj>

<members> AS <other members> BUMP <joint difference>

Purpose Define all members in terms of connected pairs of joints.

Usage This table is mandatory. There may be no more than one such table. Any row of the table may take any of the three forms shown above. Every member - from 1 to the total number of members - must be represented in the left hand column of the table, but not necessarily in numerical order.

In the first of the three forms defined above, i represents the joint number at the START end of the member; j represents the joint number at the END. The local x axis of the member runs from joint i to joint j.

In the second of the forms defined above, a sequence of members should be specified in the left most column. Numbers after the keyword CHAIN then define a sequence of pairs of joint numbers in which the END joint of one member is the START joint of the next.

The first two examples below show alternative ways of presenting the same data.

In the third form defined above, a sequence of members should be specified in the leftmost column. The four numbers after keyword RANGE then define the first member (fi to fj) and the last member (li to lj) in the sequence.

Joint numbers of intermediate members are derived automatically by interpolation. The interpolation must, however, result in whole numbers, otherwise an error is reported. The last two examples below show alternative ways of presenting the same data.

In the last form of the syntax i.e.
<members> AS <other members> BUMP <joint difference>
the AS says that <members> are to have the same incidences as the other members but joint numbers bumped by the joint difference. <bump> may be positive or negative. The incidences of the 'other' members must be specified earlier in the table; forward reference is not permitted.

The last two examples below show identical data; the last example shows use of the AS. The AS command saves much time when say, a hundred untidy member incidences for the first floor has to be repeated for the second and subsequent floors with just the incidences 'bumped'.

Examples The member incidences specified below are depicted in Figure 5.31.

```
MEMBER INCIDENCES
6        15  21
7        21  3
8        3   1

MEMBER INCIDENCES
6 THRU 8  CHAIN 15,21,3,1

MEMBER INCIDENCES
19     1  5
20     6  10
21     11 15
22     16 20
23     21 25

MEMBER INCIDENCES
19 THRU 23  RANGE 1 5, 21 25

MEMBER INCIDENCES
19 THRU 21  RANGE 1 5, 11 15
22 THRU 23  AS 19 THRU 20 BUMP 15
```

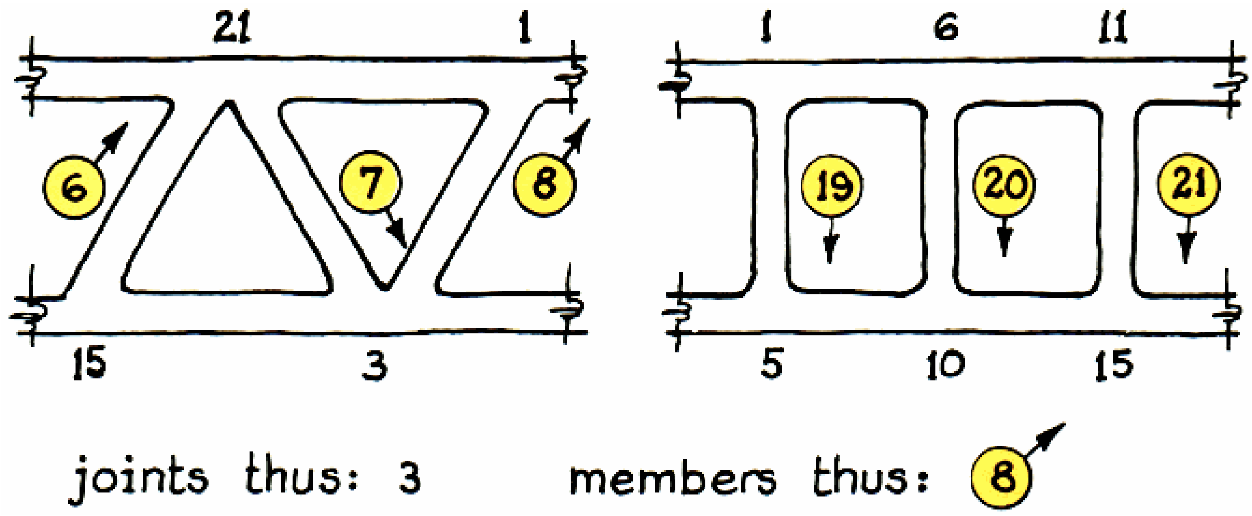


Figure 5.31: Examples of MEMBER INCIDENCES

5.7.17.1 GUI - MEMBER INCIDENCES table

Either select the menu option Geometry->MEMBER INCIDENCES table or double click/tap on any line in the MEMBER INCIDENCES table displayed in the main GUI window to switch to the MEMBER INCIDENCES table entry screen, as shown in Figure 5.32. The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

Select the "Draw Structure" button to enter the coordinates for 2D structures graphically with Draw Structure.

If you edit joints and/or members in Draw Structure, then the JOINT COORDINATES and/or the MEMBER INCIDENCES tables will be expanded out to one joint or one member per line. i.e. all logic like 1 THRU 11 STEP 2 will be expanded. This is required to prevent duplication of data. Figures 5.32 and 5.33 illustrate the before and after of editing member incidences in GUI Draw Structure.

Finally select "►" to accept changes, or "◄" to abandon changes.

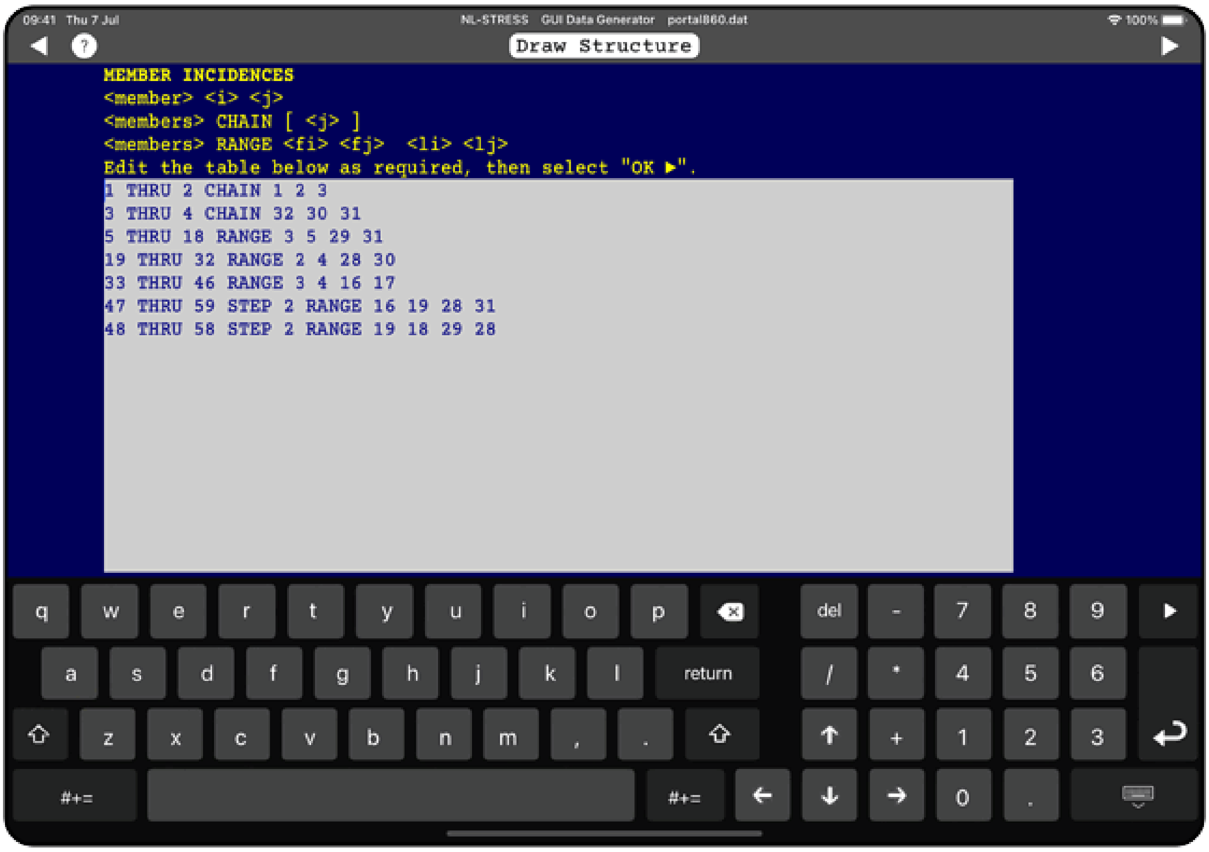


Figure 5.32: GUI MEMBER INCIDENCES table.

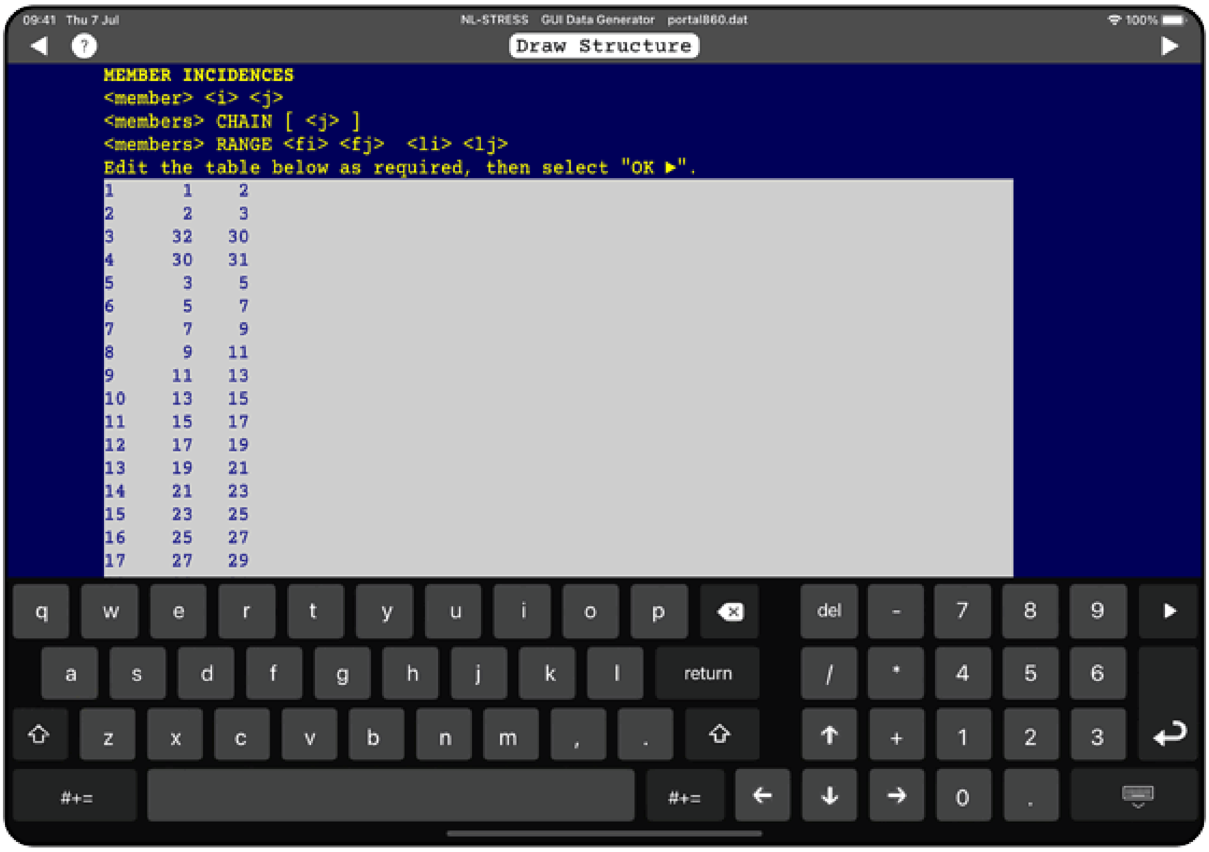


Figure 5.33: GUI MEMBER INCIDENCES table after Draw Structure.

5.7.18 The MEMBER RELEASES table

Syntax MEMBER RELEASES

<members> [START|END [FORCE|MOMENT [X|Y|Z (<spring const>)]]]

Purpose Insert a hinge or axial release at an end of a member otherwise assumed rigidly connected to its joint.

Usage This table is optional; if omitted then all members are considered rigidly connected to joints at both ends. There may be no more than one such table. Spring constants may only be given in association with the keyword MOMENT; they are measured in 'moment units' (e.g. kNm k-ft) /radian turned through. When a spring constant is applied in one of the given directions then spring constants must be applied in all given directions, just as for JOINT RELEASES.

Directions are specified with reference to local axes. Allowable directions of releases depend on the type of structure declared:

Plane truss: FORCE X at one or other end; not both

Plane frame: FORCE X at one or other end; not both
MOMENT Z at either or both ends

Plane grid: MOMENT X at one or other end, not both unless springs; MOMENT Y at either or both ends

Space truss: FORCE X at one or other end; not both

Space frame: FORCE X at one or other end, not both;
MOMENT X at one or other end, not both unless springs; MOMENT Y at either or both ends
MOMENT Z at either or both ends

Every free joint in any of these structures must have at least one unreleased member to keep it from 'spinning'.

Examples The releases specified below are depicted in Figure 5.34.

```
MEMBER RELEASES
24   START MOMENT Z, END MOMENT Z
25   END MOMENT Z
26   END FORCE X
```

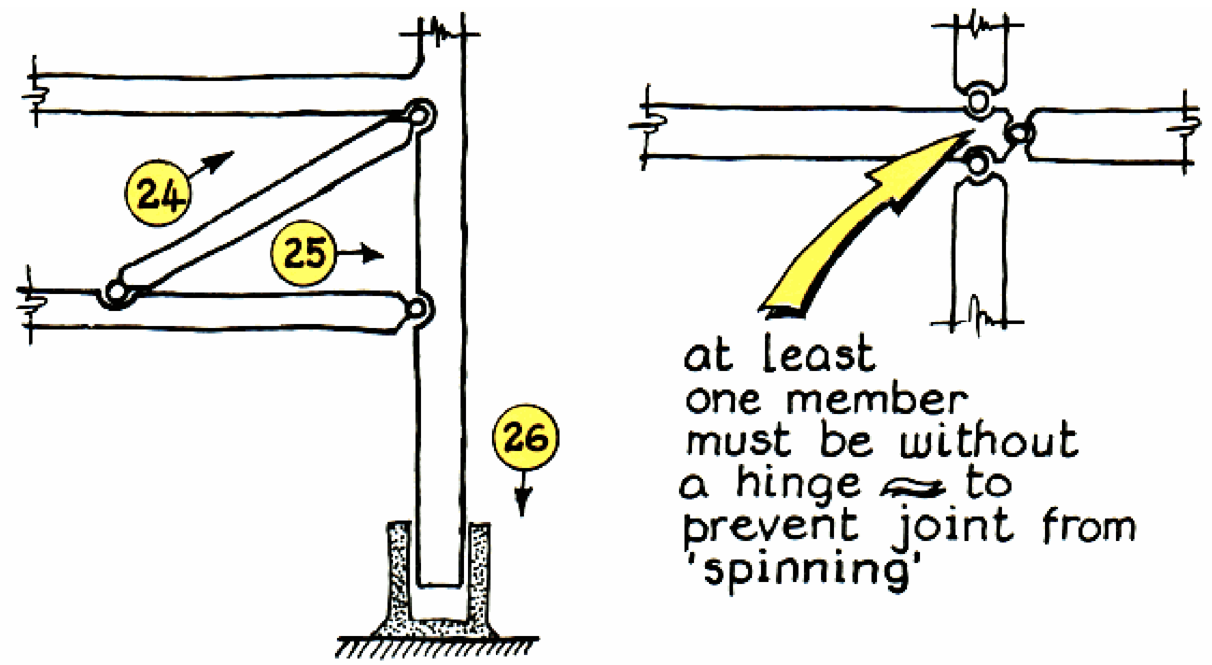


Figure 5.34: Examples of MEMBER RELEASES

5.7.19.1 GUI - MEMBER RELEASES dialog

There are two ways of entering MEMBER RELEASES in the GUI, either one release at a time using the MEMBER RELEASES dialog, or by editing the MEMBER RELEASES table.

To enter data using the MEMBER RELEASES dialog, select the menu option Geometry->MEMBER RELEASES or select the button "Switch to MEMBER RELEASES GUI" on the MEMBER RELEASES table screen.

An example of the MEMBER RELEASES dialog is shown in Figure 5.35. The releases available for your current structure type are displayed on the dialog. The available directions for each structure type are listed in the previous section.

Enter the member number data in the field provided and select the direction of member release required. If no spring constant is required, leave that field blank.

Select "►" to add the member release, or "◄" to return to the main GUI window without adding the release.

Select the "Switch to MEMBER RELEASES table" button to switch to entering data using the GUI MEMBER RELEASES table option.

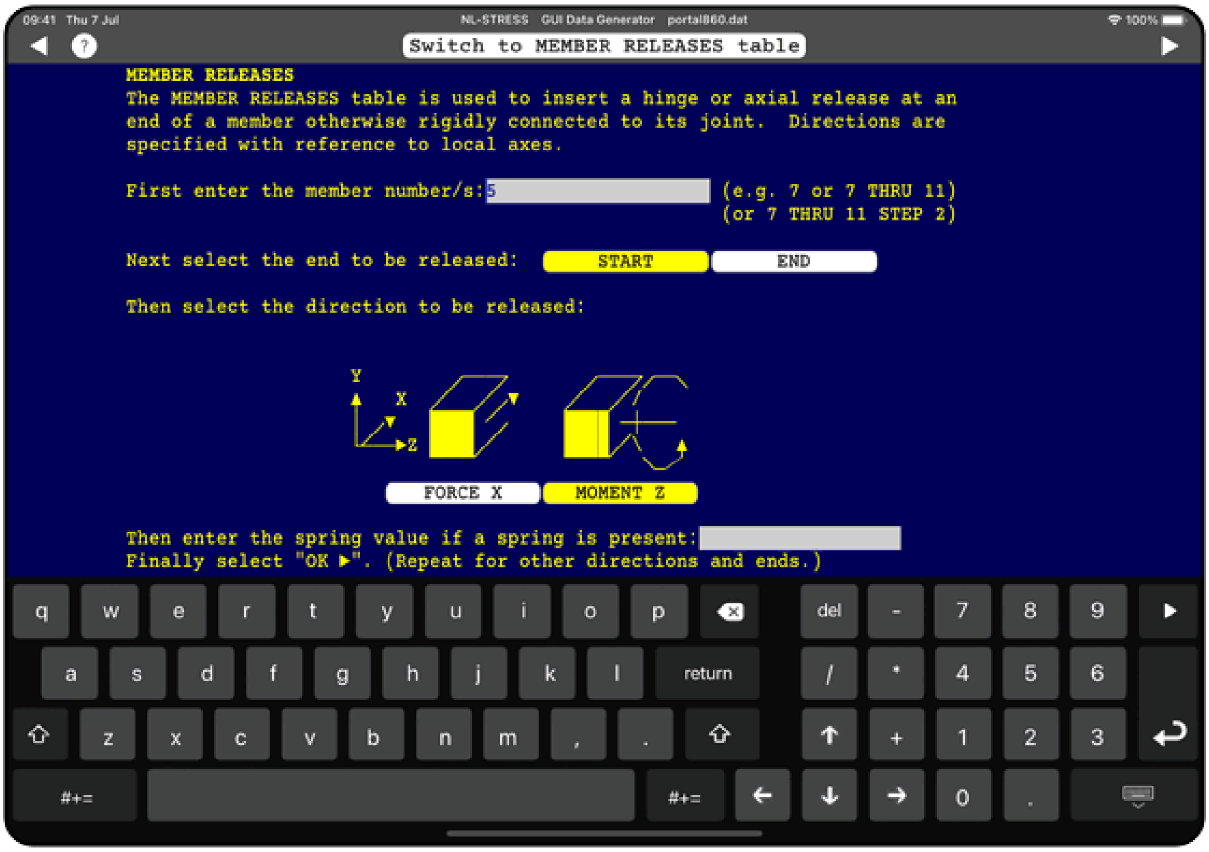


Figure 5.35: GUI MEMBER RELEASES entry for PLANE FRAME.

5.7.19.2 GUI - MEMBER RELEASES table

Either select the menu option Geometry->MEMBER RELEASES table, double click/tap on any line in the MEMBER RELEASES table displayed in the main GUI window, or select the "Switch to MEMBER RELEASES table" on the MEMBER RELEASES dialog, to switch to the MEMBER COORDINATES table entry screen, as shown in Figure 5.36.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

You may select the "Switch to MEMBER RELEASES GUI" button to switch to the MEMBER RELEASES dialog entry screen.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.36: GUI MEMBER RELEASES table.

5.7.19 The CONSTANTS command

Syntax CONSTANTS [<constant> <value> <extent>]

where <constant> is defined as:

E|G|CTE|DENSITY|YIELD|SYIELD|DIRECTION

and where <extent> may take any of the following four forms:

ALL

[<member>]

<member> THRU <member> (STEP <increment>)

ALL BUT <value> [<member>]

Purpose Assign a set of constants (elastic modulus, shear modulus, coefficient of thermal expansion, density, yield stress, shear yield stress, direction in which member can act) to members of the structure.

Usage This command is mandatory.

The keywords introduce constants as follows:

E Young's modulus of material from which member is made. Must be declared for every member. The units are force units per squared length unit (e.g. kN/m^2, k/ft^2).

G the shear modulus corresponding to E: it is denoted by G where: G = E/(2(1+P)) where P is Poisson's ratio for the material.

G is essential to the analysis of grids and space frames. G is also relevant to plane frames where members have a shape or shear area specified. The units are the same as those for E.

CTE coefficient of linear thermal expansion. Essential if the effects of temperature change are to be calculated. The units are 'per degree' (e.g. 1/degree Celsius).

DENSITY density is measured in force units per cubic length unit. Specifying density and MEMBER SELF WEIGHTS sets up a force acting in the negative direction of global axis Y for plane and space frames and in the negative direction of global axis Z for grids. For space frames it is permissible to change the direction of gravity from Y to Z.

YIELD yield stress is measured in force units per squared length unit. It is essential to specify yield stress when the METHOD PLASTIC command is used.

SYIELD shear yield stress is measured in force units per squared length unit. If not specified assumed equal to yield stress divided by SQR(3). SYIELD is used for computing the plastic collapse torsional moment for space frames.

DIRECTION permits members to be tension or compression members only. DIRECTION should be followed by +1 for compression-only members, -1 for tension-only members thus: CONSTANTS DIRECTION -1 13 14 15
sets members 13, 14 and 15 as tension only members.

The implementation of DIRECTION facility is rigorous and it will be necessary to analyse structures as METHOD SWAY and give the NUMBER OF INCREMENTS command. In some structures it may be that because of non-linear effects (e.g. lift-off at a support) a member defined as a tension only member goes into compression (and thus carries no axial load) and at a higher loading level once again becomes a tension member. NL-STRESS will handle such cases.

For values of DIRECTION >0 and <1, NL-STRESS prevents the nominated member carrying tension if positive fraction or compression if negative fraction and multiplies remaining member stiffness by the fraction given, thus:
DIRECTION 0.1 ALL
would cause all members which go into tension to carry no tension, and have their various stiffness reduced to 10% of that given in the member properties table, leaving all members which do not go into tension unchanged. Similarly:
DIRECTION -0.2 ALL
would cause all members which go into compression to carry no compression, and have their various stiffness reduced to 20% of that given in the member properties table, leaving all members which do not go into compression unchanged. As with all such modelling devices, it is up to the engineer to satisfy him/herself that the device is appropriate for the structure being analysed. In both cases the constants E & G are multiplied by the absolute value of the fraction given for the current loading.

There may be several CONSTANTS commands in a set of data. However, it is an error to specify a particular constant for a particular member more than once. The keyword ALL says 'all that are not yet set'.

```
Examples  CONSTANTS  E 28E6  ALL
          CONSTANTS  G 11.2E6  1,2,3,5,7,9   G 5.6E6  4,6,8
          CONSTANTS  E 205E6  1 THRU 81 STEP 2  E 28E6 2 THRU 80 STEP 2
          CONSTANTS  DENSITY 24 ALL,  BUT 25.5  1,2,3
          CONSTANTS  DIRECTION 0.5 1 THRU 270
```

5.7.19.1 GUI - CONSTANTS dialog

There are two ways of entering CONSTANTS data in the GUI, either using the CONSTANTS dialog, or by editing the CONSTANTS table.

To enter data using the CONSTANTS dialog, select the menu option Geometry-CONSTANTS or select the button "Switch to CONSTANTS GUI" on the CONSTANTS table screen.

An example of the CONSTANTS dialog is shown in Figure 5.37.

Clicking on a constants button will update the window.

Select "►" to add the CONSTANTS data, or "◄" to return to the main GUI window without adding the data.

Select the "Switch to CONSTANTS table" button to switch to entering data using the GUI CONSTANTS table option.

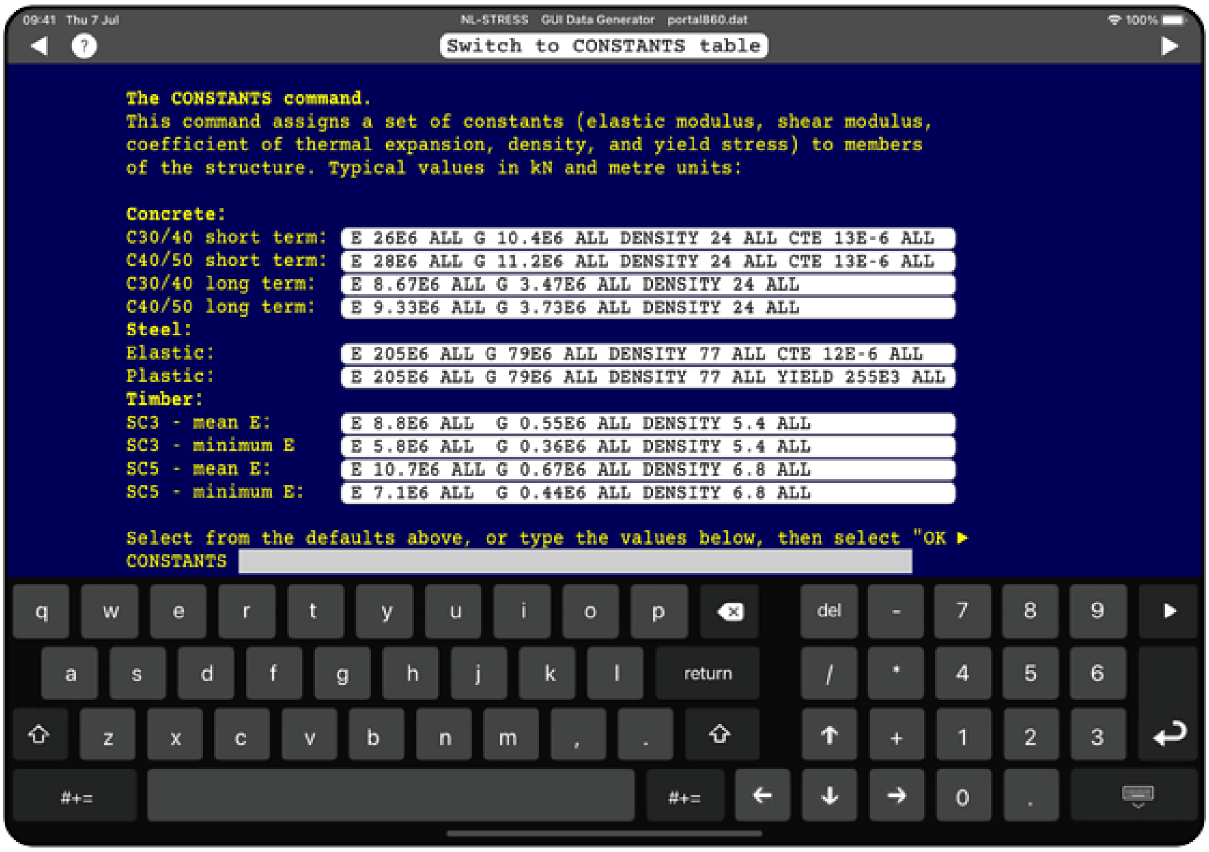


Figure 5.37: GUI CONSTANTS entry.

5.7.19.2 GUI - CONSTANTS table

Either select the menu option Geometry->CONSTANTS table, double click/tap on any CONSTANTS line in the main GUI window, or select the "Switch to CONSTANTS table" on the CONSTANTS dialog, to switch to the CONSTANTS table entry screen, as shown in Figure 5.38.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

You may select the "Switch to CONSTANTS GUI" button to switch to the CONSTANTS dialog entry screen.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.38: GUI CONSTANTS table.

5.7.20 The MEMBER PROPERTIES table

Syntax MEMBER PROPERTIES

<members> [<property> <value>]

<members> <shape> [<dimension> <value>]

<members> AS <other member>

where <property> is defined as:

AX|AY|AZ|IX|IY|IZ|C|CX|CY|CZ|BETA|FXP|MXP|MYP|MZP

and <shape> is defined as:

SIZE|RECTANGLE|CONIC|OCTAGON|ISECTION|TSECTION|HSECTION

and <dimension> is defined as:

D|DL|DY|DYL|DZ|DZL|T|TY|TZ|R|C|CX|CY|CZ|CL|CXL|CYL|CZL|BETA

Purpose Define the properties of the cross section of every member - either by giving section properties directly or by specifying the shape of cross section and supplying leading dimensions.

Usage This table is mandatory and there may be more than one. Several lines may be used to specify the section properties for one member. Duplicated properties are faulted.

Properties may be given directly, in which case enough properties must be given to make the structure stand up.

The significance of each of the keywords is explained below where X, Y and Z refer to the member's local axes:

AX cross-sectional area

AY,AZ shear areas relative to the principal axes Y and Z (five sixths of the actual area in the case of a rectangle; taken as infinitely large by default) If Poisson's ratio=1E-12 then shear areas are omitted for the member.

IX torsion constant (polar I in the case of a circular section)

IY,IZ second moment of area (moment of inertia) about principal axes Y and Z respectively

FXP squash load

MXP,MYP,MZP plastic moments about axes X, Y and Z

BETA angle of twist as defined in Figure 5.6 and measured in degrees. A positive angle makes local z dip below the horizon through the START end. BETA is zero by default, and may only be specified for space frames.

R Internal radius at corner of RECTANGLE. If thickness T is given <= 20% of the least external dimension, a steel hollow section is assumed having an external

radius of 1.25R and internal radius of R, typical of hot rolled structural hollow sections.
R Root radius for an ISECTION.

Properties are more conveniently given indirectly, by specifying a shape and leading dimensions. Figure 5.39 and Figure 5.40 define shapes and dimensions for non-tapered members. 'L' ending a keyword refers to the Last value for a set of tapered members. If DYL is given in the data then DZL must also be given.

The keyword SIZE specifies that for the member numbers given and for reasons of space, only some of the dimensions are included. The subsequent appearance of RECTANGLE|CONIC etc. completes the data. The fifth example below is for tapered members 1 THRU 3.
Whenever DY=DZ a single dimension D may be given.
Whenever TY=TZ a single dimension T may be given.
Whenever CY=CZ a single dimension C may be given.
Whenever DYL=DZL a single dimension DL may be given.
Whenever CYL=CZL a single dimension CL may be given.

NL-STRESS recognises a cross-sectional area of 1E-12 as an instruction to omit that member from results, and plots.

Given a shape and dimensions NL-STRESS is able to derive all properties relevant to the type of structure being analysed. In this case CY and CZ are taken as distances from the centroid to the outermost edge along Y and Z respectively, CX to the outermost corner, and BETA is set to zero. These items may be specifically set to other values.

When doing elastic-plastic analysis it is simpler to define cross sections by shape and leading dimensions - not by giving AX, IZ etc. directly - because plastic moments and 'squash loads' can be automatically derived from dimensions of the cross section.

It is permissible to omit labels to the section properties and give only a set of values in the following order:
Plane frame & plane truss: AX AY IZ CY
Plane grid: IX IY AZ CZ AX
Space frame & space truss: AX AY AZ IX IY IZ CX CY CZ BETA

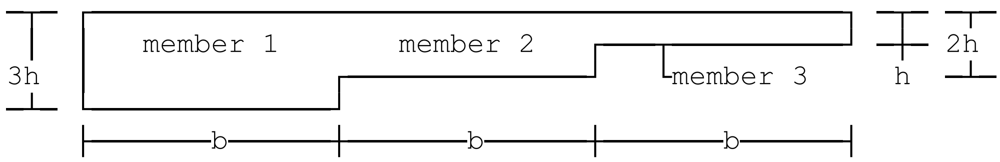
Trailing zeros may be omitted, thus for a space frame for which stresses are not required & BETA=0, only six values corresponding to AX AY AZ IX IY IZ need be given in order.

Member properties which have a taper in the Y and/or Z directions may be specified as a group using THRU and STEP in association with the dimensions DL, DYL & DZL, and positions for computation of stresses CL, CXL, CYL or CZL, all of which refer to the dimensions for the last member in the group. The third example below specifies a tapered square hollow section. In the third example, member 3 has its depth and width specified as 0.2, member 5 as 0.3, member 7 as 0.4; all three members have a constant thickness of 0.01. If in doubt about which dimensions have been used for any member, remember that the keyword SUMMARY following the PRINT command will provide a summary of the section properties computed for the analysis; a brief check on the AX will confirm the depth & width of the section.

The fourth example below specifies a tapered I section. In the fourth example, the SIZE keyword is used to specify dimensions which are common to members 1 THRU 3, the depth is specified as 0.465, width as 0.153, web thickness as 0.0107, flange thickness as 0.018. The second line of the data i.e. 1 THRU 3 ISECTION DYL 0.565 DZL 0.253 tells NL-STRESS that an I section is being defined for members 1 THRU 3 and that the depth & width of member 3, the last member, is 0.565 & 0.253 respectively.

When specifying tapered members, each sub-member is assigned uniform properties throughout its length even if it is segmented. The first member has properties computed from the start dimensions, the last member from DYL, DZL...
All members between the first and last members are computed assuming a linear change in dimensions between the first and last members. It is not permissible to taper a single member even if it is segmented - segmented members are provided for within-member stability analyses. Generally two or three 'THRU' members will give engineering accuracy for the modelling of tapered members, the engineer must make a judgement dependent upon the structure being analysed and the accuracy required. The figure shows a tapered member having three segments.

Tapered section having section depths 3h, 2h & h.
Each, of three segments shown, has an equal length b.



Assuming that the parameters b & h have been assigned, then for a rectangular section of width w, the data would be:

```
1 THRU 3 RECTANGLE DY 3*h DYL h DZ b
```

If the section width also tapered from 3*w to w, the data would be:

```
1 THRU 3 RECTANGLE DY 3*h DYL h DZ 3*w DZL w
```

Were the section to be an I-section, then the number of parameters may exceed the line length of 80 characters. For this situation, NL-STRESS permits the member properties to be spread over two lines, the first containing the keyword SIZE, the second line containing the keyword ISECTION, see the fourth example which follows. For assurance, it is recommended that the keyword SUMMARY is included in the PRINT command so that the checking engineer can verify that the section properties are as expected.

It is not permissible to follow INCLUSIVE with AS thus 3 2 17 INCLUSIVE AS 4 THRU 6 will be faulted.

Examples

```
MEMBER PROPERTIES
1 THRU 3  AX .19 IZ .045
4 RECTANGLE DY .25 DZ .57 BETA 90

MEMBER PROPERTIES
1 THRU 3 .18 0 .045
5 .2 0 .05

MEMBER PROPERTIES
1 THRU 2 ISECTION DY 0.465 DZ 0.153 TZ 0.0107 TY 0.0189
3 THRU 7 STEP 2 RECTANGLE D 0.2 DL 0.4 T 0.01
```

```
MEMBER PROPERTIES
1 THRU 3 SIZE DY 0.465 DZ 0.153 TZ 0.0107 TY 0.018
1 THRU 3 ISECTION DYL 0.565 DZL 0.253

MEMBER PROPERTIES
1 THRU 5 RECTANGLE D 0.8
6 9 BOTH RECTANGLE DY 0.6 DZ 0.8
7 8 10 11 12 INCLUSIVE AS 1
```

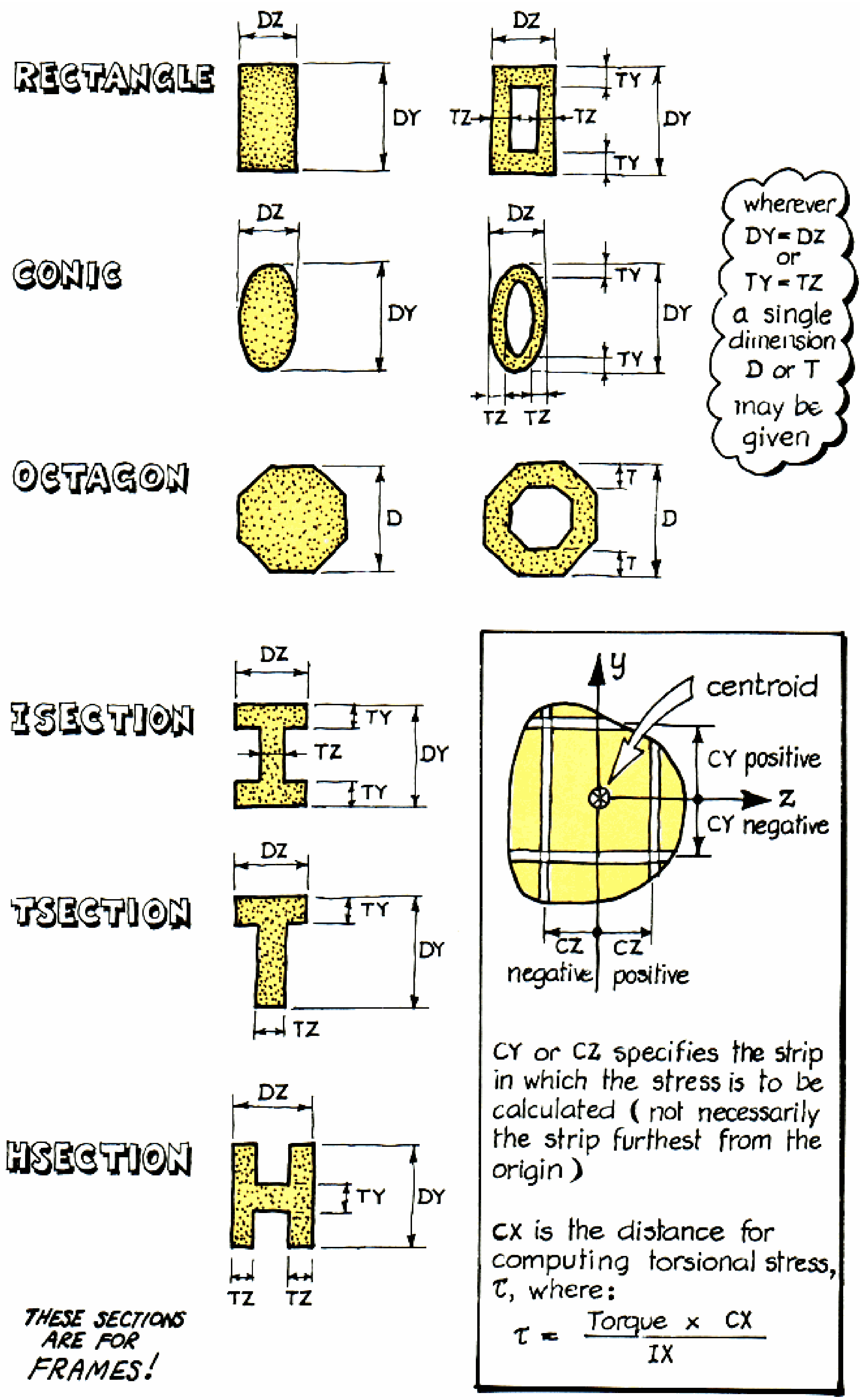


Figure 5.39: Member cross sections (frames)

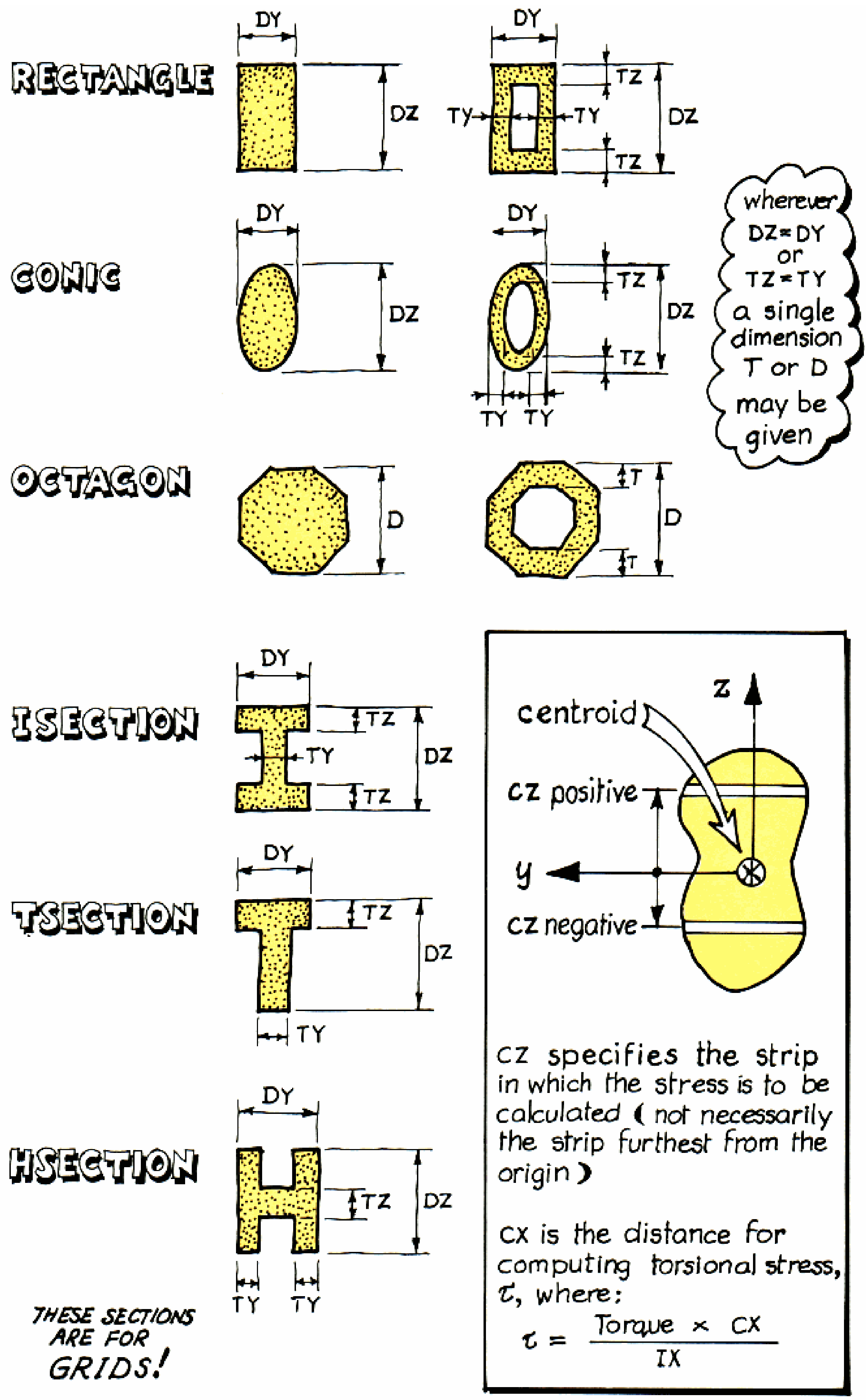


Figure 5.40: Member cross sections (grids)

5.7.20.1 GUI - MEMBER PROPERTIES data entry

There are many ways of entering MEMBER PROPERTIES data in the GUI.

The main methods are listed in the Geometry menu, as shown in Figure 5.41.

These methods are:

- MEMBER PROPERTIES dialog
- MEMBER PROPERTIES CONIC
- MEMBER PROPERTIES HSECTION
- MEMBER PROPERTIES ISECTION
- MEMBER PROPERTIES OCTAGON
- MEMBER PROPERTIES TSECTION
- MEMBER PROPERTIES RECTANGLE
- MEMBER PROPERTIES table

And an additional option available from the MEMBER PROPERTIES dialog is:

- MEMBER PROPERTIES steel section tables.

These methods are described in the following sections.

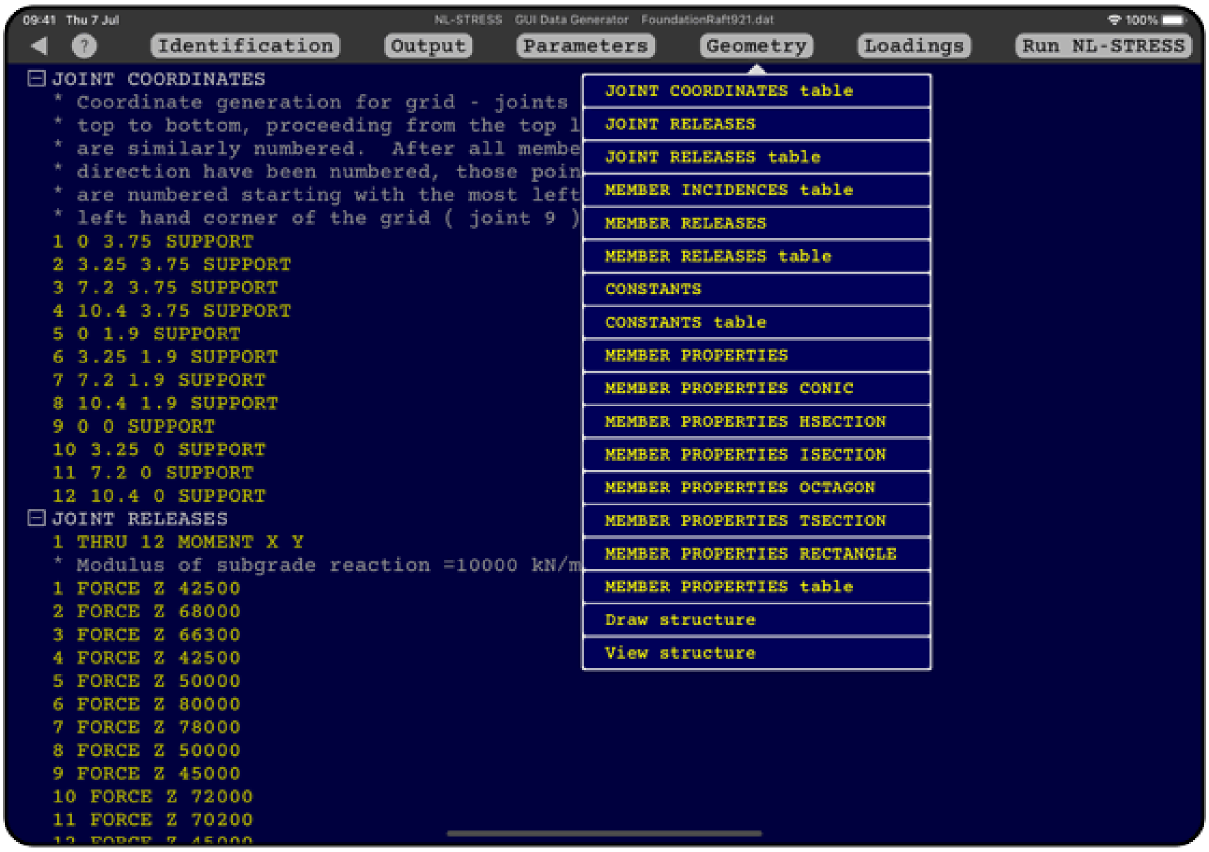


Figure 5.41: GUI Geometry menu.

5.7.20.2 GUI - MEMBER PROPERTIES dialog

To enter data using the MEMBER PROPERTIES dialog, select the menu option "Geometry->MEMBER PROPERTIES" or select the button "Switch to MEMBER PROPERTIES GUI" on the MEMBER PROPERTIES table screen.

An example of the MEMBER PROPERTIES dialog is shown in Figure 5.42.

First enter the member number information at the top of the screen.

Then choose one of the following options:

1) SECTION PROPERTIES

Enter values for the fields AX, AY, AZ, IX. IY. IZ, CX, CY, CZ, C, BETA, FXP, MXP, MYP, MZP as required then select the "►" button to finish.

2) SHAPE

Select one of the shapes, RECTANGLE, ISECTION, OCTAGON, TSECTION, CONIC, HSECTION, then enter data into the fields on the dialog which appears. See Section 5.7.20.4 for further details.

3) AS ANOTHER MEMBER

Click the "AS" Another Member button and enter the member number of the other member. Select the "►" button to finish.

4) FROM STEEL SECTION TABLES

Select the "Steel Section Tables" button and select a line from the table which appears. See section 5.7.20.5 for more details.

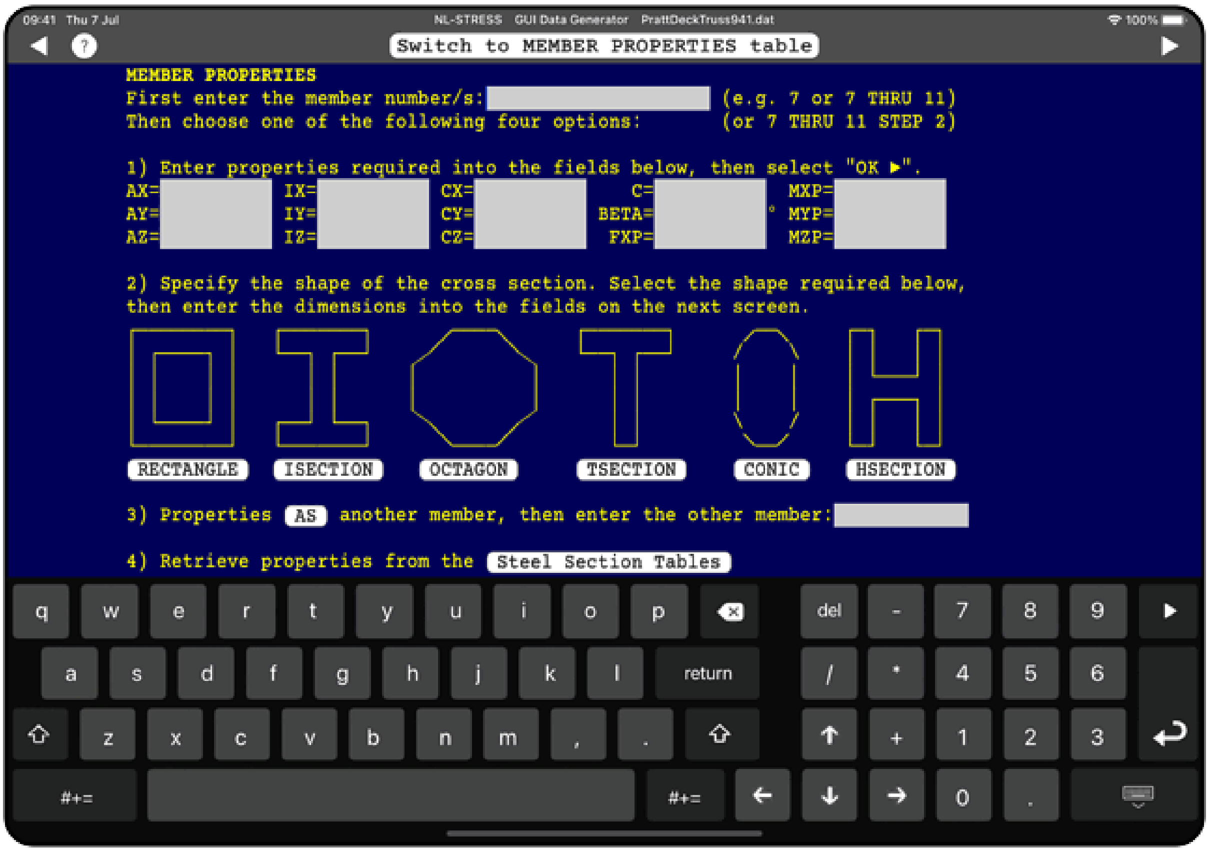


Figure 5.42: GUI MEMBER PROPERTIES entry.

5.7.20.3 GUI - MEMBER PROPERTIES table

Either select the menu option "Geometry->MEMBER PROPERTIES table", double click/tap on any MEMBER PROPERTIES line in the main GUI window, or select the "Switch to MEMBER PROPERTIES table" on the MEMBER PROPERTIES dialog, to switch to the MEMBER PROPERTIES table entry screen, as shown in Figure 5.43.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the table provided.

You may select the "Switch to MEMBER PROPERTIES GUI" button to switch to the MEMBER PROPERTIES dialog entry screen.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.43: GUI MEMBER PROPERTIES table.

5.7.20.4 GUI - MEMBER PROPERTIES section shapes

Either select one of the menu options "Geometry->MEMBER PROPERTIES CONIC, HSECTION, ISECTION, OCTAGON, TSECTION or RECTANGLE", or select the desired button on the MEMBER PROPERTIES dialog, to switch to the dialog entry screen for the chosen shape.

Figures 5.44 and 5.45 illustrate the ISECTION dialogs for Frames/Trusses and Grids respectively. Note the DY and DZ, and the TY and TZ are transposed. You will be presented with the appropriate dialog depending on the structure type chosen.

Enter the required data in the field provided. Any value not required should be left blank.

You may select the "Switch to MEMBER PROPERTIES GUI" button to switch to the MEMBER PROPERTIES dialog entry screen.

Finally select "►" to accept changes, or "◄" to abandon changes.

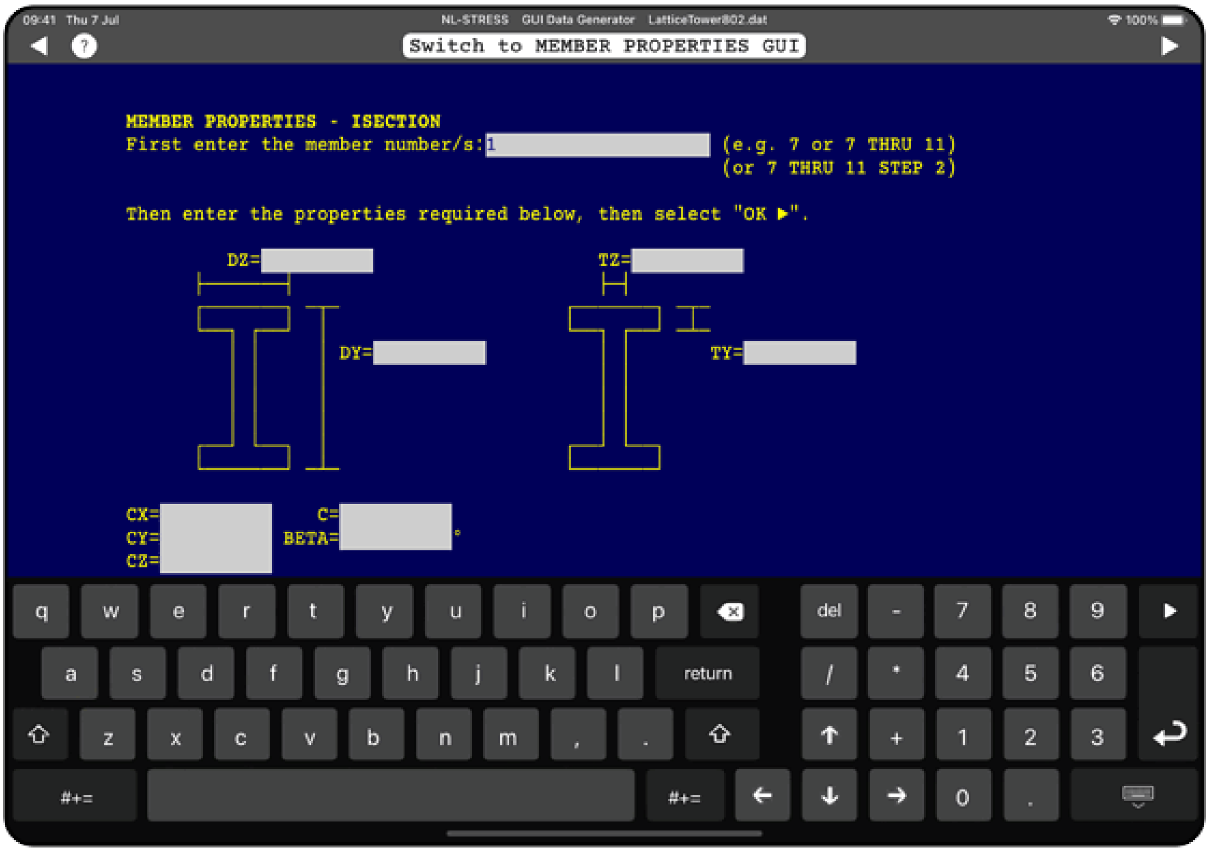


Figure 5.44: GUI MEMBER PROPERTIES I section (Frame/Truss).

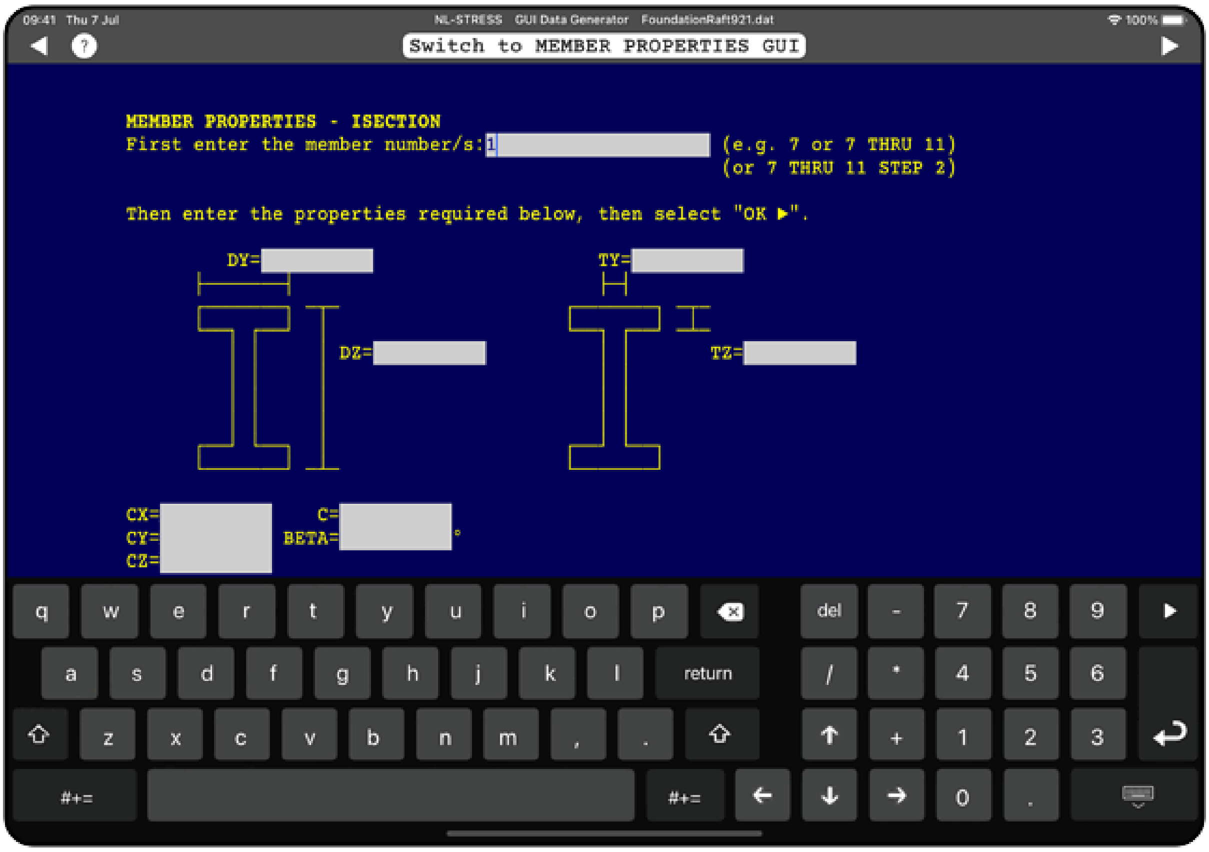


Figure 5.45: GUI MEMBER PROPERTIES I section (Grid).

5.7.20.5 GUI - MEMBER PROPERTIES Steel Section Tables

Enter the required member numbers at the top of the MEMBER PROPERTIES dialog and then select the button "Steel Section Tables" to switch to the MEMBER PROPERTIES steel section tables screen, as shown in Figure 5.46.

Select the required line in the table provided, or select "◄" to not add new properties.

You may select the "Switch to MEMBER PROPERTIES GUI" button to switch back to the the MEMBER PROPERTIES dialog entry screen.

Note, different tables are presented for FRAME/TRUSS and for GRID to accommodate the different axis directions for between the structural types.

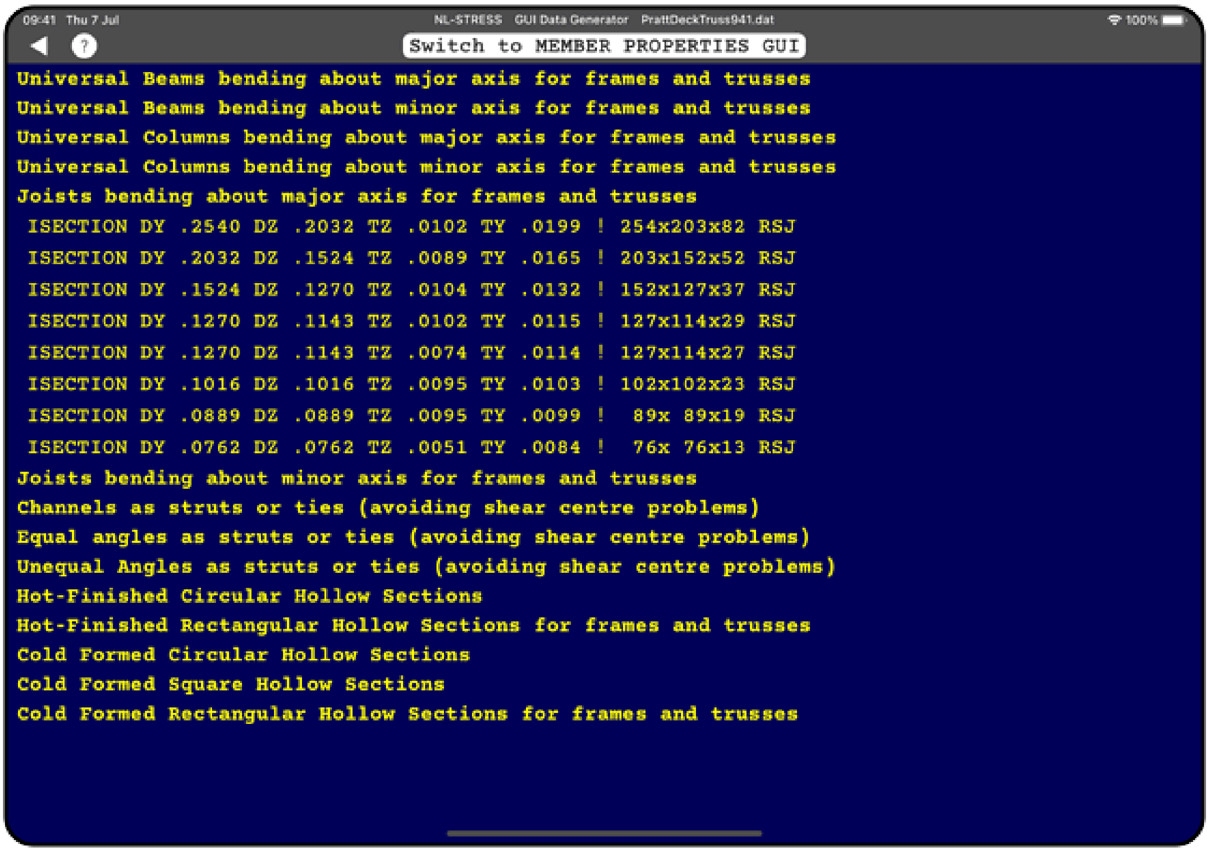


Figure 5.46: GUI MEMBER PROPERTIES steel section tables (Frame/Truss).

5.7.21 The LOADING command

Syntax LOADING <title>
LOADING DYNAMIC <g>

Purpose Introduce a set of data for a basic loading condition or for a combination; reproduce the given title at the head of every page of results for that loading condition or combination.

The second form of the command tells NL-STRESS that Raleigh's method is to be used to compute the natural frequency for the loading case using an acceleration due to gravity given by <g>. Care must be taken with the tabulated natural frequencies that a realistic mode shape is implied especially when considering cases of primary beams supporting secondary beams.

Usage — This command is mandatory before each basic loading condition and before each combination. There should be no more than fifty characters in <title>.

After this command may come a TABULATE command. In the absence of a TABULATE command introduced locally, the global one applies. (The global TABULATE command is placed before the JOINT COORDINATES table.) In the absence of a global or local command, TABULATE DISPLACEMENTS FORCES REACTIONS applies by default.

In the second example below, the acceleration due to gravity is given as 9.80665 (which is obviously m/sec2 units) implying that all units are in kN & m and combinations thereof. In the third example below, variables a & b have been preset e.g. a=23 & b=2.8 which would cause the LOADING title to be displayed in the results as:
LOADING Applied to member 23 of length 2.8
It is also permissible to include an assignment e.g. +c=2*b in the loading title; this would be displayed as: c=2*b=5.6

Example

```
LOADING 1.1 x Dead + 1.5 x Live
TABULATE FORCES

LOADING DYNAMIC 9.80665

LOADING Applied to member +a of length +b
```

5.7.21.1 GUI - Load case management

The first four options on the GUI's "Loadings" menu are:

- Select current load case
- Edit current load case title
- Add load case before current
- Add load case after current

There will always be a current load case set when using the GUI, this load case will displayed on the top line of any loading screen. The four options above provide a straightforward of managing and switching between the load cases.

To delete a load case, you will currently need to go to the menu option "Identification->Edit the data file" and delete the lines manually.

5.7.21.2 GUI - Select current load case

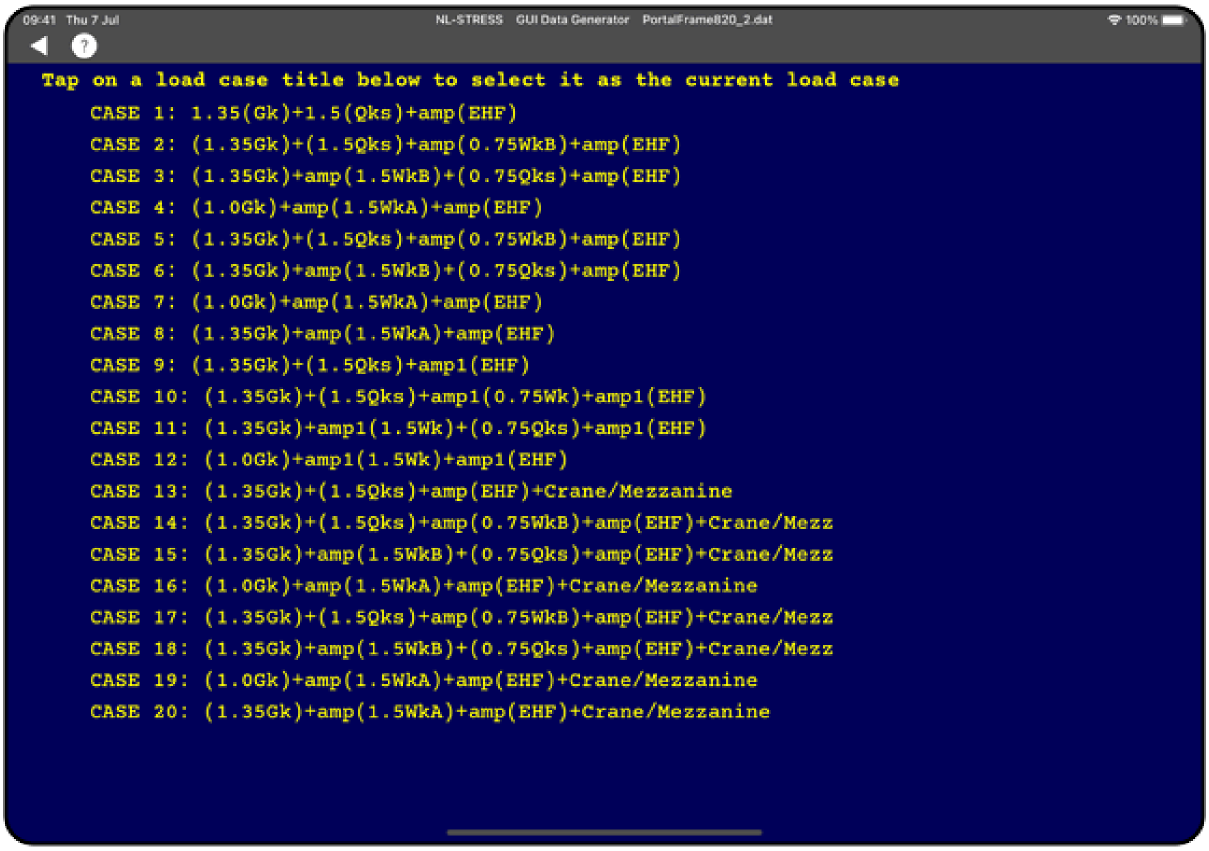


Figure 5.47: GUI - Select current load case.

5.7.21.3 GUI - Edit current load case title

Select the menu options "Loadings->Edit current load case title" and then edit the title text in the dialog, as shown in Figure 5.48.

Select "►" to accept changes, or "◄" to abandon changes.

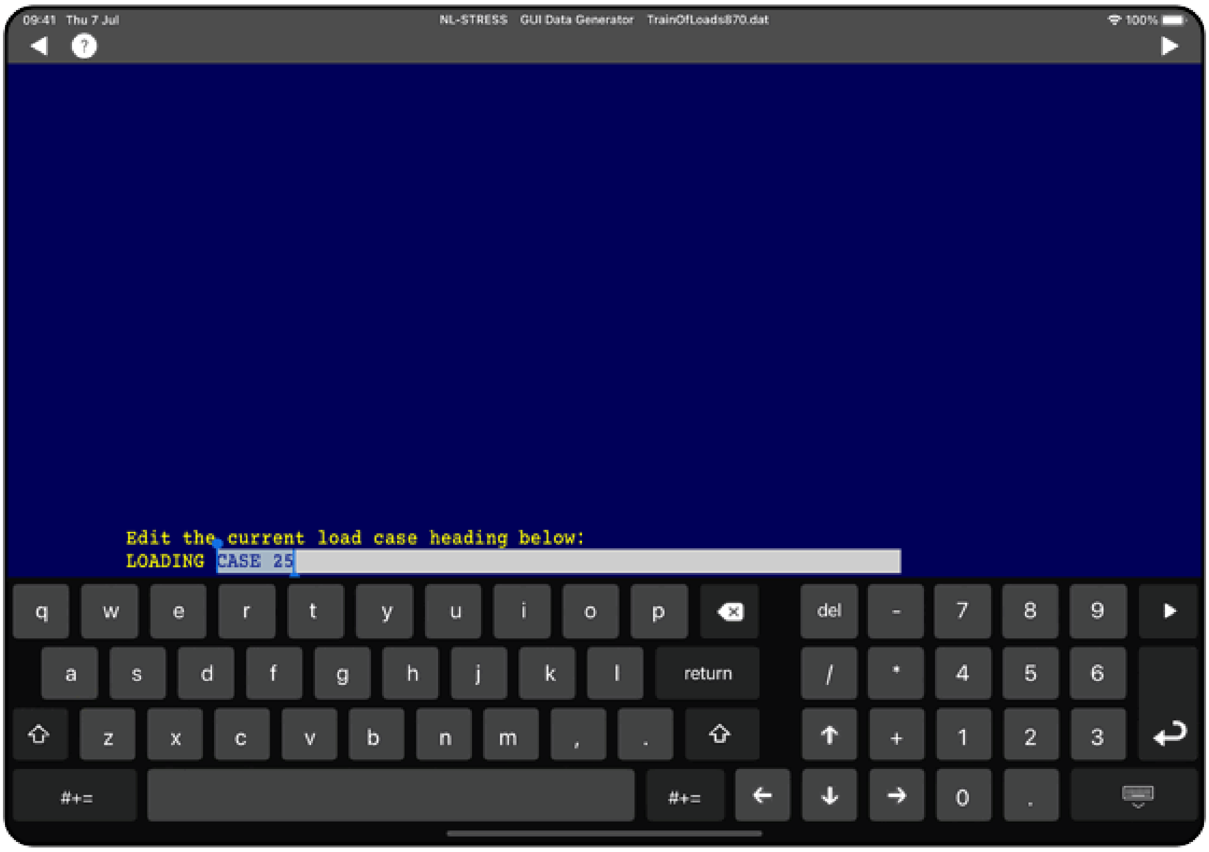


Figure 5.48: GUI - Edit current load case title.

5.7.21.4 GUI - Add load case before/after current

The two menu options, "Loadings->Add load case before current" and "Loadings->Add load case after current" work in the same way.

The dialog displays the name of the current load case for reference, as shown in Figure 5.49.

Enter a title for the new load case, then select the "▶" button. A new load case will be created and inserted either before or after the current load case.

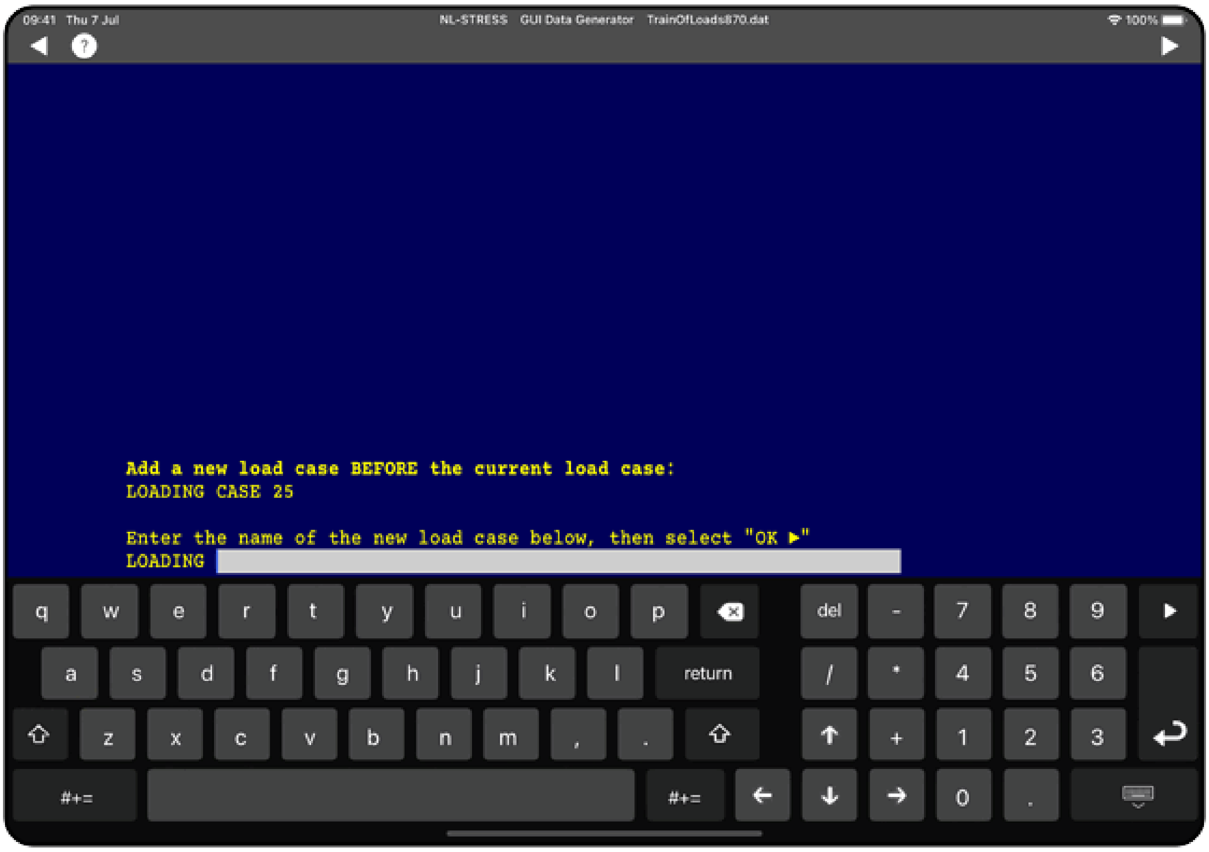


Figure 5.49: GUI - Add load case before current.

5.7.22 The JOINT LOADS table

Syntax JOINT LOADS

<joints> [FORCE|MOMENT [X|Y|Z <value>]]

<j> AREA <k> FORCE Z <value> [strips]

0 FORCE Z <value> <X-coord> <Y-coord>

Purpose Apply point loads or point moments to joints of the structure. Apply an AREA load to the joints of a plane grid where <j> and <k> are any two diagonally opposite joint numbers of the rectangular area over which the area load is to be applied, and [strips] is an optional integer number specifying the number of strips in each of two directions into which the area load is to be divided, see section 5.11.7. In the third form of the command, joint 0 (zero) denotes that the joint number/s are undefined and that <value> located at coordinates X-coord,Y-coord should be shared to the nearest joints of the grid using the principles described in section 5.11.7. When using the second and third forms of the joint loads command, it is incumbent on the engineer to inspect the sum of applied loads - as listed at the end of the results - to ensure that the total loading applied is as expected.

Usage This table is optional and there may be more than one of them among the data for a loading condition. Joint loads appropriate to each type of structure are as follows where X, Y and Z refer to global axes:

Plane truss:	FORCE X,Y	
Plane frame:	FORCE X,Y	MOMENT Z
Plane grid:	MOMENT X,Y	FORCE Z
Space truss:	FORCE X,Y,Z	
Space frame:	FORCE X,Y,Z	MOMENT X,Y,Z

Forces are measured in force units (e.g. kN, kips) and moments in force units times length units (e.g. kNm, k-ft). Area loads are measured in force units per unit area (e.g. kN/m^2, k/ft^2). AREA loads are only supported on structures of type PLANE GRID. The procedure for sharing area loads to the joints is described in section 5.11.7.

It is permissible to omit labels to the joint loads and give only a set of values in the above order (using zero when no load is required in that particular direction). Trailing zeros may be omitted. The second example below provides the same data as the first example (providing that both examples are for a plane frame).

The third example below applies an area load of -10 (force per unit area) to a rectangular area which has its lower left corner at joint 4 and its upper right corner at joint 24.

The fourth example below applies an point load of -18.2 located at coordinates 12.5+2*a,38.5+2*b shared to the nearest joints of the grid. Parameters a & b must have been set previously. The use of parameters allows one load case to be set up for a set of wheel loads and then duplicated for different vehicle positions, only requiring

the engineer to amend the parameters. Of course the expressions used to represent the coordinates may be more complicated than the simple example shown.
When a member is segmented, then additional joints (above those declared in the NUMBER OF JOINTS command) are added at the end of each segment. For a 2 span beam having 2 members and 3 joints & 1 segment, the joint numbering is as follows.

```
1                               2                  3
═════════════════════════════════════════════════════
▲           Member 1            ▲      Member 2
```

For NUMBER OF SEGMENTS 4, the joint numbering is as follows.

```
1       4       5       6       2    7    8    9    3
═════════════════════════════════════════════════════
▲           Member 1            ▲      Member 2
```

The number of joints given by the engineer for the unsegmented case i.e. NUMBER OF SEGMENTS 1, are kept, but additional joint numbers are added to segment the members as shown. For 4 segments, 3 additional joints will be required for each member. For 'ns' segments, 'ns-1' additional joints will be required for each member. Simple arithmetic is all that is needed to work out the additional joint numbers within each member. There are some situations for which the ability to apply loading to the additional joints is appropriate; for such cases it will be necessary to add the keyword TRACE at the end of NUMBER OF SEGMENTS command. If the keyword TRACE is omitted and loading is applied to the additional joints, the error message 'Data out of range at line -' will be displayed.

Examples

```
JOINT LOADS
1 THRU 3 FORCE Y -123.45
4 6 8 10 15 INCLUSIVE FORCE X 55, MOMENT Z 234.56
5 7 9 11 14 INCLUSIVE FORCE X 55
12 13 BOTH MOMENT Z 234.56

JOINT LOADS
1 THRU 3 0 -123.45
4 55 0 234.56

JOINT LOADS
4 AREA 24 FORCE Z -10

JOINT LOADS
0 FORCE Z -18.2 12.5+2*a 38.5+2*b
```

5.7.22.1 GUI - JOINT LOADS dialog

There are two ways of entering JOINT LOADS data in the GUI, either using the JOINT LOADS dialog, or by editing the JOINT LOADS table.

To enter data using the JOINT LOADS dialog, select the menu option "Loadings->JOINT LOADS" or select the button "Switch to JOINT LOADS GUI" on the JOINT LOADS table screen.

An example of the JOINT LOADS dialog is shown in Figure 5.50.

The dialog will present the appropriate directions only for the current structure type. The directions appropriate for each structure type are listed in the previous section.

Select "►" to add the JOINT LOADS data, or "◄" to return to the main GUI window without adding the data.

Select the "Switch to JOINT LOADS table" button to switch to entering data using the GUI JOINT LOADS table option.

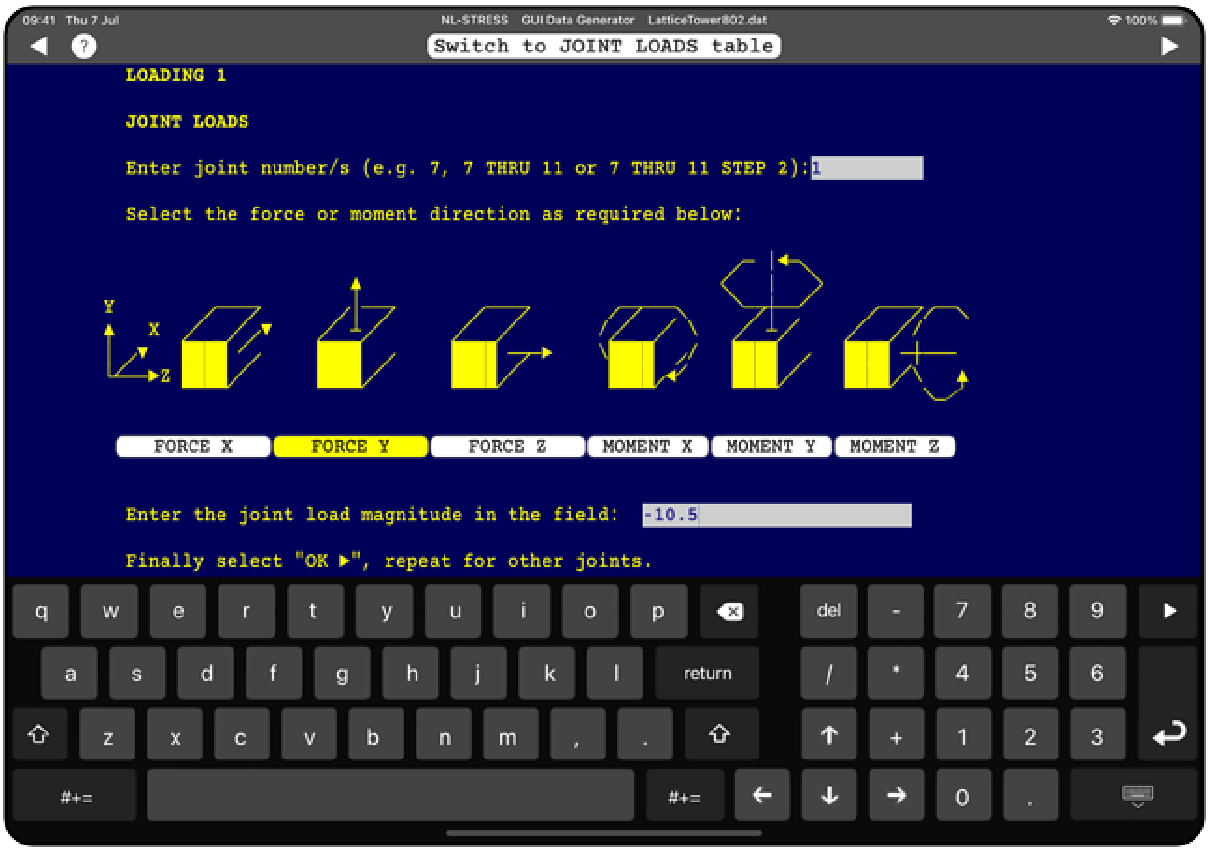


Figure 5.50: GUI JOINT LOADS dialog.

5.7.22.2 GUI - JOINT LOADS table

Either select the menu option "Loadings->JOINT LOADS table", double click/tap on any JOINT LOADS line in the main GUI window, or select the "Switch to JOINT LOADS table" on the JOINT LOADS dialog, to switch to the JOINT LOADS table entry screen, as shown in Figure 5.51.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

You may select the "Switch to JOINT LOADS GUI" button to switch to the JOINT LOADS dialog entry screen.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.51: GUI JOINT LOADS table.

5.7.23 The JOINT DISPLACEMENTS table

Syntax JOINT DISPLACEMENTS

```
<joints> [ DISPLACEMENT|ROTATION [ X|Y|Z <value> ] ]
```

Purpose Specify displacements and rotations of supported joints - typically to study the effects of structural settlement.

Usage This table is optional and there may be more than one of them among the data for a loading condition.

Displacements and rotations may be applied only to joints that are supported. If a supported joint is released in a certain direction then no displacement may be applied in that particular direction.

Directions of displacement appropriate to each type of structure are specified relative to global axes as follows:

```
Plane truss:  DISPLACEMENT X,Y
Plane frame:  DISPLACEMENT X,Y    ROTATION Z
Plane grid:   ROTATION X,Y        DISPLACEMENT Z
Space truss:  DISPLACEMENT X,Y,Z
Space frame:  DISPLACEMENT X,Y,Z  ROTATION X,Y,Z
```

Linear displacements are measured in length units; rotations in radians.

It is permissible to omit labels to the joint displacements and give only a set of values in the above order (using zero when no displacement is required in that particular direction). Trailing zeros may be omitted. The second example below provides the same data as the first example (providing that both examples are for a plane frame).

Examples

```
JOINT DISPLACEMENTS
1 THRU 2 DISPLACEMENT Y -0.002
3 ROTATION Z 0.001
4 7 11 INCLUSIVE DISPLACEMENT X 0.007
5 6 8 9 10 INCLUSIVE DISPLACEMENT X 0.014

JOINT DISPLACEMENTS
1 THRU 2 0 -0.002
3 0 0 0.001
```

5.7.23.1 GUI - JOINT DISPLACEMENTS dialog

There are two ways of entering JOINT DISPLACEMENTS data in the GUI, either using the JOINT DISPLACEMENTS dialog, or by editing the JOINT DISPLACEMENTS table.

To enter data using the JOINT DISPLACEMENTS dialog, select the menu option "Loadings->JOINT DISPLACEMENTS" or select the button "Switch to JOINT DISPLACEMENTS GUI" on the JOINT DISPLACEMENTS table screen.

An example of the JOINT DISPLACEMENTS dialog is shown in Figure 5.52.

The dialog will present the appropriate directions only for the current structure type. The directions appropriate for each structure type are listed in the previous section.

Select "►" to add the JOINT DISPLACEMENTS data, or "◄" to return to the main GUI window without adding the data.

Select the "Switch to JOINT DISPLACEMENTS table" button to switch to entering data using the GUI JOINT DISPLACEMENTS table option.

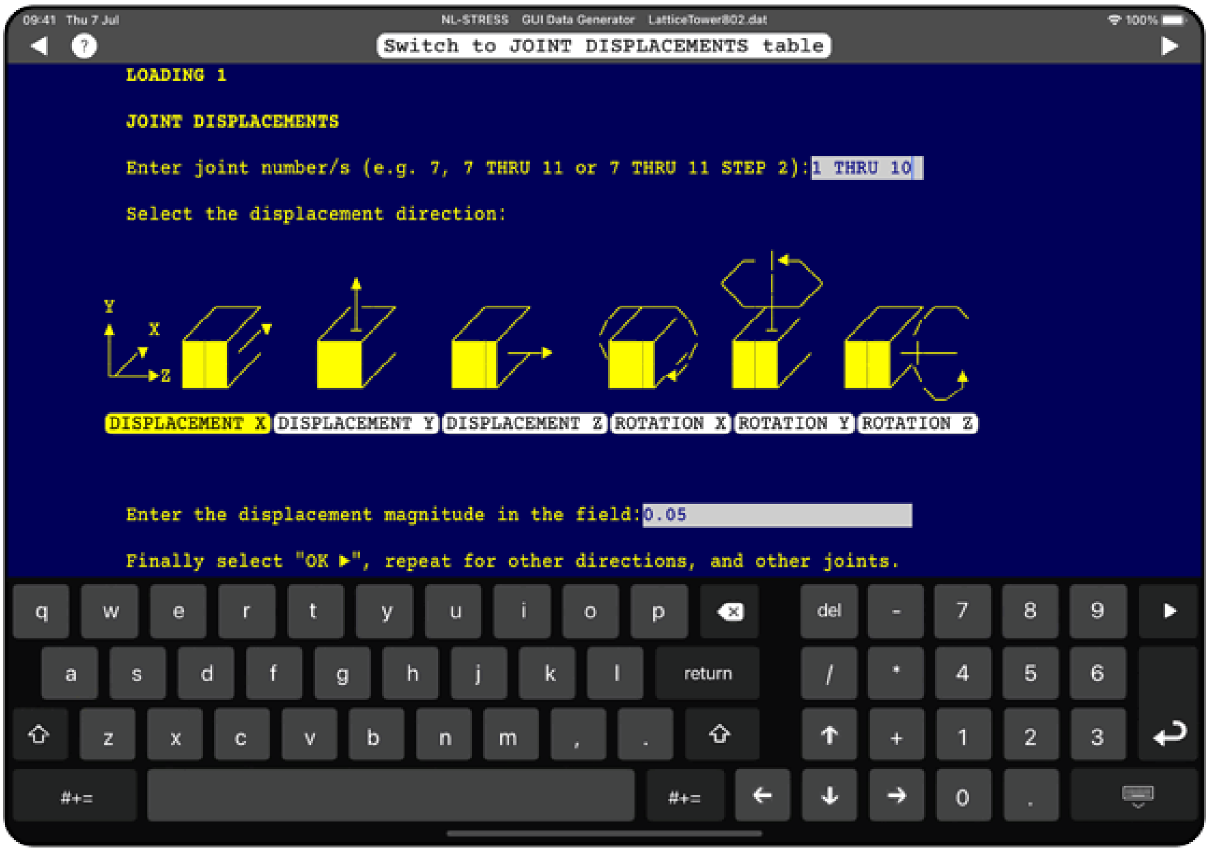


Figure 5.52: GUI JOINT DISPLACEMENTS dialog.

5.7.23.2 GUI - JOINT DISPLACEMENTS table

Either select the menu option "Loadings->JOINT DISPLACEMENTS table", double click/tap on any JOINT DISPLACEMENTS line in the main GUI window, or select the "Switch to JOINT DISPLACEMENTS table" on the JOINT DISPLACEMENTS dialog, to switch to the JOINT DISPLACEMENTS table entry screen, as shown in Figure 5.53.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

You may select the "Switch to JOINT DISPLACEMENTS GUI" button to switch to the JOINT DISPLACEMENTS dialog entry screen.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.53: GUI JOINT DISPLACEMENTS table.

5.7.24 The MEMBER LOADS table

Syntax

```
<members> FORCE|MOMENT X|Y|Z (GLOBAL) CONCENTRATED [ P|L <value> ]

<members> FORCE|MOMENT X|Y|Z (GLOBAL|PROJECTED) UNIFORM [ W|LA|LB
                                                          <value> ]

<members> FORCE|MOMENT X|Y|Z (GLOBAL|PROJECTED) LINEAR [ WA|WB|LA|LB
                                                          <value> ]
```

Purpose — Apply a concentrated, uniformly-distributed, or linearly-distributed load or moment to a member. A distributed load or moment may be applied over any part of the length of the member.

Usage — This table is optional and there may be more than one of them among the data for a loading condition. The loads specified in the examples are depicted in Figure 5.54.

Omission of both FORCE and MOMENT implies FORCE by default. Omission of keywords GLOBAL and PROJECTED signifies that the nominated X, Y or Z is a local axis of the loaded member.

The difference between values for LB and LA gives the loaded length of member directly.

The keyword GLOBAL signifies that the nominated X, Y or Z is a global axis.

The keyword PROJECTED also signifies that the nominated X, Y or Z is a global axis. But the loaded length of member is found by projecting the length of member between LA and LB onto a plane normal to the nominated global axis.

Keyword P signifies a point load (force units) or point moment (moment units: force times length). Keywords W, WA, WB introduce intensities of a distributed load (force units per length unit) or distributed moment (moment units per length unit: i.e. force units).

The keywords L, LA, LB signify distance to point load, to start of distributed load, to end of distributed load respectively. All are measured in length units.

Omission of WA signifies zero (a triangular load with maximum intensity WB) and similarly for WB.

Omission of LA implies that the load starts where the member starts; omission of LB that the load ends where the member ends.

For linear elastic analysis member loading is defined in relation to the undisturbed geometry of the structure. For non linear analysis NL-STRESS treats member loading in the following way:

- concentrated loads specified at a distance from the start of the member have their position varied so that the ratio of L to original member length remains constant. Thus midpoint loads remain at the midpoint regardless of change in member length

- uniform and linear loads, and global uniform and linear loads, have their positions varied as above and have their magnitudes varied so that the total load applied remains constant

- member self weights are adjusted so that the self weight does not change with variation in length of member

- projected uniform and linear loads have their magnitudes adjusted to compensate for change in member length but the projected length over which they act changes with change in frame geometry as would be expected.

Examples

```
MEMBER LOADS
1 FORCE Y CONCENTRATED P 100 L 2.0
2 FORCE Y GLOBAL CONCENTRATED P 100 L 2.0
3 MOMENT Z CONCENTRATED P 100 L 2.0
4 FORCE Y UNIFORM W 12
5 FORCE Y GLOBAL UNIFORM W 12
6 FORCE Y PROJECTED UNIFORM W 12
7 FORCE Y UNIFORM W 12 LA .7 LB 2.5
8 FORCE Y GLOBAL LINEAR WB 12 LA .7 LB 2.5
9 FORCE Y PROJECTED LINEAR WA 12 LA .7 LB 2.5
```

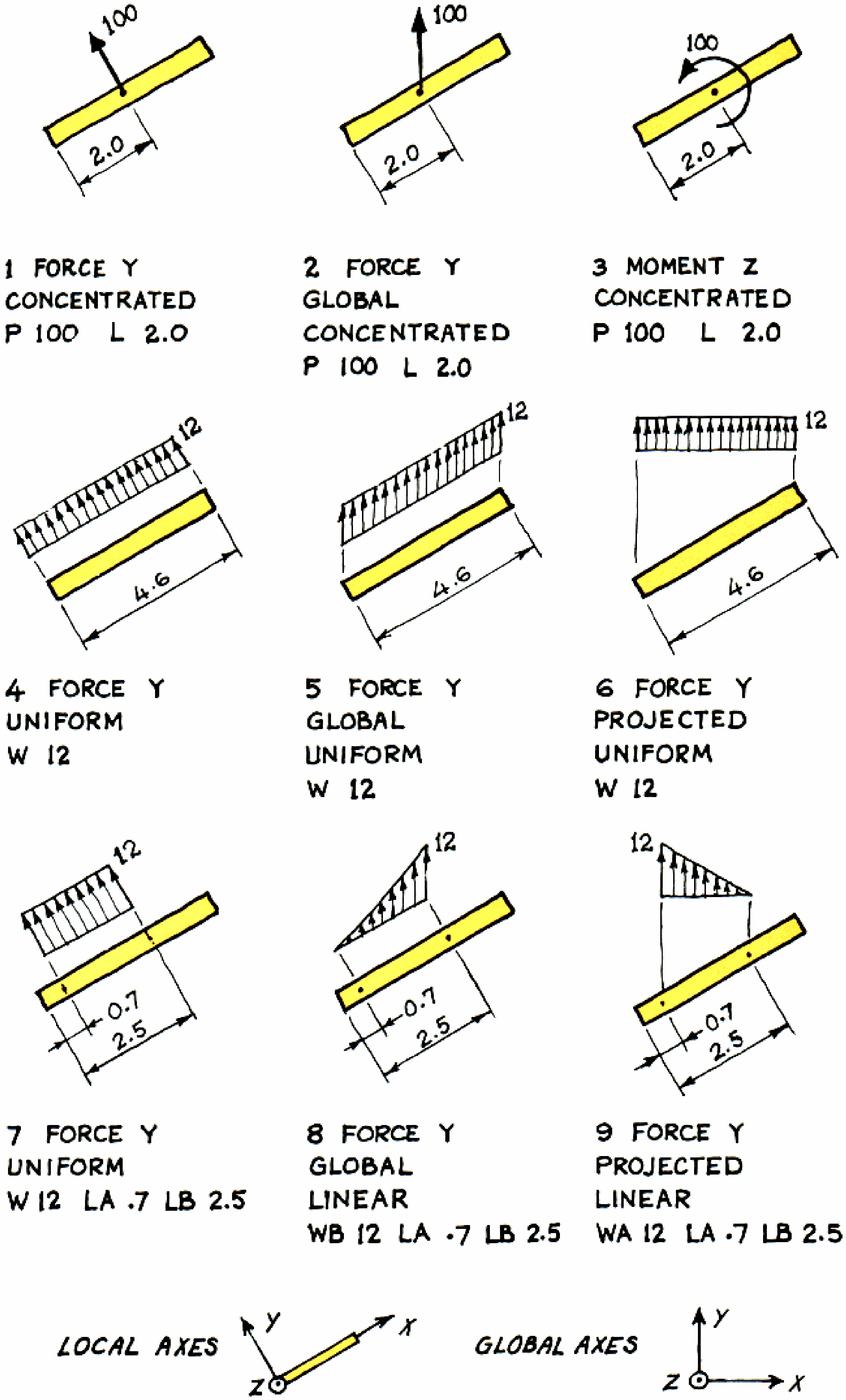


Figure 5.54: Examples of MEMBER LOADS

5.7.24.1 GUI - MEMBER LOADS dialog

There are two ways of entering MEMBER LOADS data in the GUI, either using the MEMBER LOADS dialog, or by editing the MEMBER LOADS table.

To enter data using the MEMBER LOADS dialog, select the menu option "Loadings->MEMBER LOADS" or select the button "Switch to MEMBER LOADS GUI" on the MEMBER LOADS table screen.

An example of the MEMBER LOADS dialog is shown in Figure 5.55.

The dialog will present the appropriate directions only for the current structure type.

Select "►" to add the MEMBER LOADS data, or "◄" to return to the main GUI window without adding the data.

Select the "Switch to MEMBER LOADS table" button to switch to entering data using the GUI MEMBER LOADS table option.

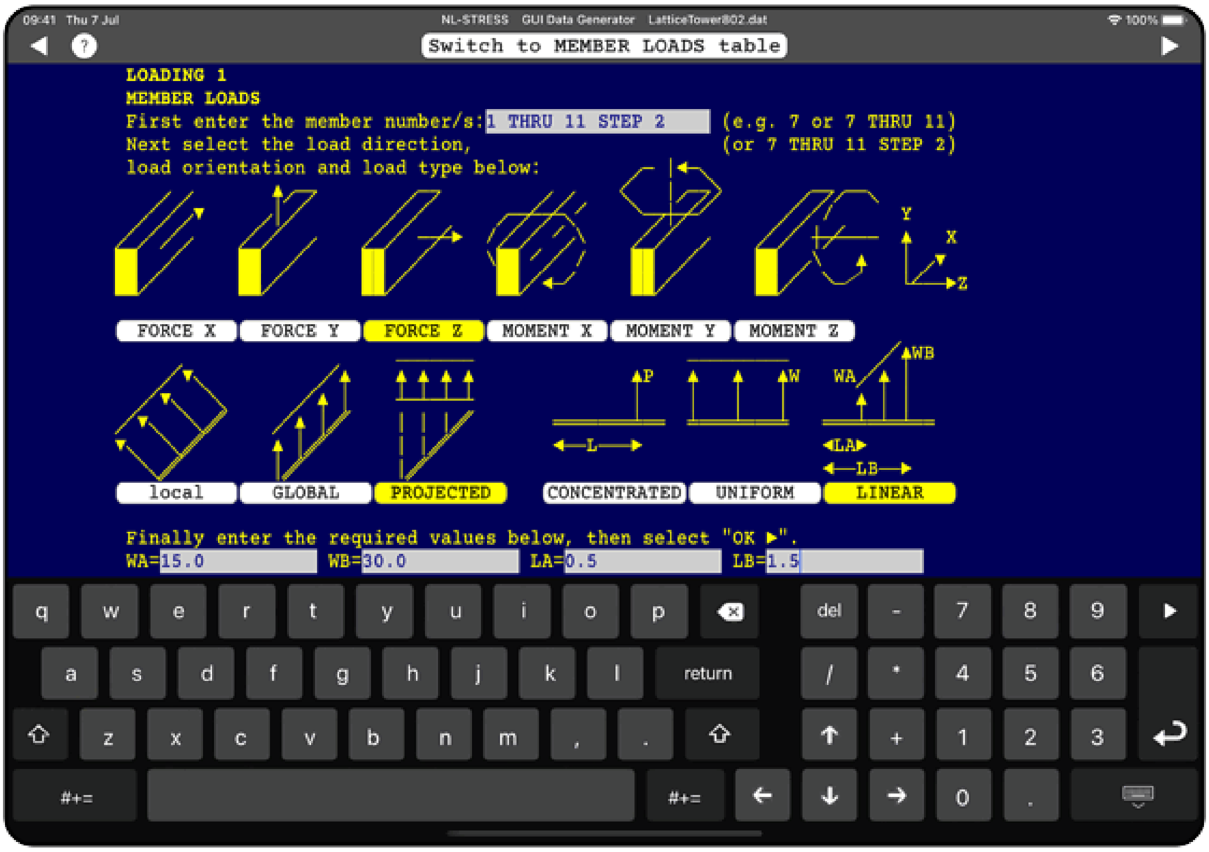


Figure 5.55: GUI JOINT LOADS dialog.

5.7.24.2 GUI - MEMBER LOADS table

Either select the menu option "Loadings->MEMBER LOADS table", double click/tap on any MEMBER LOADS line in the main GUI window, or select the "Switch to MEMBER LOADS table" on the MEMBER LOADS dialog, to switch to the MEMBER LOADS table entry screen, as shown in Figure 5.56.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

You may select the "Switch to MEMBER LOADS GUI" button to switch to the MEMBER LOADS dialog entry screen.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.56: GUI MEMBER LOADS table.

5.7.25 The MEMBER DISTORTIONS table

Syntax MEMBER DISTORTIONS

```
&lt;members&gt; [ DISTORTION|ROTATION [ X|Y|Z &lt;value&gt; ] ]
```

Purpose Chiefly to study 'lack of fit' problems. The member is distorted in the directions specified, then 'clamped' into the structure at its ends and let go.

Usage This table is optional and there may be more than one of them among the data for a loading condition. Distortions are measured in length units, rotations in radians, and the directions are the directions of local axes. The example is for a member compressed before building into the structure.

Directions of displacement appropriate to each type of structure are specified relative to local axes as follows:

```
Plane truss:  DISTORTION X,Y
Plane frame:  DISTORTION X,Y    ROTATION Z
Plane grid:   ROTATION X,Y      DISTORTION Z
Space truss:  DISTORTION X,Y,Z
Space frame:  DISTORTION X,Y,Z  ROTATION X,Y,Z
```

It is permissible to omit labels to the joint displacements and give only a set of values in the above order (using zero when no distortion is required in that particular direction). Trailing zeros may be omitted. The second example below provides the same data as the first example (providing that both examples are for a plane frame).

Examples

```
MEMBER DISTORTIONS
1 DISTORTION X 0.001
2 THRU 8 STEP 2 DISTORTION X 0.005
3 5 7 9 10 INCLUSIVE DISTORTION X 0.007

MEMBER DISTORTIONS
1 0.001
```

5.7.25.1 GUI - MEMBER DISTORTIONS dialog

There are two ways of entering MEMBER DISTORTIONS data in the GUI, either using the MEMBER DISTORTIONS dialog, or by editing the MEMBER DISTORTIONS table.

To enter data using the MEMBER DISTORTIONS dialog, select the menu option "Loadings->MEMBER DISTORTIONS" or select the button "Switch to MEMBER DISTORTIONS GUI" on the JOINT LOADS table screen.

An example of the MEMBER DISTORTIONS dialog is shown in Figure 5.57.

The dialog will present the appropriate directions only for the current structure type. The directions appropriate for each structure type are listed in the previous section.

Select "►" to add the MEMBER DISTORTIONS data, or "◄" to return to the main GUI window without adding the data.

Select the "Switch to MEMBER DISTORTIONS table" button to switch to entering data using the GUI MEMBER DISTORTIONS table option.

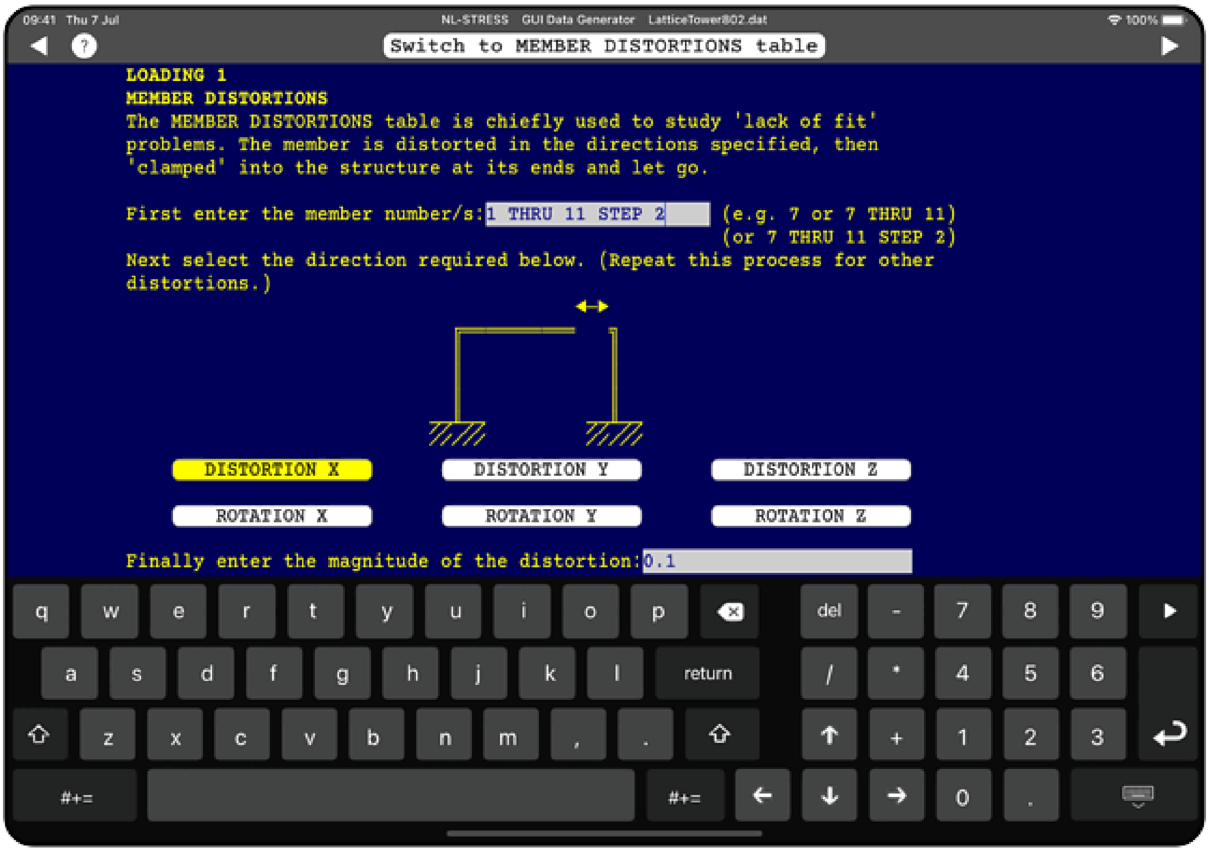


Figure 5.57: GUI MEMBER DISTORTIONS dialog.

5.7.25.2 GUI - MEMBER DISTORTIONS table

Either select the menu option "Loadings->MEMBER DISTORTIONS table", double click/tap on any MEMBER DISTORTIONS line in the main GUI window, or select the "Switch to MEMBER DISTORTIONS table" on the MEMBER DISTORTIONS dialog, to switch to the MEMBER DISTORTIONS table entry screen, as shown in Figure 5.58.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

You may select the "Switch to MEMBER DISTORTIONS GUI" button to switch to the MEMBER DISTORTIONS dialog entry screen.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.58: GUI MEMBER DISTORTIONS table.

5.7.26 The MEMBER TEMPERATURE CHANGES table

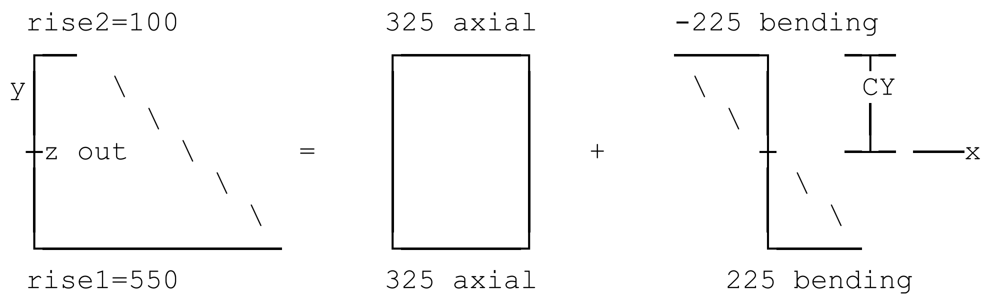
Syntax   MEMBER TEMPERATURE CHANGES

<members> <rise1> (<rise2>)

Purpose   Investigate the effect of temperature change. A rise in temperature is given positively, a fall negatively.

Usage   This table is optional and there may be more than one of them among data for a loading condition. For this table to have any effect, the constant with keyword CTE must have been set for all members referred to. Structures of type PLANE GRID do not consider axial loads in the members and setting CTE for grids produces no effect.

For a plane frame/truss <rise1> refers to the temperature at the face of the beam nearest the local origin, <rise2> refers to the temperature at the face of the beam furthest from the local origin. If <rise2> equals <rise1>, then <rise2> may be omitted and the temperature change will only cause an axial stress in the member, as in the first example below. If <rise2> is not equal to <rise1>, as in the second example below, then the member/s will be subjected to bending stress in addition to axial stress (unless of course the members are unrestrained). For the second example the two temperatures cause axial and bending effects thus:



For the above temperature changes on a plane frame member an unrestrained beam would curve in the arc of a circle, the radius of the circle depending on the depth of the beam. For temperature stresses, NL-STRESS assumes that the depth of the beam is twice CY; thus for temperature stresses, CY must be given a value in the member properties table, directly, or indirectly by defining the section geometry e.g. RECTANGLE DY 0.6 DZ 0.3.

Space frame members are treated in a similar manner to the above; but in addition to, or instead of, bending about the Z axis as in a plane frame, the engineer may wish to consider bending about the Y axis. For a space frame, if NL-STRESS finds that CZ has been set, then bending about the Y axis is considered. The setting or not setting of CY and CZ thereby control whether bending about the Z or Y axis respectively are taken into consideration for temperature changes.

Example

```
MEMBER TEMPERATURE CHANGES
1 THRU 36 -15
37,41,48 INCLUSIVE 550 100
38 39 40 42 43 44 46 INCLUSIVE 450 150
```

```
45 47 BOTH 450 150
48 THRU 49 400
50 51 BOTH 390
52 53 INCLUSIVE 380
```

5.7.26.1 GUI - MEMBER TEMPERATURE CHANGES table

Either select the menu option "Loadings->MEMBER TEMPERATURE CHANGES", or double click/tap on any MEMBER TEMPERATURE CHANGES line in the main GUI window, to switch to the MEMBER TEMPERATURE CHANGES table entry screen, as shown in Figure 5.59.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.59: GUI MEMBER TEMPERATURE CHANGES table.

5.7.27 The MEMBER SELF WEIGHTS table

Syntax MEMBER SELF WEIGHTS

<members> <factor> (<direction>)

Purpose Let NL-STRESS compute self weights of members from given densities, given joint coordinates and given cross-sectional areas.

NL-STRESS applies the resulting gravitational forces in a direction opposite that of the global Y axis for plane and space frames and opposite that of the global Z axis for grids. For space frames it is permissible to change the direction from the global Y axis to the global Z axis.

Usage This table is optional and there may be more than one among data for a loading condition.

The constant with keyword DENSITY must have been set for all members which are to contribute their weight.

<factor> is a multiplying factor by which NL-STRESS multiplies calculated self weights. The factor allows account to be taken of cladding and finishes (12% extra weight is allowed for in the example below).

<direction> =2 or 3 for space frames applies the resulting gravitational forces in the direction opposite that of the global Y or Z axes respectively.

<direction> should be omitted for plane frames & grids. NL-STRESS applies the resulting gravitational forces in a direction opposite that of the global Y axis for plane frames and opposite that of the global Z axis for grids.

Example

```
MEMBER SELF WEIGHTS
1 THRU 36 1.12
37 39 BOTH 1.20
38 40 41 42 INCLUSIVE 1.12
```

5.7.27.1 GUI - MEMBER SELF WEIGHTS table

Either select the menu option "Loadings->MEMBER SELF WEIGHTS", or double click/tap on any MEMBER SELF WEIGHTS line in the main GUI window, to switch to the MEMBER SELF WEIGHTS table entry screen, as shown in Figure 5.60.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.60: GUI MEMBER SELF WEIGHTS table.

5.7.28 The MEMBER LENGTH COEFFICIENTS table

Syntax MEMBER LENGTH COEFFICIENTS

<members> <strain>

Purpose Study the effect of creep and shrinkage.

Usage This table is optional and there may be more than one of them among data for a loading condition.

The example below shows an alternative way of expressing the data in the earlier example of MEMBER DISTORTIONS, assuming the original length of member 1 was 0.77

Example

```
MEMBER LENGTH COEFFICIENTS
1       0.001/0.77
2 4 6   0.0054
3 5     BOTH 0.006
```

5.7.28.1 GUI - MEMBER LENGTH COEFFICIENTS table

Either select the menu option "Loadings->MEMBER LENGTH COEFFICIENTS" or double click/tap on any MEMBER LENGTH COEFFICIENTS line in the main GUI window, to switch to the MEMBER LENGTH COEFFICIENTS table entry screen, as shown in Figure 5.61.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.61: GUI MEMBER LENGTH COEFFICIENTS table.

5.7.29 The COMBINE command

Syntax COMBINE [<basic condition> <factor>]

Purpose Specify a loading condition comprising a combination of basic loading conditions. This facility is not available for non-linear analysis.

Usage This command must be preceded by a LOADING command and optionally a TABULATE command.

The first LOADING command is denoted 1, the second denoted 2, and so on. This is the number nominated by <basic condition>.

<factor> is a multiplying factor by which all basic loads in a basic loading condition are multiplied as they are assembled to make the combined loading condition.

There may be several COMBINE commands in the data for a combination, but COMBINE commands may not be mixed with any other loading data. NL-STRESS places a limit of 25 numbers/line of data so if 14 loadcases must be combined then two combine commands will be needed as in the example.

Example

```
LOADING  Ultimate dead, live and reverse wind
COMBINE  1 1.4 2 1.4 3 1.4 4 1.4 5 1.4 6 1.4 7 1.4 8 1.4
COMBINE  12 1.6 13 1.6 14 1.6 17 1.6 18 1.6, 22 -1.0
```

5.7.29.1 GUI - COMBINE command

Either select the menu option "Loadings->COMBINE" or double click/tap on any COMBINE line in the main GUI window, to switch to the COMBINE table entry screen, as shown in Figure 5.62.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.62: GUI COMBINE command.

5.7.30 The MAXOF command

Syntax MAXOF [<basic condition>]

Purpose Specify a loading condition to be a maximum set of values picked from a selection of basic loading conditions; i.e. to build the upper bound of a moment or shear envelope at each segment end.

Usage This command must be preceded by a LOADING command and optionally a TABULATE command.

The first LOADING command is denoted 1, the second denoted 2, and so on. This is the number nominated by <basic condition>.

There may be several MAXOF commands in the data for a loading condition, but MAXOF commands may not be mixed with any other loading data. NL-STRESS places a limit of 25 numbers/line of data so that if the maximum values of 36 loadcases must be extracted then two commands will be needed as in the example.

Example

```
LOADING Max envelope
MAXOF 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20
MAXOF 21 22 23 24 25 26 27 28 29 30 31 32 33 34 35 36
```

5.7.30.1 GUI - MAXOF command

Either select the menu option "Loadings->MAXOF" or double click/tap on any MAXOF line in the main GUI window, to switch to the MAXOF table entry screen, as shown in Figure 5.63.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.63: GUI MAXOF command.

5.7.31 The MINOF command

Syntax MINOF [<basic condition>]

Purpose Specify a loading condition to be a minimum set of values picked from a selection of basic loading conditions; i.e. to build the lower bound of a moment or shear envelope at each segment end.

Usage This command must be preceded by a LOADING command and optionally a TABULATE command.

The first LOADING command is denoted 1, the second denoted 2, and so on. This is the number nominated by <basic condition>.

There may be several MINOF commands in the data for a loading condition, but MINOF commands may not be mixed with any other loading data. NL-STRESS places a limit of 25 numbers/line of data so that if the minimum values of 36 loadcases must be extracted then two commands will be needed as in the example.

Example

```
LOADING Min envelope
MINOF 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20
MINOF 21 22 23 24 25 26 27 28 29 30 31 32 33 34 35 36
```

5.7.31.1 GUI - MINOF command

Either select the menu option "Loadings->MINOF" or double click/tap on any MINOF line in the main GUI window, to switch to the MINOF table entry screen, as shown in Figure 5.64.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.64: GUI MINOF command.

5.7.32 The ABSOF command

Syntax ABSOF [<basic condition>]

Purpose Specify a loading condition comprising the numerically largest values from a set of basic loading conditions.

This loading condition is useful for picking out severest stresses from a set of load cases.

Usage This command must be preceded by a LOADING command and optionally a TABULATE command.

The first LOADING command is denoted 1, the second denoted 2, and so on. This is the number nominated by <basic condition>.

When STRESSES have been requested for an 'ABSOF' loading case, NL-STRESS shows the member numbers of those members which have maximum stresses in each component direction.

There may be several ABSOF commands in the data for a loading condition, but ABSOF commands may not be mixed with any other loading data. NL-STRESS places a limit of 25 numbers/line of data so that if the maximum stresses of 36 loadcases must be extracted, then two commands will be needed as in the example.

Example

```
LOADING Max stresses
ABSOF 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20
ABSOF 21 22 23 24 25 26 27 28 29 30 31 32 33 34 35 36
```

5.7.32.1 GUI - ABSOF command

Either select the menu option "Loadings->ABSOF" or double click/tap on any ABSOF line in the main GUI window, to switch to the ABSOF table entry screen, as shown in Figure 5.65.

The cursor will be moved to the line in the table corresponding to the line selected in the main GUI window.

Windows: the table may also be added using the right mouse button popup menu.

Enter the required data in the field provided.

Finally select "►" to accept changes, or "◄" to abandon changes.

Figure 5.65: GUI ABSOF command.

5.7.33 The SOLVE command

Syntax SOLVE

Purpose Ensure a solution; omission causes the data to be checked but no solution to be attempted.

Usage Place immediately before FINISH at the end of the data.

Example SOLVE

5.7.34 The FINISH command

Syntax FINISH

Purpose Mark the end of the data.

Usage This command must be included at the end of the data. NL-STRESS assumes that FINISH marks the end of the data and therefore ignores any data which follow.

Example FINISH

5.8 Quick reference

The syntax of all commands and tables is reproduced below, together with a section reference for each.

```
Capital letters indicate keywords.
Words in pointed brackets < > describe the kind of data needed.
Vertical bars | say 'or'.
( ) signify optional data.
[ ] include one or more items from within square brackets.
BLOCK sta end, copy lines sta-end.  #filename, inc. external file.
<joints> or <members> may be expressed e.g. 6   or   6 THRU 15
   or  6 THRU 15 STEP 3  or  6 31 BOTH  or  6 12 13 132 INCLUSIVE.
Conditionals: IF ... THEN ... (ENDIF)  IF ...  ...  ...  ENDIF
Structured looping: n=0  REPEAT  n=n+1 ...  UNTIL n=nj  ENDREPEAT.
Unstructured looping: n=0  :100  n=n+1 ...  IF n<nj GOTO 100.
```

```
TITLE <title: max 50 characters>                                 5.7.1
MADEBY <title: max 8 characters>                                 5.7.2
DATE <title: max 8 characters>                                   5.7.3
REFNO <title: max 8 characters>                                  5.7.4
TABULATE ([ FORCES|REACTIONS|DISPLACEMENTS|STRESSES ] | ALL)     5.7.5
PRINT [ DATA|RESULTS|COLLECTION|DIAGRAMS ] (FROM <page>
                                          (LENGTH <length>)) 5.7.6
TYPE  PLANE|SPACE  TRUSS|FRAME|GRID                              5.7.7
METHOD  ELASTIC|SWAY|PLASTIC  (JOINTS|NODES)                     5.7.8
NUMBER OF JOINTS <j>  ( LISTING <extent> )                       5.7.9
NUMBER OF MEMBERS <m> ( LISTING <extent> )                      5.7.10
NUMBER OF SUPPORTS <s>                                          5.7.11
NUMBER OF LOADINGS <l>                                          5.7.12
NUMBER OF INCREMENTS <i> (<accuracy>) (TRACE)                   5.7.13
NUMBER OF SEGMENTS <g> (TRACE)                                  5.7.14
JOINT COORDINATES (SYMMETRY X|Y|Z (<distance>))                 5.7.15
<joint> <x-coord> <y-coord> (<z-coord>) (S|SUPPORT)
<joints> SYMMETRY <other joint> (S|SUPPORT)
<joints> [ X|Y|Z|XL|YL|ZL <coord> ] (S|SUPPORT)
JOINT RELEASES                                                  5.7.16
<joints> [ FORCE|MOMENT  [ X|Y|Z (<spring const>) ] ]
MEMBER INCIDENCES                                               5.7.17
<member> <i> <j>
<members> CHAIN [ <j> ]
<members> RANGE <fi> <fj>  <li> <lj>
MEMBER RELEASES                                                 5.7.18
<members> [ START|END [ FORCE|MOMENT [ X|Y|Z (<spring const>) ] ] ]
CONSTANTS [ <constant> <value> <extent> ]                       5.7.19
where: <constant> is E|G|CTE|DENSITY|YIELD
and: <extent> is ALL | [ <member> ] | ALL BUT <value> [ <member>]
MEMBER PROPERTIES                                               5.7.20
<members> [ <property> <value> ]
<members> <shape> [ <dimension> <value> ]
<members> AS <other member>
where: <property> is AX|AY|AZ|IX|IY|IZ|C|CX|CY|CZ|BETA|FXP|MXP|MYP|MZP
    and: <shape> is RECTANGLE|CONIC|OCTAGON|ISECTION|TSECTION|HSECTION
    and: <dimension> is D|DY|DZ|T|TY|TZ|C|CX|CY|CZ|BETA
```

```
LOADING <title: max 50 characters>                                 5.7.21
JOINT LOADS                                                        5.7.22
<joints> [ FORCE|MOMENT [ X|Y|Z <value> ] ]
JOINT DISPLACEMENTS                                                5.7.23
<joints> [ DISPLACEMENT|ROTATION [ X|Y|Z <value> ] ]
MEMBER LOADS                                                       5.7.24
<members> FORCE|MOMENT X|Y|Z (GLOBAL) CONCENTRATED [ P|L <value> ]
<members> FORCE|MOMENT X|Y|Z (GLOBAL|PROJECTED) UNIFORM
                                                [ W|LA|LB <value>]
<members> FORCE|MOMENT X|Y|Z (GLOBAL|PROJECTED) LINEAR
                                            [ WA|WB|LA|LB <value>]
MEMBER DISTORTIONS                                                 5.7.25
<members> [ DISTORTION|ROTATION [ X|Y|Z <value> ] ]
MEMBER TEMPERATURE CHANGES                                         5.7.26
<members> <rise>
MEMBER SELF WEIGHTS                                                5.7.27
<members> <factor>
MEMBER LENGTH COEFFICIENTS                                         5.7.28
<members> <strain>
COMBINE [ <basic condition> <factor> ]                             5.7.29
MAXOF [ <basic condition> ] | ALL                                  5.7.30
MINOF [ <basic condition> ] | ALL                                  5.7.31
ABSOF [ <basic condition> ] | ALL                                  5.7.32
SOLVE                                                              5.7.33
FINISH                                                             5.7.34
```

5.9 Error messages

1	Check complete, number of errors is -
2	Combination only allowed with method elastic not allowed with non-linear at line -
4	Data out of range at line -
5	Data wrong for structure TYPE at line -
6	Displaced joint unsupported at line -
7	Element geometry error at line -
8	Error at line -
9	Errors prevent continuation past line -
10	Excessive deflection or rotation or unity factor
11	Expression or stack or syntax failure at line -
13	Instability corrected by soft spring
14	Invalid section property at line -
15	Invalid constants for member -
16	Isolated joint number -
17	Increments wrong for method
18	Invalid section property for member -
20	Member distortion must not be segmented at line -
21	Member/s not connected to structure
22	Missing command or table before line -
23	Missing data at or before line -
24	Mechanism at stiffness matrix row -
25	Missing section property for member -
28	Number of load cases found is wrong
29	Number of supports found is wrong
30	Number of joints found is wrong
31	Number of members found is wrong
33	Other load data may not be included at line -
34	Out of data at line -
38	Repeated command at line -
39	Repeated data at line -
40	Release not possible at line -
41	Renumbering not possible
42	Syntax check complete; number of errors -
43	Tabulate stresses not permitted for method plastic
44	TYPE not installed
45	Unstable direction number -
46	Unstable at joint number -
48	Virtual memory error
50	Zero length for member -

When NL-STRESS finds that the structure being analysed is a mechanism, it gives the joint number at which the failure occurred and a direction number as follows:

Plane frames & trusses: 1 = FORCE X, 2 = FORCE Y, 3 = MOMENT Z
Plane grids: 1 = MOMENT X, 2 = MOMENT Y, 3 = FORCE Z
Space frames: 1,2,3,4,5,6 correspond to FORCE X Y Z, MOMENT X Y Z

Because off-diagonal terms affect the analysis, the joint number and direction given may not be exact, but they are a good place to start.

As an example of an error, enter the name of the supplied file nlkcmg.dat; edit the data for the properties of member 4 to be:

```
4 AX 1E-4 IZ 238E18
```

on running the data the following error messages are displayed:

```
UNSTABLE AT JOINT NUMBER    2
UNSTABLE IN DIRECTION NUMBER    2
MECHANISM AT STIFFNESS MATRIX ROW    5
```

The forgoing is an example of incompatible data which causes a negative or zero value to appear on the leading diagonal of the stiffness matrix.

5.10 External files and linking

A file of data for NL-STRESS may contain instructions for inputting section properties from the NL-STRESS Standards' File, inputting blocks of data from external files, decoding and inputting selected lines of data from external files, piping data to named external files, copying and deleting files, and running SCALE. All these features enable the engineer to automate design processes.

5.10.1 The standards' file

Installed with NL-STRESS is a standards file called nls.sta. This is a text file and may be displayed to reveal that it contains section sizes for bending about major and minor axes (ISECTION & HSECTION) with each line containing a reference number: 1 for the first line; 2 for the second ....

Inclusion of a record number anywhere in the data file preceded by @ with/without a gap between e.g. @10 will cause (in this case) the tenth line of the standards file to replace the @10. Thus a line in the section properties table:

```
1 THRU 8 @10
```

would cause the section properties contained in the tenth line of the standards file to be assigned to members 1 to 8. To avoid uncertainty the line with the replaced section properties is printed if the PRINT DATA command is given and an integer number follows the @. It is permissible to use a variable name for the record number e.g.

```
a=10
1 THRU 8 @ a
```

would behave as before, but for this case - as it is possible that the value of the variable 'a' has been changed before the results are printed - the section properties are not displayed and the line of data is printed unchanged. When a variable name is given, allow one space between the @ and the variable name as shown; although NL-STRESS will accept the data with/without the space, if you use SCALE option 676/7 (to preprocess the data and replace all variables) the space will be necessary.

The current standards file - as supplied - follows; the number to the right before the section size is the record number referred to above. The line of data: 1 THRU 8 @3
will extract the 3rd record thus:

```
1 THRU 8 ISECTION DY .9266 DZ .3077 TZ .0195 TY .0320
```

Please note that the line of data: 1 THRU 8 ISECTION @3
will cause the word ISECTION to be duplicated thus causing an error.

```
ISECTN   DY 1.0361 DZ .3085 TZ .0300 TY .0541 !   1 :1016 x 305 x 487 UB
ISECTN   DY 1.0259 DZ .3054 TZ .0269 TY .0490 !   2 :             x 437 UB
ISECTN   DY 1.0160 DZ .3030 TZ .0244 TY .0439 !   3 :             x 393 UB
ISECTN   DY 1.0081 DZ .3020 TZ .0211 TY .0400 !   4 :             x 349 UB
ISECTN   DY 1.0000 DZ .3000 TZ .0191 TY .0359 !   5 :             x 314 UB
ISECTION DY .9901 DZ .3000 TZ .0165 TY .0310 !   6 :             x 272 UB
ISECTION DY .9802 DZ .3000 TZ .0165 TY .0260 !   7 :             x 249 UB
ISECTION DY .9703 DZ .3000 TZ .0160 TY .0211 !   8 :             x 222 UB
ISECTION DY .9210 DZ .4205 TZ .0214 TY .0366 !   9 : 914 x 419 x 388 UB
```

```
ISECTION DY .9118 DZ .4185 TZ .0194 TY .0320 !  10 :           x 343
ISECTION DY .9266 DZ .3077 TZ .0195 TY .0320 !  11 : 914 x 305 x 289 UB
ISECTION DY .9184 DZ .3055 TZ .0173 TY .0279 !  12 :           x 253
ISECTION DY .9104 DZ .3041 TZ .0159 TY .0239 !  13 :           x 224
ISECTION DY .9030 DZ .3033 TZ .0151 TY .0202 !  14 :           x 201
ISECTION DY .8509 DZ .2938 TZ .0161 TY .0268 !  15 : 838 x 292 x 226 UB
ISECTION DY .8407 DZ .2924 TZ .0147 TY .0217 !  16 :           x 194
ISECTION DY .8349 DZ .2917 TZ .0140 TY .0188 !  17 :           x 176
ISECTION DY .7698 DZ .2680 TZ .0156 TY .0254 !  18 : 762 x 267 x 197 UB
ISECTION DY .7622 DZ .2667 TZ .0143 TY .0216 !  19 :           x 173
ISECTION DY .7540 DZ .2652 TZ .0128 TY .0175 !  20 :           x 147
ISECTION DY .7500 DZ .2644 TZ .0120 TY .0155 !  21 :           x 134
ISECTION DY .6929 DZ .2558 TZ .0145 TY .0237 !  22 : 686 x 254 x 170 UB
ISECTION DY .6875 DZ .2545 TZ .0132 TY .0210 !  23 :           x 152
ISECTION DY .6835 DZ .2537 TZ .0124 TY .0190 !  24 :           x 140
ISECTION DY .6779 DZ .2530 TZ .0117 TY .0162 !  25 :           x 125
ISECTION DY .6358 DZ .3114 TZ .0184 TY .0314 !  26 : 610 x 305 x 238 UB
ISECTION DY .6202 DZ .3071 TZ .0141 TY .0236 !  27 :           x 179
ISECTION DY .6124 DZ .3048 TZ .0118 TY .0197 !  28 :           x 149
ISECTION DY .6172 DZ .2302 TZ .0131 TY .0221 !  29 : 610 x 229 x 140 UB
ISECTION DY .6122 DZ .2290 TZ .0119 TY .0196 !  30 :           x 125
ISECTION DY .6076 DZ .2282 TZ .0111 TY .0173 !  31 :           x 113
ISECTION DY .6026 DZ .2276 TZ .0105 TY .0148 !  32 :           x 101
ISECTION DY .5445 DZ .2119 TZ .0127 TY .0213 !  33 : 533 x 210 x 122 UB
ISECTION DY .5395 DZ .2108 TZ .0116 TY .0188 !  34 :           x 109
ISECTION DY .5367 DZ .2100 TZ .0108 TY .0174 !  35 :           x 101
ISECTION DY .5331 DZ .2093 TZ .0101 TY .0156 !  36 :           x  92
ISECTION DY .5283 DZ .2088 TZ .0096 TY .0132 !  37 :           x  82
ISECTION DY .4672 DZ .1928 TZ .0114 TY .0196 !  38 : 457 x 191 x  98 UB
ISECTION DY .4634 DZ .1919 TZ .0105 TY .0177 !  39 :           x  89
ISECTION DY .4600 DZ .1913 TZ .0099 TY .0160 !  40 :           x  82
ISECTION DY .4570 DZ .1904 TZ .0090 TY .0145 !  41 :           x  74
ISECTION DY .4534 DZ .1899 TZ .0085 TY .0127 !  42 :           x  67
ISECTION DY .4658 DZ .1553 TZ .0105 TY .0189 !  43 : 457 x 152 x  82 UB
ISECTION DY .4620 DZ .1544 TZ .0096 TY .0170 !  44 :           x  74
ISECTION DY .4580 DZ .1538 TZ .0090 TY .0150 !  45 :           x  67
ISECTION DY .4546 DZ .1529 TZ .0081 TY .0133 !  46 :           x  60
ISECTION DY .4498 DZ .1524 TZ .0076 TY .0109 !  47 :           x  52
ISECTION DY .4128 DZ .1795 TZ .0095 TY .0160 !  48 : 406 x 178 x  74 UB
ISECTION DY .4094 DZ .1788 TZ .0088 TY .0143 !  49 :           x  67
ISECTION DY .4064 DZ .1779 TZ .0079 TY .0128 !  50 :           x  60
ISECTION DY .4026 DZ .1777 TZ .0077 TY .0109 !  51 :           x  54
ISECTION DY .4032 DZ .1422 TZ .0068 TY .0112 !  52 : 406 x 140 x  46 UB
ISECTION DY .3980 DZ .1418 TZ .0064 TY .0086 !  53 :           x  39
ISECTION DY .3634 DZ .1732 TZ .0091 TY .0157 !  54 : 356 x 171 x  67 UB
ISECTION DY .3580 DZ .1722 TZ .0081 TY .0130 !  55 :           x  57
ISECTION DY .3550 DZ .1715 TZ .0074 TY .0115 !  56 :           x  51
ISECTION DY .3514 DZ .1711 TZ .0070 TY .0097 !  57 :           x  45
ISECTION DY .3534 DZ .1260 TZ .0066 TY .0107 !  58 : 356 x 127 x  39 UB
ISECTION DY .3490 DZ .1254 TZ .0060 TY .0085 !  59 :           x  33
ISECTION DY .3104 DZ .1669 TZ .0079 TY .0137 !  60 : 305 x 165 x  54 UB
ISECTION DY .3066 DZ .1657 TZ .0067 TY .0118 !  61 :           x  46
ISECTION DY .3034 DZ .1650 TZ .0060 TY .0102 !  62 :           x  40
ISECTION DY .3110 DZ .1253 TZ .0090 TY .0140 !  63 : 305 x 127 x  48 UB
ISECTION DY .3072 DZ .1243 TZ .0080 TY .0121 !  64 :           x  42
ISECTION DY .3044 DZ .1234 TZ .0071 TY .0107 !  65 :           x  37
ISECTION DY .3127 DZ .1024 TZ .0066 TY .0108 !  66 : 305 x 102 x  33 UB
ISECTION DY .3087 DZ .1018 TZ .0060 TY .0088 !  67 :           x  28
ISECTION DY .3051 DZ .1016 TZ .0058 TY .0070 !  68 :           x  25
ISECTION DY .2596 DZ .1473 TZ .0072 TY .0127 !  69 : 254 x 146 x  43 UB
ISECTION DY .2560 DZ .1464 TZ .0063 TY .0109 !  70 :           x  37
ISECTION DY .2514 DZ .1461 TZ .0060 TY .0086 !  71 :           x  31
ISECTION DY .2604 DZ .1022 TZ .0063 TY .0100 !  72 : 254 x 102 x  28 UB
ISECTION DY .2572 DZ .1019 TZ .0060 TY .0084 !  73 :           x  25
```

```
ISECTION DY .2540 DZ .1016 TZ .0057 TY .0068 !  74 :             x  22
ISECTION DY .2068 DZ .1339 TZ .0064 TY .0096 !  75 : 203 x 133 x  30 UB
ISECTION DY .2032 DZ .1332 TZ .0057 TY .0078 !  76 :             x  25
ISECTION DY .2032 DZ .1018 TZ .0054 TY .0093 !  77 : 203 x 102 x  23 UB
ISECTION DY .1778 DZ .1012 TZ .0048 TY .0079 !  78 : 178 x 102 x  19 UB
ISECTION DY .1524 DZ .0887 TZ .0045 TY .0077 !  79 : 152 x  89 x  16 UB
ISECTION DY .1270 DZ .0760 TZ .0040 TY .0076 !  80 : 127 x  76 x  13 UB

HSECTN  DZ 1.0361 DY .3085 TY .0300 TZ .0541 ! 101 :1016 x 305 x 487 UB
HSECTN  DZ 1.0259 DY .3054 TY .0269 TZ .0490 ! 102 :             x 437 UB
HSECTN  DZ 1.0160 DY .3030 TY .0244 TZ .0439 ! 103 :             x 393 UB
HSECTN  DZ 1.0081 DY .3020 TY .0211 TZ .0400 ! 104 :             x 349 UB
HSECTN  DZ 1.0000 DY .3000 TY .0191 TZ .0359 ! 105 :             x 314 UB
HSECTION DZ .9901 DY .3000 TY .0165 TZ .0310 ! 106 :             x 272 UB
HSECTION DZ .9802 DY .3000 TY .0165 TZ .0260 ! 107 :             x 249 UB
HSECTION DZ .9703 DY .3000 TY .0160 TZ .0211 ! 108 :             x 222 UB
HSECTION DZ .9210 DY .4205 TY .0214 TZ .0366 ! 109 : 914 x 419 x 388 UB
HSECTION DZ .9118 DY .4185 TY .0194 TZ .0320 ! 110 :             x 343
HSECTION DZ .9266 DY .3077 TY .0195 TZ .0320 ! 111 : 914 x 305 x 289 UB
HSECTION DZ .9184 DY .3055 TY .0173 TZ .0279 ! 112 :             x 253
HSECTION DZ .9104 DY .3041 TY .0159 TZ .0239 ! 113 :             x 224
HSECTION DZ .9030 DY .3033 TY .0151 TZ .0202 ! 114 :             x 201
HSECTION DZ .8509 DY .2938 TY .0161 TZ .0268 ! 115 : 838 x 292 x 226 UB
HSECTION DZ .8407 DY .2924 TY .0147 TZ .0217 ! 116 :             x 194
HSECTION DZ .8349 DY .2917 TY .0140 TZ .0188 ! 117 :             x 176
HSECTION DZ .7698 DY .2680 TY .0156 TZ .0254 ! 118 : 762 x 267 x 197 UB
HSECTION DZ .7622 DY .2667 TY .0143 TZ .0216 ! 119 :             x 173
HSECTION DZ .7540 DY .2652 TY .0128 TZ .0175 ! 120 :             x 147
HSECTION DZ .7500 DY .2644 TY .0120 TZ .0155 ! 121 :             x 134
HSECTION DZ .6929 DY .2558 TY .0145 TZ .0237 ! 122 : 686 x 254 x 170 UB
HSECTION DZ .6875 DY .2545 TY .0132 TZ .0210 ! 123 :             x 152
HSECTION DZ .6835 DY .2537 TY .0124 TZ .0190 ! 124 :             x 140
HSECTION DZ .6779 DY .2530 TY .0117 TZ .0162 ! 125 :             x 125
HSECTION DZ .6358 DY .3114 TY .0184 TZ .0314 ! 126 : 610 x 305 x 238 UB
HSECTION DZ .6202 DY .3071 TY .0141 TZ .0236 ! 127 :             x 179
HSECTION DZ .6124 DY .3048 TY .0118 TZ .0197 ! 128 :             x 149
HSECTION DZ .6172 DY .2302 TY .0131 TZ .0221 ! 129 : 610 x 229 x 140 UB
HSECTION DZ .6122 DY .2290 TY .0119 TZ .0196 ! 130 :             x 125
HSECTION DZ .6076 DY .2282 TY .0111 TZ .0173 ! 131 :             x 113
HSECTION DZ .6026 DY .2276 TY .0105 TZ .0148 ! 132 :             x 101
HSECTION DZ .5445 DY .2119 TY .0127 TZ .0213 ! 133 : 533 x 210 x 122 UB
HSECTION DZ .5395 DY .2108 TY .0116 TZ .0188 ! 134 :             x 109
HSECTION DZ .5367 DY .2100 TY .0108 TZ .0174 ! 135 :             x 101
HSECTION DZ .5331 DY .2093 TY .0101 TZ .0156 ! 136 :             x  92
HSECTION DZ .5283 DY .2088 TY .0096 TZ .0132 ! 137 :             x  82
HSECTION DZ .4672 DY .1928 TY .0114 TZ .0196 ! 138 : 457 x 191 x  98 UB
HSECTION DZ .4634 DY .1919 TY .0105 TZ .0177 ! 139 :             x  89
HSECTION DZ .4600 DY .1913 TY .0099 TZ .0160 ! 140 :             x  82
HSECTION DZ .4570 DY .1904 TY .0090 TZ .0145 ! 141 :             x  74
HSECTION DZ .4534 DY .1899 TY .0085 TZ .0127 ! 142 :             x  67
HSECTION DZ .4658 DY .1553 TY .0105 TZ .0189 ! 143 : 457 x 152 x  82 UB
HSECTION DZ .4620 DY .1544 TY .0096 TZ .0170 ! 144 :             x  74
HSECTION DZ .4580 DY .1538 TY .0090 TZ .0150 ! 145 :             x  67
HSECTION DZ .4546 DY .1529 TY .0081 TZ .0133 ! 146 :             x  60
HSECTION DZ .4498 DY .1524 TY .0076 TZ .0109 ! 147 :             x  52
HSECTION DZ .4128 DY .1795 TY .0095 TZ .0160 ! 148 : 406 x 178 x  74 UB
HSECTION DZ .4094 DY .1788 TY .0088 TZ .0143 ! 149 :             x  67
HSECTION DZ .4064 DY .1779 TY .0079 TZ .0128 ! 150 :             x  60
HSECTION DZ .4026 DY .1777 TY .0077 TZ .0109 ! 151 :             x  54
HSECTION DZ .4032 DY .1422 TY .0068 TZ .0112 ! 152 : 406 x 140 x  46 UB
HSECTION DZ .3980 DY .1418 TY .0064 TZ .0086 ! 153 :             x  39
HSECTION DZ .3634 DY .1732 TY .0091 TZ .0157 ! 154 : 356 x 171 x  67 UB
HSECTION DZ .3580 DY .1722 TY .0081 TZ .0130 ! 155 :             x  57
HSECTION DZ .3550 DY .1715 TY .0074 TZ .0115 ! 156 :             x  51
```

```
HSECTION DZ .3514 DY .1711 TY .0070 TZ .0097 ! 157 :           x  45
HSECTION DZ .3534 DY .1260 TY .0066 TZ .0107 ! 158 : 356 x 127 x  39 UB
HSECTION DZ .3490 DY .1254 TY .0060 TZ .0085 ! 159 :           x  33
HSECTION DZ .3104 DY .1669 TY .0079 TZ .0137 ! 160 : 305 x 165 x  54 UB
HSECTION DZ .3066 DY .1657 TY .0067 TZ .0118 ! 161 :           x  46
HSECTION DZ .3034 DY .1650 TY .0060 TZ .0102 ! 162 :           x  40
HSECTION DZ .3110 DY .1253 TY .0090 TZ .0140 ! 163 : 305 x 127 x  48 UB
HSECTION DZ .3072 DY .1243 TY .0080 TZ .0121 ! 164 :           x  42
HSECTION DZ .3044 DY .1234 TY .0071 TZ .0107 ! 165 :           x  37
HSECTION DZ .3127 DY .1024 TY .0066 TZ .0108 ! 166 : 305 x 102 x  33 UB
HSECTION DZ .3087 DY .1018 TY .0060 TZ .0088 ! 167 :           x  28
HSECTION DZ .3051 DY .1016 TY .0058 TZ .0070 ! 168 :           x  25
HSECTION DZ .2596 DY .1473 TY .0072 TZ .0127 ! 169 : 254 x 146 x  43 UB
HSECTION DZ .2560 DY .1464 TY .0063 TZ .0109 ! 170 :           x  37
HSECTION DZ .2514 DY .1461 TY .0060 TZ .0086 ! 171 :           x  31
HSECTION DZ .2604 DY .1022 TY .0063 TZ .0100 ! 172 : 254 x 102 x  28 UB
HSECTION DZ .2572 DY .1019 TY .0060 TZ .0084 ! 173 :           x  25
HSECTION DZ .2540 DY .1016 TY .0057 TZ .0068 ! 174 :           x  22
HSECTION DZ .2068 DY .1339 TY .0064 TZ .0096 ! 175 : 203 x 133 x  30 UB
HSECTION DZ .2032 DY .1332 TY .0057 TZ .0078 ! 176 :           x  25
HSECTION DZ .2032 DY .1018 TY .0054 TZ .0093 ! 177 : 203 x 102 x  23 UB
HSECTION DZ .1778 DY .1012 TY .0048 TZ .0079 ! 178 : 178 x 102 x  19 UB
HSECTION DZ .1524 DY .0887 TY .0045 TZ .0077 ! 179 : 152 x  89 x  16 UB
HSECTION DZ .1270 DY .0760 TY .0040 TZ .0076 ! 180 : 127 x  76 x  13 UB

ISECTION DY .4746 DZ .4240 TZ .0476 TY .0770 ! 201 : 356 x 406 x 634 UC
ISECTION DY .4556 DZ .4185 TZ .0421 TY .0675 ! 202 :           x 551
ISECTION DY .4366 DZ .4122 TZ .0358 TY .0580 ! 203 :           x 467
ISECTION DY .4190 DZ .4070 TZ .0306 TY .0492 ! 204 :           x 393
ISECTION DY .4064 DZ .4030 TZ .0266 TY .0429 ! 205 :           x 340
ISECTION DY .3936 DZ .3990 TZ .0226 TY .0365 ! 206 :           x 287
ISECTION DY .3810 DZ .3948 TZ .0184 TY .0302 ! 207 :           x 235
ISECTION DY .3746 DZ .3747 TZ .0165 TY .0270 ! 208 : 356 x 368 x 202 UC
ISECTION DY .3682 DZ .3726 TZ .0144 TY .0238 ! 209 :           x 177
ISECTION DY .3620 DZ .3705 TZ .0123 TY .0207 ! 210 :           x 153
ISECTION DY .3556 DZ .3686 TZ .0104 TY .0175 ! 211 :           x 129
ISECTION DY .3653 DZ .3222 TZ .0268 TY .0441 ! 212 : 305 x 305 x 283 UC
ISECTION DY .3525 DZ .3184 TZ .0230 TY .0377 ! 213 :           x 240
ISECTION DY .3399 DZ .3145 TZ .0191 TY .0314 ! 214 :           x 198
ISECTION DY .3271 DZ .3112 TZ .0158 TY .0250 ! 215 :           x 158
ISECTION DY .3205 DZ .3092 TZ .0138 TY .0217 ! 216 :           x 137
ISECTION DY .3145 DZ .3074 TZ .0120 TY .0187 ! 217 :           x 118
ISECTION DY .3079 DZ .3053 TZ .0099 TY .0154 ! 218 :           x  97
ISECTION DY .2891 DZ .2652 TZ .0192 TY .0317 ! 219 : 254 x 254 x 167 UC
ISECTION DY .2763 DZ .2613 TZ .0153 TY .0253 ! 220 :           x 132
ISECTION DY .2667 DZ .2588 TZ .0128 TY .0205 ! 221 :           x 107
ISECTION DY .2603 DZ .2563 TZ .0103 TY .0173 ! 222 :           x  89
ISECTION DY .2541 DZ .2546 TZ .0086 TY .0142 ! 223 :           x  73
ISECTION DY .2222 DZ .2091 TZ .0127 TY .0205 ! 224 : 203 x 203 x  86 UC
ISECTION DY .2158 DZ .2064 TZ .0100 TY .0173 ! 225 :           x  71
ISECTION DY .2096 DZ .2058 TZ .0094 TY .0142 ! 226 :           x  60
ISECTION DY .2062 DZ .2043 TZ .0079 TY .0125 ! 227 :           x  52
ISECTION DY .2032 DZ .2036 TZ .0072 TY .0110 ! 228 :           x  46
ISECTION DY .1618 DZ .1544 TZ .0080 TY .0115 ! 229 : 152 x 152 x  37 UC
ISECTION DY .1576 DZ .1529 TZ .0065 TY .0094 ! 230 :           x  30
ISECTION DY .1524 DZ .1522 TZ .0058 TY .0068 ! 231 :           x  23

HSECTION DZ .4746 DY .4240 TY .0476 TZ .0770 ! 251 : 356 x 406 x 634 UC
HSECTION DZ .4556 DY .4185 TY .0421 TZ .0675 ! 252 :           x 551
HSECTION DZ .4366 DY .4122 TY .0358 TZ .0580 ! 253 :           x 467
HSECTION DZ .4190 DY .4070 TY .0306 TZ .0492 ! 254 :           x 393
HSECTION DZ .4064 DY .4030 TY .0266 TZ .0429 ! 255 :           x 340
HSECTION DZ .3936 DY .3990 TY .0226 TZ .0365 ! 256 :           x 287
HSECTION DZ .3810 DY .3948 TY .0184 TZ .0302 ! 257 :           x 235
```

```
    HSECTION DZ .3746 DY .3747 TY .0165 TZ .0270 ! 258 : 356 x 368 x 202 UC
    HSECTION DZ .3682 DY .3726 TY .0144 TZ .0238 ! 259 :           x 177
    HSECTION DZ .3620 DY .3705 TY .0123 TZ .0207 ! 260 :           x 153
    HSECTION DZ .3556 DY .3686 TY .0104 TZ .0175 ! 261 :           x 129
    HSECTION DZ .3653 DY .3222 TY .0268 TZ .0441 ! 262 : 305 x 305 x 283 UC
    HSECTION DZ .3525 DY .3184 TY .0230 TZ .0377 ! 263 :           x 240
    HSECTION DZ .3399 DY .3145 TY .0191 TZ .0314 ! 264 :           x 198
    HSECTION DZ .3271 DY .3112 TY .0158 TZ .0250 ! 265 :           x 158
    HSECTION DZ .3205 DY .3092 TY .0138 TZ .0217 ! 266 :           x 137
    HSECTION DZ .3145 DY .3074 TY .0120 TZ .0187 ! 267 :           x 118
    HSECTION DZ .3079 DY .3053 TY .0099 TZ .0154 ! 268 :           x  97
    HSECTION DZ .2891 DY .2652 TY .0192 TZ .0317 ! 269 : 254 x 254 x 167 UC
    HSECTION DZ .2763 DY .2613 TY .0153 TZ .0253 ! 270 :           x 132
    HSECTION DZ .2667 DY .2588 TY .0128 TZ .0205 ! 271 :           x 107
    HSECTION DZ .2603 DY .2563 TY .0103 TZ .0173 ! 272 :           x  89
    HSECTION DZ .2541 DY .2546 TY .0086 TZ .0142 ! 273 :           x  73
    HSECTION DZ .2222 DY .2091 TY .0127 TZ .0205 ! 274 : 203 x 203 x  86 UC
    HSECTION DZ .2158 DY .2064 TY .0100 TZ .0173 ! 275 :           x  71
    HSECTION DZ .2096 DY .2058 TY .0094 TZ .0142 ! 276 :           x  60
    HSECTION DZ .2062 DY .2043 TY .0079 TZ .0125 ! 277 :           x  52
    HSECTION DZ .2032 DY .2036 TY .0072 TZ .0110 ! 278 :           x  46
    HSECTION DZ .1618 DY .1544 TY .0080 TZ .0115 ! 279 : 152 x 152 x  37 UC
    HSECTION DZ .1576 DY .1529 TY .0065 TZ .0094 ! 280 :           x  30
    HSECTION DZ .1524 DY .1522 TY .0058 TZ .0068 ! 281 :           x  23

    ISECTION DY .2540 DZ .2032 TZ .0102 TY .0199 ! 301 : 254 x 203 x 82    RSJ
    ISECTION DY .2032 DZ .1524 TZ .0089 TY .0165 ! 302 : 203 x 152 x 52    RSJ
    ISECTION DY .1524 DZ .1270 TZ .0104 TY .0132 ! 303 : 152 x 127 x 37    RSJ
    ISECTION DY .1270 DZ .1143 TZ .0102 TY .0115 ! 304 : 127 x 114 x 29    RSJ
    ISECTION DY .1270 DZ .1143 TZ .0074 TY .0114 ! 305 :     x 114 x 27
    ISECTION DY .1016 DZ .1016 TZ .0095 TY .0103 ! 306 : 102 x 102 x 23    RSJ
    ISECTION DY .0889 DZ .0889 TZ .0095 TY .0099 ! 307 :  89 x  89 x 19    RSJ
    ISECTION DY .0762 DZ .0762 TZ .0051 TY .0084 ! 308 :  76 x  76 x 13    RSJ

    HSECTION DZ .2540 DY .2032 TY .0102 TZ .0199 ! 321 : 254 x 203 x 82    RSJ
    HSECTION DZ .2032 DY .1524 TY .0089 TZ .0165 ! 322 : 203 x 152 x 52    RSJ
    HSECTION DZ .1524 DY .1270 TY .0104 TZ .0132 ! 323 : 152 x 127 x 37    RSJ
    HSECTION DZ .1270 DY .1143 TY .0102 TZ .0115 ! 324 : 127 x 114 x 29    RSJ
    HSECTION DZ .1270 DY .1143 TY .0074 TZ .0114 ! 325 :     x 114 x 27
    HSECTION DZ .1016 DY .1016 TY .0095 TZ .0103 ! 326 : 102 x 102 x 23    RSJ
    HSECTION DZ .0889 DY .0889 TY .0095 TZ .0099 ! 327 :  89 x  89 x 19    RSJ
    HSECTION DZ .0762 DY .0762 TY .0051 TZ .0084 ! 328 :  76 x  76 x 13

    AX 83.5E-4 IX 61.0E-8                        ! 351 : 432 x 102 R S CHANNEL
    AX 70.2E-4 IX 46.0E-8                        ! 352 : 381 x 102 R S CHANNEL
    AX 58.8E-4 IX 35.4E-8                        ! 353 : 305 x 102 R S CHANNEL
    AX 53.1E-4 IX 27.6E-8                        ! 354 : 305 x  89
    AX 45.5E-4 IX 22.9E-8                        ! 355 : 254 x  89 R S CHANNEL
    AX 36.0E-4 IX 12.3E-8                        ! 356 : 254 x  76
    AX 41.7E-4 IX 20.4E-8                        ! 357 : 229 x  89 R S CHANNEL
    AX 33.2E-4 IX 11.4E-8                        ! 358 : 229 x  76
    AX 37.9E-4 IX 17.8E-8                        ! 359 : 203 x  89 R S CHANNEL
    AX 30.3E-4 IX 10.4E-8                        ! 360 : 203 x  76
    AX 34.2E-4 IX 15.1E-8                        ! 361 : 178 x  89 R S CHANNEL
    AX 26.5E-4 IX 8.13E-8                        ! 362 : 178 x  76
    AX 30.4E-4 IX 12.4E-8                        ! 363 : 152 x  89 R S CHANNEL
    AX 22.8E-4 IX 5.94E-8                        ! 364 : 152 x  76
    AX 19.0E-4 IX 4.92E-8                        ! 365 : 127 x  64 R S CHANNEL
    AX 13.3E-4 IX 2.55E-8                        ! 366 : 102 x  51 R S CHANNEL
    AX 8.53E-4 IX 1.23E-8                        ! 367 :  76 x  38 R S CHANNEL

    AX  163E-4 IX 652E-8                         ! 401 : 250 x 250 x 35 EQ ANGLE
    AX  150E-4 IX 502E-8                         ! 402 :             32
    AX  133E-4 IX 340E-8                         ! 403 :             28
```

```
AX  119E-4 IX 244E-8                    ! 404 :                25
AX 90.6E-4 IX 170E-8                    ! 405 : 200 x 200 x 24 EQ ANGLE
AX 76.3E-4 IX 99.9E-8                   ! 406 :                20
AX 69.1E-4 IX 73.4E-8                   ! 407 :                18
AX 61.8E-4 IX 51.9E-8                   ! 408 :                16
AX 51.0E-4 IX 53.9E-8                   ! 409 : 150 x 150 x 18 EQ ANGLE
AX 43.0E-4 IX 31.6E-8                   ! 410 :                15
AX 34.8E-4 IX 16.4E-8                   ! 411 :                12
AX 29.3E-4 IX 9.58E-8                   ! 412 :                10
AX 33.9E-4 IX 24.9E-8                   ! 413 : 120 x 120 x 15 EQ ANGLE
AX 27.5E-4 IX 13.0E-8                   ! 414 :                12
AX 23.2E-4 IX 7.58E-8                   ! 415 :                10
AX 18.7E-4 IX 3.92E-8                   ! 416 :                 8
AX 27.9E-4 IX 20.4E-8                   ! 417 : 100 x 100 x 15 EQ ANGLE
AX 22.7E-4 IX 10.6E-8                   ! 418 :                12
AX 15.5E-4 IX 3.24E-8                   ! 419 :                 8
AX 20.3E-4 IX 9.50E-8                   ! 420 :  90 x  90 x 12 EQ ANGLE
AX 17.1E-4 IX 5.58E-8                   ! 421 :                10
AX 13.9E-4 IX 2.90E-8                   ! 422 :                 8
AX 12.2E-4 IX 1.96E-8                   ! 423 :                 7
AX 10.6E-4 IX 1.24E-8                   ! 424 :                 6
AX 15.1E-4 IX 4.91E-8                   ! 425 :  80 x  80 x 10 EQ ANGLE
AX 12.3E-4 IX 2.56E-8                   ! 426 :                 8
AX 9.35E-4 IX 1.10E-8                   ! 427 :                 6
AX 13.1E-4 IX 4.25E-8                   ! 428 :  70 x  70 x 10 EQ ANGLE
AX 10.6E-4 IX 2.22E-8                   ! 429 :                 8
AX 8.13E-4 IX 0.954E-8                  ! 430 :                 6
AX 11.1E-4 IX 3.58E-8                   ! 431 :  60 x  60 x 10 EQ ANGLE
AX 9.03E-4 IX 1.87E-8                   ! 432 :                 8
AX 6.91E-4 IX 0.810E-8                  ! 433 :                 6
AX 5.82E-4 IX 0.474E-8                  ! 434 :                 5
AX 7.41E-4 IX 1.53E-8                   ! 435 :  50 x  50 x  8 EQ ANGLE
AX 5.69E-4 IX 0.666E-8                  ! 436 :                 6
AX 4.80E-4 IX 0.39E-8                   ! 437 :                 5
AX 3.89E-4 IX 0.203E-8                  ! 438 :                 4
AX 2.96E-4 IX 0.0866E-8                 ! 439 :                 3
AX 5.09E-4 IX 0.5940E-8                 ! 440 :  45 x  45 x  6 EQ ANGLE
AX 4.30E-4 IX 0.3490E-8                 ! 441 :                 5
AX 3.49E-4 IX 0.1810E-8                 ! 442 :                 4
AX 2.66E-4 IX 0.0776E-8                 ! 443 :                 3
AX 4.48E-4 IX 0.5220E-8                 ! 444 :  40 x  40 x  6 EQ ANGLE
AX 3.79E-4 IX 0.3070E-8                 ! 445 :                 5
AX 3.08E-4 IX 0.1600E-8                 ! 446 :                 4
AX 2.35E-4 IX 0.0686E-8                 ! 447 :                 3
AX 2.78E-4 IX 0.2240E-8                 ! 448 :  30 x  30 x  5 EQ ANGLE
AX 2.27E-4 IX 0.1170E-8                 ! 449 :                 4
AX 1.74E-4 IX 0.0506E-8                 ! 450 :                 3
AX 2.26E-4 IX 0.1820E-8                 ! 451 :  25 x  25 x  5 EQ ANGLE
AX 1.85E-4 IX 0.0959E-8                 ! 452 :                 4
AX 1.42E-4 IX 0.0416E-8                 ! 453 :                 3

AX 60.0E-4 IX 73.4E-8                   ! 461 : 200 x 150 x 18 UEQ ANGLE
AX 50.5E-4 IX 42.9E-8                   ! 462 :                15
AX 40.8E-4 IX 22.2E-8                   ! 463 :                12
AX 43.0E-4 IX 31.6E-8                   ! 464 : 200 x 100 x 15 UEQ ANGLE
AX 34.8E-4 IX 16.4E-8                   ! 465 :                12
AX 29.2E-4 IX 9.58E-8                   ! 466 :                10
AX 33.9E-4 IX 24.9E-8                   ! 467 : 150 x  90 x 15 UEQ ANGLE
AX 27.5E-4 IX 13.0E-8                   ! 468 :                12
AX 23.2E-4 IX 7.58E-8                   ! 469 :                10
AX 31.6E-4 IX 23.2E-8                   ! 470 : 150 x  75 x 15 UEQ ANGLE
AX 25.7E-4 IX 12.1E-8                   ! 471 :                12
AX 21.6E-4 IX 7.08E-8                   ! 472 :                10
AX 22.7E-4 IX 10.6E-8                   ! 473 : 125 x  75 x 12 UEQ ANGLE
```

```
AX 19.1E-4 IX 6.25E-8                   ! 474 :                10
AX 15.5E-4 IX 3.24E-8                   ! 475 :                 8
AX 19.7E-4 IX 9.21E-8                   ! 476 : 100 x  75 x 12 UEQ ANGLE
AX 16.6E-4 IX 5.41E-8                   ! 477 :                10
AX 13.5E-4 IX 2.81E-8                   ! 478 :                 8
AX 15.6E-4 IX 5.08E-8                   ! 479 : 100 x  65 x 10 UEQ ANGLE
AX 12.7E-4 IX 2.64E-8                   ! 480 :                 8
AX 11.2E-4 IX 1.79E-8                   ! 481 :                 7
AX 10.6E-4 IX 2.22E-8                   ! 482 :  80 x  60 x  8 UEQ ANGLE
AX 9.38E-4 IX 1.50E-8                   ! 483 :                 7
AX 8.11E-4 IX 0.954E-8                  ! 484 :                 6
AX 9.41E-4 IX 1.96E-8                   ! 485 :  75 x  50 x  8 UEQ ANGLE
AX 7.19E-4 IX 0.846E-8                  ! 486 :                 6
AX 8.60E-4 IX 1.79E-8                   ! 487 :  65 x  50 x  8 UEQ ANGLE
AX 6.58E-4 IX 0.774E-8                  ! 488 :                 6
AX 5.54E-4 IX 0.453E-8                  ! 489 :                 5
AX 5.08E-4 IX 0.5940E-8                 ! 490 :  60 x  30 x  6 UEQ ANGLE
AX 4.29E-4 IX 0.3490E-8                 ! 491 :                 5
AX 2.46E-4 IX 0.1280E-8                 ! 492 :  40 x  25 x  4 UEQ ANGLE
```

The standards file is not limited to section properties; the user may extend the standards file or replace it altogether provided that each and every line in the standards file contains exactly 80 characters (78 text plus carriage return and line feed). For example, firms who trade under several styles could include their styles in the standards file, then merely give the relevant line number following every TITLE command.

For access to the standards file, there is no need to specify the name all the engineer need do is specify the line number following the @ as described above.

5.10.2 Input of data from named external files

There are occasions when a 'family' of data files all have an identical section of data. To save repeating the section in every file, it is permissible to save the section in a named file and include the section in any data file which needs it. The first rule of computing is 'never store the same bit of information in more than one file' for - as sure as God made little apples - when the information is updated, some file/s will be missed. Suppose there is a possibility that either the name of the job or the firm, may change; then save the page heading information (excluding the date) in a file of the same name/number as the job e.g. if J1234567 contains:

```
TITLE ROBERT FITZROY - CAPTAIN OF THE BEAGLE,
TITLE SOMETIME GOVERNOR OF NEW ZEALAND, CREATOR
TITLE OF THE WEATHER FORECAST, FORGOTTEN HERO.
TITLE STRESS ANALYSIS OF MAST IN STORM FORCE 9.
MADEBY RF
REFNO J1234567
```

then include the file name prefixed by # (hash) at the start of a line of data for every data file for the job viz:

```
#J1234567
```

thus ensuring the page headings are consistent for every analysis and that if the file J1234567 is changed, all subsequent NL-STRESS analyses will show the amended page heading in the results. Please note that the # <filename> command pulls the nominated file in for the analysis only; thus on exit from NL-STRESS the data file will contain the

<filename> command, and not the contents of <filename>.

Of course use of the # <filename> command is not limited to just the page headings, the command can be useful in the middle of the data or at the end. Another rule of computing is 'wring as much useful information as is possible from the data supplied'. Section 5.11.8 describes post-processing, commands and logic, which come between the SOLVE and FINISH commands. A file called 'wring.ndf' is supplied with NL-STRESS which will cause maximum deflections, moments etc. to be computed and displayed in the results. To use 'wring.ndf' (or other post-processing procedures) include the line: #wring.ndf on its own separate line - with the # as the first character - after the SOLVE command and before the FINISH command.

5.10.3 Piping data to external files

On encountering a line starting with *>, NL-STRESS will cause that line to be sent to the file public.stk. This feature is particularly useful when NL-STRESS is being run in batch mode for it allows information from certain runs out of several hundred, to be reported. If public.stk does not exist then it will be created; if it exists then lines starting with *> will be appended to the end of PUBLIC.STK but with their control characters *> omitted. Although this facility is primarily for use in post-processing between the SOLVE & FINISH commands, it is permissible to use the facility before the SOLVE, but if this is done the values printed will be the values on the stack at the end of processing as piping is not done until all analysis and results are being written. When it is required that variables have to be replaced e.g.

```
    *> +a +b
```

leave a space after the *>. If you require values to be shown as the data file is being processed, use the | facility in section 5.4.11. Once public.stk has been created, if editing is needed, use NLE32.EXE to preserve the file structure, further data piped to a file having a different protocol will be ignored.

There are occasions when it is necessary to write data or results to a named file e.g. so that SCALE may process that data. To do this, append the name of the file to '>', leave a space after the named file and follow with the data e.g.

```
>vmres.stk h= fsc=
```

where the = is interpreted by NL-STRESS as fill in the current value of the parameter before the equals e.g. if h=-23.6 & fsc=28, then

```
h=-.236000E+02 fsc=0.280000E+02
```

will be written to the file vmres.stk. When using this feature, the output is E format with 6 decimal digits. Text may also be written to a named external file e.g. NOT OK; if more functionality is required then SCALE may be invoked from NL-STRESS.

The form of the data that can be output/piped to the file public.stk is different to that which can be output to an named file. The two forms of the data are as described above. For the first form, everything on the line following *> is written to public.stk; for the second form, everything on the line following the name of the file is written to the named file.

5.10.4 Manipulating files

It is possible to copy or delete files by including the commands COPY and DEL after a '%' as the first character on a line e.g.
%COPY fil.nam fil.sav
will copy the file fil.nam to the file fil.sav. There must be no space following the %.

5.10.5 Running SCALE from NL-STRESS

To run SCALE from NL-STRESS include its name after a '&' as the first character on a line i.e.
&SCALE
will run SCALE. There must be no space following the &.

There are many examples of SCALE proformas invoking NL-STRESS e.g. sc678.pro. The following is an example of NL-STRESS invoking SCALE. In the NL-STRESS data file vm453.dat which is a checking aid for ridged portal frames, it is necessary to obtain the C1 factor by two way interpolation. Those familiar with SCALE will know that many of the SCALE proformas contain tables. It is convenient to devise a proforma containing the tables given in DEVERSEMENT ELASTIQUE D'UNE POUTRE A SECTION BI-SYMETRIQUE SOUMISE A DES MOMENTS D'EXTREMITE ET UNE CHARGE REPARTIE OU CONCENTREE, reference STA-CAL 1-02, CENTRE TECHNIQUE INDUSTRIEL DE LA CONSTRUCTION METALLIQUE.

Let us call the SCALE proforma which contains the tables extracted from the above publication: vm453a.pro.
As we are in an NL-STRESS data file, we need to pass data to proforma vm453a.pro, invoke SCALE to run the proforma and then pick up the results from vm453.cal. In the following seven lines, the first line contains two assignments. Lines 2-4 contain two commands, and values for mu & psi piped to the file pub.stk. Lines 5-7 contain three commands which are self explanatory. The eigth line pipes characters: 'c702.dat/bvm453a.pro' to the file pub.stk. The ninth line copies the characters to fil.nam, SCALE is invoked and the file pub.stk is cleared. The twelth line copies the calculations contained in c702.cal to c702a.cal. The reason for this is to produce a new file for import into this data file. The copy command then reinstates the original file name, finally the last line displays the result.

```
mu=-2 psi=-.2
%del pub.stk
%del vm453a.stk
>pub.stk mu= +mu psi= +psi
%copy pub.stk vm453a.stk
%copy fil.nam fil1.nam
%del pub.stk
>pub.stk c702.dat/bvm453a.pro
%copy pub.stk fil.nam
&SCALE
%del pub.stk
%copy c702.cal c702a.cal
#c702a.cal
%copy fil1.nam fil.nam
val= +val
```

The above may be used as a model for NL-STRESS to invoke SCALE to run a calculation the results of which can then be used by NL-STRESS.

5.10.6 Recycling the data

The character < at the start of a line, normally placed between SOLVE and FINISH, directs NL-STRESS to re-start the data; the characters << behave similarly but omit the initial display of the data and the opportunity of editing.

For automatic design, it is necessary to modify the data from a first analysis and rerun the analysis using the modified data. A less-than, as the first character on a line, commands NL-STRESS to recycle the data. Of course there is little point in running the same data for a second or subsequent analysis, but the previous features in this section in association with parametric data and post processing permit for example for a space frame:

- analysis assuming unity for section properties for all members
- computation of section properties between the SOLVE and FINISH to carry the member forces from the previous analysis
- piping of computed section properties to a named file
- << to rerun the data dependent on the variable NLOOP
- # to import the new section properties for a subsequent analysis.

NLOOP takes the value 1 the first time a set of data is run, 2 the second and so on. A simplified example of data starting with the Y ordinate =3.2 for NLOOP=1 and incrementing it to 3.7 for NLOOP=2 and and incrementing it to 4.2 for NLOOP=3 follows. An explanation is given to the right of the data.

```
y=3.2
JOINT COORDINATES
IF NLOOP=1
%DEL cc924.stk
ENDIF
#cc924.stk
1 1.8 y
...
...
SOLVE
y=y+0.5
>cc924.stk y=
IF NLOOP<3
<<
ENDIF
FINISH
```

For the first analysis, y is set to 3.2 and as NLOOP=1 the file cc924.stk will be deleted, thus the #cc924.stk will not import any data and the coordinates for joint 1 will be: 1.8,3.2. Following the SOLVE, y is incremented to 3.7, the '>cc924.stk y=' pipes the value y=3.7 to the file cc924.stk. If NLOOP is less than 3, the << causes the data to be rerun such that #cc924.stk imports the value y=3.7 for NLOOP=2, thus overwriting the initial setting.

5.11 Advanced topics

This section contains information on the following advanced topics:

5.11.1 Interleaving two languages

NL-STRESS combines two languages:

- the NL-STRESS high level language which consists of MIT STRESS keywords such as: JOINT COORDINATES, MEMBER INCIDENCES, JOINT LOADS, MEMBER LOADS etc. greatly extended. ("STRESS: A User's Manual, A Problem-Orientated Computer Language for Structural Engineering", S.J. Fenves, R.D. Logcher, S.P. Mauch, K.F. Reinschmidt, The Department of Civil Engineering MIT, 1964.)
- a programming language similar to PRAXIS (PRAXIS: A Computer Program for Reproducing Proforma design calculations", ALCOCK D.G. & BROWN D.W., The Computer Journal, Vol 33, No.4, 1990) which provides 'looping' by the programming structure REPEAT-UNTIL-ENDREPEAT, and conditionals such as: IF... ENDIF; IF... THEN; IF... GOTO... etc.

As in any software, bugs are possible, more likely if two languages are interleaved. The reason that two languages are interleaved is to provide a set of data which permits:

- Structural analysis
- Pre-processing in accordance with codes of practice e.g. Eurocode 3 for the computation of member imperfection e0 thus:

```
IF stg=235
IF tf<=.04 THEN fy=235E3
IF tf>.04 THEN fy=215E3
ENDIF
IF stg=275
IF tf<=.04 THEN fy=275E3
IF tf>.04 THEN fy=255E3
ENDIF
IF stg=355
IF tf<=.04 THEN fy=355E3
IF tf>.04 THEN fy=335E3
ENDIF
IF stg=460
IF tf<=.04 THEN fy=460E3
IF tf>.04 THEN fy=430E3
ENDIF
* Shear yield +fys=fy/SQR(3) kN/m² , +L'=L/(nr+1) m
* Eurocode 3 Table 6.2: Selection of buckling curve for a X-section:
* curve names +cu0=$(a0) +cu1=$(a) +cu2=$(b) +cu3=$(c) +cu4=$(d)
IF h/b>1.2 AND tf<=0.04 AND fy<=420E3 THEN curve=2
IF h/b>1.2 AND tf<=0.04 AND fy>420E3 THEN curve=0
```

and so on.

- Post-processing in accordance with structural theory e.g.

```
* In Professor Horne's Interaction Formulae, coincident values for
* Ned, Mxed, Myed & Mzed & Nrd, Mxrd, Myrd & Mzrd are substituted:
* Twist factor +t=SQR(1-(Mxed/Mxrd)^2) +alph=h*tw/A
```

```
   * Normalised axial effect +n=Ned/(Nrd*t)
   IF n<=alph
   * n<=alph so +mpyp=t*Myrd*(1-n^2/(alph*(2-alph))) kNm
   *        and +mpzp=t*Mzrd kNm
   IF n>alph THEN * n>alph so +mpyp=t*Myrd*2*(1-n)/(2-alph) kNm
   IF n>alph THEN * and +mpzp=t*Mzrd*(1-2*alph+n)*(1-n)/(1-alph)^2 kNm
   * Horne's eqn. +mrh=DE2((Myed/mpyp)^2+Mzed/mpzp)
   *
   * EN 1993-1-1:2005 (E) Clause 6.2.1. gives conservative interaction
   * formula +unc=DE2(Ned/Nrd+Myed/Myrd+Mzed/Mzrd)
```

and so on.

Structural engineers will be able to follow the above examples of pre-processing and post-processing. Pre-processing instructions come before the keyword SOLVE; post-processing instructions come after the keyword SOLVE at which time the analysis will have been completed i.e. displacements, forces, stresses & reactions... will be available for extraction using the ARR() function e.g.

```
   * COLLAPSE ANALYSIS
   * Collapse may be due to failure at supports or centre of the span.
   * NL-STRESS has available all results for every loading increment;
   * when collapse occurs, the triad 'ucz ucy fcx' (see below) may be
   * factored down to comply with Eurocode 3 requirements, which take
   * precedence over forces, moments & stresses computed by NL-STRESS.
   * Collapse loading increment +lli=ARR(12,4,2) +nrow=nli*nm
   * Applied loads uaz= +uaz uay= +uay fax= +fax
   * UDL carried Z direction    +ucz=ABS(uaz*lli/nli) kN/m
   * UDL carried Y direction    +ucy=ABS(uay*lli/nli) kN/m
   * Axial load carried         +fcx=ABS(fax*lli/nli) kN
```

and so on.

The combined languages have been tested extensively, including the reassignment of variables e.g. +a=SQR(a)*27*(a-1). Nevertheless it is recommended that the reassignment of symbolic names - colloquially variables - is avoided. Currently there may be up to 32000 different names for variables so there should be no shortage.

There are several passes through the data:

- the first to read and store all the structural data up to the first LOADING command
- the second to read and store the data following the LOADING command up to the keyword STORE
- the third to read any post processing commands which come between the keywords SOLVE and FINISH
- the fourth to write the data before the keywords SOLVE to the results with any expressions and assignments evaluated
- the fifth to write the post processing commands with any expressions and assignments evaluated.

At the time that the summary of the input data is written to the results file, the first line 'a=3 b=2 nr=0' below, will be sent to the results file after assigning values to the variables: a, b and nr. Further comparisons follow:

```
   APPEARING IN THE DATA            WRITTEN TO THE RESULTS
   a=3 b=2 nr=0                     a=3 b=2 nr=0
   IF a>b THEN * +c=a +c=c^2        IF a>b THEN * c=a=3 c=c^2=9
   IF b<=a THEN * +c=b +c=c^3       IF b<=a THEN * c=b=2 c=c^3=8
   IF nr=0 THEN * +c=c/2 +c=c^4     IF nr=0 THEN * c=c/2=4 c=c^4=256

   APPEARING IN THE DATA            WRITTEN TO THE RESULTS
   +a=3 +b=2 +nr=0                  a=3 b=2 nr=0
   IF a>b THEN +c=a +c=c^2          IF a>b THEN c=a=3 c=c^2=9
   IF b<=a THEN +c=b +c=c^3         IF b<=a THEN c=b=2 c=c^3=8
```

```
   IF nr=0 THEN +c=c/2 +c=c^4       IF nr=0 THEN c=c/2=4 c=c^4=256

   APPEARING IN THE DATA            WRITTEN TO THE RESULTS
   * +a=3 +b=2 +nr=0                * a=3 b=2 nr=0
   IF a>b THEN * +c=a +c=c^2        IF a>b THEN * c=a=3 c=c^2=9
   IF b<=a THEN * +c=b +c=c^3       IF b<=a THEN * c=b=2 c=c^3=8
   IF nr=0 THEN * +c=c/2 +c=c^4     IF nr=0 THEN * c=c/2=4 c=c^4=256
```

The purpose of the asterisk is to tell NL-STRESS that the line may contain general text as well as assignments e.g.

```
  IF n<=alph THEN * n<=alph so +mpyp=t*Myrd*(1-n^2/(alph*(2-alph))) kNm
```

which contains text such as 'n<=alph so' and 'kNm' and the assignment

```
   +mpyp=t*Myrd*(1-n^2/(alph*(2-alph)))
```

The line which follows is typical of that written to the results

```
  * n<=alph so mpyp=t*Myrd*(1-n^2/(alph*(2-alph)))=4746.2991 kNm
```

In much the same way as PRAXIS (PRAXIS: A Computer Program for Reproducing Proforma design calculations", ALCOCK D.G. & BROWN D.W., The Computer Journal, Vol 33, No.4, 1990), a line commencing with an asterisk may contain a mixture of text and assignments. There now follows further examples.

```
   APPEARING IN THE DATA            WRITTEN TO THE RESULTS
   * +a=2 +b=14                     * a=2 b=14
   IF a<b AND b>10                  IF a<b AND b>10
   * +b=b/2                         * b=b/2=7
   ENDIF                            ENDIF
   * b= +b                          * b=7

   APPEARING IN THE DATA            WRITTEN TO THE RESULTS
   * +a=12 +b=15                    * a=12 b=15
   IF b>a                           IF b>a
   * b= +b                          * b=15
   * Calcs +b=b*2 +b=SQR(b)         * Calcs b=b*2=30 b=SQR(b)=5.4772
   ENDIF                            ENDIF

   APPEARING IN THE DATA            WRITTEN TO THE RESULTS
   * +a=12 +b=15                    * a=12 b=15
   IF b<a
   * b= +b                          The 4 lines to the left are
   * Calcs +b=b*2 +b=SQR(b)         omitted as b is not < a.
   ENDIF

   APPEARING IN THE DATA            WRITTEN TO THE RESULTS
   a1=VEC(2500)*10                  a1=VEC(2500)*10
   +a1=VEC(2500)*10                 a1...=(2500)*10
   * +a1=VEC(2500)*10               * a1...=(2500)*10
```

RULES TO AVOID BUGS WHEN THE INPUT DATA IS BEING WRITTEN TO THE RESULTS FILE

- Logic, such as that directly above, is carried out e.g. the 'IF b<a' is evaluated; if false then the IF-ENDIF programming structure is omitted in its entirety.
  The programming structure IF Boolean THEN ... is treated similarly to the IF-ENDIF programming structure. If the IF Boolean THEN... is true, assignments following the THEN are evaluated. If an IF Boolean THEN... is false, assignments following the THEN are not evaluated; the engineer may include such lines by setting 'sense=6' which causes such lines to be included followed by an 'x' added to the end of the last assignment, to tell the engineer that the Boolean is false.
- Lines commencing with an asterisk have their assignments and expressions which commence with a plus sign, evaluated and written to the results file regardless of whether the line is internal or

external to a programming structure.

- Structural loops e.g. IF-UNTIL-ENDREPEAT or alternatively: 'label' IF 'condition' GOTO 'label' are copied directly to the results file with substitutions for assignments and expressions which are inside the structural loop and commence with a plus sign, evaluated.
- When a structural analysis has completed, a stack of variables and their values, used in processing the NL-STRESS data from the page heading commands up to the keyword SOLVE, is available for checking. Set 'sense=5' near the start of the data to cause a copy of the stack to be sent to the results file.

- Assignment to a VEC function is a special case. If the line containing the VEC function does not start with an asterisk then the VEC function is written to the results with a minor cosmetic change, but the VEC function is not evaluated. If the line containing the VEC function starts with an asterisk, then the VEC function is evaluated and written to the results file with a minor cosmetic change.

- NL-STRESS ensures that after the input data has been read, the results of all assignments in the input data are contained in a stack of values available for use in any post-processing. After the keyword SOLVE has been invoked, and the analysis carried out, NL-STRESS re-reads the input data and writes it to the start of the results - assuming the keyword DATA is present in the PRINT command. It is straight forward to write an exact copy of the input data; but more meaningful to the engineer and the checking engineer if the input data is written with all expressions and assignments evaluated. It is important that the evaluation of assignments does not corrupt the stack of values, for corruption may cause bugs in in the post-processing. As an example, if the input data contains y(1)=VEC(0)*20 to initialise y ordinates y(1) to y(20) with zero, and selected y ordinates are subsequently re-assigned values say: y(1)=2.7 y(3)=3.7 y(5)=4.7 and so on, then if the VEC function clears y(1) to y(20) when the y(1)=VEC(0)*20 is written to the results file, but the y(1)=2.7 y(3)=3.7 y(5)=4.7 and so on, are written to the results file but not assigned; then the y ordinates will be cleared but the odd numbered coordinates will remain as zero. To avoid confusion, three simple rules follow:

- Lines of the input data which start with an asterisk have all their assignments carried out and thus update the main stack of variables and their values.
- Lines which start with an IF-THEN true followed by an asterisk have all their assignments carried out and thus update the main stack of variables and their values e.g. IF a>b THEN * +c=a +c=c^2
- Lines which start with an IF-THEN false followed by an asterisk are not ignored, an 'x' is added to the end of the text. The 'x' tells the engineer that the Boolean returns false. If an engineer wishes to suppress an IF-THEN... for which the Boolean returns false; recast the logic within an IF-ENDIF programming structure.

The easiest way of testing the foregoing is to create a short file of data, and to experiment with adding logic, assignments and expressions before and after the SOLVE command e.g.

```
PRINT DATA, RESULTS, FROM 1
TABULATE ALL
TYPE PLANE FRAME
NUMBER OF JOINTS 2
NUMBER OF SUPPORTS 1
NUMBER OF MEMBERS 1
NUMBER OF LOADINGS 1
JOINT COORDINATES
```

```
*  /│Y    (1)     │          Units are kN & m and combinations.
* 1/╪══════┴═══════▼2 ──X    Beam is laterally restrained.
* Pre-processing in here.
1 0 0 SUPPORT
2 3 0
MEMBER INCIDENCES
1 1 2
CONSTANTS E 210E6 ALL G 81E6 ALL
MEMBER PROPERTIES
1 ISECTION DY 0.3104 DZ 0.1669 TZ 0.0079 TY 0.0137 R 0.0089
LOADING CASE 1
JOINT LOADS
2 FORCE Y -50
SOLVE
* Post-processing in here.
FINISH
```

It is assumed that after the structural analysis has been completed, the main stack of variables and their values contain: y1=0 & y2=.15 Note that when an expression such as +y2 is replaced by its current value held in the stack, any leading zero is omitted and the replacement value is moved one space to the left to join the equals. Examples of input data and results which would be shown in the pre-processing.

```
INPUT DATA             RESULTS              NOTES
y1=VEC(0)*2            y1=VEC(0)*2          Both y1 & y2 are not reassigned
y2=0.15                y2=0.15              therefore their values are
* y1= +y1 y2= +y2      * y1=0 y2=.15        those held in the stack at the
                                            end of the structural analysis.

+y1=VEC(0)*2           y1...=(0)*2          y1 & y2 are reassigned to zero
y2=0.15                y2=0.15              but y2 is not reassigned thus
* y1= +y1 y2= +y2      * y1=0 y2=0          y1=0 & y2=0.

+y1=VEC(0)*2           y1...=(0)*2          y1 & y2 are reassigned to zero.
+y2=0.15               y2=0.15              y2 is reassigned to 0.15, thus
* y1= +y1 y2= +y2      * y1=0 y2=.15        y1=0 & y2=1.5.

* +y1=VEC(0)*2         * y1...=(0)*2        y1 & y2 are reassigned to zero.
+y2=0.15               y2=0.15              y2 is reassigned to 0.15, thus
* y1= +y1 y2= +y2      * y1=0 y2=.15        y1=0 & y2=1.5.

a=3 b=4                a=3 b=4
IF a<b                 IF a<b
* +y1=VEC(0)*2         * y1...=(0)*2        y1 & y2 are reassigned to zero.
+y2=0.15               y2=0.15              y2 is reassigned to 0.15, thus
* y1= +y1 y2= +y2      * y1=0 y2=.15        y1=0 & y2=1.5.
ENDIF                  ENDIF

a=3 b=4                a=3 b=4              All that appears in the results
IF a>b                                      is just the first line.  As
* +y1=VEC(0)*2                              the Boolean 'IF a>b' is false,
+y2=0.15                                    then all contained within the
* y1= +y1 y2= +y2                           IF-ENDIF is omitted.
ENDIF

a=3 b=4                a=3 b=4              Similar result to that above.
IF a>b                                      Note that if the line
* +y1=VEC(0)*2                              * y1= +y1 y2= +y2
* +y2=0.15                                  followed the ENDIF, an error
* y1= +y1 y2= +y2                           would occur as y1 & y2 had not
ENDIF                                       been assigned previously.
```

```
a=3 b=4                  a=3 b=4                  As the y2=0.15 following
* +y1=VEC(0)*2           * y1...=(0)*2            the THEN is neither
IF a<b THEN y2=0.15      IF a<b THEN y2=0.15      prefixed with a plus nor
* y1= +y1 y2= +y2        * y1=0 y2=0              prefaced with an asterisk,
                                                  y2 is not reassigned.

a=3 b=4                   a=3 b=4                 As the y2=0.15 following
* +y1=VEC(0)*2            * y1...=(0)*2           the THEN is prefaced with
IF a<b THEN * +y2=0.15 IF a<b THEN * y2=0.15 an asterisk, then it is
* y1= +y1 y2= +y2         * y1=0 y2=.15           reassigned to 0.15.

a=3 b=4                   a=3 b=4                 As the y2=0.15 following
* +y1=VEC(0)*2            * y1...=(0)*2           the THEN is prefixed with
IF a<b THEN +y2=0.15      IF a<b THEN y2=0.15     a plus, then it is
* y1= +y1 y2= +y2         * y1=0 y2=.15           reassigned to 0.15.
```

WRITING THE POST-PROCESSING DATA AT THE END OF THE RESULTS
Writing the post processing data at the end of the results is more straightforward as the NL-STRESS language is not contained between the SOLVE and FINISH commands. For typical examples of post processors see: vmecg.ndf vmecp.ndf vmecs.ndf vmmoj.ndf vmper.ndf ... Inspection will reveal that 95% of each post processor contains logic, the remaining 5% comprises lines which start with an asterisk. The lines in post processors which start with an asterisk are those which are written at the end of the results. An example of post-processing follows:

```
i=0                                               Occasionally there is a
:3                                                need to loop and list
i=i+1 y(i)=0              * y( 7 )=1.6            selected values. Looping
IF i<6000 GOTO 3          * y( 758 )=2.1          and listing selected
y7=1.6 y758=2.1           * y( 5964 )=-2.78       values causes confusion
y5964=-2.78                                       in the input data, but may
i=0                                               be carried out in the post
:8                                                processing which only
i=i+1                                             displays lines commencing
IF y(i)<>0                                        with an asterisk.
* y( +i )= +y(i)
ENDIF
IF i<6000 GOTO 8
```

5.11.2 Plastic and non-linear analysis

For the plastic analysis of plane frames, NL-STRESS gives the end rotation of members after plastic hinges form. This enables the plastic rotation to be simply computed by subtracting the member end rotation from the joint rotation. Trace of formation of plastic hinges is given by reference to the segment number rather than original member number. If a member is not segmented then the member number will be the same as the segment number. If members have 10 segments, then member 1 will go from segment 1 to segment 10, member 2 will go from segment 11 to segment 20, and so on.

The plastic section properties of an I section are given in section 7.2.6 of the NL-STRESS Reference Manual, interaction formulae for plane frames are given in section 7.6.2 of the NL-STRESS Reference Manual. Inspection of the interaction formulae shows that shear forces are not included in the formulae. For an I section carrying heavy shear forces, the shear force does affect the plastic moment. NL-STRESS permits the engineer to specify that shear forces be taken into account for I sections, by assigning 'sense=2' near the start of the data. If 'sense' is not assigned, or not assigned to 2, then shear forces will not be included in the interaction formula for I sections. Benchmark PL05.BMK is an example for a reversing plastic hinge; setting 'sense=0' or omitting to set 'sense' as in benchmark PL05.BMK will cause the I section interaction formula for plane frames to be used but excluding shear forces.

NL-STRESS permits the engineer to control whether Professor Horne's Q forces (see section 7.3.3 in the NL-STRESS Reference Manual) should be taken into account. Normally these will be taken into account. For members with an axial release at one end, Q forces are not appropriate and are ignored. For cantilevers, Q forces can give rise to instability. To suppress Q forces from being applied to just cantilevers, assign 'sense=-3' near the start of the data.

If Q forces are not required for any members, assign 'sense=3' near the start of the data. Q forces were devised by Professor Horne to improve equilibrium and compatibility for redundant structures. For simply supported beams and other simple structures, Q forces may be omitted if the EQUILIBRIUM CHECK gives good agreement between the applied forces & reactions.

For non linear analysis, NL-STRESS cycles for up to 500 cycles trying to achieve satisfaction of equilibrium and compatibility for every member/segment. NL-STRESS permits the engineer to control whether the cycling may continue for the next load increment, or not. Assign 'sense=4' near the start of the data to stop cycling continuing after 500 cycles in the previous load increment. If 'sense' is not assigned, or not assigned to 4, then cycling will be allowed to continue for subsequent loading increments until the analysis becomes unstable.

When two languages are interleaved e.g. NL-STRESS for structural analysis, and a subset of PRAXIS for post-processing (see Section 5.11.1) then it is helpful to see the main stack of variables & their values. Assign 'sense=5' near the start of the data to cause the main stack to be written to the results file after the analysis has completed.
Normally lines such as: IF Boolean THEN false
are omitted from the data at the start of the results. To include such lines, assign 'sense=6' near the start of the data to cause
IF Boolean THEN false
lines to be included in the data at the start of the results. When such lines are included, an 'x' is added to end of the line to tell the engineer that the Boolean returns 'false'.

(Section 5.1.10 gives another example of the use of the sense switch for changing the format of numerical output.)

5.11.3 False mechanisms

For all versions of NL-STRESS prior to 2.2: after each increment of loading was applied to the structure, NL-STRESS located each member/segment end which would be plastic under the next loading increment, and released the member/segment end by applying equal and opposite plastic moments about the release (the values of the plastic moments being computed from Professor Horne's interaction equations).

This treatment worked well when no more that one new plastic hinge appeared within a loading increment, but when more than one hinge appeared it was apparent that for a small number of structures, had the first hinge been inserted in isolation then the second one would not have formed. This phenomenon is usually referred to as the 'false mechanism' problem.

Considering a single bay ridged portal frame having hinged or pinned feet and subject to a plan UDL, for a symmetrical structure under symmetrical loading, hinges at the eaves will form at the same time. The four hinges will constitute a mechanism and failure will be reported. In a real structure, because of differences in rolling margins, a hinge will form firstly at just one of the eaves and after it has formed, plastic rotation will take place there, the structure swaying towards that eave and thereby preventing a plastic hinge forming at the opposite eave. Further loading will eventually cause the next plastic hinge to form near the ridge giving the collapse mechanism.

NL-STRESS prevents false mechanisms forming by adding only one new hinge at a time in any loading increment. This 'adding one new hinge at a time' is implemented in plane frames, grids, and space frames.

5.11.4 Plastic hinges

There is yet another phenomenon that can occur in plastic analysis i.e. unloading plastic hinges. This is a rare phenomenon, but occasionally because of plastic hinges developing in one member of a structure, plastic hinges in another part start to reverse i.e. unload. NL-STRESS models the effect for plane frames (grids do not normally have the problem). The theory (due to Professor Horne) is given in the NL-STRESS Reference Manual, but in summary: when a plastic hinge of value Mp starts to reverse, having reached a hinge angle i, NL-STRESS replaces the pin by a rotational spring of stiffness b = 100.Mp/i, and introduces equal and opposite external moments of value 99.Mp.

Plastic moments of resistance are computed from interaction formulae devised by Professor MR Horne and others. Horne & Morris (Plastic Design of Low-Rise Frames, 1981, Collins, London), give the following guidance on the effect of strain hardening.

"After plastic strains of the order of 6-10 times the elastic strain at yield have taken place, structural steels do show an increase of stress, referred to as strain-hardening. The initial slope of the strain-hardening stress-strain relationship is of the order of 5% of that during the elastic range. Although this rate of increase of stress with strain is not large, it is sufficient to prevent 'infinite curvature' from developing at those cross-sections where the plastic moment capacity is theoretically attained.

The overall effect of the strain-hardening phenomenon in rigid frames is that, at the plastic collapse load predicted by simple theory, bending moments in regions of high moment gradient rise somewhat above the theoretical plastic hinge value, but remain somewhat below in regions of low moment gradient. The bending moments tend, because of strain-hardening, to lie up to about 10% above and below the nominal plastic values."

NL-STRESS permits the engineer to model the hinge stiffness remaining after a plastic hinge has formed by specifying a percentage of the plastic moment following the METHOD command e.g. METHOD PLASTIC 5 which would specify that 5% of the plastic moment be used as the hinge stiffness. If the percentage is omitted NL-STRESS assumes a percentage of 200/(number of loading increments) i.e. 2% for a loading applied in 100 increments.

Recapping, when carrying out plastic analysis, NL-STRESS applies the loading in a NUMBER OF INCREMENTS. Cycling takes place at each loading level, introducing no more than one plastic hinge at a time to ensure:

- that false mechanisms do not occur
- that any further plastic hinge/s, often caused by the formation of a plastic hinge at the same loading level, are introduced. Cycling continues at constant load level until all hinges which should form at the loading level, do form.

5.11.5 Limits

Current limits set in NL-STRESS are 254 loadings (load cases), 32000 joints including additional joints introduced by segmenting members, 32000 supports, 32000 members or segments, 500 for the number of increments in which each loading is applied for non-linear analysis, 100 for the number of segments in each member.

Internally, NL-STRESS has two stacks for variables: VSTAK() for general variables of all types; VAR() for variables va(1:8000), vb(1:8000), vc(1:8000), vd(1:8000). Read/write access to these special arrays is quicker than access to the general stack. These special arrays may be used as singly or doubly subscripted variables, as described in 4.5.

NL-STRESS data has a current maximum of 80 characters/line, but this is not a restriction as the CONSTANTS, COMBINE, MAXOF, MINOF & ABSOV commands may all be given on several lines, coupled with the ability to use variables in the data mean that long lines such as:

```
1 THRU 7 STEP 2 FORCE X GLOBAL CONCENTRATED P -(12.3+13.75)/2
                                       L 0.5*SQR(3.4^2+4.7^2)
```

may be written on two lines thus:

```
a=-(12.3+13.75)/2  b=0.5*SQR(3.4^2+4.7^2)
1 THRU 7 STEP 2 FORCE X GLOBAL CONCENTRATED P a L b
```

5.11.6 Keeping to the syntax

The syntax of the NL-STRESS language as fully described in sections 5.5, 5.6 & 5.7 describes what constitutes a valid line of data. It is not permissible to depart from the syntax given, for example a line of joint coordinates or member incidences table may start with
1 THRU 5 STEP 2 (meaning 1 to 5 in steps of 2 i.e. members 1,3 & 5)
or 1 3 5 INCL (meaning 1 3 & 5 inclusive) but not e.g.

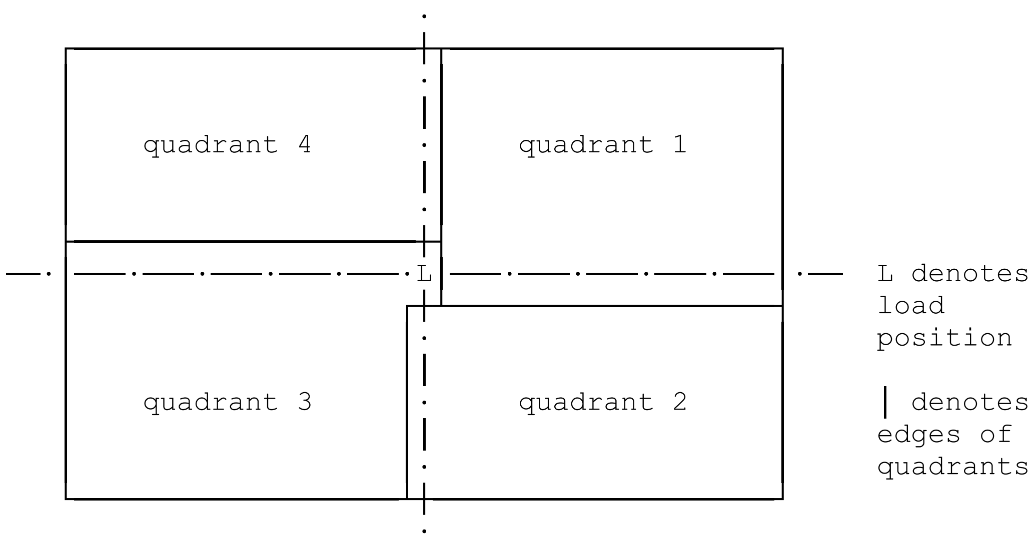
```
1 THRU 5 7 12 FORCE Y -3
```

which will be reported as an error.

5.11.7 Sharing area loads to the joints

Firstly NL-STRESS divides the rectangular area into [strips] in the X and Y directions. Each, of the strips^2 cells created, attracts 1/strips^2 times the total load on the rectangle and each component load is considered to be a concentrated load acting at the centre of its cell. If the optional parameter [strips] is omitted from the command (see JOINT LOADS) then [strips] is assumed as 64, i.e. the area load is modelled as 64^2 = 4096 point loads, each acting at the centre of each cell created by the strips.

Secondly NL-STRESS finds the nearest four joints in the NE,SE,SW,NW quadrants - numbered 1 to 4 in order - which surround each load point at (XL,YL). The software then locates any members joining the joints in quadrants 1-2, 2-3, 3-4, 4-1.



If a joint is found beneath the load position, all the load is applied to that joint. If a member is found beneath the load position, the load is shared to the joints at the ends of the member in the usual way.

If two 'approximately horizontal' members are found joining quadrants 2-3 and 4-1, then the load is firstly shared to these members in proportion to the perpendicular distances from these members, and thence to the joints at the end of these 'approximately horizontal' members.

If two 'approximately vertical' members are found joining quadrants 1-2 and 3-4, then the load is firstly shared to these members in proportion to the perpendicular distances from these members, and thence to the joints at the end of these 'approximately vertical' members.

If neither two 'approximately horizontal' members nor two 'approximately vertical' members are found, but joints are found in three or four segments, then the load is shared to the nearest three joints; loads lying in the same vertical or horizontal line with two joints, are located and treated as a special case.

The load carried by each of the three joints is computed from three simultaneous equations satisfying: vertical equilibrium
moments about the X axis
moments about the Y axis.

If the nearest joints are found in just two quadrants then the load is shared to the two joints in inverse proportion to the distance between the load and the joint. If the nearest joint is found in less than two quadrants then the load is ignored.

5.11.8 Arrays and post-processing

To trace the analysis process give the keyword TRACE following the PRINT command which causes the various arrays to be included in the results at appropriate stages in the analysis. The arrays are summarised below. Between the commands SOLVE and FINISH, a data file may post-processed by use of the ARR() function.

This section give a formal description of the structure of the .arr (ARRays) file, for those engineers who wish to include their own post-processing commands and statements after the SOLVE command, firstly key symbolic names:

```
NJORG  No-of-joints-originally   given after NUMBER OF JOINTS command
NMORG  No-of-members-originally  given after NUMBER OF MEMBERS ......
NSPM   No-of-segments-per-member ..................... SEGMENTS .....
NLSORG No-of-loading-systems     ..................... LOADINGS .....
NINC   No-of-loading-increments  ..................... INCREMENTS ...
NLSB   No-of-loading-systems-basic, excluding combinations
NLSC   No-of-loading-systems-combinations =NL-NLSB
NM     No-of-members-in-analysis    =NMORG*NSPM
NJ     No-of-joints-in-analysis     =NJORG+NMORG*(NSPM-1)
NLS    No-of-loadings-in-analysis   =NLSORG*NINC
NDJ    No-of-displacements/joint    =3 for 2D, =6 for 3D structures
NDM    No-of-displacements/member   =2*NDJ
NPD    No-of-possible-displacements =NDJ*NJ
MML    Maximum-member-loads on any member
Array
No.
 1  For each of NM rows, columns:
    1=end node start, 2=end, 3=member release No., 4=member loads
    counter, 5=order for posting to stiffness matrix.  Additionally
    for plane frame: 6-7 for MZ springs at start and end nodes
        plane grid:  6-9 for MX,MY springs at start and end nodes
        space grid: 6-11 for MX,MY,MZ springs at start and end nodes.
 2  For each of NM rows, columns:
    1-9=rotation matrix, 10=new member length, 11=original member
    length, 12=axial correcting load, 13=member results required
    (1=Yes, 0=No).
 3  For each of NM rows, columns:
    1-13 for upper triangle member stiffnesses for plane truss/frames
```

```
     1-26 ....................................... space ...........
  4  For each of NPD rows & NLS columns: artificial actions at joints
  5  ................................. actions at joints
  6  ................................. joint displacements
  7  For each of NPD rows, 3 columns elastic, ((9 non-linear)) give:
     restraint list, cumulative restraint list, current combined
     joint load vector ((col4=predicted displacement for last but 2,
     5=previous 6=current incr. for non-linear case, 7=old current,
     8=previous array 4 to the current incr., 9=previous array 5 to
     the current increment for possible increment repeating case)).
  8  For each of NJ rows, 6 columns hold: joint and node Nos. 3 coords
     column 6 holds 1 if joint results reqd, else =0.  For 2d plastic,
     z coordinate used for: number of members attached, then for
     plane frame plastic holds 0 if not exactly 2 members & free moment
     spring; or 1.0D95 if 2 members & free moment spring; or
     plastic spring value - used to save adding hinges on either side
     of a midspan joint, especially for pseudo mechanism problem
     before unloaded joint is detected.
  9  For each on NJ or NM rows (whichever is the maximum) for non-
     linear, columns 1-15 variously hold:
     For members: 1=FXP 2=MXP 3=MYP 4=MZP 5=ALPHA1 6=ALPHA2 7=formula
                FXH MXH MYH MZH (start end) FXH MXH MYH MZH (end end).
     For joints: Lowest unity factor at joint, member No. with lowest
                unity factor, number of plastic hinges this increment.
 10  For each loading increment & each member, NLS*NM rows: columns
     1=FXC 2=MXC 3=MYC 4=MZC (start), 5=FXC 6=MXC 7=MYC 8=MZC (end).
 11  For each of NM rows, 22 columns hold: 1=AX 2=AY 3=AZ 4=IX 5=IY
     6=IZ 7=CX 8=CY 9=CZ 10=BETA 11=E 12=G 13=CTE 14=DENS 15=YIELD
     16=SYIELD 17=DIRECTION 18=section ref 19=DY 20=DZ 21=TY 22=TZ.
 12  4 lines of common +
     4 line page heading + load case titles & tabulate requirements.
     Array 12 holds four sets of COMMON in the first four records.
     These may be accessed from SCALE etc. e.g. +nm=ARR(12,1,2)
     will extract the number of members for the current problem and
     store it in the variable 'nm'.  The contents of the first four
                                     records of array 12 follow.
     RECORD 1:
       ┌── Column number 'n' in ARR(12,1,n)
       1   NJ   No. of joints.
       2   NM   No. of members.
       3   NSUP No. of supports.
       4   NLS  No. of loading systems.
       5   ITAB(1)   Default for tabulate forces   =1 if reqd, else=0.
       6       (2)   .................... stresses  ..................
       7       (3)   .................... displac.  ..................
       8       (4)   .................... reactions ..................
       9   IUBW   Upper band width ( NDJ * ( joint diff +1 ) ).
      10   ISOLVE =1 if solve reqd, else =0.
      11   NLSB   No. of basic loading systems.
      12   NLSC   No. of combined loading systems.
      13   MML    Max No. of loads on any member for current structure.
      14   LCASE  Current load case No.
      15   LINOL  Line No. in data file for start of loading data.
      16   ISTYP  Structure type 1 -> 5 for plane truss, plane frame
                  grid, space truss, space frame.
      17   NDJ    No. of displacements / joint (3 for 2D, 6 for 3D)
      18   NDM    ...................... member (2*NDJ).
      19   NCOOR  No. of coordinates (2 for 2D, 3 for 3D).
      20   IPRNT(1)   1 if data reqd,  else =0.
      21         2    1 if trace reqd, else =0.
      22         3    1 if collection, else =0.
      23   LCSTA   Start load case number.
      24   LCEND   End   load case number.
      25   NLSORG  Number of loading systems originally specified.
```

26 NERR No. of errors.
27/28 NPD No. of possible displacements (NJ*NDJ).
29/30 N No. of degrees of freedom (NPD - No. of restraints).

RECORD 2:
┌── Column number 'n' in ARR(12,2,n)
1 LINO Line No. in output file.
2 LUSED No. of lines to be used on a page (typically 58).
3 NPFLG New page flag =1 if new page reqd., else =0.
4 LPP Lines per page (typically 66).
5 IPNUM Current page number in results file.
6-65 IHEAD() Default screen & firm heading.

RECORD 3:
┌── Column number 'n' in ARR(12,3,n)
1 .dat file name without the .dat.
->26
27->31 File handling switches.

RECORD 4:
┌── Column number 'n' in ARR(12,4,n)
1 NINC Number of increments in which load is applied.
2 INCN Current increment number.
3 ITRACI Trace of increments if =1.
4 NSPM Number of segments per member.
5 ITRACS Trace of segments if <>0.
6 NMORG Number of members originally set.
7 NJORG .... joints ....
8 NJC Number of joints currently set.
9 METH Method 1=elastic 2=sway 3=plastic.

13 For each loading increment & each member, NLS*NM rows: columns 1-NDM member end forces, cols NDM+1 to 2*NDM member end stresses.
14 For each of NPD rows & NLS columns: support reactions.

If node renumbering required, temporary arrays 15-17 are built then after checking load data, re-initialised in RDLCD as follows.
15 For NLSB*NM rows, columns give member load information in 5*MML columns i.e. 5 columns for each member load set giving load type and start & end positions & magnitudes.
16 For each of NLSC rows of combined loadings: column 1 holds the combination type (COMBINE MAXOF MINOF ABSOF), and data for up to 10 lines of the command at 25 numbers/line.
17 For each of NLS*NM rows: columns 1-NDM hold member end reactions and scratch array for build-up of array in load case combinations.

18 The structure stiffness matrix, initialised in BLDSM, has N or 2*N rows and IUBW columns where:
N Number of degrees of freedom
IUBW Upper band width of structure stiffness matrix
If there is only one loading, or the engineer has set the /M switch, then the number of rows =N; else the number of rows=2*N for computational efficiency.

The following describes post-processing contained in the Barrel Vault NDF file.

NL-STRESS holds NODE displacements for NLS loadings in ARRay 6 with the last element being ARR(6,NPD,NLS) where NPD is the Number of Possible Displacements and NLS the Number of Loading Systems. For a space frame NPD=6 degrees of freedom times nj joints, thus to extract the central deflection in the Y direction we need to 'look up' row=(cn-1)*6+2 where 'cn' is the node number at the centre of the roof.

Following the METHOD command, optionally comes keywords JOINT or NODES. Omission of the keyword JOINTS as well as the keyword NODES makes the software allocate 'node' numbers to joints in the order in which joint numbers are presented in the data. For example, if the order of joint numbers in the JOINT COORDINATES table reads 2, 4, 3, 1,... then joint 2 becomes node 1, joint 4 becomes node 2, joint 3 becomes node 3, joint 1 becomes node 4,... The keyword JOINTS signifies that joint numbers are to be treated as node numbers. The keyword NODES tells NL-STRESS to derive a correspondence between joint numbers and node numbers such as to reduce the 'band width' to a suitably small value. The 'band width' may be found by looking at every member and finding the difference between the node numbers at its ends. The biggest difference establishes the 'band width'. The smaller the band width, and the more efficiently NL-STRESS analyses the frame.

If at the start of this analysis the keyword JOINT followed the METHOD command, deflections could be extracted directly from ARRay 6 as the joint numbers and node number would be the same, and in the same order. For the general case it will first be necessary to find the node number corresponding to the required joint number before ARRay 6 can be 'looked up'.

NL-STRESS holds joint and node Nos., coordinates and a print flag (1 if joint results are required, else =0) in ARRay 8, with the last element being ARR(8,NJ,6). Thus the node number 'n' corresponding to joint 'j' will be found from: n=ARR(8,j,2)

```
Central joint No. +cj=(nj+1)/2 midspan edge joint +ej=cj+nex/2
Node number at centre of roof +nc=ARR(8,cj,2) thus to extract
centre deflection from ARRay 8, 'look up' row +rc=(nc-1)*6+2
hence NL-STRESS central joint deflection +dc=ARR(6,rc,1)
Node number at midspan edge +ne=ARR(8,ej,2) thus to extract
edge deflection from ARRay 8, 'look up' row +re=(ne-1)*6+2
hence midspan edge vertical joint defln +de=ARR(6,re,1)
```

As mentioned above, Array 12 holds four sets of COMMON in the first four records. These may be accessed as described above, e.g. +nm=ARR(12,1,2)
will extract the number of members for the current problem and store it in the variable 'nm'.

If NL-STRESS finds that the file ~segs is present when it is run, it is taken as an instruction to write results back to the file ~segs. As an example, the file nlkcmg.dat when converted to a space frame (to include bending in the XZ plane) takes the following structure. All deflections, bending moments & shear forces relate to the local axes.

```
          ┌ 21 Deflections in 5 lines      ┐              ┐
          │ 21 Bending moments in 5 lines  │              │ Loadcase 1
          │ 21 Shear forces in 5 lines     │              ┘
          │ 21 Deflections in 5 lines      │ Bending      ┐
          │ 21 Bending moments in 5 lines  │ in XY        │ Loadcase 2
          │ 21 Shear forces in 5 lines     │ plane        ┘
          │ 21 Deflections in 5 lines      │              ┐
          │ 21 Bending moments in 5 lines  │              │ Loadcase 3
Member 1  │ 21 Shear forces in 5 lines     ┘              ┘
          │ 21 Deflections in 5 lines      ┐              ┐
          │ 21 Bending moments in 5 lines  │              │ Loadcase 1
          │ 21 Shear forces in 5 lines     │              ┘
          │ 21 Deflections in 5 lines      │ Bending      ┐
          │ 21 Bending moments in 5 lines  │ in XZ        │ Loadcase 2
          │ 21 Shear forces in 5 lines     │ plane        ┘
```

```
           │ 21 Deflections in 5 lines    │             ┐
           │ 21 Bending moments in 5 lines│             │ Loadcase 3
           └ 21 Shear forces in 5 lines   ┘             ┘
           ┌ 21 Deflections in 5 lines    ┐             ┐
           │ 21 Bending moments in 5 lines│             │ Loadcase 1
           │ 21 Shear forces in 5 lines   │             ┘
           │ 21 Deflections in 5 lines    │ Bending     ┐
           │ 21 Bending moments in 5 lines│ in XY       │ Loadcase 2
           │ 21 Shear forces in 5 lines   │ plane       ┘
           │ 21 Deflections in 5 lines    │             ┐
           │ 21 Bending moments in 5 lines│             │ Loadcase 3
 Member 2  │ 21 Shear forces in 5 lines   ┘             ┘
           │ 21 Deflections in 5 lines    ┐             ┐
           │ 21 Bending moments in 5 lines│             │ Loadcase 1
           │ 21 Shear forces in 5 lines   │             ┘
           │ 21 Deflections in 5 lines    │ Bending     ┐
           │ 21 Bending moments in 5 lines│ in XZ       │ Loadcase 2
           │ 21 Shear forces in 5 lines   │ plane       ┘
           │ 21 Deflections in 5 lines    │             ┐
           │ 21 Bending moments in 5 lines│             │ Loadcase 3
           └ 21 Shear forces in 5 lines   ┘             ┘
 And so on.

 NB  For plane frames - bending in the XZ plane is omitted,
         plane grids  - bending in the XY plane is omitted.
```

5.11.9 Output

When the keyword DATA follows the PRINT command, then the data is included in at the start of the results. When data has been provided parametrically, it is often useful to print out the current numerical values of parameters make the data meaningful. Over 200 examples are contained in the PARAMETRIC DATA FILES & NL-STRESS VERIFIED MODELS which may be accessed by clicking on the Menu button when NL-STRESS is invoked. As a simple example, in the two lines which follow, the +e0 is replaced by the value previously computed.

```
Data───┬* Maximum amplitude of the member imperfection e0= +e0 , which
       └* is used to set a bow of e0= +e0 in both the XY & XZ planes.
```

Assuming the value e0 equals 0.08 then the above two lines would be shown in the results as:

```
Results┬* Maximum amplitude of the member imperfection e0=.08, which
       └* is used to set a bow of e0=.08 in both the XY & XZ planes.
```

Cosmetic changes, to improve the appearance of the results, include: moving the numerical value forward to abut the equals sign and moving the comma forward in the first line of the two lines. Although the above example is simple, it will be obvious that the printing of the numerical value for e0 will be of considerable help to the checking engineer; other examples, taken from data & results follow.

```
Data───┬* Buckling force about yy  +Ncy=PI^2*E*Iy/L^2 kN
       └* Buckling force about zz  +Ncz=PI^2*E*Iz/L'^2 kN
Results┬* Buckling force about yy  Ncy=PI^2*E*Iy/L^2=58262.7702 kN
       └* Buckling force about zz  Ncz=PI^2*E*Iz/L'^2=3678.7681 kN

Data───┬* Local bow imperfect. elastic: +lbe0=VEC(350,300,250,200,150)
       └* Local bow imperfect. plastic: +lbp0=VEC(300,250,200,150,100)
Results┬* Local bow imperfect. elastic: lbe0...=(350,300,250,200,150)
       └* Local bow imperfect. plastic: lbp0...=(300,250,200,150,100)

Data───┬* Resist. to mmts about x-x  +Mxrd=ARR(9,mn,2) kNm
       │* Resist. to mmts about y-y  +Myrd=ARR(9,mn,3) kNm
       └* Resist. to mmts about z-z  +Mzrd=ARR(9,mn,4) kNm
```

```
Results┬* Resist. to mmts about x-x  Mxrd=ARR(9,mn,2)=105.0381 kNm
       │* Resist. to mmts about y-y  Myrd=ARR(9,mn,3)=4151.3299 kNm
       └* Resist. to mmts about z-z  Mzrd=ARR(9,mn,4)=783.2257 kNm

Data───┬* For simply supported beam carrying a udl: +C1=1.127 +C2=0.454
       │* +k1=C1*PI^2*E*Iz/(k*L')^2 kN +k2=C2*zg m
       │* +k3=SQR((k/kw)^2*Iw/Iz+k^2*L'^2*G*It/(PI^2*E*Iz)+(C2*zg)^2) m
       └* Critical moment for LTB +Mcr=k1*(k3-k2) kNm
Results┬* For simply supported beam carrying a udl: C1=1.127 C2=0.454
       │* k1=C1*PI^2*E*Iz/(k*L')^2=4145.9717 kN k2=C2*zg=.2091 m
       │* k3=SQR((k/kw)^2*Iw/Iz+k^2*L'^2*G*It/(PI^2*E*Iz)+(C2*zg)^2)
       │                                                      =.7889 m
       └* Critical moment for LTB Mcr=k1*(k3-k2)=2404.1036 kNm

Data───┬* SECTION CLASSIFICATION Cross-section class= +class
       │* Eurocode 3, Clause 6.2.1. is a conservative interaction
       └* formula +uns=ABS(Ned)/Nrd+ABS(Myed)/Myrd+ABS(Mzed)/Mzrd
Results┬* SECTION CLASSIFICATION Cross-section class=1
       │* Eurocode 3, Clause 6.2.1. is a conservative interaction
       └* formula uns=ABS(Ned)/Nrd+ABS(Myed)/Myrd+ABS(Mzed)/Mzrd=.5703

Data───┬JOINT COORDINATES
       │j=0 xinc=-L/nsg x=xinc
       │:10
       x=x+xinc j=j+1
       │z=e0-4*e0*(x-L/2)^2/L^2 j x -z -z
       └IF j<nsg+1 GOTO 10
Results┬JOINT COORDINATES
       │j=0 xinc=-L/nsg x=xinc
       │:10
       │x=x+xinc j=j+1
       │z=e0-4*e0*(x-L/2)^2/L^2 j x -z -z
       └IF j<nsg+1 GOTO 10
```

The last examples of data & result show JOINT COORDINATES as being identical. It would be possible to insert a line within the loop e.g.

```
Data───┬* +z=e0-4*e0*(x-L/2)^2/L^2 j x -z -z
Result └* z=e0-4*e0*(x-L/2)^2/L^2=-0.9 j x -z -z
```

to show the z-ordinate, but not very useful as the value provided would only be for the last pass through the loop. Note that the result =-0.9 for the z ordinate has its sign reversed by the -z i.e. the last z ordinate =0.9. If the intention was to provide the numerical values of coordinates, then inclusion of the keyword SUMMARY following the PRINT command would produce a summary of MEMBER DATA, JOINT DATA and LOAD DATA, similar to that which follows.

```
SUMMARY OF MEMBER/SEGMENT DATA
(FIXITY IN THE ORDER FX FY FZ MX MY MZ: 1 FOR SOME FIXITY, 0 WHEN FREE)

MEMB   END FIXITY    AX       AY       AZ       IX       IY       IZ
STRT END    CX       CY       CZ       E        G        CTE    DENSITY
                                                                  BETA
           YIELD    SYIELD   FXP      MXP      MYP      MZP     FORMULA

   1 111111111111 .494E-01 .257E-01 .172E-01 .175E-04 .720E-02 .454E-03
   1    2 .506E+00 .210E+00 .461E+00 .210E+09 .808E+08 .000E+00 .000E+00
                                                               .000E+00
        .235E+06 .136E+06 .115E+05 .105E+03 .415E+04 .783E+03 .300E+01

   2 111111111111 .494E-01 .257E-01 .172E-01 .175E-04 .720E-02 .454E-03
   2    3 .506E+00 .210E+00 .461E+00 .210E+09 .808E+08 .000E+00 .000E+00
                                                               .000E+00
        .235E+06 .136E+06 .115E+05 .105E+03 .415E+04 .783E+03 .300E+01
```

```
    3 111111111111 .494E-01 .257E-01 .172E-01 .175E-04 .720E-02 .454E-03
    3    4 .506E+00 .210E+00 .461E+00 .210E+09 .808E+08 .000E+00 .000E+00
                                                                .000E+00
          .235E+06 .136E+06 .115E+05 .105E+03 .415E+04 .783E+03 .300E+01

    4 111111111111 .494E-01 .257E-01 .172E-01 .175E-04 .720E-02 .454E-03
    4    5 .506E+00 .210E+00 .461E+00 .210E+09 .808E+08 .000E+00 .000E+00
                                                                .000E+00
          .235E+06 .136E+06 .115E+05 .105E+03 .415E+04 .783E+03 .300E+01

    5 111111111111 .494E-01 .257E-01 .172E-01 .175E-04 .720E-02 .454E-03
    5    6 .506E+00 .210E+00 .461E+00 .210E+09 .808E+08 .000E+00 .000E+00
                                                                .000E+00
          .235E+06 .136E+06 .115E+05 .105E+03 .415E+04 .783E+03 .300E+01

    6 111111111111 .494E-01 .257E-01 .172E-01 .175E-04 .720E-02 .454E-03
    6    7 .506E+00 .210E+00 .461E+00 .210E+09 .808E+08 .000E+00 .000E+00
                                                                .000E+00
          .235E+06 .136E+06 .115E+05 .105E+03 .415E+04 .783E+03 .300E+01

    7 111111111111 .494E-01 .257E-01 .172E-01 .175E-04 .720E-02 .454E-03
    7    8 .506E+00 .210E+00 .461E+00 .210E+09 .808E+08 .000E+00 .000E+00
                                                                .000E+00
          .235E+06 .136E+06 .115E+05 .105E+03 .415E+04 .783E+03 .300E+01

    8 111111111111 .494E-01 .257E-01 .172E-01 .175E-04 .720E-02 .454E-03
    8    9 .506E+00 .210E+00 .461E+00 .210E+09 .808E+08 .000E+00 .000E+00
                                                                .000E+00
          .235E+06 .136E+06 .115E+05 .105E+03 .415E+04 .783E+03 .300E+01

 SUMMARY OF JOINT DATA
 (RESTRAINT IS 0 WHEN FREE, -1 WHEN FIXED, OTHERWISE SPRING VALUE)

 JOINT  NODE   X COOR    Y COOR    Z COOR
               FX REST   FY REST   FZ REST   MX REST   MY REST   MZ REST

     1      1 -.400E+01 0.100E+00 0.100E+00
              -.100E+01 -.100E+01 -.100E+01 -.100E+01 0.000E+00 0.000E+00

     2      2 -.600E+01 0.165E+00 0.165E+00
              0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

     3      3 -.800E+01 0.240E+00 0.240E+00
              0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

     4      4 -.100E+02 0.325E+00 0.325E+00
              0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

     5      5 -.120E+02 0.420E+00 0.420E+00
              0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

     6      6 -.140E+02 0.525E+00 0.525E+00
              0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

     7      7 -.160E+02 0.640E+00 0.640E+00
              0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

     8      8 -.180E+02 0.765E+00 0.765E+00
              0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

     9      9 -.200E+02 0.900E+00 0.900E+00
              0.000E+00 -.100E+01 -.100E+01 -.100E+01 0.000E+00 0.000E+00
```

SUMMARY OF JOINT LOAD DATA
LOADING CASE 1: AXIAL LOAD & UDL's IN Y & Z DIRECTION

JOINT	FX	FY	FZ	MX	MY	MZ
1	0.3678E+04	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
9	-.3678E+04	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

For the reason given above - as a general principle - avoid including results within any loop. For similar reasons, avoid including results within IF-ENDIF, REPEAT-UNTIL-ENDREPEAT and single and nested loops formed by labels and conditional GOTO's. For further examples, see section 5.11.10 for AVOIDING ERRORS IN DATA.

N.B. The SUMMARY of joint load data includes loads applied directly to joints and fixed end moments from udl's applied to members connected to joints. Thus, in the previous table, if a member 8 which was connected to joint 9, contained a udl then the fixed end moment MZ (or MY) resulting from the udl would be included under heading MZ (or MY).

Over the years, many features have been added to the NL-STRESS language language to provide flexibility and control the output. The three commands TABULATE, PRINT, LISTING are described formally in section 5.7; examples of their use follow. These three commands are used in association with keywords such as: COLLECTION, SUMMARY, TRACE, DISPLACEMENTS, FORCES, STRESSES, REACTIONS, ALL, FROM, THRU, STEP, BOTH, INCLUSIVE.

TABULATE
Generally the results of an analysis are presented as tables, the TABULATE command tells NL-STRESS which tables have to be tabulated. Out of the three commands which control output, the TABULATE command is different as it is the only command which may be used locally as well as globally.

TABULATE FORCES REACTIONS placed near the start of the data causes tables of forces at the end of each member (or segment of a member) and reactions at supported joints to be tabulated for every LOADING or loadcase. As the keyword FORCES precedes the keyword REACTIONS in this example, the table of forces would be printed before the reactions for each and every LOADING. TABULATE ALL causes all four types of table to be printed in the order: DISPLACEMENTS FORCES STRESSES REACTIONS. TABULATE on its own means TABULATE nothing.

Suppose we have three basic loadings:
LOADING DEAD LOAD
LOADING LIVE LOAD (or IMPOSED)
LOADING WIND LOAD

and two combined loadings:
LOADING 1.4 x DEAD + 1.6 x LIVE
LOADING 1.2 x (DEAD + LIVE + WIND)

If we give the command TABULATE DISPLACEMENTS FORCES REACTIONS near the start of the data then this would cause tables of displacements, forces and reactions to be printed for all five loadings. Let us suppose that our interest in the first loading is only REACTIONS (for unfactored foundation loads) and in the second loading our interest is only DISPLACEMENTS (for serviceability), and that we have no interest in the wind acting in isolation - only in combination as in the fifth loading; then we would need to use the TABULATE command locally (on the line following the LOADING command) thus:

LOADING DEAD LOAD

```
TABULATE REACTIONS
...
LOADING LIVE LOAD
TABULATE DISPLACEMENTS
...
LOADING WIND
TABULATE
```

There would be no need to give the TABULATE locally for the last two loading cases as the global setting will suffice. A word of warning, data which follows the LOADING command has no numerical significance, the data is merely descriptive of the LOADING. NL-STRESS number the LOADINGs in the order they appear in the data. Thus if the first loading title is say: LOADING CASE 23 DEAD PLUS LIVE
this does not make the first loading case into the twenty third.

PRINT
The PRINT command may be followed by keywords: DATA, RESULTS, SUMMARY, COLLECTION, TRACE. It is recommended that the keyword DATA always be given to include a copy of the data at the start of the results, thus avoiding problems caused by interpreting the results with the wrong set of data. As the section properties are frequently changed and the structure re-analysed, it is easy - after a few weeks have elapsed - to be unsure what section properties were used if the data was not printed at the start of the results.

The keyword RESULTS is included to make the command look sensible, even if it is omitted a set of results will be produced by NL-STRESS.

NL-STRESS has facilities for parametric data generation in which the data is given in terms of parameters, the parameters being set at the front of the data. As an example suppose a multi bay portal frame may have from 2 to 10 bays then at the start of the data - following NUMBER OF commands - a line may contain:
nb=????

If you wish to analyse a four bay portal frame all you have to do is replace the ???? with the digit 4, thus: nb=4. The remainder of the data uses the parameter 'nb' e.g.

```
i=0
REPEAT
i=i+1
...
UNTIL i=nb
ENDREPEAT
```

Where the ellipsis represents parametric data which will set coordinates etc. looping the variable i from 1 to nb. When data has been given parametrically, it is reassuring to know what NL-STRESS has computed; the keyword SUMMARY will print out all the joint data, member data and loading data. Suppose that within the REPEAT-UNTIL-ENDREPEAT looping a line of data appears to set say the X co-ordinate thus:
xcoor=a+i*span

where 'a' has been previously set to 2.5m and 'span' is the span of each bay, which has been previously set to 30m. For the first bay, NL-STRESS will work out that xcoor=2.5+1*30=32.5 and use the value 32.5 in place of xcoor for the first bay; 62.5 in place of xcoor for the second bay and so on. Symbolic names (variables) such as a,i,xcoor must start with a lower case letter so as not to confuse with NL-STRESS keywords such as REPEAT, UNTIL...

As mentioned above, the keyword DATA - following the PRINT command - ensures that the data is included at the start of the results. When this command is given, the line xcoor=a+i*span will appear as written. As also mentioned above the keyword SUMMARY will summarise the data; from the summary the engineer may check that the X coordinate is the one intended for use. There are occasions when the engineer will want to show the value used, directly in the data file. There are two ways of doing this: (1) to pre-process the NL-STRESS data using SCALE option 677 which removes the REPEAT-UNTIL etc. and replaces symbolic names such as xcoor with their numerical value; (2) to use the plus sign as a prefix as now described.

When the data file is included at the start of the results, then assignments such as xcoor=a+i*span or expressions such as xcoor+3.2 are printed as they appear in the data; prefixing either an assignment or an expression with a plus sign tells NL-STRESS to print out the value of the assignment or expression at the time the data file is included in the results i.e. after the analysis has been completed and the results are being written.

Examples (assuming a=1.5, b=3) are given below. Although it is permissible to mix assignments such as xcoor=2*(a+b) and expressions such as (a+b)/2 on the same line, it is not recommended. An assignment such as xcoor=2*(a+b) causes the variable xcoor to be assigned the value 9. An expression such as (a+b)/2 causes the value 2.25 to be treated as a numerical item of data e.g. a coordinate - dependent on the table of data in which it occurs and its position in the line of data.

In the first example below, the value of the expression is shown in the results, left adjusted at the plus sign; the single space following the expression causes NL-STRESS to left shift the remainder of the line to preserve the single space. In the second example there is more than one space between the expressions, and NL-STRESS substitutes for each and left adjusts each to the plus signs.

In the third example below, the assignment takes place when the data is first read, thus the variable xcoor will hold the value of 9 after the data is read; when the results are written this value will be left adjusted at the plus sign and the remainder of the assignment replace by blanks.

Generally assignments and expressions are within tables such as JOINT COORDINATES, MEMBER INCIDENCES, MEMBER LOADS etc. For data files - especially those prepared parametrically - it is reassuring to give help to the checker so they know what is going on. In NL-STRESS, comment lines start with an asterisk - and although comment lines are ignored in the analysis, when the data is copied to the results, any assignments or expressions are replaced by their numerical value. The fourth example gives an example of this, note that all text which does not start with a plus sign is copied as it is; it is only text which starts with a plus sign which is assumed to be an assignment or expression.

```
       Given in the data        Shown in the results
1)     +2*(a+b) +(a+b)/2        9 2.25
2)     +2*(a+b)  +(a+b)/2       9         2.25
3)     +xcoor=2*(a+b)           9
4)     * xcoor=2*(a+b)= +xcoor  * xcoor=2*(a+b)=9
```

Normally the results for each loading follow the one before, thus the joint displacements, member forces... for loading 'one' are followed by those for loading two and so on. For large structures having over a thousand members and say fifty loadings, the results will extend to over a thousand pages. In such a case when designing say member 350, it is tedious going through all thousand pages looking for the forces on member 350 for the fifty loadings. The keyword COLLECTION gets round this problem, it instructs NL-STRESS to collect all the member forces for the fifty loadings together, thus only one or two pages will need to be scrutinised.

The keyword TRACE following the PRINT command, causes all the arrays of joint coordinates, stiffness matrix etc. to be printed in the results. Thus the keyword TRACE behaves like a super-summary, and thereby allows the engineer to trace the build-up of the various arrays and follow through the analysis to the computation of the arrays of joint deflections, member forces etc.

The keyword TRACE following the NUMBER OF SEGMENTS command, causes the results for all the segments and additional joints to be tabulated in the results. The keyword TRACE following the NUMBER OF INCREMENTS command, causes the results for all the loading increments to be tabulated in the results. Thus it is possible to trace the change in structural behaviour from elastic to plastic and then through to collapse.

LISTING

Sometimes the engineer is only interested in a small part of a large structure; the LISTING command allows the engineer to select joint and member results for this situation. Please note that the picture of the frame displayed before analysis shows the entire structure whereas after analysis the tabulated results and plots are restricted by the numbers of joints and members specified in any LISTING command. If the joints requested in the LISTING command bear no relation to the members, then a strange looking plot will be produced.

The LISTING command following the NUMBER OF JOINTS command specifies those joint numbers for which results are to be included. The joint numbers may be sequential, or not, but in either case should be given in ascending order. Similarly the LISTING command following the NUMBER OF MEMBERS command specifies those member numbers for which results are to be included. The member numbers may be sequential, or not, but in either case should be given in ascending order.

Let us suppose we are only interested in the results for members: 1,3,5,7,9 of a structure which has 230 members, then the LISTING command would be:

```
NUMBER OF MEMBERS 230 LISTING 1 THRU 9 STEP 2
```

If we were also interested in members 105 and 106 then the LISTING command would be:

```
NUMBER OF MEMBERS 230 LISTING 1 3 5 7 9 105 106 INCLUSIVE
```

If we are interested in over a dozen members then there will be insufficient room on the line of NL-STRESS data - which has a limit of 80 characters - to list all the member numbers. In this situation NL-STRESS will read the numbers from an external file. Suppose we have a file called 'numbers' containing:

```
1 3 5 7 9 105 106 107 108 109 110 111 112 113 114 115 200 210
202 203 204 205 206 207 208 209 210 211 212 213 214
```

then the listing command would be:

```
NUMBER OF MEMBERS 230 LISTING #numbers
```

The # (hash) sign tells NL-STRESS to open the file whose name follows the #, and read the member (or joint) numbers which follow. The numbers should be in ascending order and the file may not contain keywords, only valid member (or joint) numbers. There should be no gap between the # and the file name.

5.11.10 Avoiding errors in data

NL-STRESS reads a file of data written in the NL-STRESS language. The language includes 'reserved words' written with upper case letters and parameters or variables starting with lower case letters (to distinguish the parameters from the 'reserved words'). Only reserved words are recognised by NL-STRESS, other words are not e.g. TABULATE

```
   FORCES DISPLACEMENTS
```

is recognised as a command to tabulate member forces and joint displacements; the command

```
   TABULATE FORCES DEFLECTIONS
```

will cause an error as DEFLECTIONS is not recognised. Of course it would not be difficult to make NL-STRESS recognise DEFLECTIONS. The keyword DISTORTIONS is used as it includes both deflections and rotations. Similar errors will be avoided if the engineer prepares data in accordance with the syntax defined and described in this manual.

Working in kN & m units, the stresses are tabulated in kN/m², so to convert to the more familiar N/mm², we need to divide by a thousand. A stress shown as 120855.559 is therefore interpreted as 121 N/mm². It is all too easy to conclude that the design is OK; engineers know that stresses have to be combined and that at the intersections of members, higher local stresses are produced.

All popular structural analysis software known to the author, assume that the Y axis is upwards, with X going to the right (just as we were taught in school). Thus engineers know that gravity loads are negative because the force of gravity is downwards. With parametric data we would assign a gravity udl for example:

```
   ut=-5.6              !   UDL on each top chord.
```

When we use the parameter later, we remember gravity loads are downwards so we insert a minus sign in front of 'ut' e.g.

```
      nh+1 THRU 3*nh FORCE Y UNIFORM W -ut
```

This is an error for we now have made gravity forces upwards. Some will say "no problem if all loads are up", but it is a problem, for member self weights are applied downwards therefore reducing the applied loading. The moral, avoid putting a minus sign in front of a parameter; if the value has to be negative, assign the parameter with the negative value and use the parameter unsigned.

With the exception of the BETA angle (which was traditionally measured in degrees) all angles in NL-STRESS are measured in radians, as generally used in engineering mathematics. Do not convert degrees to radians, if the angle has been input as say 'a' degrees; either work throughout using trigonometrical functions such as SIN(RAD(a)), or more simply assign ar=RAD(a), and use as SIN(ar). In this example we have added the 'r' to remind ourselves and the checker that the angle has been converted to radians. Mathematically SIN(30) is acceptable, sine and cosine are continuous functions of period 2*PI. Thirty radians divided by 2*PI =4.77465 thus 0.22535 cycles before the start of the sixth cycle. 0.22535*2*PI =1.416 radians before the start of the sixth cycle or 81.136° before the start of sixth cycle and thus has value =- 0.9880. It is not sensible for NL-STRESS to fault angles by checking them against a predetermined range, for angles of 1° and 1 radian (57.3°) are equally likely to occur.

The first check that must be made is to ensure that the sum of the applied loads as printed in the EQUILIBRIUM CHECK, is as expected. The decision to advocate kN & m units as the basic set of units was made following early experiences trying N and mm; engineers happily accepted UDL's in N/mm as being identical to kN/m but then forgot that concentrated loads had to be input in N. Other errors concerning units include.

- Section sizes being input as say 230, a beam depth of 230m gives very small deflections. For such deep beams, shear deformation predominates, and the bending moment diagram looks simply supported when it should continuous.
- Elastic constants being given as say 205E3 instead of 205E6 thus causing deflections to be 1000 time greater than expected, some engineers do this deliberately to give the table of deflections in mm but do not warn the checker. Another common mistake with constants is to unthinkingly insert a minus e.g. 205E-6, this increases deflections a million-million times. NL-STRESS deliberately shows results without the E exponent as it is difficult to scan a set of results and pick out worst cases when the E exponent is used. If the deflection is too high to be printed in the width available, then NL-STRESS prints a row of asterisks.

Debugging during the development of a parametric data file (see 4.11) a | (ASCII 124) as the first character on a line followed by a time interval in seconds (0 to 60), followed by numbers, variables, or expressions will cause the values after the time interval to be displayed on the screen as a trace to keep a track of looping. Thus as a simple example, if the line

```
      | 5 i
```

were placed on a line within a loop in which the variable 'i' was being incremented by 1 starting from zero, then the values 1, 2, 3... would be displayed on the screen as NL-STRESS read the data. Each value is displayed near the top left corner of the screen. After each value is displayed, NL-STRESS pauses for the time interval (five seconds in this example as given by the first number following the | (ASCII 124) before displaying the next, so that the engineer has time to read/write the value/s. A time interval of zero is interpreted as 'pause until a box is clicked'.

When NL-STRESS has completed the analysis, if the keyword DATA follows the PRINT command in the input data, then the input data is displayed at the start of the results. Generally the data is straightforward and there are no problems, but as with any software it is possible to introduce bugs into the data; consider the following.

```
IF a<c GOTO 30
* +k1=C1*PI^2*E*Iz/(k*L)^2 kN +k2=C2*zg m
* +k3=SQR((k/kw)^2*Iw/Iz+k^2*L^2*G*It/(PI^2*E*Iz)+(C2*zg)^2) m
* Critical moment for LTB +Mcr=k1*(k3-k2) kNm
:30
```

In the above if a=3 c=5 then the computation of k1, k2, k3 & Mcr would neither be carried out nor would the five lines be displayed in the results. If a=7 c=5 then the computation of k1, k2, k3 & Mcr would be carried out and the 5 lines would be displayed in the results e.g.

```
IF a<c GOTO 30
  k1=C1*PI^2*E*Iz/(k*L)^2=13103.3178 kN k2=C2*zg=.2091 m
  k3=SQR((k/kw)^2*Iw/Iz+k^2*L^2*G*It/(PI^2*E*Iz)+(C2*zg)^2)=.6004 m
  Critical moment for LTB Mcr=k1*(k3-k2)=5127.8163 kNm
:30
```

The conditional GOTO is an unstructured programming device. It is useful for jumping forwards & backwards to provide nested loops as described in section 5.4.11; but when the looping repeats 10,000 times, it would be confusing if 10,000 passes were included in the results. As an example, the data and results for simple looping are shown below, the left two columns for a line commencing with an asterisk, the right two columns for a line omitting the asterisk. It can be seen that the rightmost results are the same as for the data except that the plus sign in the data is removed to tidy up the results. It can be seen that the leftmost results are the same as for the data except that the asterisk is deleted and the value for i^2 is that for the last looping i.e. when i=5.

```
DATA  and  their  RESULTS          DATA  and  their  RESULTS
i=0               i=0              i=0               i=0
:10               :10              :11               :11
i=i+1             i=i+1            i=i+1             i=i+1
* i^2= +i^2       i^2=25           +i^2              i^2
IF i<5 GOTO 10    IF i<5 GOTO 10   IF i<5 GOTO 11    IF i<5 GOTO 11
```

For the reason given in section 5.11.9 OUTPUT - as a general principle - avoid including results within any loop or within IF-ENDIF, REPEAT-UNTIL-ENDREPEAT and single and nested loops formed by labels and conditional GOTO's as immediately above.

To help the checking engineer make sense of the data, the following programming devices are useful:

```
IF <Boolean/s>
...
IF <Boolean/s> THEN ...
...
ENDIF
```

but avoid the inclusion of: IF <Boolean/s> GOTO <label>
within an IF-ENDIF, and avoid the inclusion of results within an IF <Boolean/s> GOTO <label>.

STAY TRUE TO THE MODEL
This is an important matter but often ignored. When a parameter varies, ensure that any property e.g. section property, which is affected by change to the parameter, properly takes the change into account.

SOME PITFALLS
During the course of an NL-STRESS analysis, several sweeps through the data are required e.g. to carry out looping; thus it is imperative that variables are initialised.

Lines which start with an asterisk may be used to combine algebra with an explanation. For simplicity, consider the computation for the area of a rectangle having sides h and b.
The calculation may be carried out by the following line:
* +h=3 +b=7 +h=h*b
which will be written in the results as:
* h=3 b=7 h=h*b=21
exactly as expected. The line "* +h=3 +b=7 +h=h*b" could be included at the start, also included just before the SOLVE and also after the SOLVE where it would become part of the post-processing. Calculations placed between the SOLVE & FINISH commands can access all the results of the analysis and are thus defined as 'post-processing'. At all three locations the line written in the results would be:
"* h=3 b=7 h=h*b=21" just as expected. But if e.g. the assignment of h was omitted from the second and third locations then the three locations would be shown in the results:
* h=3 b=7 h=h*b=21
* b=7 h=h*b=147
* b=7 h=h*b=1029

exactly as expected.

Problems can arise when assignments and substitutions for variables get mixed up on the same line e.g.

```
* +h=3 +b=7 +h=h*b
* h= +h b= +b +h=h*b
```

Because the last line above does not reassign values for h & b, it merely substitutes the current values of h & b AFTER any assignments have been carried out, the two lines would be shown in the results as:

```
* h=3 b=7 h=h*b=21
* h=147 b=7 h=h*b=147
```

For clarity for the checking engineer, it would be sensible to write

```
* +h=3 +b=7 +area=h*b
```

which would be shown in the results as:

```
* h=3 b=7 area=h*b=21
```

Occasionally, for very good reasons, an engineer applies a udl to say joints 1 to 9, when joint 1 is at the start of a member and joint 9 is at the end of the member, thus:

```
1 THRU 9 FORCE Z -wz*L/8
1 9 BOTH FORCE Z +wz*L/8/2
```

The total udl on the member =wz*L, therefore each of 9 joints will receive one eighth of the total load, thus the second line will be needed to halve the load applied to joints 1 & 9. Without thinking, the engineer includes a plus sign on the second line of data above, to remind him/herself that wz is applied upwards. As the data has been written parametrically, were the value of wz to be previously set to zero then the two lines of data would appear in the results as:

```
1 THRU 9 FORCE Z -wz*L/8
1 9 BOTH FORCE Z 0
```

The zero will be shown as prefixing an expression with a plus sign is an instruction to NL-STRESS to show the current numerical value of the expression rather than the expression.

UNITS

Although it is permissible to include assignment/s in a line of NL-STRESS data such as:

```
.Iw=Iz*(h-tf)^2/4
```

it is more helpful for the checking engineer if such an assignment has a description and units e.g.

```
* Warping constant: +Iw=Iz*(h-tf)^2/4 m6.
```

N.B. Variables, which commence with an upper case letter, must be prefixed with a full stop to distinguish the variable from the NL-STRESS language which is wholly upper case; thus the assignment

```
.Iw=Iz*(h-tf)^2/4
```

commences with a full stop. When such an assignment is included in a comment line, it must be prefixed by a plus sign to tell NL-STRESS that an assignment follows, regardless of whether the assignment commences with an upper or lower case letter. Thus for an assignment to be recognised, the variable Iw must either be prefaced by a full stop when included in a line of data, or by a plus sign when included in a comment line. The following are examples of comment lines which include assignments:

```
* Warping constant: +Iw=Iz*(h-tf)^2/4 m6
* +k1=C1*PI^2*E*Iz/(k*L')^2 kN +k2=C2*zg m
* +k3=SQR((k/kw)^2*Iw/Iz+k^2*L'^2*G*It/(PI^2*E*Iz)+(C2*zg)^2) m
* Critical moment for LTB +Mcr=k1*(k3-k2) kNm
```

6. NL-VIEW User's Manual, a 3D viewer for NL-STRESS

6.1 About NL-VIEW

NL-VIEW is a post-processing program for NL-STRESS that lets you view structures, loadings and results for NL-STRESS analyses in 3D.
NL-VIEW allows you to:

- rotate, pan and zoom, in real-time, a three dimensional model of any NL-STRESS model, showing section sizes, member principal axes, and geometry to facilitate checking the input.
- view the deflected shape of the structure in three dimensions with the actual section sizes displayed (at the time of writing only I-Sections, H-Sections and Rectangles are shown to scale) including any BETA rotation applied to the members, with the members coloured according to deflection.
- animate the deflected shape to help visualisation of the displacements, this is useful to check if any parts of the structure are not connected as expected!
- view the deflected shape of the structure with the neutral axes of the members shown with different colours and line-types for different selected loadcases.
- view bending moments and shear force diagrams in three dimensions, with the structure represented by lines or the actual section sizes.
- save a screenshot of the current view to the pdf results file.
- view the joint and member loads applied to the structure in three dimensions.
- utilise the power of the computer's graphics processors using DirectX 9.

The bending moment, shear force or deflection is plotted at 21 points equi-spaced along each member, this data is generated by the NL-STRESS analysis.

6.2 Getting started

iPad: when viewing the pdf results of an NL-STRESS analysis, or a SCALE proforma that includes an NL-STRESS analysis (as shown in Figure 6.1), tap on the "NL-VIEW" button (the 3D I-section in the top right) to switch to viewing the model in NL-VIEW.

Windows: when viewing the pdf results of an NL-STRESS analysis, or a SCALE proforma that includes an NL-STRESS analysis (as shown in Figure 6.2), click on the button labelled "NL-VIEW" to switch to viewing the model in NL-VIEW.

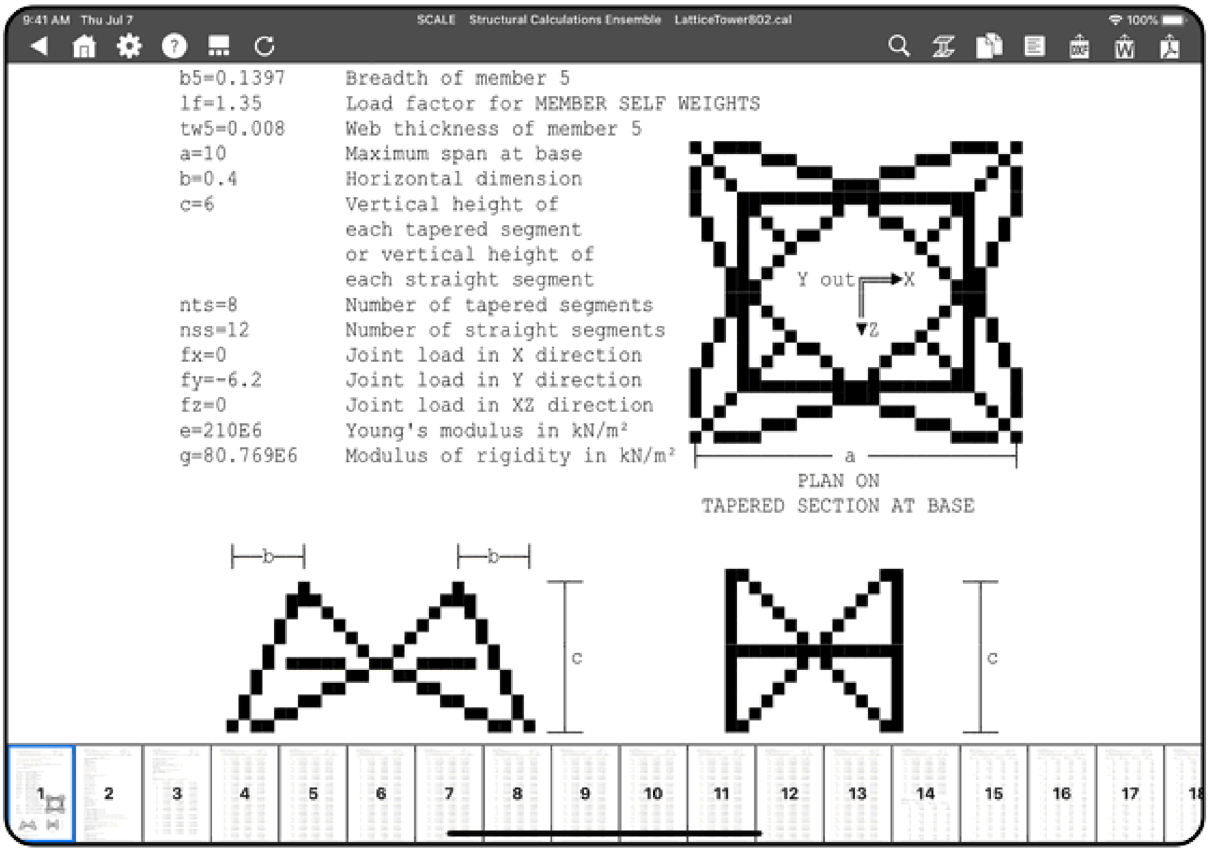


Figure 6.1: Displaying NL-STRESS results (iPad).

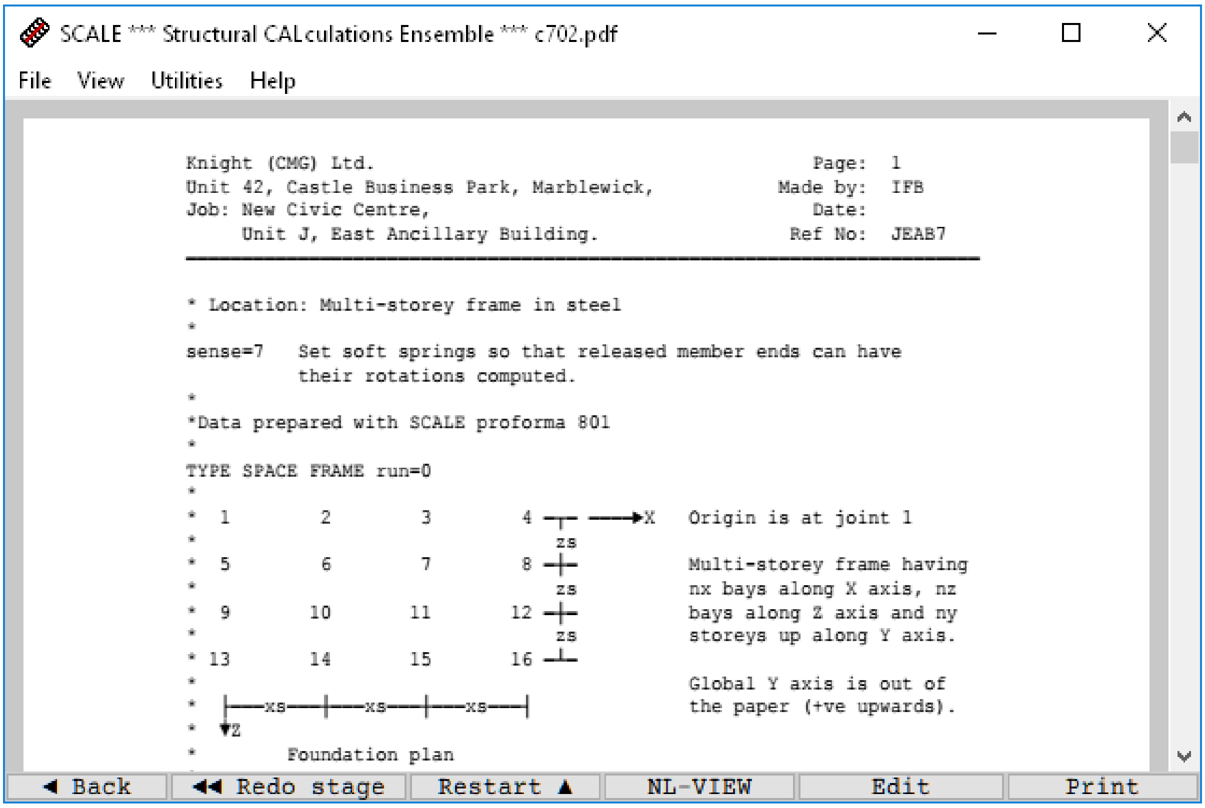


Figure 6.2: Displaying NL-STRESS results (Windows).

The NL-VIEW interface, as shown in Figure 6.3 (iPad) and Figure 6.4 (Windows), allows you to view many different aspects of the model and results. You can switch from one aspect to another by selecting items from the toolbar, or from the pop-up menu (Windows only)

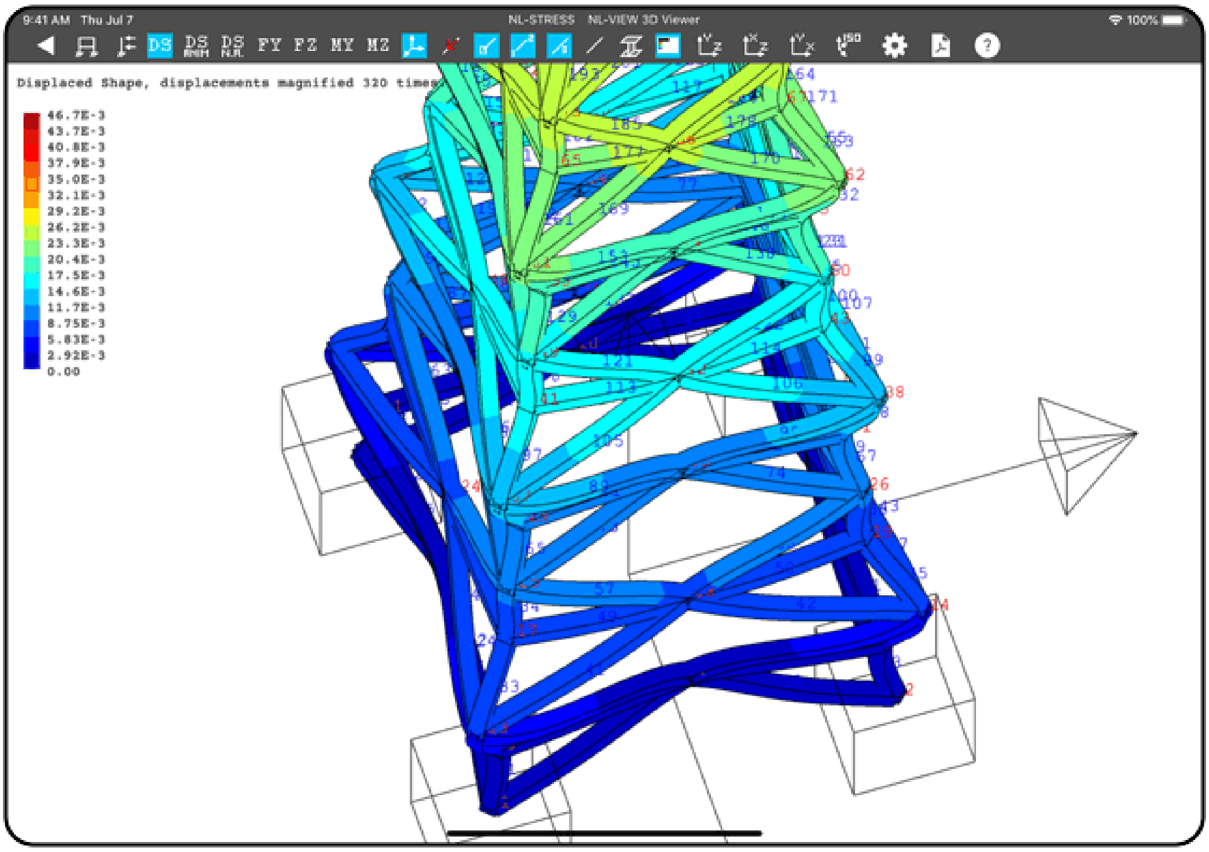


Figure 6.3: NL-VIEW 3D display of NL-STRESS analysis (iPad).

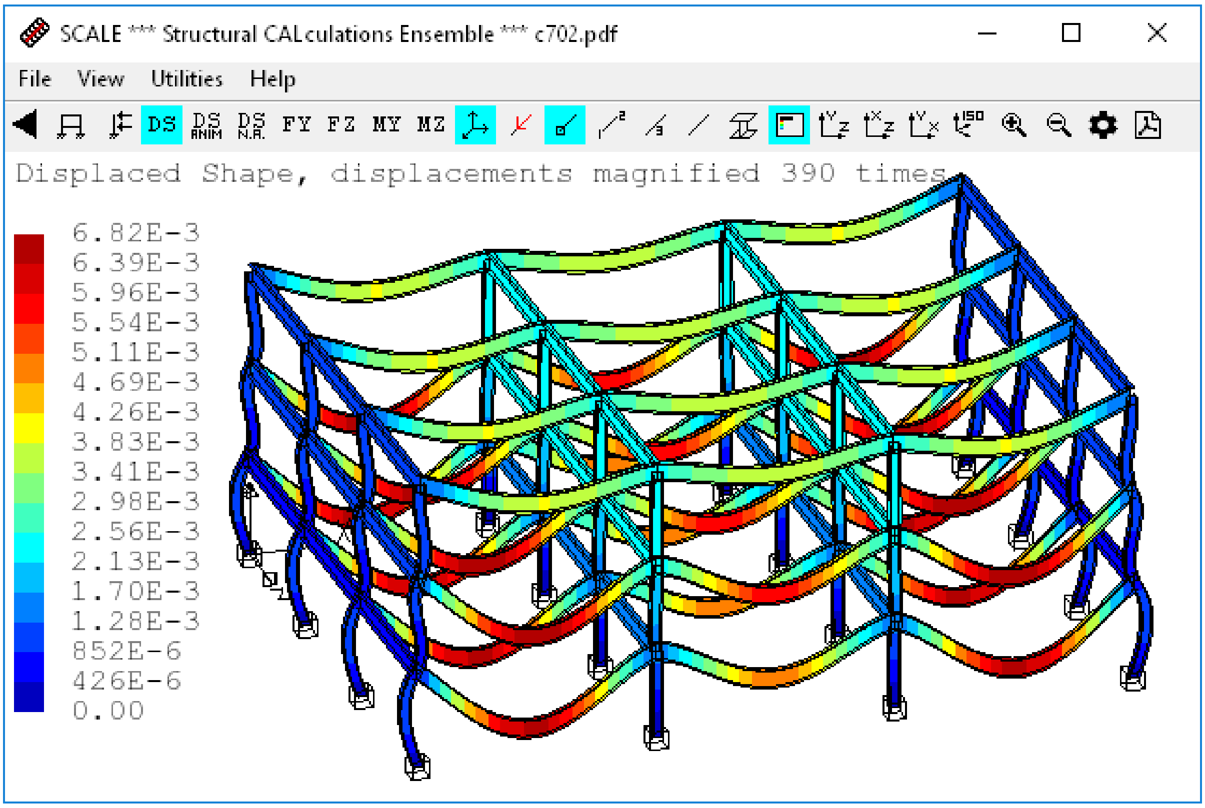


Figure 6.4: NL-VIEW 3D display of NL-STRESS analysis (Windows).

6.3 Basic navigation

The aim of the navigation system is to be as intuitive as possible.

Click and drag anywhere on the screen to rotate the view. Drag left and right to rotate the model left and right. Drag up and down to rotate the model up and down, the rotation is constrained when viewing from above and below to stop the model flipping upside down.

The path the drag takes does not affect the final rotation, to return to the initial viewpoint when dragging just drag back to where you started.

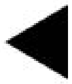
Use the mouse's scroll button to zoom in and out to make the model appear larger or smaller.

Click on both the left and right mouse buttons together and drag to pan the model around within the viewport.

6.4 Toolbar items

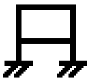
The buttons listed below are presented on the toolbar, and their use is described later in this section. Several buttons toggle an action on and off, when toggled on the button will have a light blue background.

- Back to previous screen.
- Display structure only.
- Show loads.
- Displaced shape - coloured sections.
- Animated displaced shape - coloured sections.
- Displaced shape - neutral axes.
- FY shear force. } Different combinations of
- FZ shear force. } these buttons will appear
- MY bending moment. } depending on the type of
- MZ bending moment. } structure analysed.
- Show global axes.
- Show local axes.
- Show supports.
- Show joint numbers.
- Show member numbers.
- Show neutral axes.
- Show actual sections.
- Show Key.
- View along X axis.
- View along Y axis.
- View along Z axis.
- View isometric.
- Settings.
- Save screenshot to pdf.



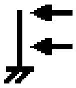
6.4.1 BACK TO PREVIOUS SCREEN

When finished viewing the results in NL-VIEW click this button to return to the display of the results.



6.4.2 DISPLAY STRUCTURE ONLY

Select Display Structure Only to display the structure without any bending moments, shear forces or deflections shown.



6.4.3 SHOW LOADS

Select Show Loads to toggle the display of the loadings applied to the model.

6.4.4 DS - DISPLACED SHAPE - COLOURED SECTIONS

Select Deflected Shape - Coloured Sections to plot the displaced shapes of all selected load cases. The Key will identify the colours which correspond to which magnitude of deflection. The displaced shapes will show the actual section properties. As the displaced shapes are not identified by load case, it is sensible to only select one load case at a time for clarity. However the option to display more than one load case is enabled for comparison between load cases.

6.4.5 DS - ANIMATED DISPLACED SHAPE - COLOURED SECTIONS

Select Animated Displaced Shape - Coloured Sections to plot an animation of the displaced shapes of all selected load cases. The Key will identify the colours which correspond to which magnitude of deflection. The displaced shapes show the actual section properties.

The animation will cycle between 0% of the displaced shape to 100% of the displaced shape in a sinusoidal manner, this is a useful as a check that all parts of the structure are deflecting as expected. By animating the deflection it is easier to see if parts of the structure are not connected, and are moving independently which can occur if there are mistakes in the data file.

As the displaced shapes are not identified by load case, it is sensible to only select one load case at a time for clarity. However the option to display more than one load case is enabled for comparison between load cases.

6.4.6 DS - DISPLACED SHAPE - NEUTRAL AXES

Select Displaced Shape - Neutral Axes to plot the displaced shape of the neutral axes with each load case being represent by a different line style and colour, which will identified in the Key. This is useful for comparing the displacements between several load cases.

The original structure may be omitted, or drawn as straight lines, or as the actual sections themselves.

6.4.7 FY SHEAR FORCE

Select FY Shear Force to plot the shear force in the local y direction along the member for all selected load cases. The shear force diagrams are connected to both ends of the members for clarity. Each load case will have a different colour, which will be identified in the Key.

The original structure will be drawn as either straight members, or as the sections themselves, as selected above. For clarity shear forces are not plotted relative to a deflected structure.

6.4.8 FZ SHEAR FORCE

Select FZ Shear Force to plot the shear force in the local z direction along the member for all selected load cases. The shear force diagrams are connected to both ends of the members for clarity. Each load case will have a different colour, which will be identified in the Key.

MY

6.4.9 MY BENDING MOMENT

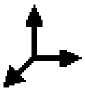
Select MY Bending Moment to plot the bending moments about the local y axis along the member for all selected load cases. The bending moment diagrams are connected to both ends of the members for clarity. Each load case will have a different colour, which will be identified in the Key.

The original structure will be drawn as either straight members, or as the sections themselves, as selected above. For clarity bending moment diagrams are not plotted relative to a deflected structure.

MZ

6.4.10 MZ BENDING MOMENT

Select MZ Bending Moment to plot the bending moments about the local z axis along the member for all selected load cases. The bending moment diagrams are connected to both ends of the members for clarity. Each load case will have a different colour, which will be identified in the Key.



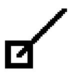
6.4.11 SHOW GLOBAL AXES

Select Show Global Axes to toggle the display a set of XYZ axes at the origin.

6.4.12 SHOW MEMBER LOCAL AXES

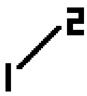
Select Show Member Local Axes to toggle the display of the local axes on each member. This is valuable as a tool to check that the axis orientations are as expected. The alignment of the member axes is described in the NL-STRESS User's Manual, the axes shown in NL-VIEW include any adjustments required for vertical members and for members with a BETA angle set.

The axes are drawn with three different lengths for clarity, the longest line represents the local x-axis, the mid-length line represents the local y-axis, and the shortest line represents the local z-axis.



6.4.13 SHOW SUPPORTS

Select Show Supports to toggle the display of the supports.

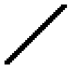


6.4.14 SHOW JOINT NUMBERS

Select Show Joint Numbers to toggle the display of the joint number next to each joint.

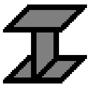
6.4.15 SHOW MEMBER NUMBERS

Select Show Member Numbers to toggle the display of the member number at the mid-point of each member.



6.4.16 SHOW NEUTRAL AXES ONLY

Select Show Neutral Axes Only to show members as single lines connecting the joints.

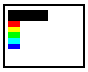


6.4.17 SHOW SECTION OUTLINES

Select Show Section Outlines to represent the sections by actual sized section properties for I-sections, H-sections and Rectangles (T-sections coming soon).

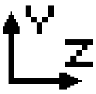
This may be used to check that the orientation of the sections is as expected, in particular for inclined and vertical members, and for members with a BETA rotation applied.

The selection of Section Outlines and Neutral Axes Only toggle each other on and off. One or other will always be displayed when displaying loadings, forces and moments. They may be individually toggled off when displaying displacements if required, if you want the displaced shape only to be shown.



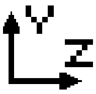
6.4.18 SHOW KEY

Select Show Key to toggle the display of a key in the top left corner of the window. The key will show which colours correspond to the loadings, displacements, loadings etc. as appropriate.



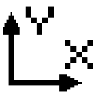
6.4.19 VIEW ALONG X AXIS

Select View Along X Axis to view the model along the global X axis.



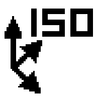
6.4.20 VIEW ALONG Y AXIS

Select View Along Y Axis to view the model along the global Y axis.



6.4.21 VIEW ALONG Z AXIS

Select View Along Z Axis to view the model along the global Z axis.



6.4.22 VIEW ISOMETRIC

Select View Isometric to view an isometric projection of the model. An isometric projection is a special case of orthographic projection where the angles between the projection of the x, y and z axes are all the same at 120 degrees. This corresponds to an elevation of arcsin(tan(30 degrees)) = +/-35.264 degrees, and an azimuth of +/-45 degrees. In an isometric projection of a cube structure, the view will be from above one corner, looking towards the opposite lower corner. The term "isometric" comes from the Greek for "equal measure" indicating that the scale along each axis of the projection is the same.

6.4.23 GO TO SETTINGS SCREEN

The settings screen is used to change the magnification factors for the plots of the displacements, shear forces, moments, point loads and distributed loads. The factor for displacement shows the number of times the actual displacement. All other factors are relative to an initially calculated starting factor of 100. Edit the factors as required and tap on the back button to return to the main NL-VIEW window.

The settings screen is used to select which loadcases to display on the model, enter a list of the loadcases required.

You can also set the azimuth and elevation settings as numbers. The screen will initially display the current values, so these can be noted for a particular viewpoint, and these values can then be entered subsequently to reproduce the same viewpoint.

6.4.24 SAVE SCREENSHOT TO PDF

Tap on this button to append a screenshot of the current view to the end of the calculations pdf file. This added page will also contain a key showing what is displayed.

6.5 POP-UP MENU

Right click in the NL-VIEW window to bring up a pop-up menu.
The pop-up menu provides handy shortcuts for selecting various viewing options. The menu also provides a list of all the loadcases and enables easy selection and deselction of the required loadcases by name simply by clicking on the corresponding menu items.

7. NL-STRESS Reference Manual

7.1 Introduction to the NL-STRESS Reference Manual

This manual supports the NL-STRESS User's Manual by giving details of formulae and procedures used in NL-STRESS.

Section 7.2 lists all the formula used by NL-STRESS for the computation of section properties when cross-sections are specified by geometry, i.e. dimensions are given rather than inertias and areas.

Sections 7.3 to 7.6 cover the non-linear aspects of the software in more detail than in the User's Manual.

Section 7.7 gives the full derivation of the stiffness matrices used by NL-STRESS, allowing for the effects of shear deformation.

7.2 Section Properties

The formulae used by NL-STRESS in the computation of elastic section properties are given. The engineer is also referred to:

- 'Formulas for Stress and Strain' by Roark, published by McGraw Hill
- 'Reinforced Concrete Designer's Manual' by Reynolds, published by Concrete Publications Ltd
- 'Steel Designers' Manual' published by Crosby Lockwood

In the formulae the following NL-STRESS labels have been substituted for the symbols used by Roark and Reynolds.

DY the overall dimension in the local y direction
DZ the overall dimension in the local z direction
TY the thickness in the local y direction
TZ the thickness in the local z direction

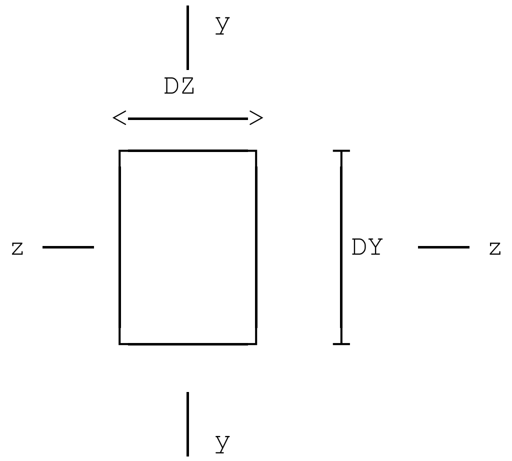
AX the cross sectional area of the member
AY the shear area of the member corresponding to shear force acting in the direction of the local y axis
AZ the shear area of the member corresponding to shear force acting in the direction of the local z axis

IX the torsional moment of inertia (or torsional constant) of the member cross-section about its longitudinal axis
IY the second moment of area (moment of inertia) of the cross-section about the local y axis
IZ the second moment of area (moment of inertia) of the cross-section about the local z axis

The axes displayed in the figures refer to the local axes.

7.2.1 Solid rectangle

Elastic section properties for solid rectangle :
Square is special case where D = DY = DZ



$$AX = DY.DZ$$

$$AY = AZ = \frac{5.AX}{6}$$

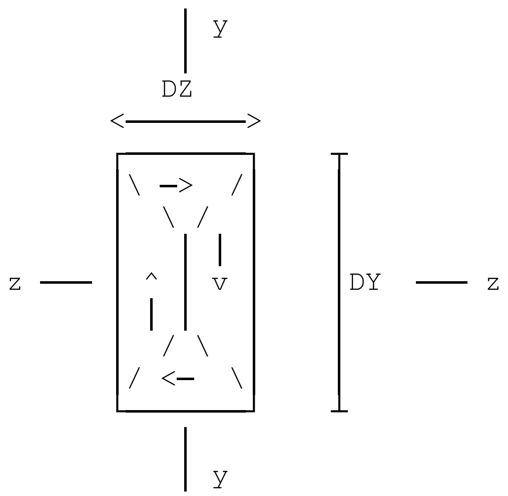
$$IY = \frac{DY.DZ^3}{12}$$

$$IZ = \frac{DZ.DY^3}{12}$$

Let A = DY/2 and B = DZ/2, where A refers to longer side, then

$$IX = A.B^3 \left[\frac{16}{3} - 3.36 \frac{B}{A} \left[1 - \frac{B^4}{12.A^4} \right] \right]$$

Plastic section properties for solid rectangle:



For plastic torque about axis x directions of shear flow shown by arrows on the cross-section

First moment of rectangle about

$$zz = DZ.\frac{DY}{2}.\frac{DY}{2} = \frac{DZ.DY^2}{4}$$

First moment of rectangle about

$$yy = DY.\frac{DZ}{2}.\frac{DZ}{2} = \frac{DY.DZ^2}{4}$$

$$\text{Plas torque const} = 2.\left[(DY-DZ).\frac{DZ}{2}.\frac{DZ}{4} \right] + \left[\frac{1}{2} \frac{(DZ)^2}{(2)^2} . \frac{2}{3} . \frac{DZ}{2} \right].4$$

$$+ 2.\left[\frac{1}{2} . \frac{DZ.DZ}{2} . \left[\frac{DY}{2} - \frac{DZ}{6} \right] \right]$$

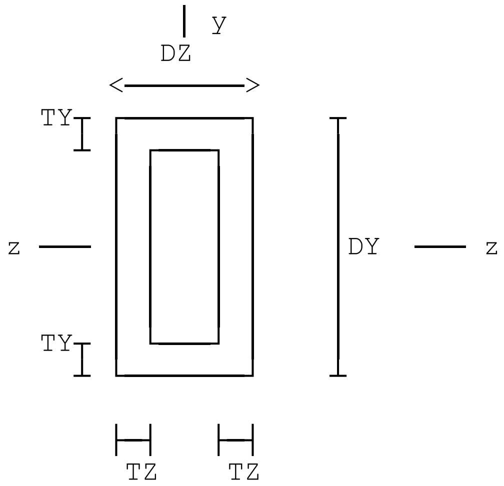
$$= DY.DZ^2 \left[\frac{1}{4} + \frac{1}{4} \right] + DZ^3 \left[\frac{-1}{4} + \frac{1}{6} - \frac{1}{12} \right]$$

$$= \frac{DY.DZ^2}{2} - \frac{DZ^3}{6}$$

When DY >>> DZ this reduces to DY.DZ^2/2 which is expression for thin strips.

7.2.2 Hollow rectangle

Elastic section properties for hollow rectangle:



RHS is special case when T = TY = TZ

Let D1 = DY - 2.TY

and D2 = DZ - 2.TZ

then AX = DY.DZ - D1.D2

$$\text{and } IY = \frac{DY.DZ^3}{12} - \frac{D1.D2^3}{12}$$

$$\text{and } IZ = \frac{DZ.DY^3}{12} - \frac{D2.D1^3}{12}$$

When TZ = 0, i.e top and bottom flanges only AY = AZ = 5.AX/6

Let A = DZ/2 and B = TY/2, then for top and bottom flanges only:

$$IX = 2.A.B^3 \left[\frac{16}{3} - 3.36 \times \frac{B}{A} \left[1 - \frac{B^4}{12A^4} \right] \right]$$

$$\text{else} \quad IX = \frac{2.TY.TZ.(DY-TY)^2 \times (DZ-TZ)^2}{DY.TY + DZ.TZ - TY^2 - TZ^2}$$

When TZ > 0, i.e walls are present: $AY = \frac{AX}{F}$

Let D2 = DY/2 & D1 = D2 - TY

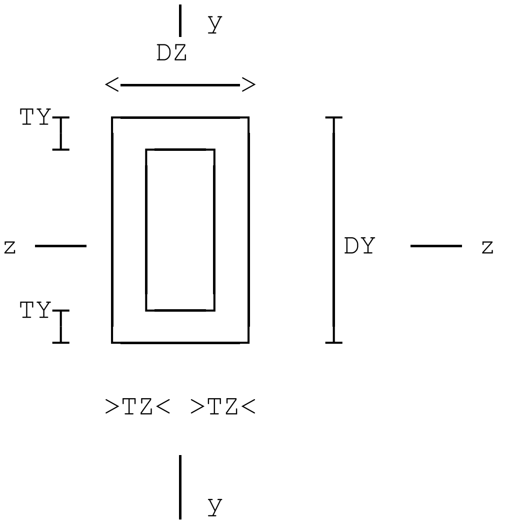
$$\text{then } F = \left[1 + \frac{3.(D2^2 - D1^2).D1}{2.D2^3} \left[\frac{DZ}{2.TZ} - 1 \right] \right] \frac{4.D2^2 \times AX}{10.IZ}$$

When TZ > 0, i.e walls are present: $AZ = \frac{AX}{F}$

Let D2 = DZ/2 & D1 = D2 - TZ

$$\text{then } F = \left[1 + \frac{3.(D2^2 - D1^2).D1}{2.D2^3} \left[\frac{DY}{2.TY} - 1 \right] \right] \frac{4.D2^2 \times AX}{10.IY}$$

Plastic section properties for hollow rectangle:



RHS is special case when
T = TY = TZ

Let b = DZ - TZ

and d = DY - TY

First moment of section about zz

$$= \frac{DZ.DY^2}{4} - \frac{(DZ-2.TZ)(DY-2.TY)^2}{4}$$

First moment of section about yy

$$= \frac{DY.DZ^2}{4} - \frac{(DY-2.TY)(DZ-2.TZ)^2}{4}$$

When TZ < TY plastic torque constant = [2.b.d.TZ]

= 2.(DZ-TZ)(DY-TY).TZ

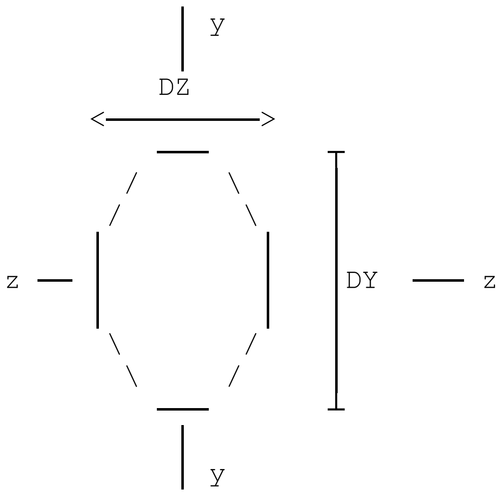
When TY < TZ plastic torque constant = [2.b.d.TY]

= 2.(DZ-TZ)(DY-TY).TY

7.2.3 Solid conic

Elastic section properties for solid conic:

Circle is special case when D = DY = DZ



$$AX = \frac{\pi.DY.DZ}{4}$$

$$AY = AZ = 0.9 \times AX$$

$$IY = \frac{\pi.DY.DZ^3}{64}$$

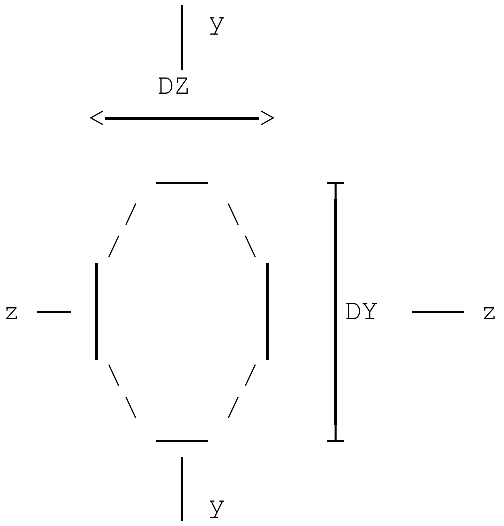
$$IZ = \frac{\pi.DZ.DY^3}{64}$$

Let A = DY/2 and B = DZ/2 then $IX = \dfrac{\pi.A^3 \times B^3}{A^2 + B^2}$

Where π = 3.141592653589793

Plastic section properties for conic:

Circle is special case when D = DY = DZ



Ellipse is projection of a circle turned through an angle, hence distance from neutral axis to centroid of half section is as for circle

$= \dfrac{2.DY}{3.\pi}$ or $\dfrac{2.DZ}{3.\pi}$ for the two axes.

First moment of ellipse about zz

$= \dfrac{\pi.DZ.DY}{8} \;.\; \dfrac{2.DY}{3.\pi} \;.\; 2 \quad \dfrac{DZ.DY^2}{6}$

First moment of ellipse about yy

$= \dfrac{\pi.DZ.DY}{8} \;.\; \dfrac{2.DZ}{3.\pi} \;.\; 2 = \dfrac{DY.DZ^2}{6}$

An accurate answer for the plastic torque constant for an ellipse is very difficult. Take as π/2 times lowest of above values to make treatment as for solid circle below:

Consider annulus of thickness dr at radius r then

$$\text{Plas torque const for solid circle} = \int_0^{D/2} r.2.\pi\,.r.dr = \left[\frac{2.\pi.r^3}{3} \right]_0^{D/2}$$

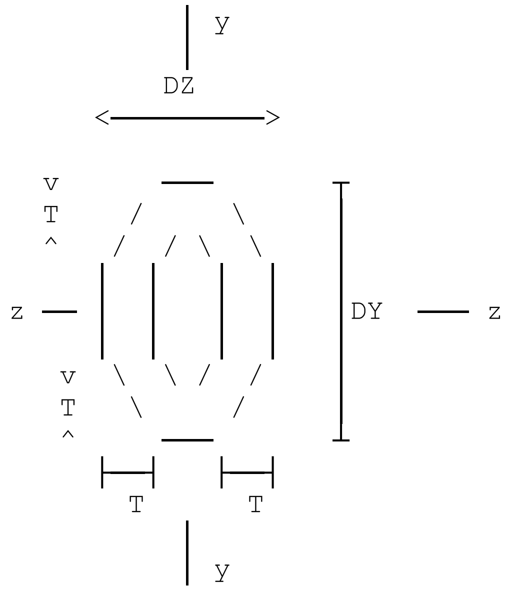
$$= \frac{\pi.D^3}{12}$$

7.2.4 Hollow conic

Elastic section properties for hollow conic sections:

NL-STRESS faults a non uniform wall thickness for hollow conic sections. The equations given here are for a uniform wall thickness in the y and z direction of T.

CHS is special case when D = DY = DZ



Let B1 = DZ -2.T

and D1 = DY -2.T

then:

$$AX = \frac{\pi}{4} (DZ.DY - B1.D1)$$

$$IY = \frac{\pi}{64} (DY.DZ^3 - D1.B1^3)$$

$$IZ = \frac{\pi}{64} (DZ.DY^3 - B1.D1^3)$$

$$AY = AZ = \frac{AX}{2}$$ (ref. Roark for thin walled hollow circle)

Let A = DY/2 and B = DZ/2 then for uniform thickness T

$$\text{and } U = \pi (A + B - T) \left[1 + 0.27 \frac{(A - B)^2}{(A + B)^2} \right]$$

$$\text{then} \quad IX = \frac{4.\pi^2 \times T (A - T/2)^2 \times (B - T/2)^2}{U}$$

where π = 3.141592653589793

For the special case of a circular section of uniform thickness:

Let R1 = D/2 and R0 = R1 - T then $IX = \frac{\pi}{2} (R1^4 - R0^4)$

Plastic section properties for circular hollow section:

Treat as difference between two solid circles viz:

$$\text{First moment about yy or zz} = \frac{D^3}{6} - \frac{(D-2.T)^3}{6}$$

For a circle D = DY = DZ

Plastic section properties for hollow ellipse:

Treat as difference between two solid ellipses

First moment of hollow ellipse about zz

$$= \frac{DZ.DY^2}{6} - \frac{(DZ-2.T)(DY-2.T)^2}{6}$$

First moment of hollow ellipse about yy

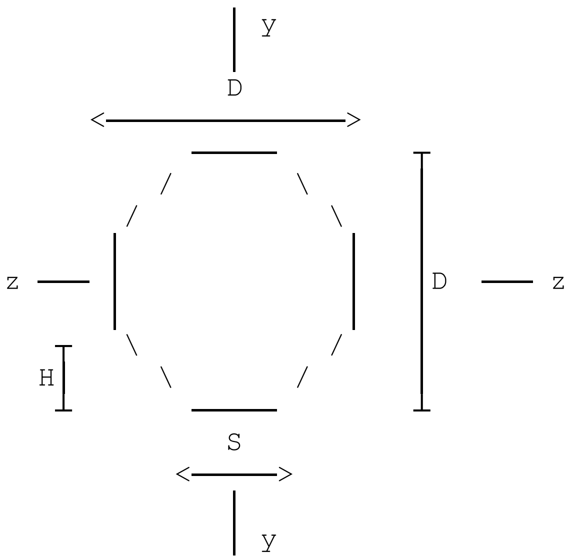
$$= \frac{DY.DZ^2}{6} - \frac{(DY-2.T)(DZ-2.T)^2}{6}$$

Plastic torque constant = 2.A.T where A is mean enclosed area

$$= 2.\left[\frac{\pi}{4}.(DY-T)(DZ-T)\right].T = \frac{\pi}{2}(DY-T)(DZ-T)T$$

7.2.5 Octagon

Not given by Roark but derived as follows:



$$S = \frac{D}{1 + 2^{0.5}}$$

$$H = \frac{D}{2 + 2^{0.5}}$$

$$AX = \left[1 - \frac{1}{(1 + 2^{0.5})^2}\right] D^2$$

$$= 0.8284271247461901 \times D^2$$

AY = AZ = 0.9 x AX (assumed the same as a circle)

$$IY = IZ = D^4 \left[\frac{1}{12} - \frac{1}{9(2+2^{0.5})^4} - \frac{1}{(1+2^{0.5})^2}\left[\frac{1}{2} - \frac{1}{6+3x2^{0.5}}\right]^2\right]$$

$$= 0.05473785412436499 \times D^4$$

$$J = IY + IZ = 0.1094757082487300 \times D^4$$

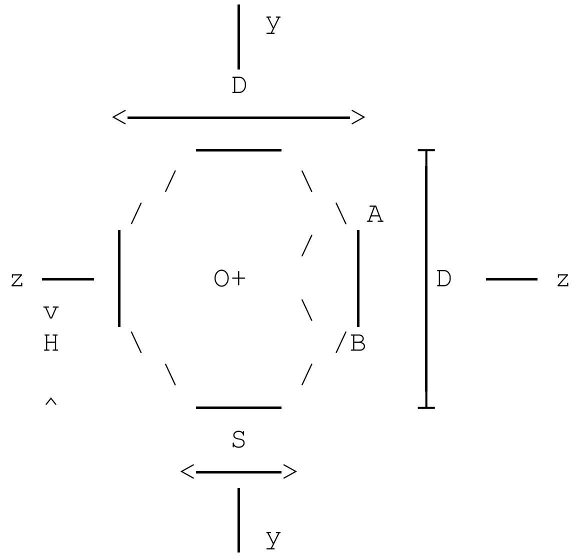
$$IX = \frac{AX^4}{40.J} \quad \text{(Roark)} \quad = \frac{0.8284271247461901^4 \times D^8}{40 \times 0.1094757082487300 \times D^4}$$

$$= 0.1075571996519179 \times D^4$$

For hollow octagon, deduct middle except for:

AY = AZ = 0.5 x AX (ref. Roark for hollow circle)

Plastic section properties for octagon:



$$S = \frac{D}{1 + 2^{0.5}}$$

$$H = \frac{D}{2 + 2^{0.5}}$$

First moment of octagon about zz = first moment about yy

$$= \frac{D}{2}.\frac{D}{2}.D - \left[\frac{D}{2+2^{0.5}}\right]^{2} \left[\left[\frac{D}{1+2^{0.5}}\right] \frac{1}{2} + \frac{2}{3} \left[\frac{D}{2+2^{0.5}}\right] \right] x2$$

$$= \frac{D^3}{4} - \left[\frac{D}{2+2^{0.5}}\right]^{2} \left[\frac{D}{1+2^{0.5}} + \frac{4.D}{3(2+2^{0.5})} \right]$$

$$= D^3 \left[\frac{1}{4} - \frac{1}{2(1+2^{0.5})^2} \left[2^{0.5} - 1 + \frac{(4-2.2^{0.5})}{3} \right] \right]$$

$$= D^3 \left[\frac{1}{4} - \frac{1}{2(1+2^{0.5})^2} \left[\frac{1 + 2^{0.5}}{3} \right] \right]$$

$$= D^3 \left[\frac{3 - 2.2^{0.5} + 2}{12} \right] = D^3 \left[\frac{5 - 2.2^{0.5}}{12} \right]$$

For plastic torque constant, in any one of 8 triangles OAB, yield stress in shear acts parallel to AB.

Plastic torque constant for segment = $\frac{1}{2} \left[\frac{D}{1+2^{0.5}} \right] \frac{D}{2} \frac{2}{3} \frac{D}{2}$

Plastic torque constant for octagon = $\frac{2}{3} (2^{0.5} - 1) D^3$

For hollow octagon, treat as difference between two solid octagons:

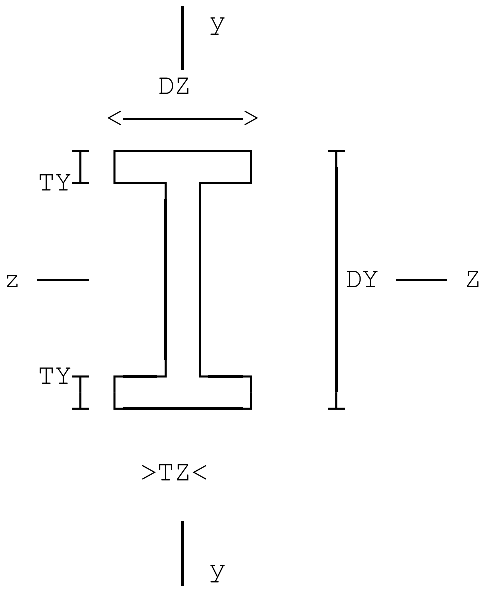
First moment of hollow octagon about yy = first moment about zz

$$= \frac{5 - 2.2^{0.5}}{12} \left[D^3 - (D - 2.T)^3 \right]$$

Plastic torque constant $= \frac{2}{3} (2^{0.5} - 1) \left[D^3 - (D-2.T)^3 \right]$

7.2.6 I Section

Elastic section properties of I Section:



Let $D1 = DY - 2.TY$

and $D2 = DZ - TZ$

then:

$AX = DY.DZ - D1.D2$

$AY = AX/F$

$$IY = \frac{2.TY.DZ^3}{12} + \frac{D1.TZ^3}{12}$$

$$IZ = \frac{DZ.DY^3}{12} - \frac{D2.D1^3}{12}$$

Let $D2 = DY/2$ and $D1 = D2 - TY$

$$\text{then } F = \left[1 + \frac{3(D2^2 - D1^2).D1}{2.D2^3} \times \left[\frac{DZ}{TZ} - 1 \right] \right] \frac{4.D2^2 \times AX}{10.IZ}$$

$$AZ = 2 \times \frac{5}{6}.DZ.TY$$

$$IX = 2.K1 + K2 + 2.A.D^4$$

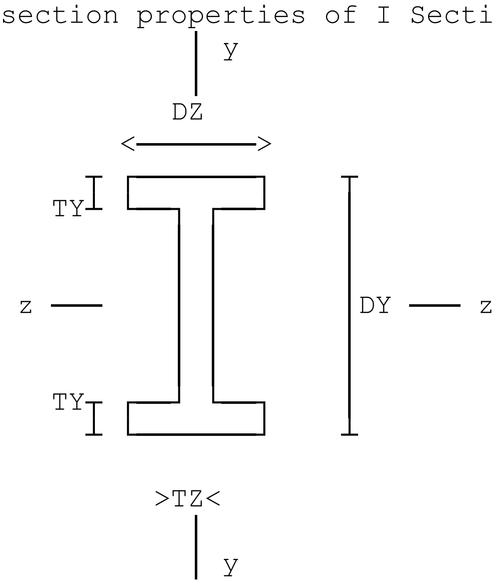
in which D is diameter of largest inscribed circle, and where

$$K1 = DZ.TY^3 \left[\frac{1}{3} - 0.21 \frac{TY}{DZ} \left[1 - \frac{TY^4}{12.DZ^4} \right] \right]$$

$$K2 = \frac{1}{3} (DY - 2.TY).TZ^3 \qquad A = 0.15 \times \frac{(\text{least of } TY \ \& \ TZ)}{(\text{greatest of } TY \ \& \ TZ)}$$

and $D = TY + \frac{TZ^2}{4.TY}$ (D is limited to TZ when TZ > 2.TW)

Plastic section properties of I Section:



First moment of I Section about zz $= DZ.TY\,(DY - TY) + \frac{(DY-2.TY)^2.TZ}{4}$

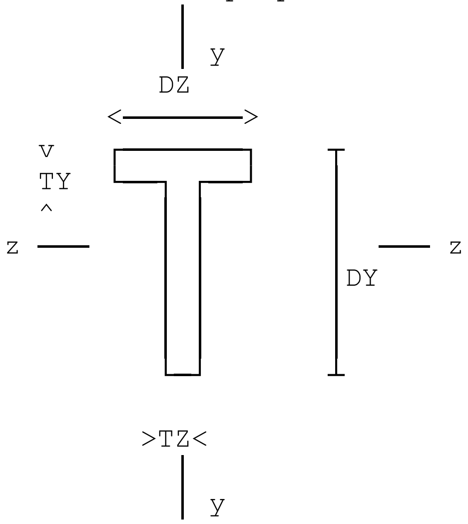
First moment of I Section about yy $= \frac{2.DZ^2.TY}{4} + \frac{TZ^2.(DY-2TY)}{4}$

$= \frac{DZ^2.TY}{2} + \frac{(DY-2.TY).TZ^2}{4}$

Plastic torque constant $= DZ.TY^2 + \frac{DY.TZ^2}{2}$ ref M.R.Horne

7.2.7 T Section

Elastic section properties of T Section:



$$AX = DZ.TY + (DY - TY).TZ$$

$$AY = \frac{5}{6}.DY.TZ \qquad AZ = \frac{5}{6}.DZ.TY$$

$$IY = TY.\frac{DZ^3}{12} + (DY-TY).\frac{TZ^3}{12}$$

$$IZ = \frac{1}{3}\left[DZ.N^3 + TZ(DY - N)^3 - (N - TY)^3 \times (DZ - TZ) \right]$$

where $N = \dfrac{TZ.DY^2 + TY^2 \times (DZ - TZ)}{2(DZ.TY + TZ(DY - TY))}$ ref Reynolds

$$IX = K1 + K2 + A.D^4$$

in which D is diameter of largest inscribed circle, and where

$$K1 = DZ.TY^3 \left[\frac{1}{3} - 0.21 \frac{TY}{DZ} \left[1 - \frac{TY^4}{12.DZ^4} \right] \right]$$

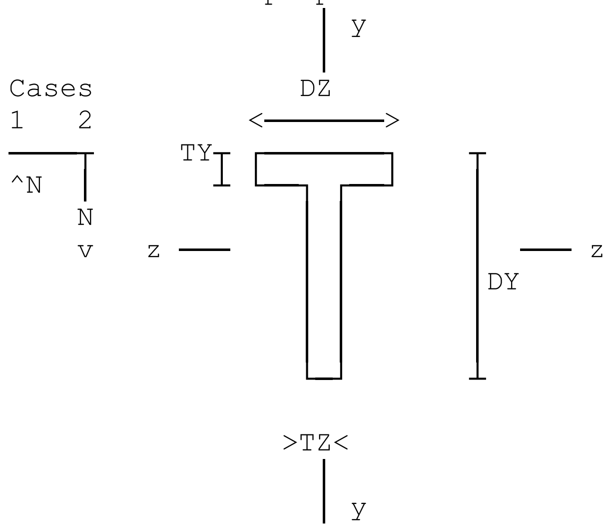
$$K2 = C.TZ^3 \left[\frac{1}{3} - 0.105 \frac{TZ}{C} \left[1 - \frac{TZ^4}{192.C^4} \right] \right]$$

where $C = DY - TY$

$$A = 0.15 \times \frac{(\text{least of } TY \,\&\, TZ)}{(\text{greatest of } TY \,\&\, TZ)} \qquad D = TY + \frac{TZ^2}{4.TY}$$

(D is limited to TW when TZ > 2.TY)

Plastic section properties of T Section:



First moment of area about yy

$$= \frac{TY.DZ^2}{4} + \frac{(DY-TY).TZ^2}{4}$$

Plastic torque constant

$$= \frac{DZ.TY^2}{2} + \frac{DY.TZ^2}{2}$$ ref M.R Horne

Case 1 neutral axis within flange

$DZ.N = DZ(TY-N) + (DY-TY).TZ$ therefore $2.DZ.N = DZ.TY + DY.TZ - TY.TZ$

$$\text{and } N = \frac{DZ.TY + DY.TZ - TY.TZ}{2.DZ}$$

First moment of T Section about zz

$$= \frac{DZ.N^2}{2} + \frac{(DY-N)^2.TZ}{2} + \frac{(DZ-TZ)(TY-N)^2}{2}$$

Case 2 neutral axis within web

$DZ.TY + (N-TY).TZ = (DY-N).TZ$ therefore $2.N.TZ = DY.TZ + TY.TZ - DZ.TY$

$$\text{and } N = \frac{DY.TZ + TY.TZ - DZ.TY}{2.TZ}$$

First moment of T Section about zz

$$= DZ.TY(N-TY) + \frac{TZ(N-TY)^2}{2} + \frac{TZ(DY-N)^2}{2}$$

7.2.8 H Section

Member Properties by reference to I Section formulae.

7.3 Finite Displacements

7.3.1 General effect

In a linear elastic analysis the structure stiffness matrix is built assuming the structure is undisplaced by applied loads. An equilibrium check on the loaded structure will reveal that reactions do not balance with applied loads (due to the fact that the load positions have changed as the structure undergoes 'finite displacements').

The general effect of finite displacements is to cause an increase in stresses throughout the structure; linear elastic analysis giving lower stresses than those actually present. Professor M.R.Horne has demonstrated how important finite displacements can be in ICE Proceedings December 1961.

NL-STRESS allows the engineer to take these finite displacements into account in the analysis simply by setting the NUMBER OF INCREMENTS command to a suitable value.

If the engineer includes the commands METHOD SWAY and NUMBER OF INCREMENTS 20, the software will build up the loading by applying it in 20 increments and after each increment use the displacement history to predict the displacements after the next increment and build the structure stiffness matrix accordingly. The prediction of next increment of deflection is due to M.R.Horne (see 7.3.2). The accuracy of the method can be verified by inspection of the Equilibrium Check.

In building the structure stiffness matrix for the next increment, NL-STRESS assumes that each member is straight between end joints; thus for members where lateral displacements are significant it will be necessary to have additional joints along each member to take these lateral displacements into account. The NUMBER OF SEGMENTS command is used to divide each and every member into the number of segments specified following the command.

Space structures present a special problem in that each member can rotate about its own axis. The current version of NL-STRESS assumes that rotations of sloping members about their own axes are negligible i.e. the beta angle specified at the start remains unaltered. For vertical members the classical assumption that local z points in the same direction as global Z works for the first cycle of the first increment; thereafter the vertical member is inclined and the classical assumption is that local z is parallel to the global xy plane (the ground). NL-STRESS computes the changes in beta that are necessary and takes them into account in SWAY and PLASTIC analyses.

Steel sections have very little torsional stiffness and the effect of changing betas at each cycle of each increment can induce torsional oscillation in the analysis. If this happens (manifest by 100 or more cycles) then it will probably be necessary to provide one or more rotational restraints in the length of the column. In all 3D non linear analyses it is essential to closely inspect the MZ plot and the equilibrium check.

7.3.2 Prediction of next increment of deflection in step-by-step analysis of non-linear structures

If a structure has the same deflection form as that corresponding to the lowest critical load factor lam'c, then deflections at the (n-1)th, nth, and (n+1)th load factor would be D'n-1, D'n, D'n+1 where:

$$D'_{n-1} = \frac{D'_0}{\frac{lam'_c}{lam'_{n-1}} - 1} \qquad D'_n = \frac{D'_0}{\frac{lam'_c}{lam'_n} - 1} \qquad D'_{n+1} = \frac{D'_0}{\frac{lam'_c}{lam'_{n+1}} - 1}$$

where D'0 = linear deflection at load factor lam'c
lam'n = load factor after (n-1)th load increment
lam'n-1 = load factor after (n-2)th load increment
lam'n+1 = load factor after (n)th load increment

With equal load increments,
lam'n+1 - lam'n = lam'n - lam'n-1 = lam say.

We wish to be able to predict (D'n+1 - D'n) from a knowledge of (D'n - D'n-1), D'n, lam and lam'n not knowing lam'c or D'0.

It may be shown with sufficient accuracy,

M = magnification factor for increments

$$= \frac{D'_{n+1} - D'_n}{D'_n - D'_{n-1}} = 1 + 2\left[\frac{D'_n - D'_{n-1}}{D'_n} - \frac{lam}{lam'_n}\right]$$

Each deflection in a structure will have its own multiplication factor M for the successive increments (D'n - D'n-1), (D'n+1 - D'n) as above.

The above formulae have been incorporated into NL-STRESS as follows:

If the number of increments is requested as 10, the first increment applies 10% of the loading to the structure with undisplaced geometry.

The second increment applies 20% of the load to the structure with geometry using the displacement from the first increment multiplied by:

$$M = \frac{n+1}{n} + 2\left(\frac{d}{D} - \frac{1}{n}\right)$$

in which n is the number of increments so far applied (=1), D is total deflection to date, d is increase from last increment thus:

M = 2 + 2(1 - 1) = 2

i.e. the stiffness matrix, member loads etc. are computed on the basis that the deflection at the end of the second increment will be twice that found after the first increment.

The third increment applies 30% of the load to the structure with geometry using displacements from the second multiplied by:

$$M = \frac{2 + 1 + 2}{2} \left[\frac{0.6}{1.1} - \frac{1}{2} \right] = 1.5 + 0.0909 = 1.5909$$

supposing total displacements to be 0.5 after the 1st increment and 1.1 after the second increment.

7.3.3 Satisfaction of equilibrium and compatibility

Application of the prediction of next increment of deflection formula give in 3.2 above will give excellent satisfaction of equilibrium requirements, allowing satisfactorily for change in geometry. This satisfaction of equilibrium can be seen from the EQUILIBRIUM CHECK following the SUPPORT REACTIONS. NL-STRESS uses double-precision arithmetic giving 16+ decimal digits of accuracy, therefore comparing applied loads with computed reactions would show the figures to be identical. So the EQUILIBRIUM CHECK compares computed reactions with applied forces in their displaced positions, and is thus a true equilibrium check. In any analysis it is also essential to satisfy 'compatibility' i.e. stresses and strains are compatible for the material constants nominated. Consider a straight, axially fully rigid member of length L, initially in the vertical position OA. It is restrained rotationally by a spring at O with a stiffness K

(i.e. restraining moment = K.theta)

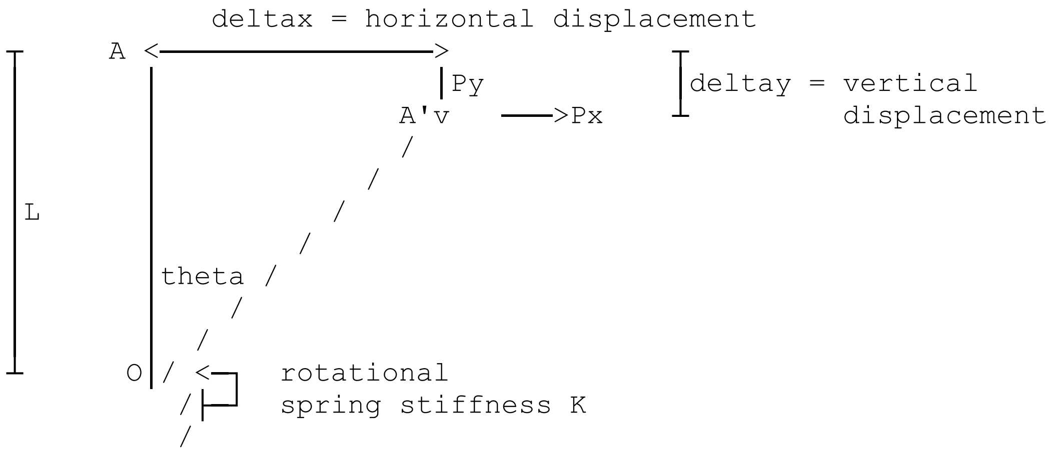
Under loads Py (vertical) and Px (horizontal), OA deflects to OA'.

Exact solution:

Equilibrium: [Px.COS(theta) + Py.SIN(theta)] = K.theta/L

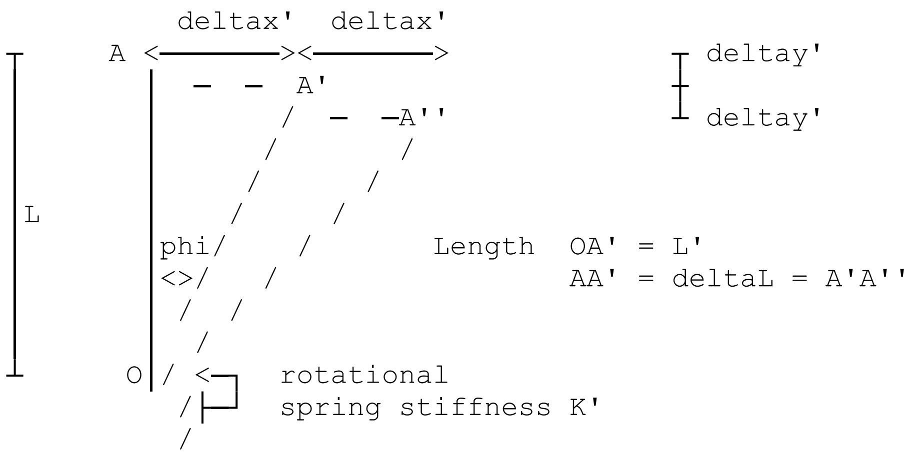
Vertical displacement of A' = deltay = L [1 - COS(theta)]

approx = L.theta^2 (1-theta^2/12)/2 approx = L.theta^2/2



Horizontal displacement of A' = L.SIN(theta)
approx = L.theta (1 - theta^2/6) approx = L.theta

Solution achieved by applying the 'prediction of next deflection formula'



A prediction is made of the final position of the member OA moved to OA', i.e. predictions are made of the horizontal displacement deltax' and the vertical displacement deltay'.

These displacements are to be the same as those obtained from the linear analysis of the displaced structure. This linear analysis gives the deflections deltaL perpendicular to the predicted final position of the member OA' i.e. A'A'' = deltaL, OA'A'' = 90 degs.

Since by this procedure, the new length of the member L', is (wrongly) different from the actual length L, we modify the spring stiffness from K to K' so that, for small rotations of OA', the incremental lateral displacement due to a force applied to A' at right angles to the member is identical with that which it would have been, had the member retained its original length.

Hence for a small force P applied at A or A' perpendicular to the member:

delta = (P.L/K).L = (P.L'/K).L' i.e. K' = (L'/L)^2.K

Applying this stiffness to the member OA', the linear deflection deltaL due to forces Px and Py is given by

$$\text{deltaL} = \left[\frac{(\text{Px.COS(phi)} + \text{Py.SIN(phi).L'}}{\text{K'}} \right] \text{L'}$$

We transform the displacement deltaL into its global coordinates, i.e.

Horizontal displacement = deltaL.COS(phi) = deltax'
Vertical displacement = deltaL.SIN(phi) = deltay'

and say that the original point A moves these distances to obtain our final position A'.

Hence, if our predicted displacements deltax' and deltay' were correct

AA' = deltaL = L.SIN(phi) = L'.SIN(phi)/COS(phi) = L'.TAN(phi)
deltax' = deltaL.COS(phi)
deltay' = deltaL.SIN(phi)

It follows that the equilibrium condition that we have satisfied is given by (on substituting deltaL = L'.TAN(phi))

$$\frac{K'.TAN(phi)}{L'} = [\ Px.COS(phi) + Py.SIN(phi)\ ]$$

or since L' = L.COS(phi), K' = (L'/L)^2.K = K.COS^2(phi),

$$[\ Px.COS(phi) + Py.SIN(phi)\ ] = \frac{K.SIN(phi)}{L}$$

$$\text{approx} = \left[\frac{K.phi}{L}\right]\left[1 - \frac{phi^2}{6}\right]$$

Hence the proportionate error in the satisfaction of the equilibrium condition is of the order (phi^2/6). Since, for practical purposes, in no case is phi going to be greater than say 1/20, the NL-STRESS error in satisfying equilibrium is not more than about 0.05%

Considering now errors in deflection,

deltax = L.SIN(theta) approx = L.theta (1 - theta^2/6)

dettax' = deltaL.COS(phi) = L.SIN(phi).COS(phi)

$$\text{approx} = L.phi\left[1 - \frac{2.phi^2}{3}\right]$$

Since theta is very close to phi (because of the accuracy of the equilibrium condition), the error in deltax is only of the order (proportionately) of about phi^2/2 i.e. not more than about 0.15% for phi <= 1/20.

deltay = L (1-COS(theta)) approx = L.theta^2/2

deltay' = deltaL.SIN(phi) = L.SIN^2(phi) approx = L.phi^2

Hence application of the 'prediction of next deflection formula' alone would give the error in deltay of the order of 100% for an axially rigid member. For a member of finite axial rigidity, the error will be less than 100%, since the axial contraction due to elastic strain will be calculated accurately.

Thus the changes in the distances between joints at the ends of members, projected on to the original no-load directions, are estimated

$$\text{to be } -\left[\frac{delta^2}{L} + \frac{P.L}{A.E}\right] \text{ instead of } -\left[\frac{delta^2}{2.L} + \frac{P.L}{A.E}\right] \text{ where:}$$

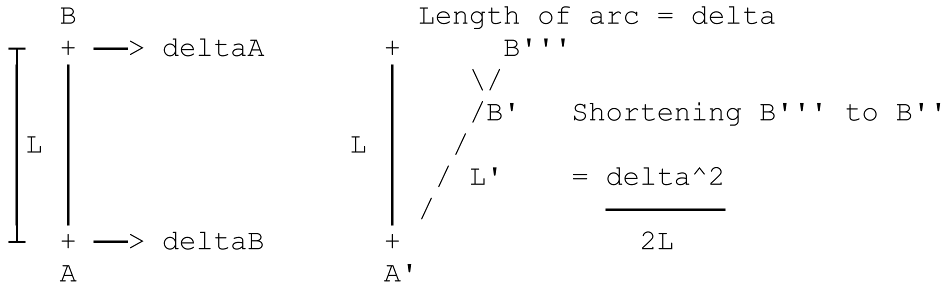
delta is the relative displacement of one end of the member relative to the other end, measured perpendicular to the member's original direction

L is the original length and A.E the axial rigidity of the member
P is the axial compressive force.

Professor Horne has explored two ways of eliminating the error. He first explored step-by-step procedures in which increments of load rather than total loads were entered as the load vector in the matrix equation for the structure in its successive deformed state.

While this satisfactorily corrected the delta^2/2.L error, it proved difficult to maintain satisfaction of equilibrium, working through the linear matrix analysis solution, and allowing correctly for the accumulated changes of geometry.

So how do we instead modify the total load - linear analysis approach to allow fully for compatibility? Actually, the solution is simple once one has seen it.



NL-STRESS estimates (using prediction of next displacement) the displacements deltaA and deltaB of the joints at the ends of a member AB at some stage of loading, so that the new position of AB within the structure is A'B'. We have these estimates for all members in the structure, at the corresponding load level, we conduct a linear analysis of the structure in its displaced geometry, for the full loading condition.

Note that deltaA & deltaB (the predicted displacements of the member) are calculated at displacements of the structure in its new geometry, and these displacements are those perpendicular to the member A'B' in its new position.

Hence, if A'B'' is parallel to AB, the rotation of the member is obtained from delta = deltaA-deltaB. The member will actually change in length due to the axial force induced in it, but ignoring this, the displaced position of AB is A'B' is defined with B''B' = delta at right angles to A'B', where A'B''=L, and hence A'B'=(L^2-delta^2)^0.5. This represents an error in that the new delineation of the member should be A'B''' where A'B''' and B''B''' is the arc of the circle, not a straight line like B''B!

How can we restore all the members to their correct length? To do this in a geometrically compatible way is not however easy, unless we can somehow get our matrix analysis to do it for us!

Now comes the trick to overcome the problem! Our solution (with the structure in its new position corresponding to A'B' for all the members) would be the correct solution for a different problem - i.e. a problem in which we had gremlins to apply compressive forces to all the members, changing (for example) the length of our member A'B''' to its shortened length A'B'.

Now imagine we had these Q forces applied to joints throughout the structure. In order to get back to the solution we want, we must apply equal and opposite forces to our Q forces to all the joints, so that the members can "breathe" themselves back to their correct state.

So we modify our linear matrix analysis by adding loads equal and opposite to the Q forces when we perform the analysis for the loading stage we have reached. We then find an induced tension force of value (say) Q' in our member, this force being the force resulting from the real forces plus the fictitious forces of form $\left[\frac{E.A}{L} . \frac{delta^2}{2.L} \right]$ acting on the complete structure.

When we print out these forces in the member we say that the actual force in member AB has become

- not Q', but Q'-Q.

We should note some important things about the fictitious force Q. It may be that it is very large - larger than the buckling load of the member - sufficient (theoretically) to produce stresses way beyond the elastic limit. But - that does not matter! It is an entirely fictitious force - which our linear analysis will treat without bothering about such considerations. The actual force is (Q'-Q), and where Q is very large, (Q'-Q) may be of quite reasonable value.

Note also that since the members will have their correct lengths (allowing only for changes of length due to actual longitudinal forces), there is no need to fudge their stiffness properties to allow for fictitious changes in length. Thus in our simple example described above, there is not need to have a modified K' not= K.

Let us call the deflections as obtained from applying the 'prediction of next displacement formula' at the nth load increment the 'first order deflections'. These are the deflections which are obtained by linear matrix analysis, assuming that the structure has deflected into the shape given by those deflections applied to the initial undeformed state of the structure. As shown above, these deflections are incompatible with the true deformed state because we are expecting the members to undergo a shortening of $\frac{EA}{2} \left[\frac{delta}{L} \right]^2$ beyond any actual elastic change of length they really undergo. We refer to the matrix analysis performed in the NL-STRESS software to calculate these deflections as the 'basic analysis'.

The additional deformations we apply to the structure in order to correct the incompatibility in the deformed state we term the 'second order deflections', and the matrix analysis performed to introduce these deflections we term the 'second order analysis'. Note that we are now dividing the analysis into the two components. Under the scheme described above the two analyses were carried out simultaneously, but this is of course only one possibility.

We term the actual total applied loads at the nth load increment the 'applied loads'. The fictitious axial loads (symbolised by axial compressive forces Q introduced into the members) we term the 'compatibility forces', and the forces, equal and opposite to the sums of the forces Q, applied to the joints, we term the 'compatibility loading'. The tensile forces Q' induced in the respective members by the compatibility loading we term the 'induced compatibility forces'.

We should note that the induced compatibility forces Q' will be almost equal numerically to the compatibility forces Q, but not quite, the difference between them being due to the small amounts of flexural deformations undergone by the members of the frame in accommodating the corrections to the axial lengths of the members. The flexural deformations referred to are equal and opposite to those parts of the total flexural deformations given by the basic analysis that are ascribable to the incorrectly introduced member shortenings,

$$\frac{E.A}{2}\left[\frac{\text{delta}}{L}\right]^{2}$$

In the simple illustrative model of a rotationally restrained member OA in Fig 1, the real structure follows the curve AD, and by assuming it goes instead to B, we have shortened the members by the amount DB. In the physical structure analysed we imagine these shortenings to have required the introduction of the compressive forces Q, which deformations would themselves be accompanied by the flexural deformations that we are now reversing. (These 'secondary' flexural deformations due to false member shortening are not represented in our simple model in Fig 1 below, because of its consisting of only one member). After the second order analysis, we are required to eliminate the fictitious compressive compatibility forces Q by deducting quantities Q from the tensile 'induced' compatibility forces Q' derived in the second order analysis.

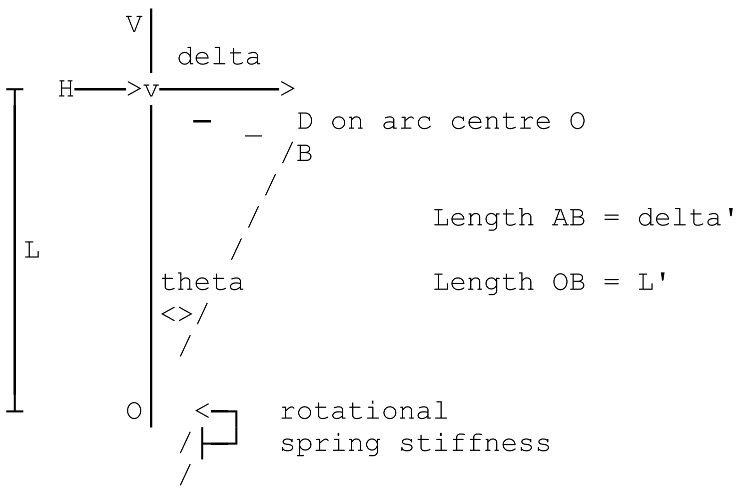
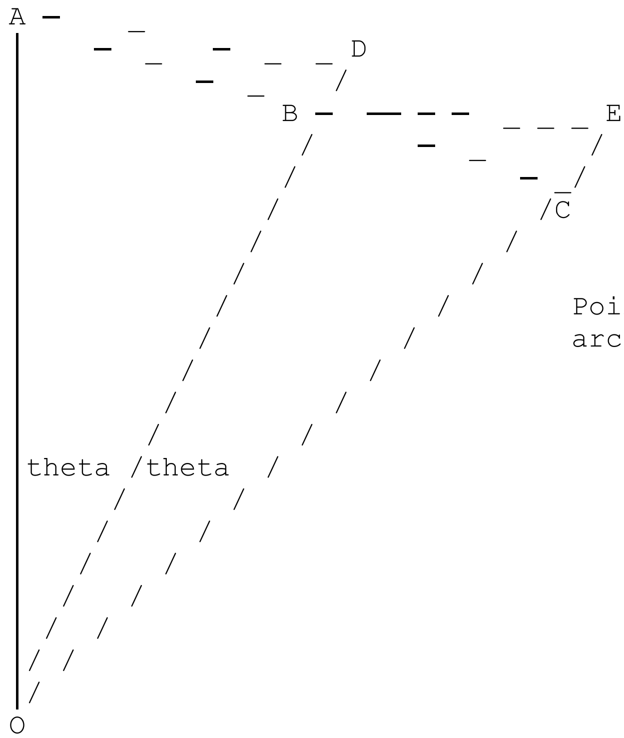


Fig 1



Points A, D, and C lie on an arc centre O

Fig 2

Now let us follow through the analytical procedures, being careful to distinguish exactly how we are satisfying both the compatibility and the equilibrium conditions at the various stages. We will follow what we are doing by reference to the simple elastically rotationally restrained, flexurally rigid member OA in Fig 1. We estimate the deflected state of our member making it lie along the line OD, defined by the rotation theta. The first order deflections are represented by BC in Fig 2, and since these are calculated by linear (small deflection) analysis, the direction BC must be at right angles to the displaced member direction OD. (Note - for simplicity of exposition, the member OA is assumed to be axially rigid with respect to axial forces due to the applied loads V & H, but axially flexible under forces Q.) Taking these deflections now as referring to the deflections undergone from the original state of the member OA, we can see that the top end of the member must move from A to B where AB has the same magnitude and lies in the same direction as BC. Hence the triangles OBA and OBC are mirror images about OB, and the new length of the length of the member has incorrectly been changed from L to L.COS(theta).

What our basic analysis has given us therefore, is the 'real' solution to a structure in which, as we rotate the member OA through the angle theta, we introduce progressively a gradually increasing compatibility force Q applied directly to the member to achieve the necessary shortening. The equilibrium condition satisfied in our basic analysis between the applied forces and the internal forces (apart from the Q forces) is unaffected, since we introduce imaginary Q forces applied directly to each member, exactly in equilibrium with the compressive compatibility forces 'in' those members. We note also that, for a structure in its final deformed condition OB, our linear matrix analysis is giving the correct answer for relationships between applied forces, internal forces and the elastic deformations, because the direction of the incremental deformation at B 'is' correctly at right angles to OB' in our simplified model.

By combining the basic analysis and second order analysis, one gets the displacement vector BE in Fig 2, which we transfer to AD to get the total displacements measured from the no-load position of the structure. Now, the displacement vector for small displacements from the position OD of our member is in the direction AD (=BE), i.e. no longer at right angles to OD. While we will still obtain correct equilibrium conditions if we have induced equilibrium forces that are exactly correct for the compatibility loads that we have applied, equilibrium will not be satisfied otherwise. This in turn requires us to have exactly predicted the correct rotations of the members. Since the compatibility forces can be quite high (in fact, well above the elastic limit compressive loads), we can see that it will be difficult to ensure satisfaction of the equilibrium conditions.

To overcome this difficulty, the current version of NL-STRESS carries out the basic analysis and the second order analysis in one operation, but iterates the calculation of the compatibility forces at each increment of load. However, because the compatibility forces may be large, errors may 'throw' the calculation procedure off course, and the calculations could go wild. If this does happen, an increase in the NUMBER OF INCREMENTS will help.

Refined calculation of compatibility forces:

Referring to Fig 1, we have two methods of obtaining theta.

From $$\frac{delta}{L} = SIN(theta).COS(theta) = \frac{SIN(2.theta)}{2}$$

or $$\frac{delta'}{L'} = TAN(theta)$$

where:
- delta = primary deflection of one end of member relative to the other, parallel to original direction of member
- delta' = primary deflection of one end of member relative to the other, parallel to displaced direction of member
- L = original length
- L' = displaced length

Using delta/L:

SIN(2.theta) = 2.delta/L

$$COS(2.theta) = \left[1 - 4\left[\frac{delta}{L}\right]^2 \right]^{.5} = 1 - 2\left[\frac{delta}{L}\right]^2 + 2\left[\frac{delta}{L}\right]^4$$

= 2.COS^2(theta) - 1

$$COS(theta) = \left[1 - \left[\frac{delta}{L}\right]^2 \left[1 + \left[\frac{delta^2}{L^2}\right]^2 \right] \right]^{.5}$$

$$\text{Hence since } Q = \frac{E.A\ (BD)}{L'} = \frac{EA(1-COS(theta))}{COS(theta)} = EA\left[\frac{1}{COS(theta)} - 1\right]$$

$$Q = EA \left[\frac{1}{\left[1 - \left[\frac{delta}{L} \right]^2 \left[1 + \left[\frac{delta}{L} \right]^2 \right] \right]^{.5}} - 1 \right]$$

$$Q = EA \left[1 + \frac{1}{2} \left[\frac{delta}{L} \right]^2 + \frac{1}{2} \left[\frac{delta}{L} \right]^4 + \frac{3}{8} \left[\frac{delta}{L} \right]^4 - 1 \right]$$

$$\text{i.e.} \quad Q = \frac{EA}{2} \left[\frac{delta}{L} \right]^2 \left[1 + \frac{7}{4} \left[\frac{delta}{L} \right]^2 \right]$$

Using delta'/L', which approach is used by NL-STRESS:

$Q = EA\ [\ \ [\ 1 + TAN^2(theta)\ ]\ ^{.5} - 1\ ]$ and expanding,

$$Q = \frac{EA}{2} \left[\frac{delta'}{L'} \right]^2 \left[1 - \frac{1}{4} \left[\frac{delta'}{L'} \right]^2 + \frac{1}{8} \left[\frac{delta'}{L'} \right]^4 - \frac{5}{64} \left[\frac{delta'}{L'} \right]^6 + \frac{7}{128} \left[\frac{delta'}{L'} \right]^8 \right]$$

7.4 Stability

Two stability effects are of interest in the analysis of frames viz:

7.4.1 Sway stability

Frames sway in the main due to the application of horizontal forces and the horizontal displacement of applied vertical loads produces secondary moments. This effect is generally known as the P-delta effect. At the elastic critical condition a very small increase in loading produces a large horizontal deflection of the frame which loses stability and collapses as a whole.

By carrying out a modified linear analysis as described in 'Finite Displacements', NL-STRESS allows for the stability effect associated with sway.

7.4.2 Within member stability

Engineers are familiar with the bending stiffness of structural members as used in moment distribution. Euler showed that under a critical axial load Pe, the bending stiffness of a structural member was zero and the member buckled for a very small increase in loading. The effect of buckling of the individual members of a structural frame is known as 'within member stability'.

There is a fundamental difficulty in allowing for within member stability merely by modification of the stiffnesses at end of members (i.e. in relation to rotation). This is because, not only does the stiffness change, but also the 'carry-over' factor (which is 0.5 for a uniform member in the absence of axial thrust). Since the carry-over factor of 0.5 is fundamentally part of any standard linear analysis software, one cannot allow at all sensibly for member stability merely by modifying the stiffness of the member.

However there is a way out - by adding a joint at the mid-length of each member. If the mid-point is treated as a node in the analysis (so that the member becomes two members end to end) and allowance is made for change of position of the node (as is done for the other nodes) then the prime cause of instability is allowed for.

The adjustment is not quite correct for only one internal node at the mid-point of the member, but may be made nearly exact by internal nodes at quarter points which will allow for both single and double curvature bending.

In NL-STRESS additional nodes internal to each member are introduced by the NUMBER OF SEGMENTS command. If the number of segments is set to 2, then the member is divided into two segments by the addition of one internal node at mid-point. If the number of segments is set to 4, then each member is divided into four segments by the addition of three internal nodes at quarter points. These additional internal nodes are normally transparent; if you wish to examine the displacement or forces at these internal nodes you should add the word TRACE to the end of the NUMBER OF SEGMENTS command.

Just as there is a reduction of bending stiffness for members in compression, so there is an increase in bending stiffness for members in tension. Again the addition of 3 or more internal nodes will allow for the increase for members subjected to both single and double curvature bending.

7.5 Elastic-plastic analysis

Plastic design of a frame, formerly known as the Collapse Method of design but changed to avoid alarming the architect:

- increases the working load by a factor (1.4 for DL, 1.6 for Live)
- applies this factored loading to the frame
- introduces plastic hinges in the frame at positions where the plastic bending moment capacity is exceeded
- checks to make sure that the frame does not collapse as a mechanism for member sizes selected

Section 7.5.1 describes how this procedure may be carried out using a linear elastic analysis. Section 7.5.2 describes the elastic-plastic method of NL-STRESS.

7.5.1 Elastic-plastic analysis using linear elastic software

An elastic-plastic analysis may be carried out using standard linear analysis software by the following procedure:

a) Carry out linear analysis at given load level

b) At certain positions the plastic moment Mp (reduced as necessary for effects of axial load) will have been exceeded. Suppose that at a particular position it is M > Mp.

c) Repeat the analysis (as a new linear analysis) but introduce a pin at the position where M > Mp (or positions where this applies). At the same time, introduce at the pin equal and opposite pairs of moments Mp as additional external loads. In this second analysis include the modified geometry defined by the deflections from the first analysis - this will allow for non linear elastic effects.

d) Repeat c) if further hinges appear.

Of course, if a load level above the collapse load is chosen, there will not be convergence to a solution; more and more hinges popping up until the structure becomes a mechanism. The mechanism tells you that you are above the collapse load. NL-STRESS uses a step-by-step increase in the loading. The number of steps is set by the NUMBER OF INCREMENTS command. After each increment has been added NL-STRESS goes through the above procedure automatically.

7.5.2 Elastic-plastic analysis of NL-STRESS

NL-STRESS automates the elastic-plastic method of analysis described in Section 7.5.1.

To carry out an elastic plastic analysis in NL-STRESS it is necessary that the engineer:

- specifies METHOD PLASTIC (see the method command in the User's Manual).
- specifies NUMBER OF INCREMENTS as typically between 10 and 50. If 20 is used and the loading is factored by 2.0, then each increment puts a tenth of the working load on the structure. Thus results of the tenth increment correspond to the working load condition (10 x 0.1 = 1.0) and it would be expected that at least 17 increments of loading are sustained before the structure fails as a mechanism. (Seventeen increments of loading corresponds to a collapse load factor of 17 x 0.1 = 1.7.)

For all versions of NL-STRESS prior to 2.2: after each increment of loading was applied to the structure, NL-STRESS located each member/ segment end which would be plastic under the next loading increment, and released the member/segment end by applying equal and opposite plastic moments about the release (the values of the plastic moments being computed from Professor Horne's interaction equations). This treatment worked well when no more that one new plastic hinge appeared within a loading increment, but when more than one hinge appeared it was apparent that for a small number of structures, had the first hinge been inserted in isolation then the second one would not have formed. This phenomenon is usually referred to as the 'false mechanism' problem.

NL-STRESS now prevents false mechanisms forming by adding only one new hinge at a time in any loading increment. This 'adding one new hinge at a time' is implemented in plane frames, grids, and space frames.

There is yet another phenomenon that can occur in plastic analysis i.e. unloading plastic hinges. This is a rare phenomenon, but occasionally because of plastic hinges developing in one member of a structure, plastic hinges in another part start to reverse i.e. unload. NL-STRESS models the effect for plane frames (grids do not normally have the problem). The theory (Professor Horne): when a plastic hinge of value Mp starts to reverse, having reached a hinge angle i, replace the pin by a rotational spring of stiffness b =

100.Mp/i, and introduce equal and opposite external moments of value 99.Mp.

6.5.3 Elastic-plastic analysis of compression members

In rigid-frame (non-triangulated) structures, bending moments arise as 'primary moments', whereas in rigid-jointed triangulated frames, loaded only at the joints, any bending moments arise only as 'secondary moments'. Results obtained from 'unit shape factor analysis', in which plastic deformation is assumed to be confined to discrete plastic 'hinges', have been found, both analytically and experimentally, to be of fully sufficient accuracy for rigid frames. For triangulated frames, in which the structural action of the members lies essentially in their resistance to axial forces, unit shape factor analysis may give results which are not sufficiently reliable. The reasons are as follows.

a) The presence of initial imperfections and residual stresses has an appreciable influence on the carrying capacities of members loaded in axial compression, whereas for members loaded primarily in bending (even when some compressive axial loading is also present) their effect explored analytically is small, and certainly overshadowed by the many variable effects due to other factors which arise in practice and experimentally.

b) Compression members may, at their maximum capacity load, have attained a state of only limited plasticity, without the formation anywhere of a plastic hinge. This is because partially penetrating plastic zones have so reduced the stiffness of the members of a frame that a state of critical buckling has been reached.

Structural analyses which take account of the spread of plastic zones, allowing for the incidence of partial plasticity, have been developed and widely used, although mainly for purposes of research, and then mainly in relation to the behaviour of discrete members. When used for the analysis of complete frames, the demands made on memory capacity soon become enormous. Even such software, applied to the analysis of triangulated rigid-joints frames, will not however give acceptable results unless due allowance is made for the effects of imperfections and residual stresses.

The essential step in using NL-STRESS to make possible the reliable analysis of triangulated frames is therefore to explore how such an allowance can best be made. Since imperfections and residual stresses are variable and for the purpose of analysis unknown, it cannot be claimed for any form of analysis that it will give a uniquely 'correct' result. Nevertheless, the design of members that may be subject to the effects of instability and thereby to varying degrees sensitive to the effects of imperfections and residual stresses is covered in codes of practice, and is moreover so covered as to represent the 'wisdom' gathered from much experience and from many investigations including test results. In the majority of structural codes, the capacity of members in compression is expressed by formulae based on the theoretical attainment of the limiting elastic stress in an eccentrically loaded strut or strut with an initial out-of- straightness. However, the level of eccentricity or lack of straightness is chosen empirically to give a statistically justified allowance for the effects of initial imperfections and residual stresses on the 'collapse load' and not on the load at which yield is first reached. The use of formulae based theoretically on elastic behaviour (such as the Perry-Robertson formula used, in a modified form, in BS5950) is merely a convenience. It would have been equally legitimate to have used, as a means of expressing the load capacity of members in compression, formulae based on the attainment of full plasticity in initially eccentrically loaded or curved members, provided a suitable adjustment is made in the values chosen for the degrees of eccentricity or lack of straightness.

A suitable means of using NL-STRESS to deal with compression members is therefore to introduce, into the members of the frame, imperfections of such magnitude that the knowledge that has been accumulated over the years, and which is represented in the requirements of structural codes in relation to the failure loads of compression members in practice, is exploited. An analysis for the derivation of such geometrical imperfections, using the requirements for compression members in BS5950, is represented in section 7.5.4 below.

If the engineer introduces geometric imperfection of the magnitudes so derived, NL-STRESS will, using unit form factor analysis, predict the deformation of plastic hinges at earlier stages in the analysis than would be the case without the introduction of those imperfections, thereby making appropriate allowance for the spread of plasticity as well as for the imperfections and residual stresses actually present in practice. The fact that allowance is made for all these effects follows because the magnitudes of the imperfections have been derived from code requirements (although in the code itself, only for single, pin-ended members) which themselves effectively do so.

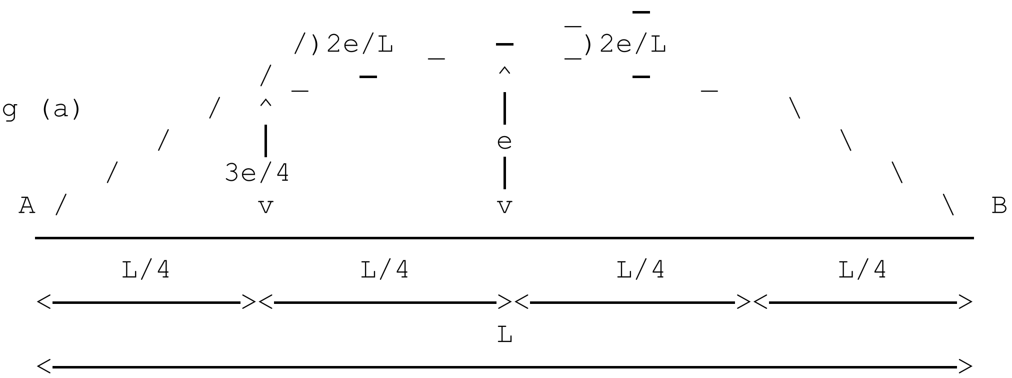
It is proposed that for the sake of convenience, the recommended imperfections be introduced for all members, in all types of frames. The introduction of imperfections into the tension members of triangulated frames will have a completely negligible effect. For members subjected to primary bending moments and not carrying axial compressive forces that are at all comparable with their capacities as pure compression members (as is common in rigid-jointed sway frames), the effect of the introduced imperfections will be small. For members where both the lateral loads and the axial compressive forces are significant as causes of failure (in, for example, 'beam-columns'), the introduction of these same imperfections will have an appropriate effect.

7.5.4 Derivation of member imperfection values for use in analysis

Angular discontinuities should be introduced at all internal nodal points of any member, adopting a parabolic distribution of nodal deflections relative to the ends of the member.

Initially assumed deflected forms for three internal nodes would therefore be as shown in Fig (a)

Fig (a)

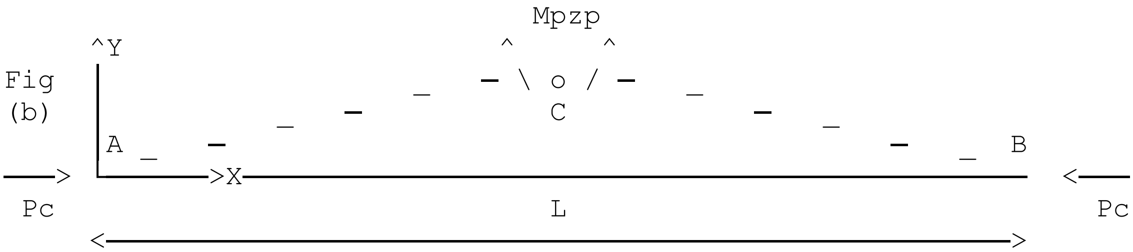


In order to derive suitable values for the initial displacement e, the collapse loads as obtained by applying a unit form-factor elastic-plastic analysis to members treated as pin-ended struts are equated to the collapse loads of nominally straight members as given by the strut formulae in BS5950.

With a large number of nodes, the initial shape approaches a smooth parabola with central deflection e_∞ (where ∞ = infinity).

A convenient analysis is obtained if the initial shape is assumed to be sinusoidal, this being a close approximation to a parabola. The state of the strut at the point of collapse is as shown in Fig (b).

Fig (b)



At the point of collapse, the effect of the axial thrust Pc = A.pc is to have increased the central deflection at C (Fig b) from the initial vale e_∞ to a value y given by:

$$y = \frac{Mpzp}{Pc} = \frac{e_\infty}{1 - pc/pe}$$

Hence it is found that:

$$\frac{e_\infty}{L} = \frac{1}{lambda} \cdot \frac{Spz}{A.rz} \cdot \frac{Spzp}{Spz} \cdot \frac{py.(pe - pc)}{pc.pe}$$

Derived values of imperfections

The values e_∞/L for members of hollow circular section, and of rolled Universal Beams (254 x 102 at 22 kg/m) are given in column 2 of Tables 1, 2 and 3 respectively. Typically, it has been assumed that py=350 N/mm² and E=205E3 N/mm².

TABLE 1

1	2	3
lambda	10^3*e∞ / L	10^3*e∞ / L-Lo
0	-	-
15.2	0	-
30	1.40	2.83
50	1.96	2.81
80	2.19	2.70
100	2.14	2.53
140	2.00	2.25
200	1.88	2.03
300	1.81	1.90

ASSUMED IMPERFECTIONS - HOLLOW CIRCULAR SECTION

TABLE 2

1	2	3
lambda	10^3*e∞ / L	10^3*e∞ / L-Lo
0	-	-
15.2	0	-
30	1.25	2.54
50	1.76	2.52
80	2.02	2.49
100	2.08	2.46
140	2.03	2.28
200	1.92	2.08
300	1.85	1.95

ASSUMED IMPERFECTIONS - ISECTION ABOUT MAJOR AXIS

TABLE 3

1	2	3
lambda	10^3*e∞ / L	10^3*e∞ / L-Lo
0	-	-
15.2	0	-
30	3.85	7.80
50	4.86	6.98
80	4.08	5.04
100	3.27	3.86
140	2.64	2.96
200	2.37	2.56
300	2.27	2.39

ASSUMED IMPERFECTIONS - ISECTION ABOUT MINOR AXIS

Since in BS5950, imperfections are assumed to be zero for members of slenderness less than lambda0 = 0.2.π(E/py)^0.5, values of e/L become zero for lambda<= lambda0. Hence in column 3 of Tables 1,2 and 3 are shown values of e/(L-L0) where L0=lambda0.rz=0.2.π.rz.(E/py)^0.5.

Except for I-sections bent about the minor axis, the values of e/(L-L0) do not vary greatly. The greatest effects of imperfections occur for slendernesses for which pe=py, i.e. for the struts of Tables 1,2 and 3, when lambda approx=75. At values of lambda greater than about twice this limit, the effects become proportionally much smaller, while for low values of lambda, pc in any case approaches or even reaches py.

The values of e/(L-L0) are necessarily closely related to the values of the "Robertson constant" a. Considering all these factors, and the results given in Tables 1,2 and 3, it is recommended that for all members in an NL-STRESS elastic-plastic analysis, there should be introduced an initial mid-point lateral deflection of approximately e=1.4a(L-L0)/1000 where a is the Robertson constant in BS 5950 and L0=0.2.π.rz.(E/py)^0.5. Imperfections at nodal points should be on a parabola. A minimum of four segments per member is recommended.

Professor M R Horne has done a few outline calculations on the effect of having only one intermediate node in a compression member and has concluded that it would be unsatisfactory to rely on a single intermediate node as inconsistent results can be obtained at high slendernesses.

Since it cannot be claimed that the imperfection allowance is other than empirical and inevitably approximate, there is no point in choosing other than rounded values in the expression for e. The finally recommended formula is:

e = an(L - 0.6.r.(E/py)^0.5)/1000

where 'an' depends on the type of cross-section and axis of bending and r is the radius of gyration about that axis. The classification of cross-sections is as in Table 25 of BS5950, according to the BS5950 strut Table to which reference is there made, as follows.

Strut Table	an
27(a)	3
27(b)	5
27(c)	8
27(d)	12

It may be noted that the maximum proportion by which compressive stress capacity for an=3 exceeds that for an=12 (for the same slenderness) is as much as 40%, indicating the importance of including variable values of 'an' according to the type of member.

Incorporation of imperfections:

In plane frames in which it can be assumed that members will not buckle out-of-plane, only in-plane imperfections need be postulated. However, if there is any possibility of member failure by buckling out-of-plane, it is necessary to assume imperfections about both axes. In space frames, imperfections should be introduced about both principal axes.

The question of what imperfection to take for space frames requires careful consideration. Obviously, if one had a member pin-ended about both axes at both ends, one ought to take 0.707e about each of the two axes. To take e about each axis, giving a total imperfection of

1.414e, would result in a maximum underestimate of strength of the order of up to about 10% (such an underestimate being possible for a compression member of slenderness of the order of:

$$\lambda = \pi \left[\frac{E}{py} \right]^{0.5}$$

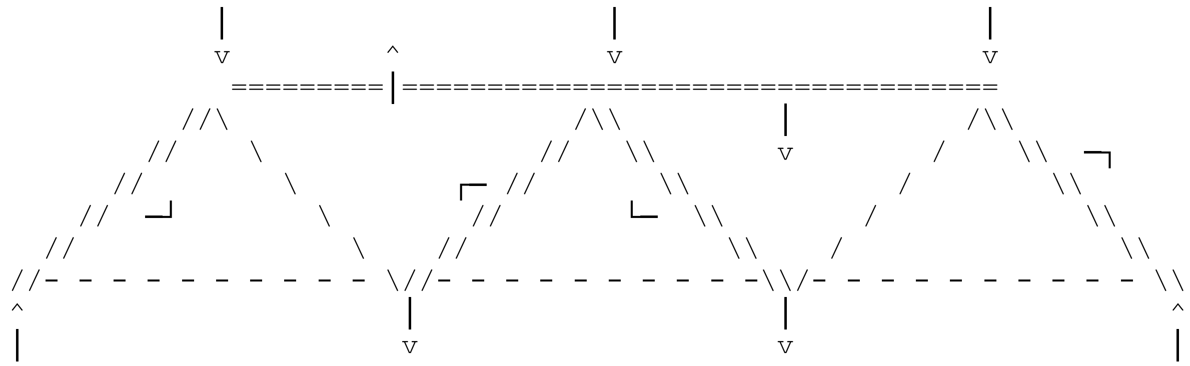
with smaller underestimates for lower and higher slendernesses). However if the member were not pin-ended about both axes, and unless it had degrees of restraint at the ends almost identical about both respective axes, it would fail predominantly about one axis or the other, so that the effect of adopting an imperfection of 0.707e about each axis could result in an overestimate of strength, this overestimate having a maximum possible value in unfavourable circumstances again of the order of 10%. On the other hand, with even slightly differing degrees of restraint and/or imposed terminal end moments or transverse loading conditions, the distinct tendency to fail about the most unfavourable axis would mean that, by adopting an imperfection about each axis of e, the cases in which one would be likely to make any significant underestimate of strength would be fairly few. Hence take e about each axis.

For members which have one radius of gyration significantly less than that at right-angles (e.g. a rolled I-section), one should for simplicity adopt the respective e value about each axis, using the correct respective lengths between points of effective displacement support. The respective length may of course differ for the two axes.

For members neither of whose principal axes coincide with local frame axes (such as is normally the case for members of angle section) one should use the appropriate e values calculated relative to the member's principal axes, not as would be calculated using local frame axes.

The directions of imperfections should be chosen so that the tendency towards buckling of two compression members meeting at a point will tend to rotate that joint in the same direction. This is illustrated by reference to a simple Warren truss in Fig (c), where the members in compression are shown by double lines. Out-of-plane imperfections of a plane truss should all be in the same direction.

It should also be noted that, for slender members in bending, even when there is little axial load, it is important to include the lateral imperfections so that NL-STRESS will be enabled to include the possible effects of lateral-torsional buckling.



Direction of imperfections in compression members shown by arrows
(Directions immaterial for tension members)

Fig (c)

7.5.5 Rotations at plastic hinges

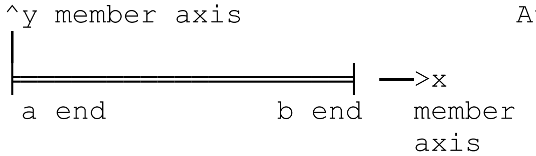
As stated in section 7.5.1, NL-STRESS models a plastic hinge by introducing a pin at the end of a member or segment, with equal and opposite pairs of moments Mp as additional external loads. The plastic rotation at the end of the member or segment is not directly available as the computed joint rotations are for the ends of members or segments which are unreleased at the joint. To compute the rotation at the end of a member or segment there are two possible approaches:

- knowing the end moments and positions, apply all the member loads to the member and thereby compute the end rotation/s
- factor up the the end forces and displacements for the first load case (in which no plastic hinges can occur) and modify the resulting known member forces and displacements for the change in moments and displacements due to the plasticity thus finding the end rotations at any plastic joint.

The second approach is more computationally efficient though it needs more theory which now follows.

7.5.5.1 Carry over factor

The carry over factor for rotation at end a (pinned) caused by rotation at end b is of course 0.5 when shear deformation is ignored. NL-STRESS takes shear deformation into account and the carry over factor is now derived.



At the end of the first increment let
da be y displacement at a end
ra .. z rotation at a end
db be y displacement at b end
rb .. z rotation at b end
ma .. moment at a end

Let 'n' be the current increment number, then for no plasticity at either end of the member or segment the final displacements and rotations will be:

n.da	y displacement at a end
n.ra	z rotation at a end
n.db	y displacement at b end
n.rb	z rotation at b end
n.ma	moment at a end

After the n'th increment we know

Da	y displacement at a end
Db	y displacement at b end
Ma	moment at a

and if no hinge at end b

Rb	z rotation at b end

Writing shear constant $S = \frac{3EI}{L} \left[\frac{1}{1 + \frac{12EI}{L^2.G.Ay}} \right]$ for 'b' continuous.

and section constant $T = EI/L$ then from 7.7.1.4.

Moment at 'a' due to rotation at b: $Ma = (-T + S).(Rb - n.rb)$

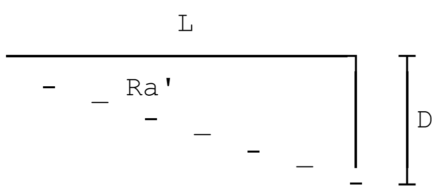
Rotation at a due to moment at a: $Raa = \frac{Ma}{T + S}$ again from 7.7.1.4.

Rotation at a due to rotation at b: $Rab = c .(Rb - n.rb)$

Where carry over factor for rotation: $c = \frac{-T + S}{T + S}$

7.5.5.2 Rotation at 'a' due to y displacement at 'b'

A plastic joint may form at one or both of the ends so we must modify the n.da, n.ra etc. to take into account the changes in end moments and displacements. It is simpler to consider the modification in stages rather than all together. Sections 7.5.5.3 and 7.5.5.4 cover for the case of a plastic hinge at 'a' only, and plastic plastic hinges at 'a' and 'b'. For both cases, changes to rotation at 'a' caused by displacement at 'b' are common and now given.



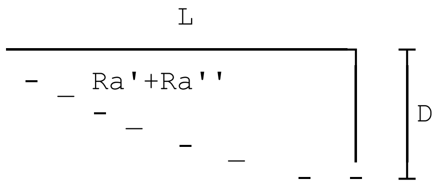
D = (Db-n.db) - (Da-n.da)

Changes in displacement:
At a end = Da - n.da
b = Db - n.db

$$Ra' = \frac{(Db-n.db) - (Da-n.da)}{L}$$

Where Ra' = change in rotation at 'a' due to rotation of the member caused by the plasticity. Now this rotation of the member causes a rotation of the end b = Ra' which in turn induces a rotation at 'a'.

Ra'' = c.Ra' where c is carry over factor for rotation given in 7.5.5.1.



Thus combined rotation at 'a' due to displacement 'D' with end 'b' fixed against rotation:

Ra' + Ra'' = Ra'.(1 + c)

7.5.5.3 Plastic hinge at end 'a' only

After load increment 'n' has been applied we know the rotation at 'a' = n.ra to which we need to add the rotational increases at 'a' due to:

- change in y displacement with all else suppressed (see 7.5.5.1 above)
- change in rotation at 'b' with all else suppressed
- change in moment at 'a' with all else suppressed.

As 'b' remains attached to the joint then the end rotation 'Rb' of the member is known. Thus: Ra'''=-c.(Rb-n.rb) where Ra''' is the rotation at 'a' due to the change in rotation at 'b'. Change in sign is due to the fact that a positive rotation at 'b' will cause a negative rotation at 'a'.

Finally we must add the change in rotation at 'a' due to the change in moment at 'a' with all else suppressed. Again from 7.7.1.4,

rotation at 'a' due to change in moment at 'a': $$Ra'''' = \frac{Ma - n.ma}{T + S}$$

Combining the rotation components then change in rotation at 'a' given

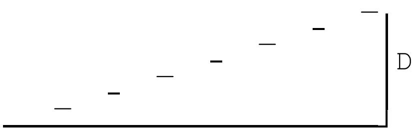
by: $$Ra = \left[\frac{(Db - n.db) - (Da - n.da)}{L} \right].(1+c) - c.(Rb - n.rb) + \left[\frac{Ma - n.ma}{T + S} \right]$$

Thus final rotation at 'a': Raf = R2 - n.ra - Ra

7.5.5.4 Plastic hinge at ends 'a' and 'b'

After load increment 'n' has been applied we know the rotation at 'a' = n.ra to which we need to add the rotational increases at 'a' due to:

- change in y displacement with member having pins at both ends
- change in moment at 'b' with 'a' pinned
- change in moment at 'a'.



L

D = (Db-n.db) - (Da-n.da)

For a member having pins at both ends, a positive displacement D causes a positive rotation at 'a':

$$Ra' = \frac{D}{L} = \frac{(Db-n.db) - (da-n.da)}{L}$$

Ra'' = c.Rb' where c is carry over factor for rotation given in 7.5.5.1.

End b has a plastic hinge of value Mb which causes a rotation at 'b' given by:

$$Rb' = \frac{Mb-n.mb}{\dfrac{3EI}{L}\left[\dfrac{1}{1+\dfrac{3EI}{L^2.G.Ay}}\right]}$$

see 7.7.1.1. Writing S for the shear factor i.e. the divisor, and substituting in Ra' = c.Rb' gives:

$$Ra'' = \frac{c.(Mb-n.mb)}{S}$$ where S is the shear factor for pinned end at 'b'.

Rotation at 'a' due to change in moment at 'a':

$$Ra''' = \frac{Ma-n.ma}{\dfrac{3EI}{L}\left[\dfrac{1}{1+\dfrac{3EI}{L^2.G.Ay}}\right]}$$

see 7.7.1.1. Writing S for the shear factor as before

$$Ra''' = \frac{Ma-n.ma}{S}$$

Combining the rotation components then rotation at end a =

$$Ra = \left[\frac{(Db-n.db) - (Da-n.da)}{L}\right] + \frac{c.(Mb-n.mb)}{S} + \frac{Ma-n.ma}{S}$$

Thus final rotation at 'a': Raf = R2 - n.ra - Ra

7.5.6 Dealing with unloading hinges

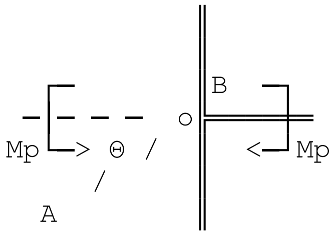
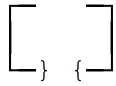


Fig (1) Load lambda(1)

Applied moments denoted thus:

Internal moments denoted thus: 

Suppose that, as analysed at load level lambda(1), there is a plastic hinge at the end B of member AB, with hinge rotation Θ.

We have simulated the plastic hinge by introducing a pin in place of the rigid continuity in the original structure, plus EXTERNAL applied moments Mp.

Suppose that, at load level lambda(1), we detect hinge reversal; we now replace the pine at B by a rotational spring of stiffness k, see Fig.2. This simulates the loaded structure, again at load level lambda(1).

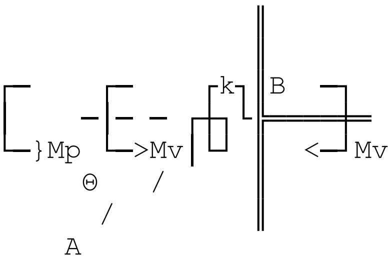
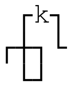


Fig (2) Load lambda(1)

Spring denoted by: 

The plastic moment Mp now becomes and INTERNAL moment, and we must add a VIRTUAL (external) moment Mv to achieve the built-in-hinge angle Θ. The total; moment acting on the spring is this Mv + Mp, so that

$$Mv + Mp = k.\Theta$$

To analyse for load level lambda(2) > lambda(1), we now operate on the unloaded, initially unstressed structure (built-in rotation Θ cancelled) shown in Fig (3)

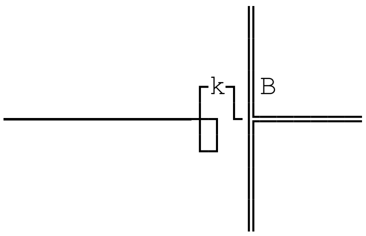


Fig (3)

lambda=0

We know that, when we load the structure at load level lambda(1) > lambda(2), the internal moment at B in member AB will be some value M < Mp (Fig 4).

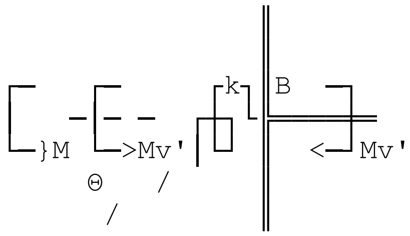


Fig (4)

We would like to retain EXACTLY the built-in rotation Θ. To achieve this, we COULD apply external virtual moments Mv' as required by the condition:

$$Mv' + M = k.\Theta$$

Since we do not know M until we have completed the analysis, suppose we use the SAME virtual moments Mv. The built-in rotation will now be Θ' where:

$$Mv + M = k.\Theta'$$

The proportionate error e in the built-in plastic discontinuity will therefore be:

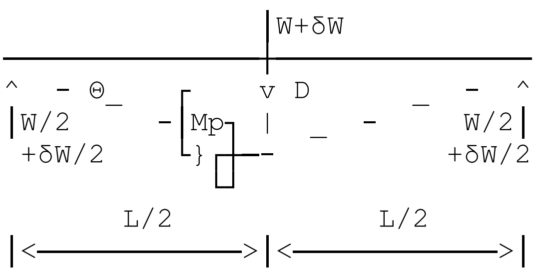
$$e = \frac{\Theta - \Theta'}{\Theta} = \frac{Mp - M}{Mv + Mp} = \frac{1 + M/Mp}{1 + s} \quad \text{where } Mv = s.Mp \text{ and } b = \frac{(1+ s).Mp}{\Theta}$$

Professor Horne conjectures that (1-M/Mp) is unlikely to exceed about 1/3 in value. Hence, if we make s=99, k= 100Mp/Θ, the error e will not be greater than 1/300.

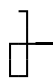
In summary: when a plastic hinge of value Mp starts to reverse, having reached a hinge angle Θ, replace the pin by a rotational spring of stiffness b = 100.Mp/Θ, and introduce equal and opposite external moments of value 99.Mp.

7.5.6.1 Carrying loads on pseudo mechanisms

Unloading of plastic hinges is usually preceded by a local mechanism; example PL05 in this manual gives an example. In order to be able to get past the local mechanism stage, some stiffness has to be introduced at plastic hinges. The plastic hinge model of a pin at the end of a member/segment is replaced by a weak member end spring with equal and opposite moments applied on either side of the spring.



The beam is in equilibrium with the load W and plastic hinge Mp = W.L/4 when extra load δW is added.

 denotes a spring

Moment carried by spring $\delta M = \frac{\delta W.L}{4} = \frac{W.L}{4.N}$ kNm

Assuming a spring stiffness of K kNm/rad, then rotation across the

$$\text{spring } \Theta = \frac{\delta M}{K} = \frac{W.L}{4.N.K} = \frac{Mp}{N.K} \qquad \text{Rearranging} \quad K = \frac{Mp}{N.\Theta}$$

assuming a maximum acceptable spring rotation of say 0.25 rad, then:

$$K = \frac{4.Mp}{N}. \quad \text{Therefore use a WEAK spring} = \frac{4 * \text{Plastic moment}}{\text{Number of loading increments}}$$

7.6 Interaction formulae

7.6.1 General formulae

For members subjected to bending only, the introduction of a plastic hinge requires only the check of M > Mp. For most structures members will be subjected to axial load (either tension or compression) in addition to bending, and the introduction of a plastic hinge is made on the result of applying an interaction formula appropriate to the section shape and material of the member.

Interaction formulae involve a summed sequence of ratios equated to unity. If the sequence sums to a value greater than unity the formula "fails" and a plastic hinge is assumed to form.

In the general case of a member in a space frame, the member will be subjected to bending about both principal axes, axial load and axial torque and shear forces in the direction of the two principal axes. The shears are ignored; interaction formulae taking into account the remaining force components.

Each ratio is a ratio of applied force to corresponding limiting force. The word "force" is used in a general sense to mean axial load, torque or bending moment. So the possible ratios are Fx/Fxp, Mx/Mxp, My/Myp, Mz/Mzp. (The suffix "p" is for "plastic limit"; x,y,z denote local member axes with x axis going along the member; F denotes force; M denotes moment.)

The interaction formula is so named because it measures the interaction of different effects, each contributing to the formation of a plastic hinge; the more torque present the less bending moment can be carried, and so on.

Sections are to be limited to "thick-walled" ones - i.e. those which the codes permit to be treated as capable of developing plastic properties without undergoing local plate or wall buckling (the word "compact" has been used).

Notation:

- Fx denotes axial load
- Fxp denotes squash load (area x yield stress)
- Mx denotes torque (moment about local x axis)
- Mxp denotes plastic capacity about local x axis in absence of all other loads
- My denotes bending moment about local y axis
- Myp denotes plastic capacity about local y axis in absence of all other loads
- Mz denotes bending moment about local z axis
- Mzp denotes plastic capacity about local z axis in absence

of all other loads

Whereas Fx, My, Mz, all produce direct stress on the cross-section, Mx produces shear stress. At plastic limit, combination of direct stress sigma'p with shear stress tau'p at any point on cross-section to produce plasticity is given by:

sigma'p^2 + 4.tau'p^2 = f^2

where f is yield stress in tension or compression.
Now the plastic limit under torsion only (Mxp) is reached when we have a limiting shear stress over the whole cross-section of:

tau'p = f/2

If instead of Mxp we have a lesser torsional moment of Mx, then this torque can be resisted by a shear stress distributed over the cross-section in the same way as tau'p = f/2, but with a reduced shear stress of tau'p =(Mx/Mxp).tau'p = Mx.f/(Mxp.2)

Hence the simultaneous effective direct yield stress sigma'p (tensile or compressive) will be given by:

sigma'p = SQR(f^2 -4.tau'p^2) = f.SQR(1- (Mx/Mxp)^2)

where sigma'p gives the effective yield stress in tension or compression.

Under Fx and Mx alone; effective value of Fxp is $\overline{Fxp}$ where:

$$\overline{Fxp} = Fxp.\ SQR(1 - (Mx/Mxp)^2)$$

Under My and Mx alone; effective value of Myp is $\overline{Myp}$ where:

$$\overline{Myp} = Myp.\ SQR(1 - (Mx/Mxp)^2)$$

Under Mz and Mx alone; effective value of Mzp is $\overline{Mzp}$ where:

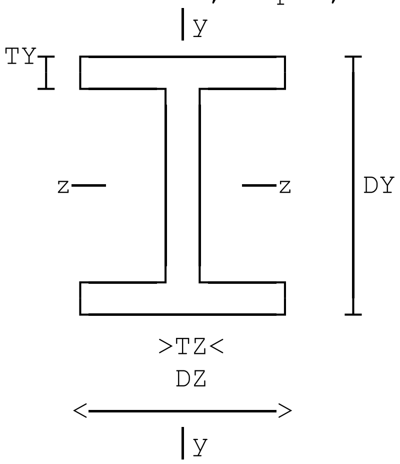
$$\overline{Mzp} = Mzp.\ SQR(1 - (Mx/Mxp)^2)$$

For cross sections having only one axis of symmetry - a T-section in NL-STRESS - the simplest (and conservative) interaction formula is employed:

$$\frac{Fx}{Fxp} + \frac{Mx}{Mxp} + \frac{My}{Myp} + \frac{Mz}{Mzp} = 1$$

For I-sections, NL-STRESS goes through a more complicated procedure as follows:

Compute the ratio, alpha, of web area to cross sectional area:



$$\text{alpha} = \frac{DY.TZ}{DY.TZ + 2(DZ-TZ)TY}$$

Compute the twist factor, t from: $t = SQR(1-(Mx/Mxp)^2)$

Compute the normalised axial effect, n, from: $n = \frac{1}{t} \frac{Fx}{Fxp}$

Depending upon the relative magnitudes of alpha and n, find a formula for Mpzp:

0 < n <= alpha ... Mpzp = t.Mzp(1-n^2/(alpha(2-alpha)))

alpha < n <=1 ... Mpzp = t.Mzp(2(1-n)/(2-alpha))

and a formula for Mpyp:

0 < n <= alpha ... Mpyp = t.Myp

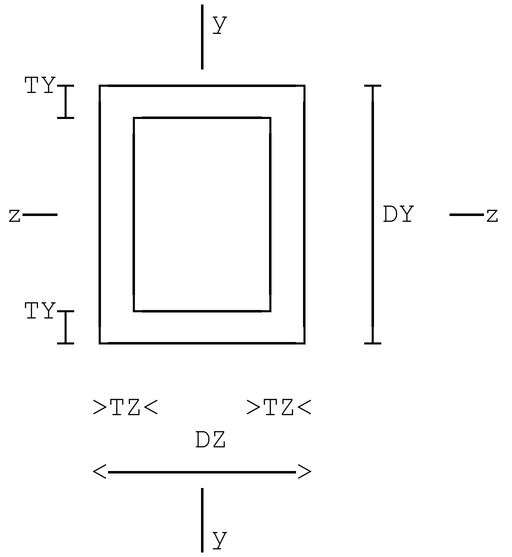
alpha < n <= 1 ... Mpyp = t.Myp((1-2alpha+n)(1-n)/(1-alpha)^2)

Then with Mpzp and Mpyp use the following interaction formula:

$$\left[\frac{Mz}{Mpzp} \right]^2 + \frac{My}{Mpyp} = 1$$

H-sections are treated the same way but with axes y and z effectively interchanged.

For rectangular sections the software adopts a procedure similar to that for I and H-sections:



$$alpha1 = \frac{DY.TZ}{(DY.TZ + DZ.TY)}$$

$$alpha2 = \frac{DZ.TY}{(DY.TZ + DZ.TY)}$$

Compute Mpzp as for an I-section but with alpha1 in place of alpha.

Compute Mpyp as though it were Mpzp for an I-section but with alpha2 in place of alpha.

These values for Mpzp and Mpyp are used in the following interaction formula:

$$\left[\frac{Mz}{Mpzp}\right]^{5/3} + \left[\frac{My}{Mpyp}\right]^{5/3} = 1$$

For circular sections the software uses the following relationship as though it were an interaction formula:

$$\frac{(Mz^2 + My^2)^{0.5}}{t.Mp.COS(n.\pi/2)} = 1$$

$$\text{where } t = SQR\left[1 - \left[\frac{Mx}{Mxp}\right]^2\right] \qquad \text{and} \quad n = \frac{1}{t} . \frac{Fx}{Fxp}$$

In the absence of special treatment, the linear formula given at the start of this section is conservative.

In using these formula we assume that no twisting occurs. Slender I—sections may buckle by combined twisting and lateral bending ("lateral torsional buckling"). Angle and channel sections will, except when bent about the axis perpendicular to the axis of symmetry, always tend to twist unless stocky, or restrained from twisting (e.g. by attachment to sheeting).

NL-STRESS makes sure that each joint has at least one member or segment connected to it without a plastic hinge. This is to provide some stiffness to the joint to prevent it from spinning. If for example four members meet at a joint and application of the interaction equations shows that all four members go plastic, then NL-STRESS simply keeps the member which had the lowest unity value connected and inserts hinges at the ends of the other three members which meet at the joint.

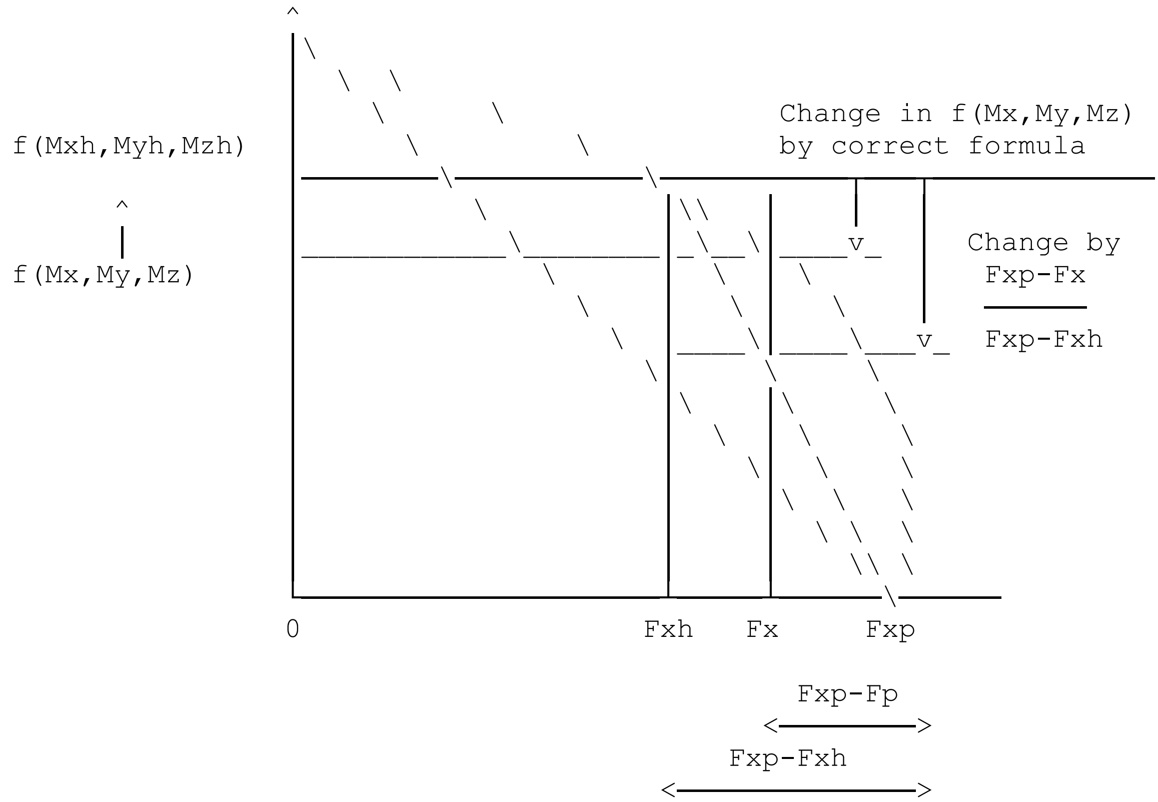
After a hinge has formed - as predicted by the terms of the relevant interaction formulae summing to greater than unity; then as further load is applied the bending stiffness will decline and in consequence the plastic moments Mx, My and Mz will reduce. Actually, the correct solution of this problem can only be described as a "stinker". Professor Horne did some work on this problem in a paper contributed to Professor Baker's "farewell" volume. The trouble is one should strictly follow what happens by applying the normality law of plasticity, which makes changes of the plastic force components (axial force and bending and twisting moments) proportionate to the corresponding changes of hinge discontinuity. This in turn means that the changes of plastic force depend on the elastic deformations of the rest of the structure i.e. you cannot consider the hinge properties as uniquely defined by what is happening at the hinge itself.

No real problem arises for uni-axial bending plus axial force - the bending moment at the hinge is simply re-estimated from the estimated new value of axial force. Near enough, one can estimate the new force as the axial force obtained from the analysis at the immediately previous load - i.e. at the (N+1)th load increment the axial force becomes (N+1)/N times the axial force at the Nth load increment. The new plastic bending moment is then obtained from the appropriate formula.

The difficulty arises when there is more than one component of bending moment (Mx,My,Mz) as well as axial force Fx. Professor Horne suggests that, as a reasonable approximation, one may assume that:

$$\frac{Mx}{Mxh} = \frac{My}{Myh} = \frac{Mz}{Mzh} = \frac{Fxp - Fx}{Fxp - Fxh}$$

where Fxh and (Mxh,Myh,Mzh) are the axial force and moment components when the plastic hinge first forms; and Fx and (Mx,My,Mz) are the axial force and moment components at any subsequent stage. If the cross- sectional shape were such that the linear formula was correct, then the assumption about changes Mxh —> Mx, Myh —> My and Mzh —> Mz would be approximate and liable to error either way. However, virtually all cross sections have a distinctly convex plastic force component interaction relationship - so that the above assumption, when Fxh increases to Fx, gives a conservative answer (see diagram). Hence Professor Horne's suggestion will be safe.



Assumed formula gives smaller f(Mx,My,Mz) - hence safe.

To summarise:

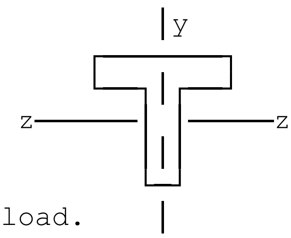
The solution given for changes in Mx, My and Mz when Fx changes is an approximation, but is acceptable when loading is increased proportionately. The complete solution could only be derived by considering the "normality" rule of plasticity, which governs the way in which deformations (angular and axial) at a plastic hinge are related to each other, and the compatibility in turn of these deformations with the total deformation of the structure. The errors arising from not undertaking such an analysis may be assumed to be sufficiently small to be neglected.

7.6.2 Interaction formulae applied to plane frames

7.6.2.1 Interaction formula 1 (sections with only one axis of symmetry)

The general formula $\frac{Fx}{Fxp} + \frac{Mx}{Mxp} + \frac{My}{Myp} + \frac{Mz}{Mzp} = 1$ reduces to

$$\frac{Fx}{Fxp} + \frac{Mz}{Mzp} = 1$$ for plane frames.



where Fxp is the squash load and Mzp is the plastic moment capacity in the absence of axial load.

Before starting next load increment, compute the ratio $\frac{Incn+1}{Incn} = r$.

At the end of the next load increment, we expect the new axial force in a member to be = r.Fx and the new moment = r.Mz.

Compute the unity factor $U = r \left[\frac{Fx}{Fxp} + \frac{Mz}{Mzp} \right]$

If U <= 1 do not introduce a plastic hinge.

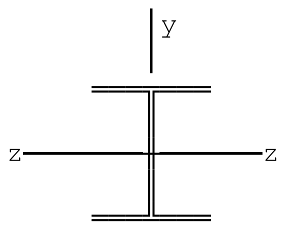
If U > 1 then compute plastic hinge value Mzh from $\frac{r.Fx}{Fxp} + \frac{Mzh}{Mzp} = 1$

rearranging $Mzh = Mzp \left[1 - \frac{r.Fx}{Fxp} \right]$

In all interaction formulae, the absolute values of moments and axial forces are used, and signs of hinges adjusted to be of the same sign as the corresponding member end forces.

7.6.2.2 Interaction formula 2 (I Sections bending about major axis)

The general formula $\left[\frac{Mz}{Mpzp} \right]^2 + \left[\frac{My}{Mpyp} \right] = 1$ reduces to



$\frac{Mz}{Mpzp} = 1$ or Mz = Mpzp

where Mpzp takes one of two values depending on the ratio alpha = web area/total area.

If $\frac{Fx}{Fxp} <= alpha$,

$$Mz = Mpzp = Mzp \left[1 - \frac{1}{alpha(2-alpha)} \left[\frac{Fx}{Fxp} \right]^2 \right]$$

rearranging $\frac{Mz}{Mzp} + \frac{1}{alpha(2-alpha)} \left[\frac{Fx}{Fxp} \right]^2 = 1$

If $\frac{Fx}{Fxp} > alpha$, $Mz = Mpzp = Mzp.2.\frac{(1-Fx/Fxp)}{(2-alpha)}$

rearranging $$\left[\frac{Mz}{Mzp}\right]\left[\frac{2-alpha}{2}\right] + \frac{Fx}{Fxp} = 1$$

Before starting next load increment, compute the ratio $\frac{Incn+1}{Incn} = r$.

At the end of the next load increment, we expect the new axial force in a member to be = r.Fx and the new moment = r.Mz.

If $\frac{r.Fx}{Fxp} <= alpha$:

Compute the unity factor $$U = \frac{r.Mz}{Mzp} + \frac{1}{alpha(2-alpha)}\left[\frac{r.Fx}{Fxp}\right]^2$$

If U <= 1 do not introduce a plastic hinge.

If U > 1 then compute the plastic hinge value Mzh from:

$$\frac{Mzh}{Mzp} + \frac{1}{alpha(2-alpha)}\left[\frac{r.Fx}{Fxp}\right]^2 = 1$$

rearranging $$Mzh = Mzp\left[1 - \frac{1}{alpha(2-alpha)}\left[\frac{r.Fx}{Fxp}\right]^2\right]$$

If $\frac{r.Fx}{Fxp} > alpha$:

Compute the unity factor $$U = r\left[\frac{Mz}{Mzp}\left[\frac{2-alpha}{2}\right] + \frac{Fx}{Fxp}\right]$$

If U <= 1 do not introduce a plastic hinge.

If U > 1 then compute the plastic hinge value Mzh from:

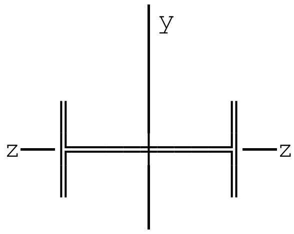
$$\left[\frac{Mzh}{Mzp}\right]\left[\frac{2-alpha}{2}\right] + \frac{r.Fx}{Fxp} = 1$$

rearranging $$Mzh = Mzp\left[\frac{2}{2-alpha}\right]\left[1 - \frac{r.Fx}{Fxp}\right]$$

In all interaction formulae, the absolute values of moments and axial forces are used, and signs of hinges adjusted to be of the same sign as the corresponding member end forces.

7.6.2.3 Interaction formula 3 (H Sections bending about minor axis)

The general formula $\left[\frac{My}{Mpyp}\right]^2 + \left[\frac{Mz}{Mpzp}\right] = 1$ reduces to



$\frac{Mz}{Mpzp} = 1$ or $Mz = Mpzp$

where Mpzp takes one of two values depending on the ratio alpha = web area/total area.

If $\frac{Fx}{Fxp}$ <= alpha, $Mz = Mpzp = Mzp$ or $\frac{Mz}{Mzp} = 1$

If $\frac{Fx}{Fxp}$ > alpha,

$$Mz = Mpzp = Mzp \left[\frac{\left[1-2.alpha+\frac{Fx}{Fxp}\right]\left[1-\frac{Fx}{Fxp}\right]}{(1-alpha)^2} \right]$$

rearranging

$$\frac{Mz}{Mzp}(1-alpha)^2 = 1 - 2.aplha + \frac{Fx}{Fxp} - \frac{Fx}{Fxp} + \frac{Fx.2alpha}{Fxp} - \left[\frac{Fx}{Fxp}\right]^2$$

or $$\frac{Mz}{Mzp}(1-alpha)^2 + 2.alpha - \frac{Fx}{Fxp}\left[2.alpha - \frac{Fx}{Fxp}\right] = 1$$

Before starting next load increment, compute the ratio $\frac{Incn+1}{Incn} = r.$

At the end of the next load increment, we expect the new axial force in a member to be = r.Fx and the new moment = r.Mz.

If $\frac{r.Fx}{Fxp}$ <= alpha; compute the unity factor $U = \frac{r.Mx}{Mzp}$

If U <= 1 do not introduce a plastic hinge.

If U > 1 then compute plastic hinge value Mzh from: Mzh = Mzp

If $\frac{r.Fx}{Fxp}$ > alpha, compute the unity factor:

$$U = \frac{r.Mz}{Mzp}.(1-alpha)^2 + 2.alpha - \frac{r.Fx}{Fxp}\left[2.alpha - \frac{r.Fx}{Fxp}\right]$$

If U <= 1 do not introduce a plastic hinge.

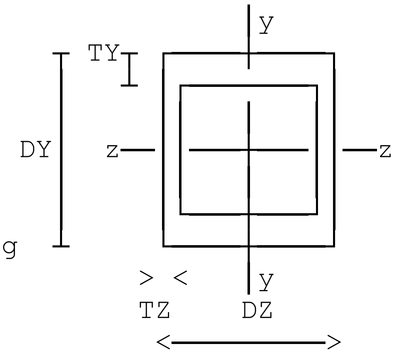
$$Mzh = Mzp\left[\frac{\left[1-2.alpha+\frac{r.Fx}{Fxp}\right]\left[1-\frac{r.Fx}{Fxp}\right]}{(1-alpha)^2}\right]$$

In all interaction formulae, the absolute values of moments and axial forces are used, and signs of hinges adjusted to be of the same sign as the corresponding member end forces.

7.6.2.4 Interaction formula 4 (Rectangular hollow sections)

The general formula $\left[\frac{Mz}{Mpzp}\right]^{5/3} + \left[\frac{My}{Mpyp}\right]^{5/3} = 1$ reduces to

$\frac{Mz}{Mpzp} = 1$ or $Mz = Mpzp$



where Mpzp takes one of two values depending on the ratio $alpha = \frac{DY.TZ}{DY.TZ + DZ.TY}$

If $\frac{Fx}{Fxp}$ <= alpha,

$$Mz = Mpzp = Mzp\left[1 - \frac{1}{alpha(2-alpha)}\left[\frac{Fx}{Fxp}\right]^2\right]$$

rearranging $\frac{Mz}{Mzp} + \frac{1}{alpha(2-alpha)}\left[\frac{Fx}{Fxp}\right]^2 = 1$

If $\frac{Fx}{Fxp}$ > alpha, $Mz = Mpzp = 2.Mzp\,\frac{(1-Fx/Fxp)}{(2-alpha)}$

rearranging $\frac{Mz}{Mzp} \frac{(2-alpha)}{2} + \frac{Fx}{Fxp} = 1$

Before starting next load increment, compute the ratio $\frac{Incn+1}{Incn} = r$.

At the end of the next load increment, we expect the new axial force in a member to be = r.Fx and the new moment = r.Mz.

If $\frac{r.Fx}{Fxp} <= alpha$:

Compute the unity factor $U = \frac{r.Mz}{Mzp} + \frac{1}{alpha(2-alpha)} \left[\frac{r.Fx}{Fxp} \right]^2$

If U <= 1 do not introduce a plastic hinge.

If U > 1 then compute the plastic hinge value Mzh from:

$$Mzh = Mzp \left[1 - \frac{1}{alpha(2-alpha)} \left[\frac{r.Fx}{Fxp} \right]^2 \right]$$

If $\frac{r.Fx}{Fxp} > alpha$, compute the unity factor:

$$U = r \left[\frac{Mz}{Mzp} \left[\frac{2-alpha}{2} \right] + \frac{Fx}{Fxp} \right]$$

If U <= 1 do not introduce a plastic hinge.

If U > 1 then compute the plastic hinge value Mzh from:

$$Mzh = \frac{2.Mzp}{(2-alpha)} \left[1 - \frac{r.Fx}{Fxp} \right]$$

The treatment for a RHS is as for an I section bending about its major axis, save for the computation of alpha.

The above formulae work for D < B for the plane frame case of axial load and bending about z.

In all interaction formulae, the absolute values of moments and axial forces are used, and signs of hinges adjusted to be of the same sign as the corresponding member end forces.

7.6.2.5 Interaction formula 5 (circular hollow sections)

The general formula

$$\frac{(Mz^2 + My^2)^{0.5}}{Mp.COS\left[\frac{\pi}{2} \cdot \frac{r.Fx}{Fxp}\right]} = 1$$

reduces to

$$\frac{Mz}{Mzp.COS\left[\frac{\pi}{2} \cdot \frac{r.Fx}{Fxp}\right]} = 1$$

or $Mz = Mzp.COS\left[\frac{\pi}{2} \cdot \frac{r.Fx}{Fxp}\right]$

Before starting next load increment, compute the ratio $\frac{Incn+1}{Incn} = r.$

At the end of the next load increment, we expect the new axial force in a member to be = r.Fx and the new moment = r.Mz.

Compute the unity factor $U = \frac{r.Mz}{Mzp.COS\left[\frac{\pi}{2} \cdot \frac{r.Fx}{Fxp}\right]}$

If U <= 1 do not introduce a plastic hinge.

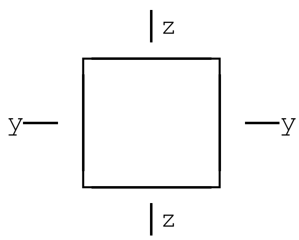
If U > 1 then compute the plastic hinge value Mzh from:

$$Mzh = Mzp.COS\left[\frac{\pi}{2} \cdot \frac{r.Fx}{Fxp}\right]$$

7.6.3 Interaction formulae applied to plane grids

For plane grids there is no axial force and no bending about the zz axis and the interaction formula for any section:

$$\left[\frac{Mx}{Mxp}\right]^2 + \left[\frac{My}{Myp}\right] = 1$$



Before starting next load increment, compute the ratio $\frac{Incn+1}{Incn} = r.$

At the end of the next load increment, we expect the new torque in a member to be = r.Mx and the new moment = r.My.

Compute the unity factor $U = \left[\left[\frac{r.Mx}{Mxp} \right]^2 + \left[\frac{r.My}{Myp} \right]^2 \right]^{0.5}$

If U <= 1 do not introduce a plastic hinge.

For first increment in which U > 1 assume plastic hinge originally formed at:

Mxh = r.Mx/U
Myh = r.My/U

where Mxh and Myh are saved as the torque and moment components when the hinge first forms.

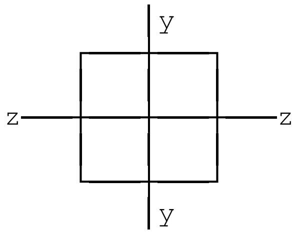
As there is no axial load, use these values of Mxh and Myh for all subsequent load increments.

7.6.4 Interaction formulae applied to space frames

7.6.4.1 Interaction formula 1 (sections with only one axis of symmetry)

The general formula $\frac{Fx}{Fxp} + \frac{Mx}{Mxp} + \frac{My}{Myp} + \frac{Mz}{Mzp} = 1$ applies.

where Fxp is the squash load and Mxp, Myp, Mzp are the plastic moment capacities about the x, y and z axes in the absence of other loads.



Before starting next load increment, compute the ratio $\frac{Incn+1}{Incn} = r.$

At the end of the next load increment, we expect the new axial force in a member to be = r.Fx and the new moments = r.Mx, r.My, r.Mz.

Compute the unity factor $U = r \left[\frac{Fx}{Fxp} + \frac{Mx}{Mxp} + \frac{My}{Myp} + \frac{Mz}{Mzp} \right]$

If U <= 1 do not introduce a plastic hinge.

For first increment in which U > 1 assume plastic hinges originally formed at:

Fxh = r.Fx/U
Mxh = r.Mx/U

Myh = r.My/U
Mzh = r.Mz/U

where Fxh and (Mxh,Myh,Mzh) are saved as the axial force and moment components when the hinge first forms.

For subsequent increments, knowing Fx as the estimated axial load,

compute the normality ratio: $nr = \dfrac{Fxp - Fx}{Fxp - Fxh}$

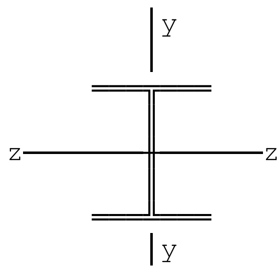
where Fxp is squash load (area x yield stress)
and Fxh is axial load when hinge first formed;
then corresponding moments to be used with Fx are: Mx = nr.Mxh
My = nr.Myh
Mz = nr.Mzh

In all interaction formulae, the absolute values of moments and axial forces are used, and signs of hinges adjusted to be of the same sign as the corresponding member end forces.

7.6.4.2 Interaction formula 2 (I Sections bending about major axis)

The general formula $\left[\dfrac{Mz}{Mpzp}\right]^2 + \left[\dfrac{My}{Mpyp}\right] = 1$

where Mpzp and Mpyp take one of two values depending on the ratio $alpha = \dfrac{\text{web area}}{\text{total area}}$



Before starting next load increment, compute the ratio $r. = \dfrac{Incn+1}{Incn}$

Compute the twist factor and normalised axial effect, n:

$$t = SQR\left[1 - \left[\frac{r.Mx}{Mxp}\right]^2\right] \qquad n = \frac{1}{t}.\frac{f.Fx}{Fxp}$$

If n <= alpha,

$$Mpzp = t.Mzp\left[1 - \frac{n^2}{alpha(2-alpha)}\right] \quad \text{and} \quad Mpyp = t.Myp$$

If n > alpha, $Mpzp = \dfrac{t.Mzp.2.(1-n)}{(2-alpha)}$

and Mpyp = t.Myp ((1 - 2.alpha) + n) (1 - n) / (1 - alpha)^2

Compute the unity factor $U = \left[\frac{r.Mz}{Mpzp} \right]^2 + \left[\frac{r.My}{Mpyp} \right]$

If U <= 1 do not introduce a plastic hinge.

For first increment in which U > 1 assume plastic hinges originally formed at:

Fxh = r.Fx/U
Mxh = r.Mx/U
Myh = r.My/U
Mzh = r.Mz/U

where Fxh and (Mxh,Myh,Mzh) are saved as the axial force and moment components when the hinge first forms.

For subsequent increments, knowing Fx as the estimated axial load,

compute the normality ratio: $nr = \frac{Fxp - Fx}{Fxp - Fxh}$

where Fxp is squash load (area x yield stress)
and Fxh is axial load when hinge first formed;

then corresponding moments to be used with Fx are: Mx = nr.Mxh
My = nr.Myh
Mz = nr.Mzh

In all interaction formulae, the absolute values of moments and axial forces are used, and signs of hinges adjusted to be of the same sign as the corresponding member end forces.

7.6.4.3 Interaction formula 3 (H Sections bending about minor axis)

Use treatment of 7.6.4.2 with y and z axes swapped.

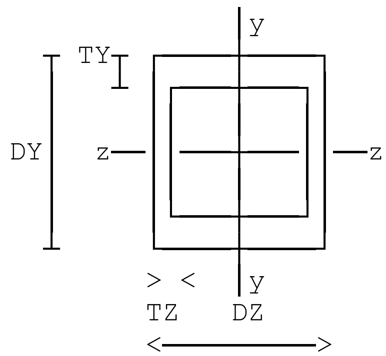
7.6.4.4 Interaction formula 4 (Rectangular hollow sections)

The general formula $\left[\frac{Mz}{Mpzp} \right]^{5/3} + \left[\frac{My}{Mpyp} \right]^{5/3} = 1$

where Mpzp and Mpyp take values depending on the ratios

$alpha1 = \frac{DY.TZ}{DY.TZ + DZ.TY}$

$alpha2 = \frac{DZ.TY}{DY.TZ + DZ.TY}$



Before starting next load increment, compute the ratio $\frac{Incn+1}{Incn} = r$.

Compute the twist factor and normalised axial effect, n:

$$t = SQR\left[1 - \left[\frac{r.Mx}{Mxp} \right]^2 \right] \qquad n = \frac{1}{t} . \frac{f.Fx}{Fxp}$$

Major axis bending:

If n <= alpha1, $Mpzp = t.Mzp \left[1 - \frac{n^2}{alpha1(2-alpha1)} \right]$

If n > alpha1, $Mpzp = \frac{t.Mzp.2.(1-n)}{(2-alpha1)}$

Minor axis bending:

If n < alpha2

Mpyp = t.Myp

If n > alpha2

Mpyp = t.Myp ((1 - 2.alpha2) + n) (1 - n) / (1 - alpha2)^2

Compute the unity factor $U = \left[\frac{r.Mz}{Mpzp} \right]^2 + \left[\frac{r.My}{Mpyp} \right]$

If U <= 1 do not introduce a plastic hinge.

For first increment in which U > 1 assume plastic hinges originally formed at:

Fxh = r.Fx/U
Mxh = r.Mx/U
Myh = r.My/U
Mzh = r.Mz/U

where Fxh and (Mxh,Myh,Mzh) are saved as the axial force and moment components when the hinge first forms.

For subsequent increments, knowing Fx as the estimated axial load,

compute the normality ratio: $nr = \frac{Fxp - Fx}{Fxp - Fxh}$

where Fxp is squash load (area x yield stress)
and Fxh is axial load when hinge first formed;

then corresponding moments to be used with Fx are: $Mx = nr.Mxh$
$My = nr.Myh$
$Mz = nr.Mzh$

In all interaction formulae, the absolute values of moments and axial forces are used, and signs of hinges adjusted to be of the same sign as the corresponding member end forces.

7.6.4.5 Interaction formula 5 (circular hollow sections)

The condition for full plasticity for a circular, hollow section under axial force r.Fx, moments r.Mx, r.My, r.Mz is:

$$\left[\left[\frac{r.Mx}{Mpzp} \right]^2 + \left[\frac{r.My}{Mpyp} \right]^2 \right]^{0.5} = 1$$

where $Mpzp = Mpyp = t.Mp.COS(n.\pi/2)$

and $t = SQR\left[1 - \left[\frac{r.Mx}{Mxp} \right]^2 \right]$ and $n = \frac{1}{t} . \frac{r.Fx}{Fxp}$

Before starting next load increment, compute the ratio $r = \frac{Incn+1}{Incn}$

At the end of the next load increment, we expect the new axial force in a member to be = r.Fx and the new moments = r.Mx, r.My, r.Mz.

Compute the unity factor $U = \frac{r}{t} . \frac{(Mz^2 + My^2)^{0.5}}{Mp.COS\left[\frac{\pi . n}{2} \right]}$

If U <= 1 do not introduce a plastic hinge.

For first increment in which U > 1 assume plastic hinges originally formed at:

Fxh = r.Fx/U
Mxh = r.Mx/U
Myh = r.My/U
Mzh = r.Mz/U

where Fxh and (Mxh,Myh,Mzh) are saved as the axial force and moment components when the hinge first forms.

For subsequent increments, knowing Fx as the estimated axial load,

compute the normality ratio: $nr = \frac{Fxp - Fx}{Fxp - Fxh}$

where Fxp is squash load (area x yield stress)
and Fxh is axial load when hinge first formed;

then corresponding moments to be used with Fx are: Mx = nr.Mxh
My = nr.Myh
Mz = nr.Mzh

In all interaction formulae, the absolute values of moments and axial forces are used, and signs of hinges adjusted to be of the same sign as the corresponding member end forces.

7.7 The stiffness method

A good description of the stiffness method of analysis may be found in 'Computer Programs for Structural Analysis' by William Weaver,Jr. published by Van Nostrand 1967.

NL-STRESS uses the stiffness method but extends it for:

- shear deformation (the effects of which are significant in short span beams and at haunches)
- finite displacements
- sway and within member stability
- elastic plastic behaviour

In the stiffness method the displacements of the joints are considered to be the basic unknowns. The procedure may be summarised:

- with the structure locked at all joints, a unit displacement is given to each joint in each possible direction of movement and the forces corresponding to each unit displacement are used to build the overall structure stiffness matrix.
- again with the structure locked, the fixed end forces due to the loads applied to each member are computed and together with the loads applied directly to the joints are used to build the combined joint load vector.
- the matrix equation - Combined joint load vector = structure stiffness matrix x joint displacements - is solved to yield the joint displacements in each possible direction of movement
- the displacements at member ends together with the fixed end forces are used to compute the member end forces
- support reactions are found by summing the contributions from members framing into the support.

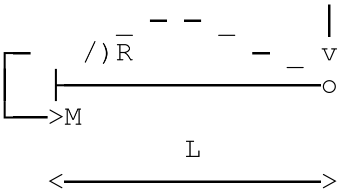
In the remainder of this section, component terms of the stiffness matrix are derived, and then used to build the stiffness matrices for 2D and 3D structures.

7.7.1 Component terms of member stiffness matrix

The original STRESS software formed the flexibility matrix for each member allowing for the effects of shear deformation, and then inverted the flexibility matrix to obtain the member stiffness matrix. In NL-STRESS the member stiffness matrix is formed directly with rigorous treatment for the effects of shear deformation.

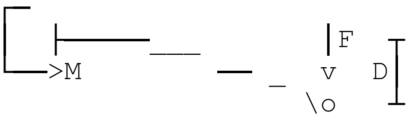
The derivation of member stiffness matrix terms involving axial forces is straightforward. For terms involving bending forces there are four basic cases viz:

(i)



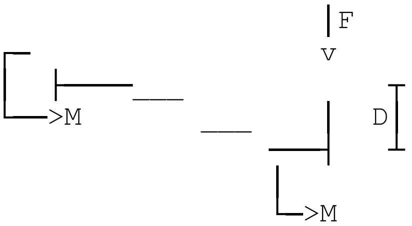
End of member pinned
An applied moment M produces a rotation R at the start of the member. The applied moment induces a force F at the end of the beam of length L and sectional properties EI and GA.

(ii)



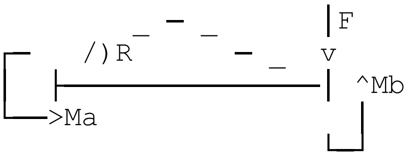
End of member pinned
An applied force F at the end of the member causes a displacement D. The applied force induces a moment M at the start of the beam of length L and sectional properties EI and GA.

(iii)



End of member fixed
An applied force F at the end of the member causes a displacement D. The applied force induces moments M at both ends of the beam of length L and sectional properties EI and GA.

(iv)

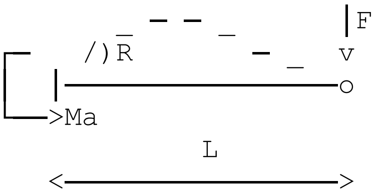


End of member fixed
An applied moment Ma produces rotation R at the start of the member. The applied moment induces a moment Mb and force F at end of the beam of length L and sectional properties EI and GA.

The well known slope deflection equation ignores the effect of shear deformation, so for each of the four basic cases, force displacement relationships will be derived using Castigliano's Theorem Method (1879) which states:

The partial differentiation of the strain energy with respect to any load or couple is a measure of the linear displacement of the point of application of that load in the direction and sense of the load or the angular rotation of the centre line at the point in the direction and sense of the couple.

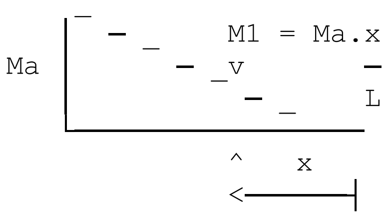
7.7.1.1 Case (i)



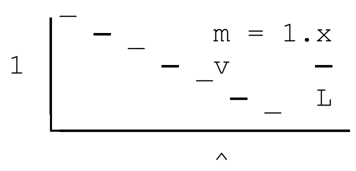
Ma = F.L and for unit Ma: 1=f.L

By Castigliano:

$$R = \int_0^L \frac{M1.m.dx}{EI} + \int_0^L \frac{F1.f.dx}{GA}$$

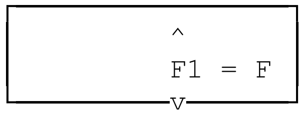


$$= \int_0^L \frac{1}{EI}\,\frac{Ma.x}{L}.\frac{x}{L}.dx + \int_0^L \frac{1}{GA}\,F.\frac{1}{L}.dx$$



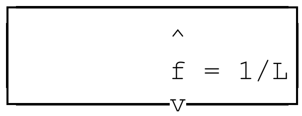
$$= \left[\frac{Ma.x^3}{3EI.L^2}\right]_0^L + \left[\frac{F.x}{L.GA}\right]_0^L$$

$$= \frac{Ma.L^3}{3EI.L^2} + \frac{F.L}{GA.L} \quad \& \text{ subs } F = \frac{Ma}{L}$$



$$R = \frac{L}{3EI}\left[\, Ma + \frac{3EI}{L}\,.\,\frac{Ma}{L.GA} \right]$$

rearranging:



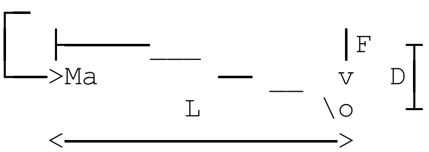
$$Ma = \frac{R.3EI}{L}\left[\frac{1}{1 + \dfrac{3EI}{L^2.GA}}\right]$$

Substituting $F = \frac{Ma}{L}$ gives

$$F = \frac{R.3EI}{L^2}\left[\frac{1}{1 + \dfrac{3EI}{L^2.GA}}\right]$$

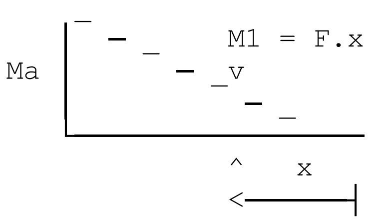
7.7.1.2 Case (ii)

Ma = F.L and for unit Ma: 1=f.L



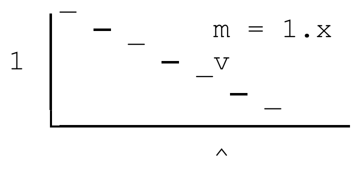
By Castigliano:

$$D = \int_0^L \frac{M1.m.dx}{EI} + \int_0^L \frac{F1.f.dx}{GA}$$

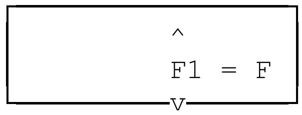


$$= \int_0^L \frac{1}{EI}\,F.x.x.dx + \int_0^L \frac{1}{GA}\,F.1.dx$$

$$= \left[\frac{F.x^3}{3EI}\right]_0^L + \left[\frac{F.x}{GA}\right]_0^L$$

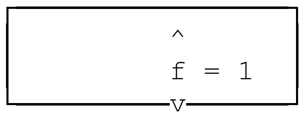


$$= \frac{F.L^3}{3EI} + \frac{F.L}{GA}$$



$$D = \frac{F.L^3}{3EI}\left[1 + \frac{3EI}{L^2.GA}\right]$$

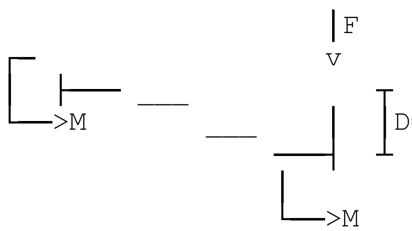
rearranging:



$$F = \frac{D.3EI}{L^3}\left[\frac{1}{1 + \frac{3EI}{L^2.GA}}\right]$$

substituting Ma=F.L gives $Ma = \frac{D.3EI}{L^2}\left[\frac{1}{1 + \frac{3EI}{L^2.GA}}\right]$

7.7.1.3 Case (iii)



Because of central point of contraflexure treat half span as for case (ii) with L = 2.Lp and

D= 2.Dp substituted in expressions on previous page thus:

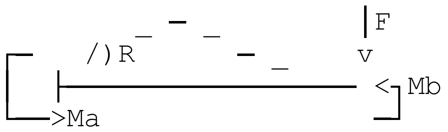
$$F = \frac{(D/2)3EI}{(L/2)^3}\left[\frac{1}{1 + \frac{3EI}{(L/2)^2.GA}}\right]$$ therefore

$$F = \frac{D.12EI}{L^3}\left[\frac{1}{1 + \frac{12EI}{L^2.GA}}\right]$$

$$Ma = \frac{(D/2)3EI}{(L/2)^2}\left[\frac{1}{1 + \frac{3EI}{(L/2)^2.GA}}\right]$$ therefore

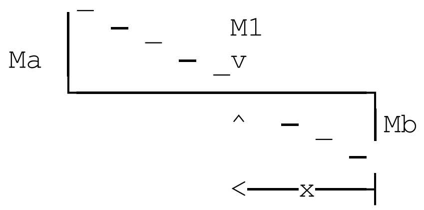
$$Ma = \frac{D.6EI}{L^2}\left[\frac{1}{1 + \dfrac{12EI}{L^2.GA}}\right]$$

7.7.1.4 Case (iv)

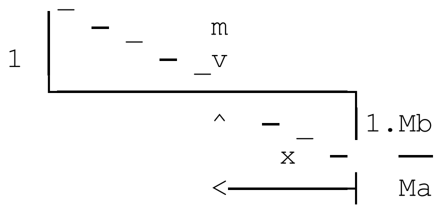


By Castigliano:

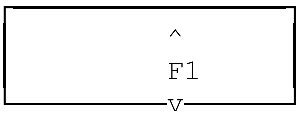
$$R = \int_0^L \frac{M1.m.dx}{EI} + \int_0^L \frac{F1.f.dx}{GA}$$



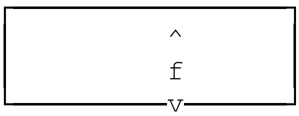
$$M1 = (Ma + Mb).\frac{x}{L} - Mb$$



$$m = \left[1 + \frac{Mb}{Ma}\right]\frac{x}{L} - \frac{Mb}{Ma}$$



$$F1 = \frac{(Ma + Mb)}{L}$$



$$f = \frac{Ma + Mb}{L.Ma}$$

Let $I1 = \int_0^L \frac{M1.m.dx}{EI}$ and $I2 = \int_0^L \frac{F1.f.dx}{GA}$ then:

$$I1 = \int_0^L \left[\frac{Ma.x}{L} + \frac{Mb.x}{L} - Mb\right]\left[\frac{x}{L} + \frac{Mb.x}{Ma.L} - \frac{Mb}{Ma}\right]\frac{dx}{EI}$$

$$= \int_0^L \left[\frac{Ma.x^2}{L^2} + \frac{Ma.x.Mb.x}{L.Ma.L} - \frac{Ma.x.Mb}{L.Ma} + \frac{Mb.x^2}{L^2} + \frac{Mb^2.x^2}{Ma.L^2} - \frac{Mb^2.x}{Ma.L} - \frac{Mb.x}{L} - \frac{Mb^2.x}{Ma.L} + \frac{Mb^2}{Ma} \right] \frac{dx}{EI}$$

$$= \frac{1}{EI} \left[\frac{Ma.x^3}{3.L^2} + \frac{2.Mb.x^3}{L^2.3} - \frac{2.Mb.x^2}{L.2} + \frac{Mb^2.x^3}{Ma.L^2.3} - \frac{2.Mb^2.x^2}{Ma.L.2} + \frac{Mb^2.x}{Ma} \right]_0^L$$

$$= \frac{1}{EI} \left[\frac{Ma.L}{3} + \frac{2.Mb.L}{3} - \frac{2.Mb.L}{2} + \frac{Mb^2.L}{Ma.3} - \frac{2.Mb^2.L}{Ma.2} + \frac{Mb^2.L}{Ma} \right]$$

$$= \frac{1}{EI} \left[\frac{Ma.L}{3} - \frac{Mb.L}{3} + \frac{Mb^2.L}{3.Ma} \right]$$

$$I2 = \int_0^L \left[\frac{Ma + Mb}{L} \right] \left[\frac{Ma + Mb}{Ma.L} \right] \frac{dx}{GA} = \left[\frac{(Ma + Mb)^2.x}{Ma.L^2.GA} \right]_0^L = \frac{1}{GA} \left[\frac{(Ma + Mb)^2}{Ma.L} \right]$$

$$R = \frac{1}{EI} \left[\frac{Ma.L}{3} - \frac{Mb.L}{3} + \frac{Mb^2.L}{3.Ma} \right] + \frac{1}{GA} \left[\frac{(Ma + Mb)^2}{Ma.L} \right] \quad ..(1)$$

For equilibrium $Ma + Mb = F.L$ ..(2)

From Maxwell:

  reaction F due to unit rotation at A in direction of Ma
= moment Ma due to unit displacement at B in direction of F

We know from previous case the moment Ma due to unit displacement in

$$\text{direction of F:} \quad Ma = \frac{6EI}{L^2} \left[\frac{1}{1 + \frac{12EI}{L^2.GA}} \right] = \frac{6EI.k}{L^2} \quad \text{where } k = \frac{1}{1 + \frac{12EI}{L^2.GA}}$$

$F = \dfrac{6EIk.R}{L^2}$ by Maxwell, and substituting in (2) gives:

$$Ma + Mb = \frac{6EIk.R}{L} \;..(3) \qquad Mb = \frac{6EIk.R}{L} - Ma \;..(4) \qquad \frac{Mb}{Ma} = \frac{6EIk.R}{L.Ma} - 1 \;..(5)$$

substituting (3),(4) & (5) into (1) to eliminate Mb gives:

$$R= \frac{1}{EI}\left[\frac{Ma.L}{3} - \frac{L}{3}\left[\frac{6EIk.R}{L} - Ma\right] + \frac{L}{3}\left[\frac{6EIk.R}{L} - Ma\right]\left[\frac{6EIk.R}{L.Ma} - 1\right]\right]$$

$$+ \frac{1}{GA}\left[\frac{36E^2.I^2.k^2.R^2}{L^3.Ma}\right]$$

$$= \frac{1}{EI}\left[\frac{Ma.L}{3} - 2EIk.R + \frac{MaL}{3} + \frac{L}{3}\,\frac{36E^2.I^2.k^2.R^2}{L^2.Ma} - \frac{L}{3}\,\frac{6EIk.R}{L}\right.$$

$$\left. - \frac{L}{3}\,\frac{6EIk.R}{L} + \frac{Ma.L}{3}\right] + \frac{36E^2.I^2.k^2.R^2}{GA.L^3.Ma}$$

Rearranging, and multiplying throughout be Ma gives:

$$Ma^2\left[\frac{L}{EI}\right] + Ma(-6R.k - R) + \frac{R^2.12EIk^2}{L}\left[1 + \frac{3EI}{L^2.GA}\right] = 0$$

which is a quadratic in Ma; with root given as: Ma =

$$\frac{6Rk + R + \sqrt{36.R^2.k^2 + R^2 + 12R^2.k - \frac{4L}{EI}.\frac{R^2.12EI.k^2}{L}\left[1 + \frac{3EI}{L^2.GA}\right]}}{\frac{2L}{EI}}$$

$$Ma = R\left[\frac{6kEI}{2L} + \frac{EI}{2L} \pm \frac{EI}{2L}\sqrt{36k^2 + 1 + 12k - 48k^2 - \frac{48k^2.3EI}{L^2.GA}}\right]$$

$$= R\left[\frac{3kEI}{L} + \frac{EI}{2L} \pm \frac{EI}{2L}\sqrt{-12k^2\left(1 + \frac{12EI}{L^2.GA}\right) + 1 + 12k}\right]$$

Substituting $1 + \frac{12EI}{L^2.GA} = \frac{1}{k}$ gives $Ma = R\left[\frac{3kEI}{L} + \frac{EI}{2L} \pm \frac{EI}{2L}\sqrt{1}\right]$

Take positive root as we need $Ma = \frac{4EI.R}{L}$ when k = 1 i.e zero shear deformation; $Ma = \left[\frac{3kEI}{L} + \frac{EI}{L} \right].R = \left[\frac{EI}{L} + \frac{3EI}{L} \left[\frac{1}{1 + \frac{12EI}{L^2.GA}} \right] \right].R$

Substituting into (4) gives $Mb = \frac{6EIk.R}{L} - \frac{3EIk.R}{L} - \frac{EI.R}{L}$ hence

$$Mb = \left[\frac{-EI}{L} + \frac{3EI}{L} \left[\frac{1}{1 + \frac{12EI}{L^2.GA}} \right] \right].R$$

7.7.2 Member stiffness matrix for plane frames

Consider member jk:
where j is 'start end'
and k is 'end end' of
the member. Let:

Fxj = force on member at start end in direction of local x axis
Fyj = ... y ....
Mzj = moment .... about z axis (z out of the page)
Fxk = force on member at end end in direction of local x axis
Fyk = ... y ....
Mzk = moment .... about z axis
Dxj = displacement at member start in direction of local x axis
Dyj = ... y ....
Rzj = rotation .... about z axis
Dxk = displacement at member end in direction of local x axis
Dyk = ... y ....
Rzk = rotation .... about z axis

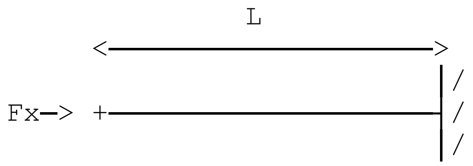
These forces and displacements are related by:

$$\begin{bmatrix} Fxj \\ Fyj \\ Mzj \\ Fxk \\ Fyk \\ Mzk \end{bmatrix} = \begin{bmatrix} (\ Sm11\ ) & (\ Sm12\ ) & Sm13 & Sm14 & Sm15 & Sm16 \\ (\ Sm21\ ) & (\ Sm22\ ) & Sm23 & Sm24 & Sm25 & Sm26 \\ (\ Sm31\ ) & (\ Sm32\ ) & Sm33 & Sm34 & Sm35 & Sm36 \\ (\ Sm41\ ) & (\ Sm42\ ) & Sm43 & Sm44 & Sm45 & Sm46 \\ (\ Sm51\ ) & (\ Sm52\ ) & Sm53 & Sm54 & Sm55 & Sm56 \\ (\ Sm61\ ) & (\ Sm62\ ) & Sm63 & Sm64 & Sm65 & Sm66 \end{bmatrix} \begin{bmatrix} Dxj \\ Dyj \\ Rzj \\ Dxk \\ Dyk \\ Rzk \end{bmatrix}$$

Vector of forces resulting from unit displacement Dxj (first column, ^)

Vector of forces resulting from unit displacement Dyj (second column, ^)

For the six displacements Dxj,Dyj,Rzj,Dxk,Dyk,Rzk the corresponding column of forces due to unit displacement will be assembled from the component terms derived in section 7.7.1 used together with Hooke's Law.



$$Dx = \frac{Fx.L}{AE} \quad \text{(Hooke's Law)}$$

>Dx< is displacement due to Fx

7.7.2.1 Both ends fixed

Consider unit displacement Dxj in isolation then for no releases:

1 = Fxj.L/Ax.E hence Fxj = E.Ax/L

Resolving along member x axis   Fxk = -Fxj = -E.Ax/L

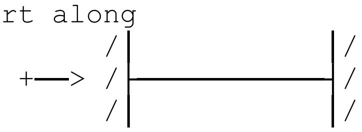
As displacements in y direction and rotations about z are all zero

Fyj = Fyk = Mzj = Mzk = 0

Therefore first column of member stiffness matrix =

$$\begin{bmatrix} EAx/L \\ 0 \\ 0 \\ -EAx/L \\ 0 \\ 0 \end{bmatrix}$$

Unit displacement at start along x axis



Consider unit displacement Dyj in isolation then for no releases:

$$Mzj = \frac{6EIz.S}{L^2} \quad \text{where} \quad S = \frac{1}{1 + \dfrac{12EIz}{L^2.GAy}}$$

(see 7.7.1.3)

Mzk = Mzj

Taking moments about k end

$$Mzj + Mzk = Fyj.L \quad \text{therefore} \quad Fyj = \frac{12EIz.S}{L^2} . \frac{1}{L} = \frac{12EIz.S}{L^3}$$

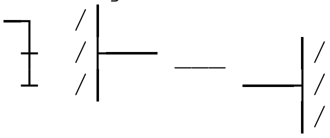
$$\text{Resolving along member y axis} \quad Fyk = -Fyj = \frac{-12EIz.S}{L^3}$$

As displacements in x direction are all zero Fxj = Fxk = 0

Therefore second column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ 12EIzS/L^3 \\ 6EIzS/L^2 \\ 0 \\ -12EIzS/L^3 \\ 6EIzS/L^2 \end{bmatrix}$$

Unit displacement at start along y axis



Consider unit rotation Rzj in isolation then for no releases:

$$Mzj = \frac{EIz}{L} + \frac{3EIz.S}{L} \quad \text{where} \quad S = \frac{1}{1 + \frac{12EIz}{L^2.GAy}}$$ (see 7.7.1.4)

$$Mzk = -\frac{EIz}{L} + \frac{3EIz.S}{L} \quad \text{where} \quad S = \frac{1}{1 + \frac{12EIz}{L^2.GAy}}$$ (see 7.7.1.4)

Taking moments about k end

$$Mzj + Mzk = Fyj.L \quad \text{therefore} \quad Fyj = \frac{6EIz.S.}{L}\frac{1}{L} = \frac{6EIz.S}{L^2}$$

Resolving along member y axis

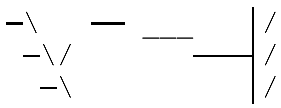
$$Fyk = -Fyj = \frac{-6EIz.S}{L^2}$$

As displacements in x direction are all zero Fxj = Fxk = 0

Therefore third column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ 6EIzS/L^2 \\ EIz/L+3EIzS/L \\ 0 \\ -6EIzS/L^2 \\ -EIz/L+3EIzS/L \end{bmatrix}$$

Unit rotation at start about z axis



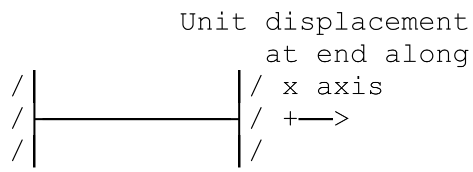
Consider unit displacement Dxk in isolation then for no releases:
1 = Fxk.L/AxE hence Fxk = E.Ax/L

Resolving along member x axis Fxj = -Fxk = -E.Ax/L

As displacements in y direction and rotations about z are all zero
Fyj = Fyk = Mzj = Mzk = 0

Therefore fourth column of member stiffness matrix =

$$\begin{bmatrix} -EAx/L \\ 0 \\ 0 \\ EAx/L \\ 0 \\ 0 \end{bmatrix}$$



Consider unit displacement Dyk in isolation then for no releases:

$$Mzk = -\frac{6EIz.S}{L^2} \quad \text{where} \quad S = \frac{1}{1 + \frac{12EIz}{L^2.GAy}}$$

(see 7.7.1.3)

Mzj = Mzk

Taking moments about j end

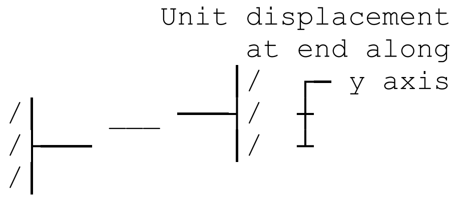
$$Mzk + Mzj = -Fyk.L \quad \text{therefore} \quad Fyk = \frac{12EIz.S}{L^2} . \frac{1}{L} = \frac{12EIz.S}{L^3}$$

$$\text{Resolving along member y axis} \quad Fyj = -Fyk = -\frac{12EIz.S}{L^3}$$

As displacements in x direction are all zero Fxk = Fxj = 0

Therefore fifth column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ -12EIzS/L^3 \\ -6EIzS/L^2 \\ 0 \\ 12EIzS/L^3 \\ -6EIzS/L^2 \end{bmatrix}$$



Consider unit rotation Rzk in isolation then for no releases:

$$Mzk = \frac{EIz}{L} + \frac{3EIz.S}{L} \quad \text{where} \quad S = \frac{1}{1 + \frac{12EIz}{L^2.GAy}}$$

(see 7.7.1.4)

$$Mzj = -\frac{EIz}{L} + \frac{3EIz.S}{L} \quad \text{where} \quad S = \frac{1}{1 + \dfrac{12EIz}{L^2.GAy}}$$

(see 7.7.1.4)

Taking moments about j end

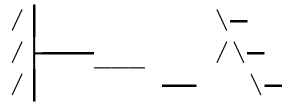
$$Mzk + Mzj = -Fyk.L \quad \text{therefore} \quad Fyk = \frac{-6EIz.S}{L} \cdot \frac{1}{L} = \frac{-6EIz.S}{L^2}$$

Resolving along member y axis $Fyj = -Fyk = \dfrac{6EIz.S}{L^2}$

As displacements in x direction are all zero $Fxk = Fxj = 0$

Therefore sixth column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ 6EIzS/L^2 \\ -EIz/L+3EIzS/L \\ 0 \\ -6EIzS/L^2 \\ EIz/L+3EIzS/L \end{bmatrix}$$



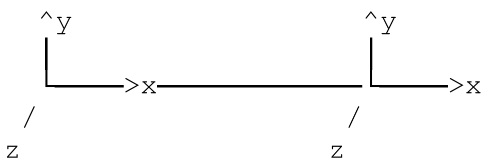
Unit rotation at end about z axis

Assembling all six columns:

$$\begin{bmatrix} \frac{EAx}{L} & 0 & 0 & \frac{-EAx}{L} & 0 & 0 \\ & \frac{12EIz.S}{L^3} & \frac{6EIz.S}{L^2} & 0 & \frac{-12EIz.S}{L^3} & \frac{6EIz.S}{L^2} \\ & & \frac{EIz(1+3S)}{L} & 0 & \frac{-6EIz.S}{L^2} & \frac{EIz(3S-1)}{L} \\ & & & \frac{EAx}{L} & 0 & 0 \\ & & & & \frac{12EIz.S}{L^3} & \frac{-6EIz.S}{L^2} \\ & & & & & \frac{EIz(1+3S)}{L} \end{bmatrix}$$

Shear deformation coefficient is:

$$S = \frac{1}{1 + \frac{12EIz}{L^2.GAy}}$$



Stiffness matrix for a member of a plane frame in member axes with full moment fixity at start and end of member (matrix is symmetrical about the diagonal)

7.7.2.2 Both ends pinned

Consider unit displacement Dxj in isolation then for pinned ends:

1 = Fxj.L/AxE hence Fxj = EAx/L

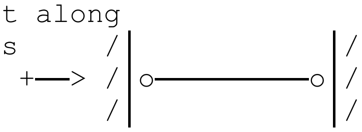
Resolving along member x axis Fxk = -Fxj = -EAx/L

As displacements in y direction and rotations about z are all zero
Fyj = Fyk = Mzj = Mzk = 0

Therefore first column of member stiffness matrix =

$$\begin{bmatrix} EAx/L \\ 0 \\ 0 \\ -EAx/L \\ 0 \\ 0 \end{bmatrix}$$

Unit displacement at start along x axis



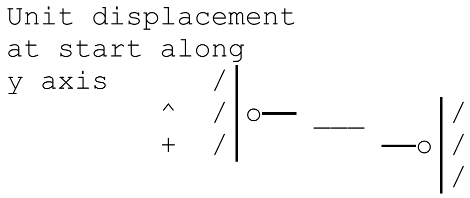
Consider unit displacement Dyj in isolation then for pinned ends:

Mzj = Mzk = Fyj = Fyk = 0

As displacements in x direction are all zero Fxj = Fxk = 0

Therefore second column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

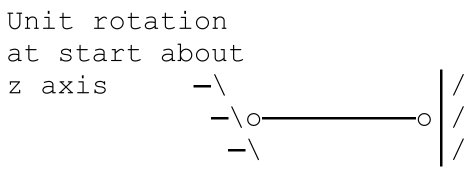


Consider unit rotation Rzj in isolation then for pinned ends:
Mzj = Mzk = Fyj = Fyk = 0

As displacements in x direction are all zero Fxj = Fxk = 0

Therefore third column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$



Consider unit displacement Dxk in isolation then for pinned ends:

1 = Fxk.L/AxE hence Fxk = EAx/L

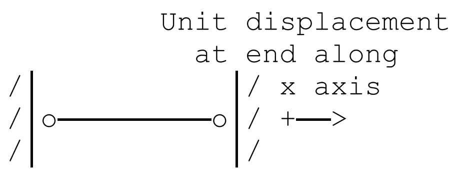
Resolving along member x axis Fxj = -Fxk = -EAx/L

As displacements in y direction and rotations about z are all zero

Fyj = Fyk = Mzj = Mzk = 0

Therefore fourth column of member stiffness matrix =

$$\begin{bmatrix} -EAx/L \\ 0 \\ 0 \\ EAx/L \\ 0 \\ 0 \end{bmatrix}$$

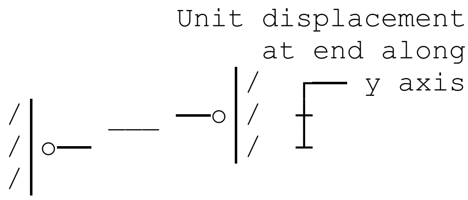


Consider unit displacement Dyk in isolation then for pinned ends:
Mzk = Mzj = Fyk = Fyj = 0

As displacements in x direction are all zero Fxk = Fxj = 0

Therefore fifth column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$



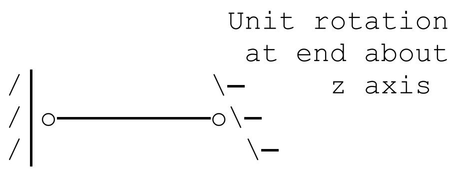
Consider unit rotation Rzk in isolation then for pinned ends:

Mzk = Mzj = Fyk = Fyj = 0

As displacements in x direction are all zero Fxk = Fxj = 0

Therefore sixth column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$



Assembling all six columns:

$$
\begin{bmatrix}
\frac{EAx}{L} & 0 & 0 & \frac{-EAx}{L} & 0 & 0 \\
 & 0 & 0 & 0 & 0 & 0 \\
 & & 0 & 0 & 0 & 0 \\
 & & & \frac{EAx}{L} & 0 & 0 \\
 & & & & 0 & 0 \\
 & & & & & 0
\end{bmatrix}
$$

```
   ^y                ^y
   |                 |
   |____>x___________|____>x
  /                 /
 z                 z
```

Stiffness matrix for a member of a plane frame in member axes with both ends pinned
(matrix is symmetrical about the diagonal)

7.7.2.3 One end pinned the other fixed

Consider unit displacement Dxj in isolation then for LH pinned and RH fixed:

$$1 = Fxj.L/AxE \text{ hence } Fxj = EAx/L$$

Resolving along member x axis Fxk = -Fxj = -EAx/L

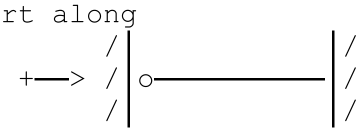
As displacements in y direction and rotations about z are all zero

Fyj = Fyk = Mzj = Mzk = 0

Therefore first column of member stiffness matrix =

$$\begin{bmatrix} EAx/L \\ 0 \\ 0 \\ -EAx/L \\ 0 \\ 0 \end{bmatrix}$$

Unit displacement at start along x axis



Consider unit displacement Dyj in isolation then for LH pinned and RH fixed:

Mzj = 0

$$Mzk = \frac{3EIz.S}{L^2}$$ (see 7.7.1.2) where $$S = \frac{1}{1 + \dfrac{3EIz}{L^2.GAy}}$$

Taking moments about k end

$$Mzk = Fyj.L \quad \text{therefore} \quad Fyj = \frac{3EIz.S}{L^2} \cdot \frac{1}{L} = \frac{3EIz.S}{L^3}$$

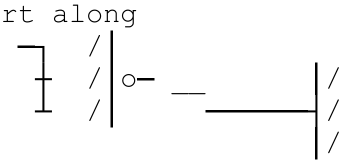
$$\text{Resolving along member y axis} \quad Fyk = -Fyj = \frac{-3EIz.S}{L^3}$$

As displacements in x direction are all zero Fxj = Fxk = 0

Therefore second column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ 3EIzS/L^3 \\ 0 \\ 0 \\ -3EIzS/L^3 \\ 3EIzS/L^2 \end{bmatrix}$$

Unit displacement at start along y axis



Consider unit rotation Rzj in isolation then for LH pinned, RH fixed:

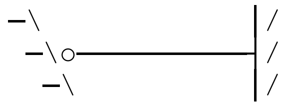
Mzj = Mzk = Fyj = Fyk = 0

As displacements in x direction are all zero Fxj = Fxk = 0

Therefore third column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

Unit rotation at start about z axis



Consider unit displacement Dxk in isolation then for LH pinned and RH fixed:

1 = Fxk.L/AxE hence Fxk = EAx/L

Resolving along member x axis Fxj = -Fxk = -EAx/L

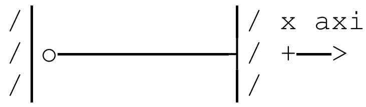
As displacements in y direction and rotations about z are all zero

Fyj = Fyk = Mzj = Mzk = 0

Therefore fourth column of member stiffness matrix =

$$\begin{bmatrix} -EAx/L \\ 0 \\ 0 \\ EAx/L \\ 0 \\ 0 \end{bmatrix}$$

Unit displacement at end along x axis



Consider unit displacement Dyk in isolation then for LH pinned and RH fixed:

Mzj = 0

$$Mzk = \frac{-3EIz.S}{L^2} \quad \text{(see 7.7.1.2)} \quad \text{where} \quad S = \frac{1}{1 + \dfrac{3EIz}{L^2.GAy}}$$

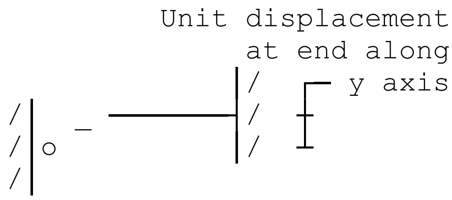
$$Fyj = \frac{-3EIz.S}{L^3} \quad \text{(see 7.7.1.2)}$$

$$\text{Resolving along member y axis} \quad Fyk = -Fyj = \frac{3EIz.S}{L^3}$$

As displacements in x direction are all zero Fxk = Fxj = 0

Therefore fifth column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ -3EIzS/L^3 \\ 0 \\ 0 \\ 3EIzS/L^3 \\ -3EIzS/L^2 \end{bmatrix}$$



Consider unit rotation Rzk in isolation then for LH end pinned and RH fixed:

$$Mzk = \frac{-3EIz.S}{L} \quad \text{(see 7.7.1.1)} \quad \text{where} \quad S = \frac{1}{1 + \frac{3EIz}{L^2.GAy}}$$

$$Mzj = 0$$

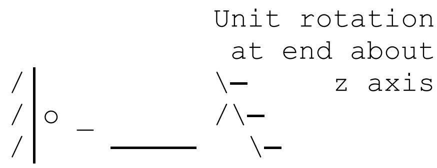
$$Fyk = \frac{-3EIz.S}{L^2} \quad \text{(see 7.7.1.1)}$$

Resolving along member y axis $Fyj = -Fyk = \frac{+3EIz.S}{L^2}$

As displacements in x direction are all zero Fxk = Fxj = 0

Therefore sixth column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ 3EIzS/L^2 \\ 0 \\ 0 \\ -3EIzS/L^2 \\ 3EIzS/L \end{bmatrix}$$

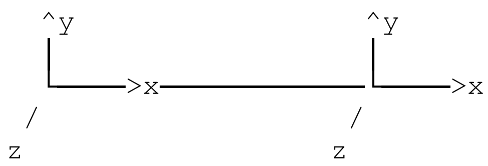


Assembling all six columns:

$$
\begin{bmatrix}
\frac{EAx}{L} & 0 & 0 & \frac{-EAx}{L} & 0 & 0 \\
 & \frac{3EIz.S}{L^3} & 0 & 0 & \frac{-3EIz.S}{L^3} & \frac{3EIz.S}{L^2} \\
 & & 0 & 0 & 0 & 0 \\
 & & & \frac{EAx}{L} & 0 & 0 \\
 & & & & \frac{3EIz.S}{L^3} & \frac{-3EIz.S}{L^2} \\
 & & & & & \frac{3EIz.S}{L}
\end{bmatrix}
$$

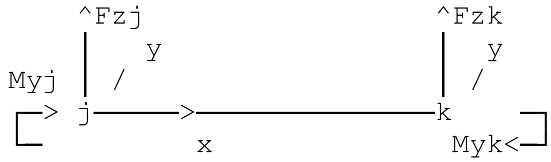
Shear deformation coefficient is:

$$S = \frac{1}{1 + \frac{3EIz}{L^2.GAy}}$$



Stiffness matrix for a member of a plane frame in member axes with LH end pinned and RH end fixed (matrix is symmetrical about the diagonal)

7.7.3 Member stiffness matrix for grids



Consider member jk: where j is 'start end' and k is 'end end' of the member.

Let:
Mxj = moment on member at start end about local x axis
Myj = ... y .... (y into paper)
Fzj = force .... in direction of local z axis
Mxk = moment on member at end end about local x axis
Myk = ... y .... (y into paper)
Fzk = force .... in direction of local z axis
Rxj = rotation at member start about local x axis
Ryj = ... y ....
Dzj = displacement .... in direction of local z axis
Rxk = rotation .... end about local x axis
Ryk = ... y ....
Dzk = displacement .... in direction of local z axis

These forces and displacements are related by:

$$\begin{bmatrix} Mxj \\ Myj \\ Fzj \\ Mxk \\ Myk \\ Fzk \end{bmatrix} = \begin{bmatrix} (\ Sm11\ ) & (\ Sm12\ ) & Sm13 & Sm14 & Sm15 & Sm16 \\ (\ Sm21\ ) & (\ Sm22\ ) & Sm23 & Sm24 & Sm25 & Sm26 \\ (\ Sm31\ ) & (\ Sm32\ ) & Sm33 & Sm34 & Sm35 & Sm36 \\ (\ Sm41\ ) & (\ Sm42\ ) & Sm43 & Sm44 & Sm45 & Sm46 \\ (\ Sm51\ ) & (\ Sm52\ ) & Sm53 & Sm54 & Sm55 & Sm56 \\ (\ Sm61\ ) & (\ Sm62\ ) & Sm63 & Sm64 & Sm65 & Sm66 \end{bmatrix} \begin{bmatrix} Rxj \\ Ryj \\ Dzj \\ Rxk \\ Ryk \\ Dzk \end{bmatrix}$$

Vector of forces resulting from unit rotation Rxj (first column)

Vector of forces resulting from unit rotation Ryj (second column)

For the six displacements Rxj,Ryj,Dzj,Rxk,Ryk,Dzk the corresponding column of forces due to unit displacement will be assembled from the component terms derived in section 7.7.1 used together with the stress:strain ratio in torsion:

$$Rx = \frac{Mx.L}{Ix.G}$$

where:

Rx is rotation of member about x axis measured in radians
Mx is torque applied to the member (about member x axis)
L is length of member
Ix is torsional constant (polar inertia for circular section)
G is modulus of rigidity (shear modulus)

7.7.3.1 Both ends fixed

Comparison of many of the terms with those previously derived for the plane frame shows a change of sign. This change of sign is due to the grid member y axis going 'into the paper' whereas the member z axis in plane frames comes 'out of the paper'.

Consider unit rotation Rxj in isolation then for no releases:

1 = Mxj.L/IxG hence Mxj = GIx/L

Resolving about member x axis Mxk = -Mxj = -GIx/L

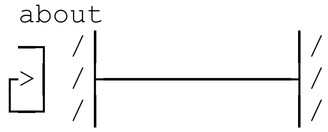
As displacements in z direction and rotations about y are all zero

Fzj = Fzk = Myj = Myk = 0

Therefore first column of member stiffness matrix =

$$\begin{bmatrix} GIx/L \\ 0 \\ 0 \\ -GIx/L \\ 0 \\ 0 \end{bmatrix}$$

Unit rotation at start about x axis



Consider unit rotation Ryj in isolation then for no releases:

$$Myj = \frac{EIy}{L} + \frac{3EIy.S}{L} \quad \text{where} \quad S = \frac{1}{1 + \frac{12EIy}{L^2.GAz}}$$ (see 7.7.1.4)

$$Myk = -\frac{EIy}{L} + \frac{3EIy.S}{L} \quad \text{where} \quad S = \frac{1}{1 + \frac{12EIy}{L^2.GAz}}$$ (see 7.7.1.4)

Taking moments about k end

$$Myj + Myk + Fzj.L = 0 \quad \text{therefore} \quad Fzj = \frac{-6EIy.S}{L} . \frac{1}{L} = \frac{-6EIy.S}{L^2}$$

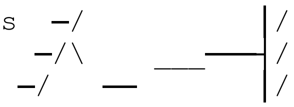
Resolving along member z axis $Fzk = -Fzj = \frac{+6EIy.S}{L^2}$

As rotations about x axis are all zero $Mxj = Mxk = 0$

Therefore second column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ EIy/L+3EIyS/L \\ -6EIyS/L^2 \\ 0 \\ -EIy/L+3EIyS/L \\ 6EIyS/L^2 \end{bmatrix}$$

Unit rotation at start about y axis



Consider unit displacement Dzj in isolation then for no releases:

$$Myj = -\frac{6EIy.S}{L^2} \quad \text{where} \quad S = \frac{1}{1 + \frac{12EIy}{L^2.GAz}}$$ (see 7.7.1.3)

$Myk = Myj$

Taking moments about k end

$$Myj + Myk + Fzj.L = 0 \quad \text{therefore} \quad Fzj = \frac{12EIy.S}{L^2} . \frac{1}{L} = \frac{12EIy.S}{L^3}$$

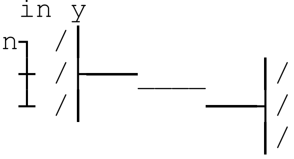
Resolving along member z axis $Fzk = -Fzj = \frac{-12EIy.S}{L^3}$

As rotations about x axis are all zero Mxj = Mxk = 0

Therefore third column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ -6EIyS/L^2 \\ 12EIyS/L^3 \\ 0 \\ -6EIyS/L^2 \\ -12EIyS/L^3 \end{bmatrix}$$

Unit displacement at start in y direction



Consider unit rotation Rxk in isolation then for no releases:
1 = Mxk.L/IxG hence Mxk = GIx/L

Resolving about member x axis Mxj = -Mxk = -GIx/L

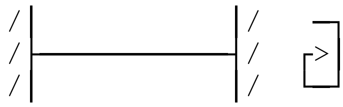
As displacements in z direction and rotations about y are all zero

Fzj = Fzk = Myj = Myk = 0

Therefore fourth column of member stiffness matrix =

$$\begin{bmatrix} -GIx/L \\ 0 \\ 0 \\ GIx/L \\ 0 \\ 0 \end{bmatrix}$$

Unit rotation at end about x axis



Consider unit rotation Ryk in isolation then for no releases:

$$Myk = \frac{EIy}{L} + \frac{3EIy.S}{L} \quad \text{where} \quad S = \frac{1}{1 + \dfrac{12EIy}{L^2.GAz}}$$ (see 7.7.1.4)

$$Myj = -\frac{EIy}{L} + \frac{3EIy.S}{L} \quad \text{where} \quad S = \frac{1}{1 + \dfrac{12EIy}{L^2.GAz}}$$ (see 7.7.1.4)

Taking moments about j end

$$Myk + Myj = Fzk.L \quad \text{therefore} \quad Fzk = \frac{6EIy.S}{L^2}$$

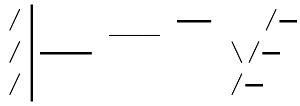
Resolving along member z axis $Fzj = -Fyk = \dfrac{-6EIy.S}{L^2}$

As rotations about x axis are all zero $Mxk = Mxj = 0$

Therefore fifth column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ -EIy/L+3EIyS/L \\ -6EIyS/L^2 \\ 0 \\ EIy/L+3EIyS/L \\ 6EIyS/L^2 \end{bmatrix}$$

Unit rotation at end about y axis



Consider unit displacement Dzk in isolation then for no releases:

$Myk = \dfrac{6EIy.S}{L^2}$ where $S = \dfrac{1}{1 + \dfrac{12EIy}{L^2.GAz}}$ (see 7.7.1.3)

$Myj = Myk$

Taking moments about j end

$Myk + Myj = Fzk.L$ therefore $Fzk = \dfrac{12EIy.S}{L^2} . \dfrac{1}{L} = \dfrac{12EIy.S}{L^3}$

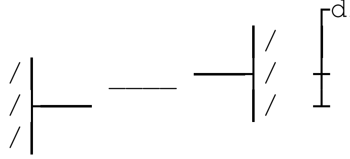
Resolving along member z axis $Fzj = -Fzk = \dfrac{-12EIy.S}{L^3}$

As rotations about x axis are all zero $Mxk = Mxj = 0$

Therefore sixth column of member stiffness matrix =

$$\begin{bmatrix} 0 \\ 6EIyS/L^2 \\ -12EIyS/L^3 \\ 0 \\ 6EIyS/L^2 \\ 12EIyS/L^3 \end{bmatrix}$$

Unit displacement at end in y directn



Proceeding in a similar manner to that for plane frames, the stiffness matrices for various combinations of end moment releases may be derived. For convenience typical matrices are given on the following pages.

$$
\begin{bmatrix}
\frac{GIx}{L} & 0 & 0 & \frac{-GIx}{L} & 0 & 0 \\
 & \frac{EIy(1+3S)}{L} & \frac{-6EIy.S}{L^2} & 0 & \frac{EIy(3S-1)}{L} & \frac{6EIy.S}{L^2} \\
 & & \frac{12EIy.S}{L^3} & 0 & \frac{-6EIy.S}{L^2} & \frac{-12EIy.S}{L^3} \\
 & & & \frac{GIx}{L} & 0 & 0 \\
 & & & & \frac{EIy(1+3S)}{L} & \frac{6EIy.S}{L^2} \\
 & & & & & \frac{12EIy.S}{L^3}
\end{bmatrix}
$$

Shear deformation coefficient is:

$$S = \frac{1}{1 + \frac{12EIy}{L^2.GAz}}$$

```
^z  y               ^z  y
|  /                |  /
|_____>x____________|_____>x
```

Stiffness matrix for a member of a plane grid in member axes
with full moment fixity at start and end of member
(matrix is symmetrical about the diagonal)

$$
\begin{bmatrix}
\frac{GIx}{L} & 0 & 0 & \frac{-GIx}{L} & 0 & 0 \\
 & 0 & 0 & 0 & 0 & 0 \\
 & & 0 & 0 & 0 & 0 \\
 & & & \frac{GIx}{L} & 0 & 0 \\
 & & & & 0 & 0 \\
 & & & & & 0
\end{bmatrix}
$$

```
^z  y               ^z  y
|  /                |  /
|_____>x____________|_____>x
```

Stiffness matrix for a member of a plane grid in member axes
with both ends pinned for bending about the y axis
(matrix is symmetrical about the diagonal)

$$
\begin{bmatrix}
\frac{GIx}{L} & 0 & 0 & \frac{-GIx}{L} & 0 & 0 \\
 & 0 & 0 & 0 & 0 & 0 \\
 & & \frac{3EIy.S}{L^3} & 0 & \frac{-3EIy.S}{L^2} & \frac{-3EIy.S}{L^3} \\
 & & & \frac{GIx}{L} & 0 & 0 \\
 & & & & \frac{3EIy.S}{L} & \frac{3EIy.S}{L^2} \\
 & & & & & \frac{3EIy.S}{L^3}
\end{bmatrix}
$$

Shear deformation coefficient is:

$$S = \frac{1}{1 + \frac{3EIy}{L^2.GAz}}$$

```
^z  y                ^z  y
|  /                 |  /
|____>x______________|____>x
```

Stiffness matrix for a member of a plane grid in member axes with LH end pinned and RH end fixed for bending about y axis (matrix is symmetrical about the diagonal)

7.7.4 Member stiffness matrix for space frames

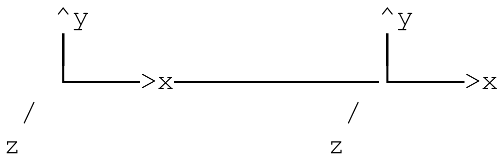
By combining the plane frame and grid member stiffness matrix terms, the space frame member stiffness matrices may be obtained for various end conditions.

For reference, typical matrices are given on subsequent pages.

$$
\begin{bmatrix}
\frac{EAx}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-EAx}{L} & 0 & 0 & 0 & 0 & 0 \\
 & \frac{12EIz.S}{L^3} & 0 & 0 & 0 & \frac{6EIz.S}{L^2} & 0 & \frac{-12EIz.S}{L^3} & 0 & 0 & 0 & \frac{6EIz.S}{L^2} \\
 & & \frac{12EIy.T}{L^3} & 0 & \frac{-6EIy.T}{L^2} & 0 & 0 & 0 & \frac{-12EIy.T}{L^3} & 0 & \frac{-6EIy.T}{L^2} & 0 \\
 & & & \frac{GIx}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-GIx}{L} & 0 & 0 \\
 & & & & \frac{EIy(1+3T)}{L} & 0 & 0 & 0 & \frac{6EIy.T}{L^2} & 0 & \frac{EIy(3T-1)}{L} & 0 \\
 & & & & & \frac{EIz(1+3S)}{L} & 0 & \frac{-6EIz.S}{L^2} & 0 & 0 & 0 & \frac{EIz(3S-1)}{L} \\
 & & & & & & \frac{EAx}{L} & 0 & 0 & 0 & 0 & 0 \\
 & & & & & & & \frac{12EIz.S}{L^3} & 0 & 0 & 0 & \frac{-6EIz.S}{L^2} \\
 & & & & & & & & \frac{12EIy.T}{L^3} & 0 & \frac{6EIy.T}{L^2} & 0 \\
 & & & & & & & & & \frac{GIx}{L} & 0 & 0 \\
 & & & & & & & & & & \frac{EIy(1+3T)}{L} & 0 \\
 & & & & & & & & & & & \frac{EIz(1+3S)}{L}
\end{bmatrix}
$$

Shear deformation coefficients are:

$$S = \frac{1}{1 + \frac{12EIz}{L^2.GAy}} \qquad T = \frac{1}{1 + \frac{12EIy}{L^2.GAz}}$$

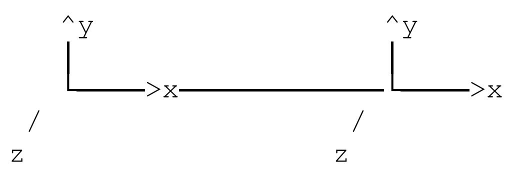


Stiffness matrix for a member of a space frame in member axes with full moment fixity at start and end of the member (matrix is symmetrical about the diagonal)

$$
\begin{bmatrix}
\frac{EAx}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-EAx}{L} & 0 & 0 & 0 & 0 & 0 \\
 & \frac{12EIz.S}{L^3} & 0 & 0 & 0 & \frac{6EIz.S}{L^2} & 0 & \frac{-12EIz.S}{L^3} & 0 & 0 & 0 & \frac{6EIz.S}{L^2} \\
 & & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 & & & \frac{GIx}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-GIx}{L} & 0 & 0 \\
 & & & & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 & & & & & \frac{EIz(1+3S)}{L} & 0 & \frac{-6EIz.S}{L^2} & 0 & 0 & 0 & \frac{EIz(3S-1)}{L} \\
 & & & & & & \frac{EAx}{L} & 0 & 0 & 0 & 0 & 0 \\
 & & & & & & & \frac{12EIz.S}{L^3} & 0 & 0 & 0 & \frac{-6EIz.S}{L^2} \\
 & & & & & & & & 0 & 0 & 0 & 0 \\
 & & & & & & & & & \frac{GIx}{L} & 0 & 0 \\
 & & & & & & & & & & 0 & 0 \\
 & & & & & & & & & & & \frac{EIz(1+3S)}{L}
\end{bmatrix}
$$

Shear deformation coefficient is:

$$S = \frac{1}{1 + \frac{12EIz}{L^2.GAy}}$$

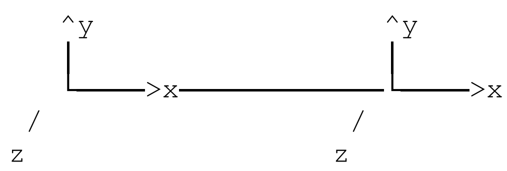


Stiffness matrix for a member of a space frame in member axes
with MOMENT Y release at start and end of the member
(matrix is symmetrical about the diagonal)

$$
\begin{bmatrix}
\frac{EA_x}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-EA_x}{L} & 0 & 0 & 0 & 0 & 0 \\
 & \frac{12EI_z.S}{L^3} & 0 & 0 & 0 & \frac{6EI_z.S}{L^2} & 0 & \frac{-12EI_z.S}{L^3} & 0 & 0 & 0 & \frac{6EI_z.S}{L^2} \\
 & & \frac{3EI_y.T}{L^3} & 0 & 0 & 0 & 0 & 0 & \frac{-3EI_y.T}{L^3} & 0 & \frac{-3EI_y.T}{L^2} & 0 \\
 & & & \frac{GI_x}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-GI_x}{L} & 0 & 0 \\
 & & & & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 & & & & & \frac{EI_z(1+3S)}{L} & 0 & \frac{-6EI_z.S}{L^2} & 0 & 0 & 0 & \frac{EI_z(3S-1)}{L} \\
 & & & & & & \frac{EA_x}{L} & 0 & 0 & 0 & 0 & 0 \\
 & & & & & & & \frac{12EI_z.S}{L^3} & 0 & 0 & 0 & \frac{-6EI_z.S}{L^2} \\
 & & & & & & & & \frac{3EI_y.T}{L^3} & 0 & \frac{3EI_y.T}{L^2} & 0 \\
 & & & & & & & & & \frac{GI_x}{L} & 0 & 0 \\
 & & & & & & & & & & \frac{3EI_y.T}{L} & 0 \\
 & & & & & & & & & & & \frac{EI_z(1+3S)}{L}
\end{bmatrix}
$$

Shear deformation coefficients are:

$$S = \frac{1}{1 + \frac{12EI_z}{L^2.GA_y}} \qquad T = \frac{1}{1 + \frac{3EI_y}{L^2.GA_z}}$$

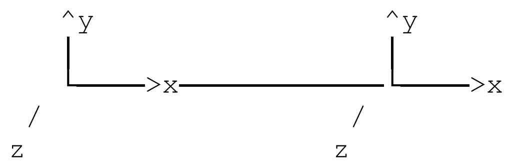


Stiffness matrix for a member of a space frame in member axes
with MOMENT Y release at start of the member
(matrix is symmetrical about the diagonal)

$$
\begin{bmatrix}
\frac{EAx}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-EAx}{L} & 0 & 0 & 0 & 0 & 0 \\
 & \frac{12EIz.S}{L^3} & 0 & 0 & 0 & \frac{6EIz.S}{L^2} & 0 & \frac{-12EIz.S}{L^3} & 0 & 0 & 0 & \frac{6EIz.S}{L^2} \\
 & & \frac{3EIy.T}{L^3} & 0 & \frac{-3EIy.T}{L^2} & 0 & 0 & 0 & \frac{-3EIy.T}{L^3} & 0 & 0 & 0 \\
 & & & \frac{GIx}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-GIx}{L} & 0 & 0 \\
 & & & & \frac{3EIy.T}{L} & 0 & 0 & 0 & \frac{3EIy.T}{L^2} & 0 & 0 & 0 \\
 & & & & & \frac{EIz(1+3S)}{L} & 0 & \frac{-6EIz.S}{L^2} & 0 & 0 & 0 & \frac{EIz(3S-1)}{L} \\
 & & & & & & \frac{EAx}{L} & 0 & 0 & 0 & 0 & 0 \\
 & & & & & & & \frac{12EIz.S}{L^3} & 0 & 0 & 0 & \frac{-6EIz.S}{L^2} \\
 & & & & & & & & \frac{3EIy.T}{L^3} & 0 & 0 & 0 \\
 & & & & & & & & & \frac{GIx}{L} & 0 & 0 \\
 & & & & & & & & & & 0 & 0 \\
 & & & & & & & & & & & \frac{EIz(1+3S)}{L}
\end{bmatrix}
$$

Shear deformation coefficients are:

$$S = \frac{1}{1 + \frac{12EIz}{L^2.GAy}} \qquad T = \frac{1}{1 + \frac{3EIy}{L^2.GAz}}$$

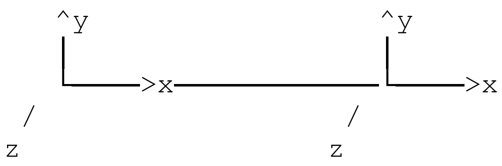


Stiffness matrix for a member of a space frame in member axes
with MOMENT Y release at end of the member
(matrix is symmetrical about the diagonal)

$$
\begin{bmatrix}
\frac{EAx}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-EAx}{L} & 0 & 0 & 0 & 0 & 0 \\
 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 & & \frac{12EIy.T}{L^3} & 0 & \frac{-6EIy.T}{L^2} & 0 & 0 & 0 & \frac{-12EIy.T}{L^3} & 0 & \frac{-6EIy.T}{L^2} & 0 \\
 & & & \frac{GIx}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-GIx}{L} & 0 & 0 \\
 & & & & \frac{EIy(1+3T)}{L} & 0 & 0 & 0 & \frac{6EIy.T}{L^2} & 0 & \frac{EIy(3T-1)}{L} & 0 \\
 & & & & & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 & & & & & & \frac{EAx}{L} & 0 & 0 & 0 & 0 & 0 \\
 & & & & & & & 0 & 0 & 0 & 0 & 0 \\
 & & & & & & & & \frac{12EIy.T}{L^3} & 0 & \frac{6EIy.T}{L^2} & 0 \\
 & & & & & & & & & \frac{GIx}{L} & 0 & 0 \\
 & & & & & & & & & & \frac{EIy(1+3T)}{L} & 0 \\
 & & & & & & & & & & & 0
\end{bmatrix}
$$

Shear deformation coefficient is:

$$T = \frac{1}{1 + \frac{12EIy}{L^2.GAz}}$$

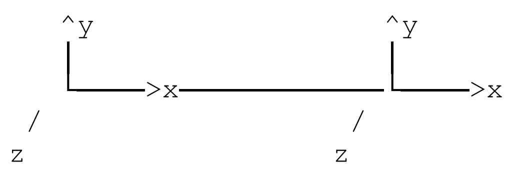


Stiffness matrix for a member of a space frame in member axes with MOMENT Z release at start and end of the member (matrix is symmetrical about the diagonal)

$$
\begin{bmatrix}
\frac{EAx}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-EAx}{L} & 0 & 0 & 0 & 0 & 0 \\
 & \frac{3EIz.S}{L^3} & 0 & 0 & 0 & 0 & 0 & \frac{-3EIz.S}{L^3} & 0 & 0 & 0 & \frac{3EIz.S}{L^2} \\
 & & \frac{12EIy.T}{L^3} & 0 & \frac{-6EIy.T}{L^2} & 0 & 0 & 0 & \frac{-12EIy.T}{L^3} & 0 & \frac{-6EIy.T}{L^2} & 0 \\
 & & & \frac{GIx}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-GIx}{L} & 0 & 0 \\
 & & & & \frac{EIy(1+3T)}{L} & 0 & 0 & 0 & \frac{6EIy.T}{L^2} & 0 & \frac{EIy(3T-1)}{L} & 0 \\
 & & & & & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 & & & & & & \frac{EAx}{L} & 0 & 0 & 0 & 0 & 0 \\
 & & & & & & & \frac{3EIz.S}{L^3} & 0 & 0 & 0 & \frac{-3EIz.S}{L^2} \\
 & & & & & & & & \frac{12EIy.T}{L^3} & 0 & \frac{6EIy.T}{L^2} & 0 \\
 & & & & & & & & & \frac{GIx}{L} & 0 & 0 \\
 & & & & & & & & & & \frac{EIy(1+3T)}{L} & 0 \\
 & & & & & & & & & & & \frac{3EIz.S}{L}
\end{bmatrix}
$$

Shear deformation coefficients are:

$$S = \frac{1}{1 + \frac{3EIz}{L^2.GAy}} \qquad T = \frac{1}{1 + \frac{12EIy}{L^2.GAz}}$$



Stiffness matrix for a member of a space frame in member axes
with MOMENT Z release at start of the member
(matrix is symmetrical about the diagonal)

$$
\begin{bmatrix}
\frac{EAx}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-EAx}{L} & 0 & 0 & 0 & 0 & 0 \\
 & \frac{3EIz.S}{L^3} & 0 & 0 & 0 & \frac{3EIz.S}{L^2} & 0 & \frac{-3EIz.S}{L^3} & 0 & 0 & 0 & 0 \\
 & & \frac{12EIy.T}{L^3} & 0 & \frac{-6EIy.T}{L^2} & 0 & 0 & 0 & \frac{-12EIy.T}{L^3} & 0 & \frac{-6EIy.T}{L^2} & 0 \\
 & & & \frac{GIx}{L} & 0 & 0 & 0 & 0 & 0 & \frac{-GIx}{L} & 0 & 0 \\
 & & & & \frac{EIy(1+3T)}{L} & 0 & 0 & 0 & \frac{6EIy.T}{L^2} & 0 & \frac{EIy(3T-1)}{L} & 0 \\
 & & & & & \frac{3EIz.S}{L} & 0 & \frac{-3EIz.S}{L^2} & 0 & 0 & 0 & 0 \\
 & & & & & & \frac{EAx}{L} & 0 & 0 & 0 & 0 & 0 \\
 & & & & & & & \frac{3EIz.S}{L^3} & 0 & 0 & 0 & 0 \\
 & & & & & & & & \frac{12EIy.T}{L^3} & 0 & \frac{6EIy.T}{L^2} & 0 \\
 & & & & & & & & & \frac{GIx}{L} & 0 & 0 \\
 & & & & & & & & & & \frac{EIy(1+3T)}{L} & 0 \\
 & & & & & & & & & & & 0
\end{bmatrix}
$$

Shear deformation coefficients are:

$$S = \frac{1}{1 + \frac{3EIz}{L^2.GAy}} \qquad T = \frac{1}{1 + \frac{12EIy}{L^2.GAz}}$$

```
   ^y                  ^y
   |                   |
   |____>x_____________|____>x
  /                   /
 z                   z
```

Stiffness matrix for a member of a space frame in member axes
with MOMENT Z release at end of the member
(matrix is symmetrical about the diagonal)

7.7.5 Forces due to member loads for plane frames

Member ends may be pinned/pinned, pinned/fixed, fixed/pinned or fixed/fixed. Less computation is required if the ends are assumed pinned/pinned and then modified for any fixity; rather than assuming them fixed/fixed and modifying them for any releases.

Member stiffness matrices have already been computed taking due allowance for shear deformation.

The procedure for the computation of member force vectors to model for an applied member load is:

a) Compute member end rotations for the pinned/pinned case

b) Use member stiffness coefficients to compute moments and shears to reduce end rotations to zero if moment fixity is present at the end/s.

c) Add computed shears to pinned-end shears to obtain the force vector for the member load under consideration.

Subsequently the force vectors for all the loads on a given member are cumulated, rotated to global axes and added to the joint force vector.

7.7.5.1 End rotations for a simply supported beam under UDL

```
            w
     A vvvvvvvvvvvvvvvvvv B
       ____________________
Ra=wL  ^                  ^
 --    |        L         |
  2    <------------------>
         x
       +----->
       ____________________
       \     |             /
         \   |M1    (+)   /
           \v            /
              - - - -
       M1=w.L.x/2-w.x^2/2

         ____________________
 ->     |    ^m      _   -
|  1    |(+) v   -
 --     | -     m=(L-x)/L

        | \  v
        |(+) \
        |-----R1-\-----------
             ^     \         |
       R1=w.L/2-w.x  \(-)    |
```

By Castigliano:

$$\text{Thet'A} = \int_0^L \frac{M1.m.dx}{EI} + \int_0^L \frac{R1.r.dx}{GA}$$

where:

M1 = bending moment at any section of the beam due to given loading

R1 = shearing force at any section of the beam due to given loading

m = bending moment at any section of the beam due to unit load applied at A in direction and sense of thet'A (reqd. defln.)

r = shearing force at any section of the beam due to unit load applied at A in direction and sense of thet'A (reqd. defln.)

```
         ┌─────v──────────────┐
    1    │     r      (−)     │
         └────────────────────┘
               ^
             r=-1/L
```

(Sign convention - sagging moments + shearing forces +ve for increasing moment)

$$\int_0^L \frac{M1.m.dx}{EIz} = \int_0^L \left[\frac{w.L.x}{2} - \frac{w.x^2}{2}\right]\left[1 - \frac{x}{L}\right]\frac{.dx}{EIz}$$

$$= \int_0^L \left[\frac{w.L.x}{2} - \frac{w.x^2}{2} - \frac{w.L.x^2}{2.L} + \frac{w.x^3}{2.L}\right]\frac{.dx}{EIz}$$

$$= \frac{1}{EIz}\left[\frac{w.L.x^2}{4} - \frac{w.x^3}{6} - \frac{w.L.x^3}{6.L} + \frac{w.x^4}{8.L}\right]_0^L$$

$$= \frac{1}{EIz}\left[\frac{w.L^3}{4} - \frac{w.L^3}{6} - \frac{w.L^3}{6} + \frac{w.L^3}{8}\right] = \frac{w.L^3}{24.EIz}$$

$$\int_0^L \frac{R1.r.dx}{GA} = \int_0^L \left[\frac{w.L}{2} - w.x\right]\left[\frac{-1}{L}\right]\frac{.dx}{GA} = \int_0^L \left[\frac{-w}{2} + \frac{w.x}{L}\right]\frac{.dx}{GA}$$

$$= \frac{1}{GA}\left[\frac{-wx}{2} + \frac{w.x^2}{2.L}\right]_0^L = \frac{1}{GA}\left[\frac{-w.L}{2} + \frac{w.L^2}{2.L}\right] = 0$$

Summary for UDL allowing for shear deformation

```
                    w ( +ve down)
                vvvvvvvvvvvvvvvvvvv
                ───────────────────
                ^\ ) thet'A     /^
        w.L/2   |  \           /  |  w.L/2
                     ─ ─ ─ ─
                         L
                <─────────────────>
```

Rotation at A: $thet'A = \frac{w.L^3}{24.EIz}$

For the fully fixed case we must apply an anti-clockwise rotation = w.L^3/(24.EIz) at A and a clockwise rotation = w.L^3/(24.EIz) at B.

i.e. $thet'ZA = +\frac{w.L^3}{24.EIz}$ and $thet'ZB = -\frac{w.L^3}{24.EIz}$

writing $S = \dfrac{1}{1 + \dfrac{12.EIz}{L^2.GAy}}$ into member stiffness matrix for plane frame fixed end case gives:

$$
\begin{bmatrix} RxA \\ RyA \\ MzA \\ RxB \\ RyB \\ MzB \end{bmatrix}
\begin{bmatrix}
\frac{EAx}{L} & 0 & 0 & \frac{-EAx}{L} & 0 & 0 \\
 & \frac{12EIz.S}{L^3} & \frac{6EIz.S}{L^2} & 0 & \frac{-12EIz.S}{L^3} & \frac{6EIz.S}{L^2} \\
 & & \frac{EIz(1+3S)}{L} & 0 & \frac{-6EIz.S}{L^2} & \frac{EIz(3S-1)}{L} \\
 & & & \frac{EAx}{L} & 0 & 0 \\
 & \text{Member stiffness} & & & \frac{12EIz.S}{L^3} & \frac{-6EIz.S}{L^2} \\
 & \text{matrix is symmetric} & & & & \\
 & \text{about diagonal} & & & & \frac{EIz(1+3S)}{L}
\end{bmatrix}
\begin{bmatrix} 0 \\ 0 \\ \frac{wL^3}{24EIz} \\ 0 \\ 0 \\ \frac{-wL^3}{24EIz} \end{bmatrix}
$$

From above:

$$MzA = \left[\frac{EIz}{L} + \frac{3.EIz.S}{L}\right]\left[\frac{w.L^3}{24EIz}\right] + \left[\frac{-EIz}{L} + \frac{3EIzS}{L}\right]\left[\frac{-wL^3}{24EIz}\right]$$

$$= \frac{wL^2}{24} + \frac{wL^2.S}{8} + \frac{wL^2}{24} - \frac{wL^2.S}{8} = \frac{wL^2}{12}$$ hence shear deflection

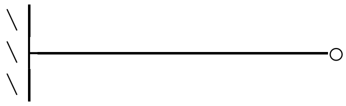
has no effect for the full fixity case.

Similarly:

$$MzB = \left[\frac{-EIz}{L} + \frac{3.EIz.S}{L}\right]\left[\frac{w.L^3}{24EIz}\right] + \left[\frac{EIz}{L} + \frac{3EIzS}{L}\right]\left[\frac{-wL^3}{24EIz}\right]$$

$$= \frac{-wL^2}{24} + \frac{wL^2.S}{8} - \frac{wL^2}{24} - \frac{wL^2.S}{8} = \frac{-wL^2}{12} \quad \text{and by inspection}$$

RyA=0 and RyB=0.

For  fixity write $S = \dfrac{1}{1 + \dfrac{3EIz}{L^2.GAy}}$

$$\begin{bmatrix} RxA \\ RyA \\ MzA \\ RxB \\ RyB \\ MzB \end{bmatrix} \begin{bmatrix} \frac{EAx}{L} & 0 & 0 & \frac{-EAx}{L} & 0 & 0 \\ & \frac{3EIz.S}{L^3} & \frac{3EIz.S}{L^2} & 0 & \frac{-3EIz.S}{L^3} & 0 \\ & & \frac{3EIz.S}{L} & 0 & \frac{-3EIz.S}{L^2} & 0 \\ & & & \frac{-EAx}{L} & 0 & 0 \\ & \text{Member stiffness} & \text{matrix is symmetric} & \text{about diagonal.} & \frac{3EIz.S}{L^3} & 0 \\ & & & & & 0 \end{bmatrix} \begin{bmatrix} 0 \\ 0 \\ \frac{wL^3}{24EIz} \\ 0 \\ 0 \\ \frac{-wL^3}{24EIz} \end{bmatrix}$$

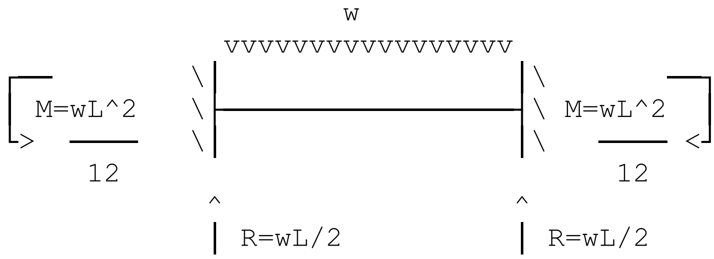
From above:

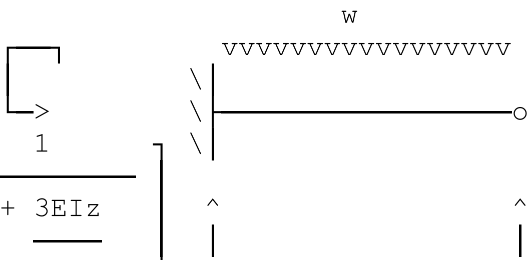
$$MzA = \frac{3EIz.S}{L} . \frac{wL^3}{24EIz} + 0 \left[\frac{-wL^3}{24EIz}\right] = \frac{wL^2.S}{8} \quad \text{and} \quad MzB=0$$

$$RyA = \frac{3EIz.S}{L^2} . \frac{wL^3}{24EIz} = \frac{wL.S}{8} \qquad RyB = \frac{-3EIz.S}{L^2} . \frac{wL^3}{24EIz} = \frac{-wL.S}{8}$$

Summing reactions $RyAtot = \dfrac{wL}{2} + \dfrac{wL.S}{8}$ and $RyBtot = \dfrac{wL}{2} - \dfrac{wL.S}{8}$

It is noted that F.E.M's allowing for effect of shear deformation:





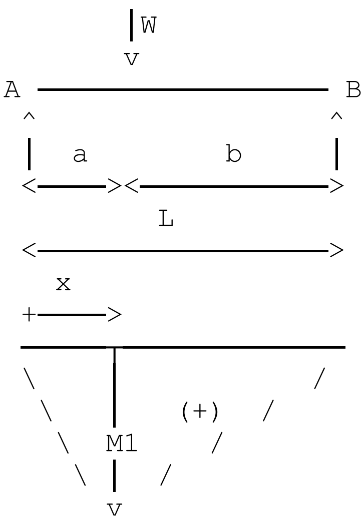
$$M = \frac{wL^2}{8}\left[\frac{1}{1 + \frac{3EIz}{L^2.GAy}}\right]$$

$$RyA = \frac{wL}{8}\left[4 + \frac{1}{1 + \frac{3EIz}{L^2.GAy}}\right] \qquad RyB = \frac{wL}{8}\left[4 - \frac{1}{1 + \frac{3EIz}{L^2.GAy}}\right]$$

It is obviously more efficient from a computational point of view to work out the rotations for the simply supported case and then obtain the force vector by multiplication with selected coefficients from the member stiffness matrix; rather than computing from the above expressions with the necessary logic to select appropriate equations.

NL-STRESS therefore works out the rotations for the simply supported case and obtains the force vector by multiplication of these rotations with the member stiffness matrix.

7.7.5.2 End rotations for a simply supported beam under point load

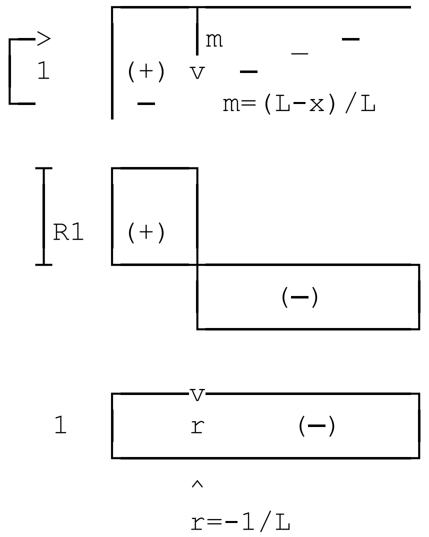


By Castigliano again:

$$Thet'A = \int_0^L \frac{M1.m.dx}{EI} + \int_0^L \frac{R1.r.dx}{GA}$$

consider LH and RH contributions and moment and shear components separately:

M1 (LHS) = W.b.x/L



M1 (RHS) = W(L-b)(L-x)/L

R1 (LHS) = W.b/L

R1 (RHS) = -W.a/L

$$\int_0^a \frac{M1.m.dx}{EIz} = \int_0^a \frac{W.b.x}{L} \cdot \frac{(L-x)}{L} \cdot \frac{dx}{EIz}$$

$$\int_0^a \left[\frac{W.B.x}{L} - \frac{W.b.x^2}{L^2} \right] \frac{dx}{EIz}$$

$$= \frac{1}{EIz} \left[\frac{W.b.x^2}{2L} - \frac{W.b.x^3}{3L^2} \right]_0^a = \frac{W}{EIz.L} \left[\frac{b.a^2}{2} - \frac{b.a^3}{3L} \right]$$

eliminating a: a=L-b a^2=L^2-2bL-b^2 a^3=L^3-3bL^2+3b^2.L-b^3

$$= \frac{W}{EIz.L} \left[\frac{L^2.b}{2} - \frac{2b^2.L}{2} + \frac{b^3}{2} - \frac{b.L^3}{3L} + \frac{b.3b.L^2}{3L} - \frac{b.3b^2.L}{3L} + \frac{b.b^3}{3L} \right]$$

$$= \frac{W}{EIz.L} \left[\frac{L^2.b}{6} - \frac{b^3}{2.L} + \frac{b^4}{3L^2} \right] = \frac{W}{6.EIz} \left[L.b - \frac{3.b^3}{L} + \frac{2.b^4}{L^2} \right]$$

$$\text{RHS:} \quad \int_a^L \frac{M1.m.dx}{EIz} = \int_a^L \frac{W}{L} \left[L^2 - b.L - x.L + b.x \right] \left[\frac{L-x}{L} \right] \frac{dx}{EIz}$$

$$= \frac{W}{EIz.L^2} \int_a^L (L^3 -b.L^2 -x.L^2 +b.x.L -L^2.x +b.x.L +x^2.L -b.x^2).dx$$

$$= \frac{W}{EIz.L^2} \int_a^L (L^3 - b.L^2 - 2.x.L^2 + 2.b.x.L + x^2.L - b.x^2).dx$$

$$= \frac{W}{EIz.L^2} \left[L^3.x - b.L^2.x - \frac{2x^2.L^2}{2} + \frac{2b.x^2.L}{2} + \frac{x^3.L}{3} - \frac{b.x^3}{3} \right]_a^L$$

Eliminating a:

$$= \frac{W}{EIz.L^2} \left[L^4 - b.L^3 - L^4 + b.L^3 + \frac{L^4}{3} - \frac{b.L^3}{3} - L^3(L-b) \right.$$

$$+ b.L^2(L-b) + L^2(L^2 - 2b.L + b^2) - b.L(L^2 - 2b.L + b^2)$$

$$\left. - \frac{L}{3}\left[L^3 - 3b.L^2 + 3b^2.L - b^3 \right] + \frac{b}{3}\left[L^3 - 3b.L^2 + 3b^2.L - b^3 \right] \right]$$

$$= \frac{W}{EIz.L^2} \left[L^4 \left[\frac{1}{3} - 1 + 1 - \frac{1}{3} \right] + L^3 \left[\frac{-b}{3} + b + b - 2b - b + b + \frac{b}{3} \right] \right.$$

$$\left. + L^2 (-b^2 + b^2 + 2b^2 - b^2 - b^2) + L \left[-b^3 + \frac{b^3}{3} + b^3 \right] - \frac{b^4}{3} \right]$$

$$= \frac{W}{EIz.L^2} \left[\frac{L.b^3}{3} - \frac{b^4}{3} \right] = \frac{W}{6EIz} \left[\frac{2b^3}{L} - \frac{2.b^4}{L^2} \right]$$

$$\text{LHS:} \int_0^a \frac{R1.r.dx}{GAy} = \int_0^a \frac{W.b}{L} \left[\frac{-1}{L} \right] \frac{.dx}{GAy} = \frac{1}{GAy} \left[\frac{-W.b.x}{L^2} \right]_0^a = \frac{-W.b.a}{L^2.GAy}$$

$$\text{RHS:} \int_a^L \frac{R1.r.dx}{GAy} = \int_a^L \frac{-W.a}{L} \left[\frac{-1}{L} \right] \frac{.dx}{GAy} = \frac{1}{GAy} \left[\frac{W.a.x}{L^2} \right]_a^L$$

$$= \frac{W}{L^2.GAy} \left[a.L - a^2 \right]$$

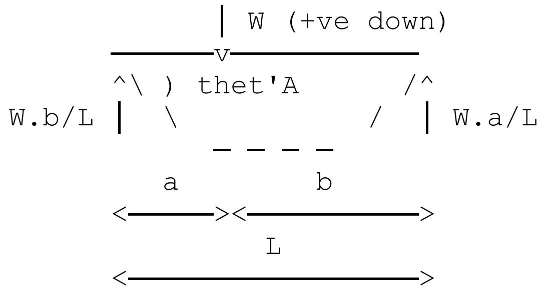
Combining all four contributions:

$$\text{Thet'A} = \frac{W}{6EIz} \left[L.b - \frac{3b^3}{L} + \frac{2b^4}{L^2} + \frac{2b^3}{L} - \frac{2b^4}{L^2} \right]$$

$$+ \frac{W}{L^2.GAy} \left[a.L - a^2 - b.a \right] = \frac{W}{6EIz} \left[L.b - \frac{b^3}{L} \right] \text{ clockwise.}$$

$$\text{Similarly:} \quad \text{Thet'B} = \frac{W}{6EIz} \left[L.a - \frac{a^3}{L} \right] \text{ anti-clockwise.}$$

Summary for point load allowing for shear deformation

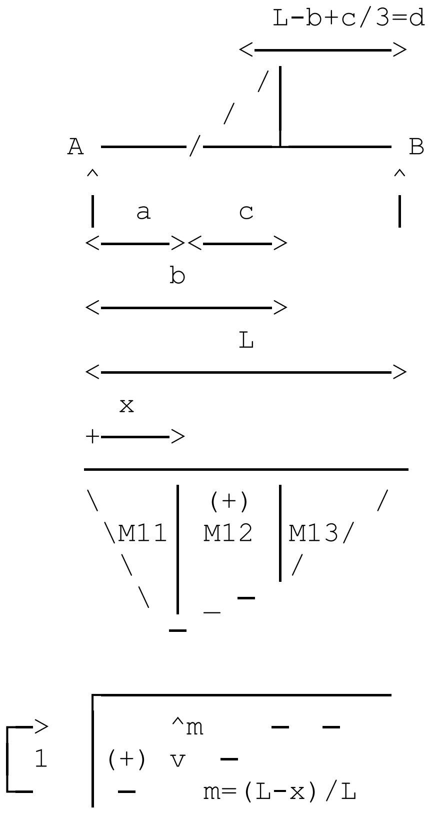


Rotation at A: $thet'A = \frac{W}{6.EIz}\left[L.b - \frac{b^3}{L} \right]$ clockwise

Rotation at B: $thet'B = \frac{W}{6.EIz}\left[L.a - \frac{a^3}{L} \right]$ anti-clockwise

For the fully fixed case we must apply an anti-clockwise rotation at A and a clockwise rotation at B of magnitudes given above.

7.7.5.3 End rotations for simply supported beam under triangular load



By Castigliano again:

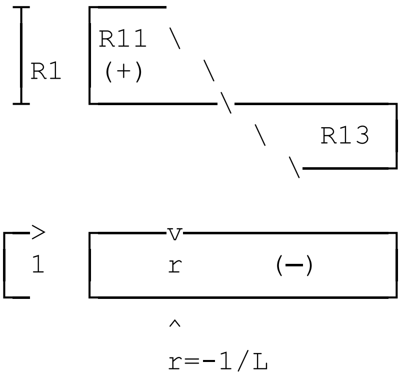
$$Thet'A = \int_0^L \frac{M1.m.dx}{EI} + \int_0^L \frac{R1.r.dx}{GA}$$

consider LH, middle, RH contributions for moment and shear components separately:

M11 is moment in LH section
M12 is moment in middle section
M13 is moment in RH section

Moments about B:

RA.L = W.d   where W = w.c/2



therefore RA = W.d/L and

RB = W*(L-d)/L

M11 = W.d.x/L

$$M12 = \frac{W.d.x}{L} - \frac{w}{c}(x-a)^3 . \frac{1}{2} . \frac{1}{3}$$

$$= \frac{W.d.x}{L} - \frac{w}{6c}(x^3 - 3a.x^2 - 3a^2.x - a^3)$$

$$R11 = \frac{W.d}{L} \qquad R12 = \frac{W.d}{L} - \frac{2W}{c^2} . \frac{(x-a)^2}{2} \qquad R13 = -\frac{W}{L}.(L-d)$$

$$M13 = \frac{W}{L}.(L-d)(L-x)$$

$$\int_0^a \frac{M11.m.dx}{EI} = \int_0^a \frac{W.d.x.(L-x).dx}{L^2.\ EI} = \frac{1}{EI}\int_0^a \left[\frac{W.d.x}{L} - \frac{W.d.x^2}{L^2} \right].dx$$

$$= \frac{1}{EI}\left[\frac{W.d.x^2}{2L} - \frac{W.d.x^3}{3L^2} \right]_0^a = \frac{1}{EI}\left[\frac{W.d.a^2}{2L} - \frac{W.d.a^3}{3L^2} \right]$$

$$\int_a^b \frac{M12.m.dx}{EI} = \int_a^b \left[\frac{W.d.x}{L} - \frac{w}{6c}(x^3 - 3a.x^2 + 3a^2.x - a^3) \right] \frac{(L-x)}{L} . \frac{dx}{EI}$$

$$= \frac{1}{EI}\int_a^b \left[\frac{W.d.x}{L} - \frac{w.x^3}{6c} + \frac{w.3a.x^2}{6c} - \frac{w.3a^2.x}{6c} + \frac{w.a^3}{6c} - \frac{W.d.x^2}{L^2} \right.$$

$$\left. - \frac{w.x^4}{6c.L} - \frac{w.a.x^3}{2c.L} + \frac{w.a^2.x^2}{2c.L} - \frac{w.a^3.x}{6c.L} \right].dx$$

$$= \frac{1}{EI}\left[\frac{W.d.x^2}{2L} - \frac{w.x^4}{24c} + \frac{w.a.x^3}{6c} - \frac{w.a^2.x^2}{4c} + \frac{w.a^3.x}{6c} - \frac{W.d.x^3}{3.L^2} \right.$$

$$+\frac{w.x^5}{30c.L}-\frac{w.a.x^4}{8c.L}+\frac{w.a^2.x^3}{6c.L}-\frac{w.a^3.x^2}{12c.L}\Bigg]_a^b$$

$$=\frac{1}{EI}\Bigg[\frac{W.d.b^2}{2L}-\frac{w.b^4}{24c}+\frac{w.a.b^3}{6c}-\frac{w.a^2.b^2}{4c}+\frac{w.a^3.b}{6c}-\frac{W.d.b^3}{3L^2}$$

$$+\frac{w.b^5}{30c.L}-\frac{w.a.b^4}{8c.L}+\frac{w.a^2.b^3}{6c.L}-\frac{w.a^3.b^2}{12c.L}-\frac{W.d.a^2}{2L}+\frac{w.a^4}{24c}$$

$$-\frac{w.a^4}{6c}+\frac{w.a^4}{4c}-\frac{w.a^4}{6c}+\frac{W.d.a^3}{3.L^2}-\frac{w.a^5}{30c.L}+\frac{w.a^5}{8c.L}-\frac{w.a^5}{6c.L}+\frac{w.a^5}{12c.L}\Bigg]$$

Collect and eliminate $w = \frac{2W}{c}$

$$=\frac{1}{EI}\Bigg[\frac{W.d.b^2}{2L}-\frac{W.b^4}{12c^2}+\frac{W.a.b^3}{3c^2}-\frac{W.a^2.b^2}{2.c^2}+\frac{W.a^3.b}{3.c^2}-\frac{W.d.b^3}{3.L^2}$$

$$+\frac{W.b^5}{15c^2.L}-\frac{W.a.b^4}{4c^2.L}+\frac{W.a^2.b^3}{3.c^2.L}-\frac{W.a^3.b^2}{6.c^2.L}-\frac{W.d.a^2}{2L}-\frac{W.a^4}{12c^2}$$

$$+\frac{W.d.a^3}{3L^2}+\frac{1}{60}.\frac{W.a^5}{c^2.L}\Bigg]$$

$$\int_b^L \frac{M13.m.dx}{EI} = \int_b^L \frac{W}{L}\left(L^2 - d.L - x.L + d.x \right)\left[1-\frac{x}{L}\right]\frac{.dx}{EI}$$

$$=\frac{1}{EI}\int_b^L\left[W.L - W.d - 2W.x + \frac{2W.d.x}{L}+\frac{W.x^2}{L}-\frac{W.d.x^2}{L^2}\right].dx$$

$$=\frac{1}{EI}\Bigg[W.L.x - W.d.x - \frac{2W.x^2}{2}+\frac{2W.d.x^2}{2.L}+\frac{W.x^3}{3.L}-\frac{W.d.x^3}{3.L^2}\Bigg]_b^L$$

$$=\frac{1}{EI}\Bigg[W.L^2 - W.d.L - W.L^2 + \frac{W.d.L^2}{L}+\frac{W.L^3}{3L}-\frac{W.d.L^3}{3.L^2} - W.L.b$$

$$+ W.d.b + W.b^2 - \frac{W.d.b^2}{L} - \frac{W.b^3}{3L} + \frac{W.d.b^3}{3L^2} \Bigg]$$

$$= \frac{1}{EI} \Bigg[\frac{W.L^2}{3} - \frac{W.d.L}{3} - W.L.b + W.d.b + W.b^2 - \frac{W.d.b^2}{L} - \frac{W.b^3}{3L}$$

$$+ \frac{W.d.b^3}{3L^2} \Bigg]$$

Combining bending contributions from the three sections:

$$\frac{theta.EI}{W} = \Bigg[\frac{d.a^2}{2L} - \frac{d.a^3}{3.L^2} + \frac{d.b^2}{2L} - \frac{b^4}{12c^2} + \frac{a.b^3}{3.c^2} - \frac{a^2.b^2}{2.c^2} + \frac{a^3.b}{3.c^2}$$

$$- \frac{d.b^3}{3.L^2} + \frac{b^5}{15.c^2.L} - \frac{a.b^4}{4.c^2.L} + \frac{a^2.b^3}{3.c^2.L} - \frac{a^3.b^2}{6.c^2.L} - \frac{d.a^2}{2.L} - \frac{a^4}{12.c^2} +$$

$$\frac{d.a\hat{}}{3.L^2} + \frac{1}{60}.\frac{a^5}{c^2.L} + \frac{L^2}{3} - \frac{d.l}{3} - Lb + d.b + b^2 - \frac{d.b^2}{L} - \frac{b^3}{3L} + \frac{d.b^3}{3.L^2} \Bigg]$$

$$= \Bigg[- \frac{d.b^2}{2L} - \frac{b^4}{12.c^2} + \frac{a.b^3}{3.c^2} - \frac{a^2.b^2}{2.c^2} + \frac{a^3.b}{3.c^2} + \frac{b^5}{15.c^2.L} - \frac{a.b^4}{4.c^2.L}$$

$$+ \frac{a^2.b^3}{3.c^2.L} - \frac{a^3.b^2}{6.c^2.L} - \frac{a^4}{12.c^2} + \frac{a^5}{60.c^2.L} + \frac{L^2}{3} - \frac{d.L}{3} - L.b + d.b$$

$$+ b^2 - \frac{b^3}{3.L} \Bigg]$$

Eliminate a by substitution:

$a = b - c$

$a^2 = b^2 - 2.b.c + c^2$

$a^3 = b^3 - 3.b^2.c + 3.b.c^2 - c^3$

$a^4 = b^4 - 4.b^3.c + 6.b^2.c^2 - 4.b.c^3 + c^4$

$a^5 = b^5 + 5.b^4.c + 10.b^3.c^2 - 10.b^2.c^3 + 5.b.c^4 - c^5$

$$\frac{theta.EI}{W} = \frac{-d.b^2}{2L} - \frac{b^4}{12c^2} + \frac{b^3.b}{3c^2} - \frac{b^3.c}{3c^2} - \frac{b^2.b^2}{2c^2} + \frac{b^2.2b.c}{2c^2}$$

$$- \frac{b^2.c^2}{2c^2} + \frac{b.b^3}{3c^2} - \frac{b.3b^2.c}{3c^2} + \frac{b.3b.c^2}{3c^2} - \frac{b.c^3}{3c^2} + \frac{b^5}{15c^2.L} - \frac{b^4.b}{4c^2.L}$$

$$+ \frac{b^4.c}{4c^2.L} + \frac{b^3.b^2}{3c^2.L} - \frac{b^3.2.b.c}{3c^2.L} + \frac{b^3.c^2}{3.c^2.L} - \frac{b^2.b^3}{6c^2.L} + \frac{b^2.3b^2.c}{6c^2.L}$$

$$- \frac{b^2.3b.c^2}{6c^2.L} + \frac{b^2.c^3}{6c^2.L} - \frac{b^4}{12c^2} + \frac{4b^3.d}{12c^2} - \frac{6b^2.c^2}{12c^2} + \frac{4b.c^3}{12c^2} - \frac{c^4}{12c^2}$$

$$- \frac{b^5}{60c^2.L} - \frac{5b^4.c}{60c^2.L} + \frac{10b^3.c^2}{60c^2.L} - \frac{10b^2.c^3}{60c^2.L} + \frac{5b.c^4}{60c^2.L} - \frac{c^5}{60c^2.L} + \frac{L^2}{3}$$

$$- \frac{d.L}{3} - L.b + d.b + b^2 - \frac{b^3}{3L}$$

$$= \frac{b^5}{c^2.L} \left[\frac{1}{15} - \frac{1}{4} + \frac{1}{3} - \frac{1}{6} + \frac{1}{60} \right] + \frac{b^4}{c^2} \left[\frac{-1}{12} + \frac{1}{3} - \frac{1}{2} + \frac{1}{3} - \frac{1}{12} \right]$$

$$+ \frac{b^4}{c.L} \left[\frac{1}{4} - \frac{2}{3} + \frac{1}{2} - \frac{1}{12} \right] + \frac{b^3}{c} \left[\frac{-1}{3} + 1 - 1 + \frac{1}{3} \right]$$

$$+ \frac{b^3}{L} \left[\frac{1}{3} - \frac{1}{2} + \frac{1}{6} - \frac{1}{3} \right] + b^2 \left[\frac{-1}{2} + 1 - \frac{1}{2} + 1 \right]$$

$$- \frac{d.b^2}{2L} - \frac{c^2}{12} + \frac{b.c^2}{12L} - \frac{c^3}{60L} - \frac{L^2}{3} - \frac{d.L}{3} - L.b + d.b$$

Eliminating d by substituting $d = l - b + c/3$

$$\frac{theta.EI}{W} = \frac{-b^3}{3L} + b^2 - \frac{b^2.L}{2L} + \frac{b^3}{2L} - \frac{b^2.c}{2L.\,3} - \frac{c^2}{12} + \frac{b.c^2}{12L}$$

$$- \frac{c^3}{60L} + \frac{L^2}{3} - \frac{L^2}{3} + \frac{L.b}{3} - \frac{L.c}{3.3} - L.b + l.b - b^2 + \frac{b.c}{3}$$

$$= \frac{b^3}{6.L} - \frac{b^2}{2} - \frac{b^2.c}{6.L} - \frac{c^2}{12} + \frac{b.c^2}{12L} - \frac{c^3}{60L} + \frac{b.L}{3} - \frac{c.L}{9} + \frac{b.c}{3}$$

$$theta = \frac{W}{6EI} \left[\frac{b^3}{L} - 3b^2 - \frac{b^2.c}{L} - \frac{c^2}{2} + \frac{b.c^2}{2L} - \frac{c^3}{10L} + 2b.L - \frac{2c.L}{3} + 2b.c \right]$$

Shear deformation components:

$$\int_0^a \frac{R11.r.dx}{GA} = \int_0^a \frac{W.d}{L}\left[\frac{-1}{2}\right]\frac{.dx}{GA} = \frac{1}{GA}\left[\frac{-W.d.x}{L^2}\right]_0^a = \frac{-W.d.a}{L^2.GA}$$

$$\int_a^b \frac{R12.r.dx}{GA} = \frac{W}{GA}\int_a^b \left[\frac{d}{L} - \frac{x^2}{c^2} + \frac{2a.x}{c^2} - \frac{a^2}{c^2}\right].\left[\frac{-1}{L}\right].dx$$

$$= \frac{-W}{L.GA}\left[\frac{d.x}{L} - \frac{x^3}{3c^2} + \frac{2a.x^2}{2c^2} - \frac{a^2.x}{c^2}\right]_a^b$$

$$= \frac{-W}{L.GA}\left[\frac{d.b}{L} - \frac{b^3}{3c^2} + \frac{a.b^2}{c^2} - \frac{a^2.b}{c^2} - \frac{d.a}{L} + \frac{a^3}{3.c^2} + \frac{a^3}{c^2} - \frac{a^3}{c^2}\right]$$

$$\int_b^L \frac{R13.r.dx}{GA} = \frac{1}{GA}\int_b^L \left[W - \frac{W.d}{L}\right].\left[\frac{-1}{L}\right].dx = \frac{W}{GA}\int_b^L \left[\frac{1}{L} - \frac{d}{L^2}\right].dx$$

$$= \frac{W}{GA}\left[\frac{x}{L} - \frac{d.x}{L^2}\right]_b^L = \frac{W}{GA}\left[1 - \frac{d}{L} - \frac{b}{L} + \frac{d.b}{L^2}\right]$$

Combining shear contributions from three sections:

$$thetaS = \frac{W}{L.GA}\left[\frac{-d.a}{L} - \frac{d.b}{L} + \frac{b^3}{3c^2} - \frac{a.b^2}{c^2} + \frac{a^2.b}{c^2} + \frac{d.a}{L} - \frac{a^3}{3c^2} + L - d - b + \frac{d.b}{L}\right]$$

Eliminate a and d by substitution:

$$thetaS = \frac{W}{L.GA}\left[\frac{b^3}{3c^2} - \frac{b^2.b}{c^2} + \frac{b^2.c}{c^2} + \frac{b.b^2}{c^2} - \frac{b.2b.c}{c^2} + \frac{b.c^2}{c^2} - \frac{b^3}{3c^2} + \frac{3b^2.c}{3.c^2} - \frac{3b.c^2}{3.c^2} + \frac{c^3}{3c^2} + L - L + b - \frac{c}{3} - b\right]$$

$$= \frac{W}{L.GA}\left[\frac{b^3}{c^2}\left[\frac{1}{3} - 1 + 1 - \frac{1}{3}\right] + \frac{b^2}{c}\left[1 - 2 + 1\right]\right] = 0$$

Rotation at B

```
 _____________________________________
   -   _      |m = x/L                |
          - v-             +          |  <-┐
        x          -   _              |    | 1
<------------->              -   _    |  --┘
```

$$\int_0^a \frac{M11.m.dx}{EI} = \int_0^a \frac{W.d.x^2.dx}{L^2.EI}$$

$$= \frac{1}{EI}\left[\frac{W.d.x^3}{3.L^2}\right]_0^a = \frac{W.d.a^3}{EI.3L^2}$$

$$\int_a^b \frac{M12.m.dx}{EI} = \int_a^b \left[\frac{W.d.x}{L} - \frac{w}{6c}\left[x^3 - 3a.x^2 + 3a^2.x - a^3\right]\right]\frac{x}{L}.\frac{dx}{EI}$$

$$= \int_a^b \left[\frac{W.d.x^2}{L^2} - \frac{w.x^4}{6c.L} + \frac{3w.a.x^3}{6c.L} - \frac{3a^2.w.x^2}{6c.L} + \frac{w.a^3.x}{6c.L}\right]\frac{dx}{EI}$$

$$= \frac{1}{EI}\left[\frac{W.d.x^3}{3.L^2} - \frac{w.x^5}{30c.L} + \frac{w.a.x^4}{8c.L} - \frac{a^2.w.x^3}{6c.L} + \frac{w.a^3.x^2}{12c.L}\right]_a^b$$

$$= \frac{1}{EI}\left[\frac{W.d.b^3}{3.L^2} - \frac{w.b^5}{30c.L} + \frac{w.a.b^4}{8c.L} - \frac{a^2.w.b^3}{6c.L} + \frac{w.a^3.b^2}{12c.L}\right.$$

$$\left. - \frac{W.d.a^3}{3.L^2} + \frac{w.a^5}{30c.L} - \frac{w.a.a^4}{8c.L} - \frac{a^2.w.a^3}{6c.L} + \frac{w.a^3.a^2}{12c.L}\right]$$

Collect like terms and substitute $w = 2W/c$:

$$= \frac{1}{EI}\left[\frac{W.d.b^3}{3.L^2} - \frac{2W.b^5}{30c^2.L} + \frac{2W.a.b^4}{8c^2.L} - \frac{2W.a^2.b^3}{6c^2.L} + \frac{2W.a^3.b^2}{12c^2.L}\right.$$

$$\left. - \frac{W.d.a^3}{3L^2} - \frac{1}{120}.2.\frac{W.a^5}{c^2.L}\right]$$

$$\int_b^L \frac{M13.m.dx}{EI} = \int_b^L \frac{W}{L} \left(L^2 - d.L - x.L + d.x \right) \frac{x}{L} \frac{.dx}{EI}$$

$$= \frac{W}{L.EI} \int_b^L \left(L.x - d.x - x^2 + \frac{d.x^2}{L} \right).dx$$

$$= \frac{W}{L.EI} \left[\frac{L.x^2}{2} - \frac{d.x^2}{2} - \frac{x^3}{3} + \frac{d.x^3}{3.L} \right]_b^L$$

$$= \frac{W}{L.EI} \left[\frac{L^3}{2} - \frac{d.L^2}{2} - \frac{L^3}{3} + \frac{d.L^3}{3L} - \frac{L.b^2}{2} + \frac{d.b^2}{2} + \frac{b^3}{3} - \frac{d.b^3}{3L} \right]$$

$$= \frac{W}{EI} \left[\frac{L^2}{6} - \frac{d.L}{6} - \frac{b^2}{2} + \frac{d.b^2}{2L} + \frac{b^3}{3L} - \frac{d.b^3}{3.L^2} \right]$$

Combining contributions from three sections:

$$\frac{theta.EI}{W} = \frac{d.a^3}{3.L^2} + \frac{d.b^3}{3.L^2} - \frac{b^5}{15.c^2.L} + \frac{a.b^4}{4.c^2.L} - \frac{a^2.b^3}{3.c^2.L} + \frac{a^3.b^2}{6.c^2.L}$$

$$- \frac{d.a^3}{3.L^2} - \frac{a^5}{60.c^2.L} + \frac{L^2}{6} - \frac{d.L}{6} - \frac{b^2}{2} + \frac{d.b^2}{2L} + \frac{b^3}{3L} - \frac{d.b^3}{3.L^2}$$

Eliminating a:

$$\frac{theta.EI}{W} = \frac{d.b^3}{3.L^2} - \frac{d.3.b^2.c}{3.L^2} + \frac{d.3.b.c^2}{3.L^2} - \frac{d.c^3}{3.L^2} + \frac{d.b^3}{3.L^2} - \frac{b^5}{15.c^2.L}$$

$$+ \frac{b^4.b}{4.c^2.L} - \frac{b^4.c}{4.c^2.L} - \frac{b^3.b^2}{3.c^2.L} + \frac{b^3.2b.c}{3.c^2.L} - \frac{b^3.c^2}{3.c^2.L} + \frac{b^2.b^3}{6.c^2.L}$$

$$- \frac{b^2.3.b^{}.c}{6.c^2.L} + \frac{b^2.3.b.c^2}{6.c^2.L} - \frac{b^2.c^3}{6.c^2.L} - \frac{d.b^3}{3.L^2} + \frac{d.3.b^2.c}{3.L^2} - \frac{d.3.b.c^2}{3.L^2}$$

$$+ \frac{d.c^3}{3.L^2} - \frac{b^5}{60.c^2.L} + \frac{5.b^4.c}{60.c^2.L} - \frac{10.b^3.c^2}{60.c^2.L} + \frac{10.b^2.c^3}{60.c^2.L} - \frac{5.b.c^4}{60.c^2.L}$$

$$+ \frac{c^5}{60c^2.L} + \frac{L^2}{6} - \frac{d.L}{6} - \frac{b^2}{2} + \frac{d.b^2}{2L} + \frac{b^3}{3L} - \frac{d.b^3}{3.L^2}$$

$$= \frac{d.b^3}{3.L^2} (1 + 1 - 1 - 1) + \frac{d.b^2.c}{L^2} (-1 + 1) + \frac{d.b.c^2}{L^2} (1 - 1)$$

$$+ \frac{d.c^3}{L^2} \left[- \frac{1}{3} + \frac{1}{3} \right] + \frac{b^5}{c^2.L} \left[- \frac{1}{15} + \frac{1}{4} - \frac{1}{3} + \frac{1}{6} - \frac{1}{60} \right]$$

$$+ \frac{b^4}{c.L} \left[- \frac{1}{4} + \frac{2}{3} - \frac{1}{2} + \frac{1}{12} \right] + \frac{b^3}{L} \left[- \frac{1}{3} + \frac{1}{2} - \frac{1}{6} + \frac{1}{3} \right]$$

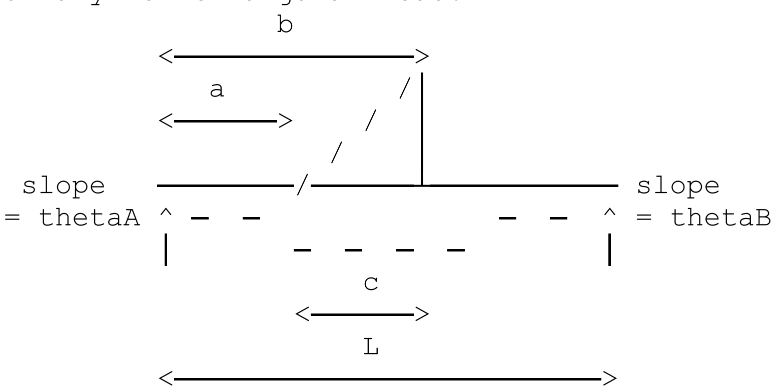
$$+ \frac{b^2.c}{L} \left[- \frac{1}{6} + \frac{1}{6} \right] - \frac{b.c^2}{12L} + \frac{c^3}{60L} + \frac{L^2}{6} - \frac{L^2}{6} + \frac{b.L}{6} - \frac{c.L}{3.6} - \frac{b^2}{2}$$

$$+ \frac{b^2.L}{2L} - \frac{b^2.b}{2L} + \frac{b^2.c}{2L.3}$$

$$= - \frac{b^3}{6L} - \frac{b.c^2}{12L} + \frac{b^2.c}{6L} + \frac{b.L}{6} - \frac{c.L}{18} + \frac{c^3}{60L}$$

$$\text{therefore} \quad \text{theta} = \frac{W}{6.EI} \left[- \frac{b^3}{L} - \frac{b.c^2}{2L} + \frac{b^2.c}{L} + b.L - \frac{c.L}{3} + \frac{c^3}{10L} \right]$$

Summary for triangular load:

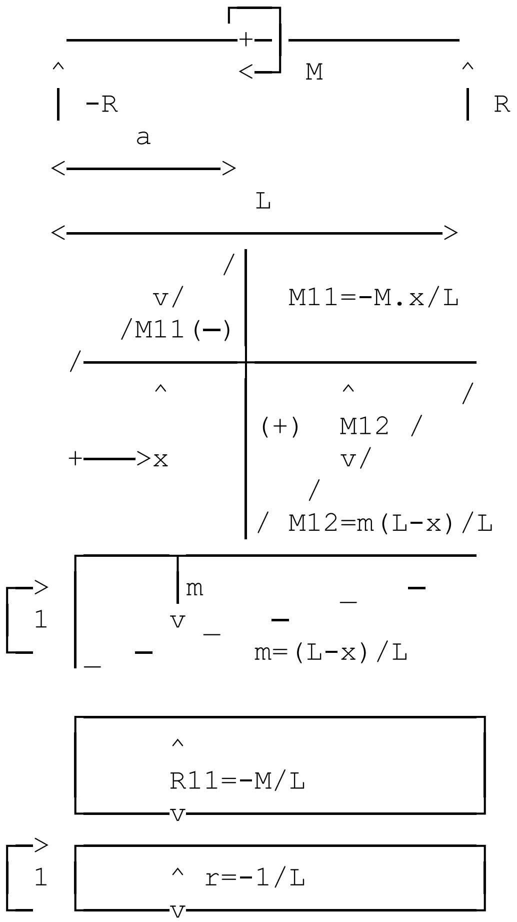


$$\text{thetaA} = \frac{W}{6EI} \left[\frac{b^3}{L} - 3b^2 - \frac{b^2.c}{L} - \frac{c^2}{2} + \frac{b.c^2}{2L} - \frac{c^3}{10L} + 2b.L - \frac{2c.L}{3} + 2b.c \right]$$

$$\text{thetaB} = \frac{W}{6.EI}\left[-\frac{b^3}{L} - \frac{b.c^2}{2L} + \frac{b^2.c}{L} + b.L - \frac{c.L}{3} + \frac{c^3}{10L} \right]$$

For reversed triangle, reverse variables.

7.7.5.4 End rotations for simply supported beam under couple



By statics M=R.L therefore

R = M/L

$$\int_0^a \frac{M11.m.dx}{EI} = \int_0^a \frac{-M.x}{L}\left[\frac{L-x}{L}\right]\frac{dx}{EI}$$

$$= \frac{1}{EI}\int_0^a \left[\frac{-M.x}{L} + \frac{M.x^2}{L^2} \right].dx$$

$$= \frac{1}{EI}\left[\frac{-M.x^2}{2L} + \frac{M.x^3}{3.L^2} \right]_0^a$$

$$= \frac{M}{EI}\left[\frac{-a^2}{2L} + \frac{a^3}{3.L^2} \right]$$

$$\int_a^L \frac{M12.m.dx}{EI} = \int_a^L \frac{M.(L-x)}{L}\left[\frac{L-x}{L}\right]\frac{.dx}{EI} = \frac{M}{L^2.EI}\left[L^2 - 2x.L + x^2 \right].dx$$

$$= \frac{M}{L^2.EI}\left[L^2.x - \frac{2.x^2.L}{2} + \frac{x^3}{3} \right]_a^L = \frac{M}{EI}\left[L - \frac{L^2}{L} + \frac{L^3}{3.L^2} - a + \frac{a^2}{L} - \frac{a^3}{3.L^2} \right]$$

$$= \frac{M}{EI}\left[\frac{L}{3} - a + \frac{a^2}{L} - \frac{a^3}{3.L^2} \right]$$

Combining two contributions from bending:

$$\text{thetaM}=\frac{M}{EI}\left[\frac{-a^2}{2L}+\frac{a^3}{3.L^2}+\frac{L}{3}-a+\frac{a^2}{L}-\frac{a^3}{3.L^2}\right]=\frac{M}{EI}\left[\frac{L}{3}-a+\frac{a^2}{2L}\right]$$

$$\int_0^L\frac{R.r.dx}{GA}=\int_0^L\frac{-M}{L}.\frac{-1}{L}\frac{.dx}{GA}=\frac{1}{GA}\left[\frac{M.x}{L^2}\right]_0^L=\frac{M}{L.GA}$$

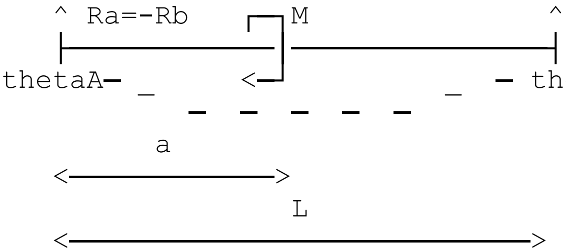
Total rotation at A: $\text{thetaA}=\frac{M}{EI}\left[\frac{L}{3}-a+\frac{a^2}{2L}\right]+\frac{M}{L.GA}$

Substitute L-a for a to get rotation at B:

$$-\text{thetaB}=\frac{M}{EI}\left[\frac{L}{3}-L+a+\frac{(L-a)^2}{2L}\right]+\frac{M}{L.GA}=\frac{M}{EI}\left[\frac{-2.L}{3}+a+\frac{L^2}{2L}\right.$$

$$\left.-\frac{2.a.L}{2L}+\frac{a^2}{2L}\right]+\frac{M}{L.GA}=\frac{M}{EI}\left[\frac{-L}{6}+\frac{a^2}{2L}\right]+\frac{M}{L.GA}$$

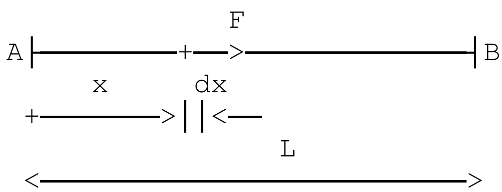
Summary with signs correct for pictorial displacement direction:



$$\text{thetaB}=\frac{-M}{EI}\left[\frac{-L}{6}+\frac{a^2}{2L}\right]-\frac{M}{L.GA}$$

$$\text{thetaA}=\frac{M}{EI}\left[\frac{L}{3}-a+\frac{a^2}{2L}\right]+\frac{M}{L.GA}$$

7.7.5.5 Concentrated load along member



Consider member AB subject to an 'in-line' force F at distance x from A applied in the direction of B.

Let the actions at A and B be Ra and Rb respectively.

Let the displacement at the point of application of x be dx.

$$Ra = \frac{dx.AE}{x} \qquad Rb = \frac{dx.AE}{(L-x)} \quad \text{eliminating } dx, \quad \frac{Ra.x}{AE} = \frac{Rb(L-x)}{AE}$$

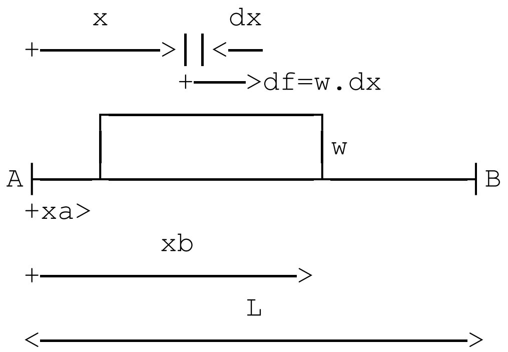
Now $F = Ra + Rb$ therefore $Ra.x = (F - Ra)(L - x)$

therefore $Ra.x = F.L - F.x - Ra.L + Ra.x$ and $Ra.L = F.L - F.x$

$$\text{and} \quad Ra = \frac{F(L-x)}{L} \quad \text{substitute back} \quad F = \frac{F(L-x)}{L} + Rb$$

$$\text{therefore} \quad Rb = F - F + \frac{F.x}{L} \qquad \text{and} \quad Rb = \frac{F.x}{L}$$

7.7.5.6 Uniform load along member



From concentrated load
From concentrated load
expression above:

$$dRa = w.dx.\frac{(L-x)}{L}$$

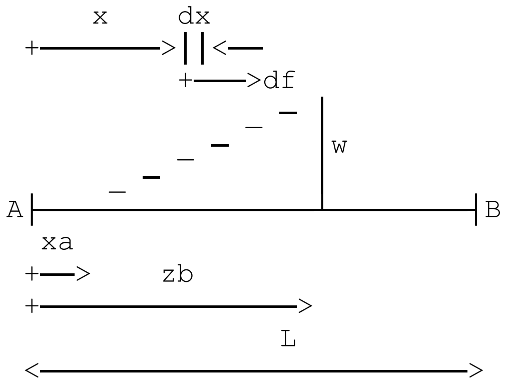
$$Ra = \int_{xa}^{xb} \frac{w.(L-x).dx}{L}$$

$$Ra = \left[w.x - \frac{w.x^2}{2L} \right]_{xa}^{xb} = w.xb - w.xa - \frac{w.xb^2}{2L} + \frac{w.xa^2}{2L}$$

$$Ra = w.(xb - xa) - \frac{w}{2L}(xb - xa)(xb + xa) = w(xb - xa)(1 - (xb + xa)/(2L))$$

write $F = w.(xb - xa)$ then $Ra = F.(1 - (xb + xa)/(2L))$ and $Rb = F - Ra$

7.7.5.7 Triangular load along member



$$df = \frac{w.(x - xa).dx}{(xb - xa)}$$

$$dRa = \frac{w.(x - xa).dx.(L-x)}{(xb-xa).L}$$

$$Ra = \int_{xa}^{xb} \frac{w.(x-xa)(L-x).dx}{L(xb-xa)} = \frac{w}{L(xb-xa)} \int_{xa}^{xb} [(x(L+xa) - xa.L - x^2].dx$$

$$\int = \left[\frac{x^2}{2}(L+xa) - xa.L.x - \frac{x^3}{3} \right]_{xa}^{xb}$$

$$= \frac{(L + xa)(xb^2 - xa^2)}{2} - xa.L.(xb - xa) - \frac{1}{3}.(xb^3 - xa^3)$$

$$= (xb - xa) \left[\frac{(L + xa)(xb + xa)}{2} - xa.L - \frac{1}{3}(xb^2 + xb.xa + xa^2) \right]$$

$$\text{Therefore } Ra = \frac{w}{2L} \left[(L + xa)(xb + xa) - 2.xa.L - \frac{2}{3}.(xb^2 + xb.xa + xa^2) \right]$$

$$= \frac{w}{2L} \left[L.xb + L.xa + xa.xb + xa^2 - 2.xa.L - \frac{2}{3}.xb^2 - \frac{2}{3}.xb.xa - \frac{2}{3}.xa^2 \right]$$

$$= \frac{w}{2L} \left[L.xb - xa.L + \frac{xa.xb}{3} + \frac{xa^2}{3} - \frac{2}{3}.xb^2 \right]$$

$$= \frac{w}{2L}(xb-xa) \left[L - \frac{(2.xb + xa)}{3} \right] \quad \text{write } F = \frac{w.(xb-xa)}{2} \text{ then:}$$

$$Ra = F.\left[1 - \frac{(2.xb + xa)}{3L} \right] \quad \text{and} \quad Rb = F - Ra.$$

7.8 Pre and post processors

NL-STRESS reads data from a text file and writes results to a text file. In carrying out the analysis of a structure, NL-STRESS produces an 'arrays' file in which are contained all the details used in the analysis. Firms who design special types of structure may write their own pre and post processors and use NL-STRESS simply as a structural workhorse. In fact the engineer need not be aware than NL-STRESS is involved in the analysis at all. A popular language for pre and post processors is BASIC which is particularly suited to the 'text processing' needed. As a simple alternative to BASIC, the engineer may use the PRAXIS notation (used in all the NL-STRESS, SCALE, LUCID and SPADE proformas) to produce both pre and post processors for NL-STRESS.

7.8.1 Pre processors

NL-STRESS works from a data file containing commands and tables of the NL-STRESS language which describe the problem to be solved. The data file may be produced by:

- editing a proforma data file written for a particular type of structure e.g. a multi-storey building of 3 bays and 5 storeys (print the NL-STRESS Proforma Data Files manual for a list of those provides)

- editing a data file of a similar frame which has been analysed already

- responding to one of the data generator proformas supplied e.g. the Question and Answer

- running a pre processor written either for a general structure or for a special type of structure e.g. a sheet piled retaining wall.

When a pre processor has been written for a special type of structure the procedure will be:

- the engineer responds to just a few questions particular to the type of structure

- the preprocessor uses the responses to produce a data file for analysis by NL-STRESS.

Proformas 560-600 are preprocessors written using the PRAXIS notation, proforma 570 for example prompts for dimensions & material properties of a portal frame and from the engineer's responses build a text file of NL-STRESS data. Commonly used procedures are included in proformas 565 and 566. Using say proforma 570 as a model, the engineer will be able to develop their own pre-processor.

7.8.2 Post processors

Results from an NL-STRESS analysis are stored in a file of the same name as the data file but ending in the extension .res. The results file is a text file which may be postprocessed by a word processor to rearrange the results, or by a specially written postprocessor to extract certain critical values which are particular to the type of structure analysed. As an alternative to a specifically written postprocessor, NL-STRESS option 1 may be run in batch mode to produce additional tables whose data may be picked up by a proforma written using the PRAXIS notation.

When option 1 is run in batch mode, additional output (over and above the bending moment and shear diagrammatic summaries) is produced thus:

- summary1.res containing Table 9999, listing in order:

NJ	Number of joints
NM	members
NSUP	supports
NLS	loadcases = NLSORG * number of loading increments
NPD	number of possible displacements NLS*NDJ
NLSB	number of basic loading cases
NLSC	combined loading cases
MML	maximum number of member loads on any member
ISTYP	structure type
NDJ	number of displacements per joint
NDM	member
NCOOR	number of coordinates
LCSTA	start pointer in NLS for current basic loading case
LCEND	end
NLSORG	original number of loading cases; same as NLS for elastic analysis.

- summary2.res containing Table 10001, listing in order: joint displacements for basic loading case 1, followed by Table 10002 for loading case 2 and so on containing:

DX	Displacement in X direction	Plane frames
DY	Y	
RZ	Rotation about Z axis	
RX	X	Grids
RY	Y	
DZ	Displacement in Z direction	
DX	Displacement in X direction	Space frames
DY	Y	
DZ	Z	
RX	Rotation about X axis	
RY	Y	
RZ	Z	

- summary3.res containing Table 11001 listing in order member end forces for basic loading case 1, followed by Table 11002 for loading case 2 and so on containing:

```
FX  Force in X direction start end ┐
FY           Y                     │
MZ  Moment about Z axis            ├ Plane frames
FX  Force in X direction end end   │
FY           Y                     │
MZ  Moment about Z axis            ┘
MX  Moment about X axis start end  ┐
MY                   Y             │
FZ  Force in Z direction           ├ Grids
MX  Moment about X axis end end    │
MY                   Y             │
FZ  Force in Z direction           ┘
FX  Force in X direction start end ┐
FY           Y                     │
FZ           Z                     │
MX  Moment about X axis            │
MY                   Y             │
MZ                   Z             ├ Space frames
FX  Force in X direction end end   │
FY           Y                     │
FZ           Z                     │
MX  Moment about X axis            │
MY                   Y             │
MZ                   Z             ┘
```

- summary4.res containing Table 12001 listing summary of:

```
21 rows giving min & max BM values at 20'th points ┐
21 rows giving min & max SF values at 20'th points │
Max & min displacements at member start            │
                                  span             │
                                  end              ├ Plane frames
AX & AY                                            │
IZ & member length                                 │
Max & min axial force at member start              │
                                  end.             ┘
```

Grids & space frames similarly follow the values given in the option 1 diagrammatic summaries.

- summary5.res containing Table 15000 listing member properties for all members.

All the above tables may be accessed by including their file name at the start of any subsequent proforma in the chain e.g. @summary2.res will include the joint displacement tables at the start of a proforma.

Any information additional to that contained in summary1.res to summary4.res required by a post-processor may be passed using the FILE command; e.g. include in the proforma the command FILE link.cal which writes any lines which start with a '%' which are subsequent to the invocation to be written to the file named 'LINK'. Again, the tables in 'link.cal' may be made available to a subsequent proforma by including @LINK at the start of the proforma.

8. TAPE User's Manual

8.1 Introduction to TAPE

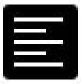
TAPE, Translator And Plot Editor, is a drawing editor that allows you to:

- edit existing drawings
- edit and annotate detail files created by LUCID or SPADE
- edit and annotate NL-STRESS plot files
- annotate NL-VIEW screenshots
- export drawings as 2D DXF files (via the DXF button on the calcs display screen)

TAPE uses the HPGL file format. HPGL is used by LUCID and SPADE for their detail drawings, by NL-STRESS for graphical representations of results, and by some SCALE proformas, e.g. for cross section calculations.

8.2 Getting started

When viewing the final calcs for SCALE, LUCID, SPADE and NL-STRESS runs, or viewing an existing calcs file with SCALE option 15, if the currently viewed page contains a diagram, then tapping the "Edit calcs" button on the right of the toolbar will automatically launch TAPE to edit the diagram/plot as shown in Figure 8.1.

The "Edit calcs" button: 

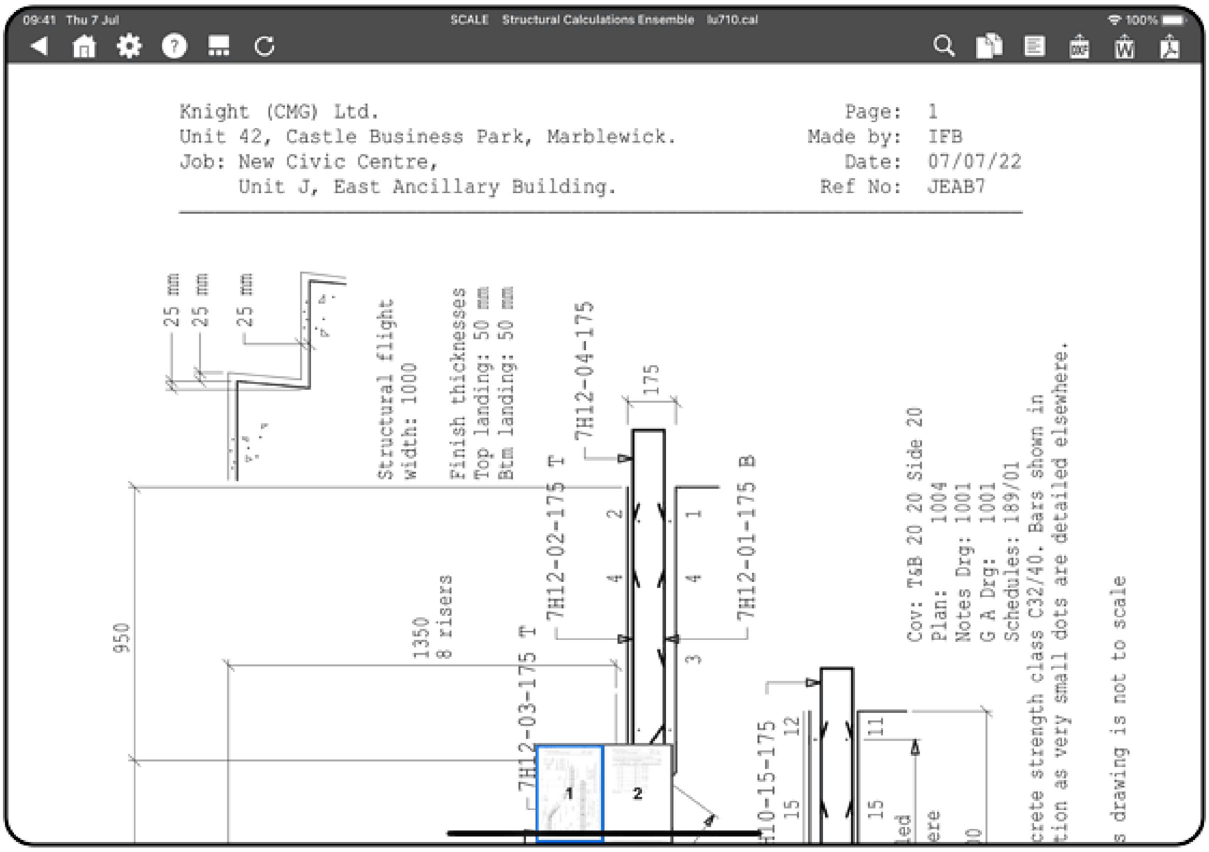


Figure 8.1: Tap the edit calcs button to launch TAPE.

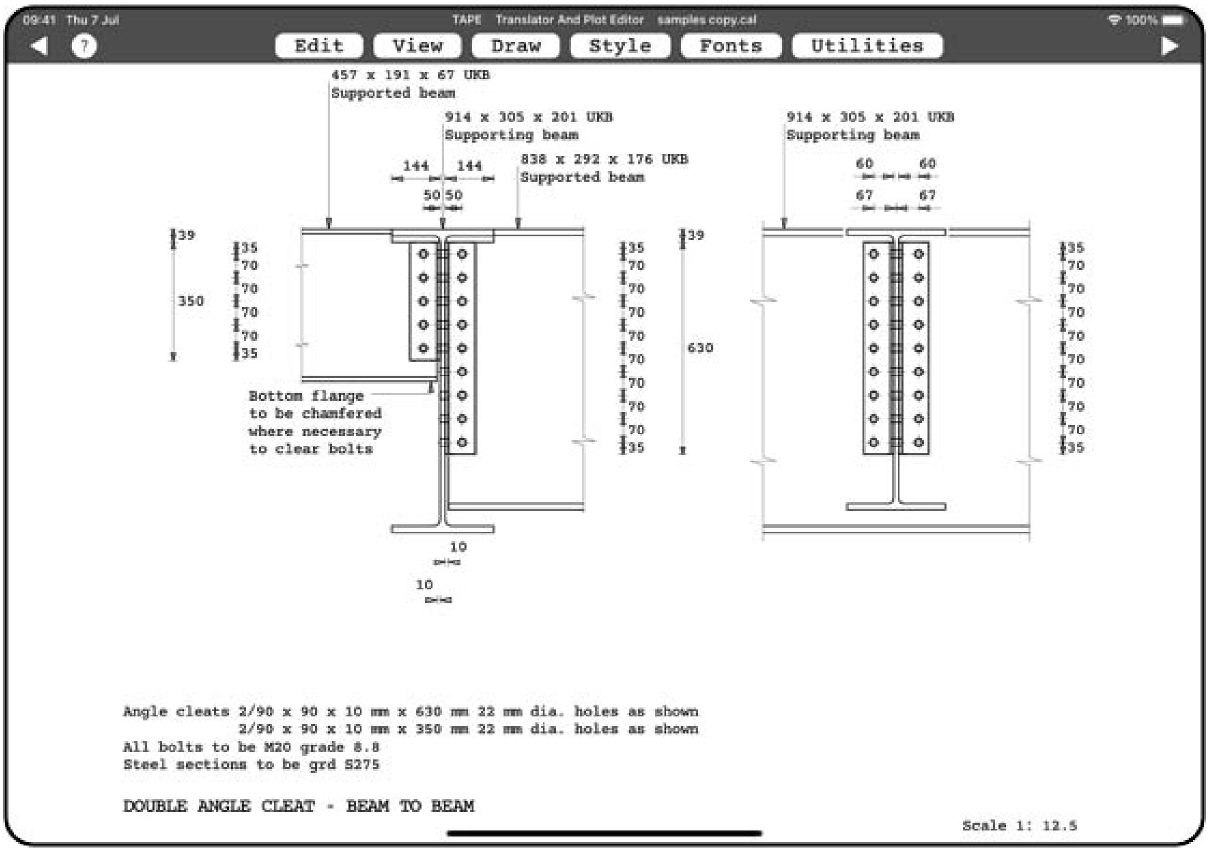


Figure 8.2: Main TAPE screen.

On switching to TAPE you are presented with the diagram or plot zoomed to fit the window, as shown in Figure 8.2.

You can zoom and pan the drawing using the View menu menu options to allow greater precision when adding entities.

On an iPad you can zoom and drag using the usual pinch and drag gestures.

To start drawing you must first select the type of graphical object you wish to add from the Draw menu.

The graphical entities available are:

- Circle - defined by its centre and a point on its circumference.
- Line - defined by its start and end points.
- Dimension Line - as Line but with end arrows.
- Box - defined by two diagonally opposite corners.
- Polyline - defined by a succession of points.
- Text - defined by a start point (bottom-left) and the entered text.

You can change the pen or line style using the Style menu or the text font by using the Fonts menu.

Windows:
TAPE rubber-bands objects as they are being added. This requires you click the mouse to start adding the object. The object is rubber-banded until you click again and the object is added to the drawing. To abandon the current object simply select Stop on the Draw menu. Polylines differ slightly in that each click ends one side and starts rubber-banding the next line. The polyline is finished by selecting Polyline again from the Draw menu. The last side being rubber-banded is not added.

iPad:
Tap to select the starting point of an operation. TAPE will temporarily identify this starting point with a target symbol. You can pinch and drag as usual to best locate the end point, then tap again to select the end point of the operation, TAPE will then add the object, and remove the temporary target symbol. To abandon the current object simply tap on "Done" on the toolbar.

The remaining drawing-related activities are:

- Edit item - click on the item you wish to modify.
- Erase - click on any items you wish to erase permanently from the drawing. Use a zoomed view to facilitate selection of the correct object.
- Pick - click on an item (or items) to mark them as selected. A second click on a selected item will de-select it. Selected items may be manipulated using the Edit menu options.

8.3 File menu (Windows only)

The file options are described below.

NEW
Start a new drawing. The model size will be the default of A4 with a scale of 1:1. All menu items will be unchecked and the initial zoom will be to show the whole page in the window. The current drawing will be closed, with a prompt to save changes if any were made. Available on toolbar.

OPEN
Use an existing drawing with an extension of ".cal" or ".plt" You can use the "List files of type" listbox on the open files dialog to show files of a particular type. The open dialog is the standard Windows dialog and supports all the usual features of double-clicking on a file to select it. The initial view is zoomed to show the extent of the existing items. The last 100 items are available in the Edit menu Undo buffer. The last ten files you worked on are available in the recent file list at the bottom of the menu. The current drawing will be closed, with a prompt to save changes if any were made. Available on toolbar.

SAVE
Save the drawing. If the drawing is being saved for the first time you are asked to supply a file name. Available on toolbar.

SAVE AS
Save the drawing in a named file. The "List files of type" listbox can be used to change the file extension. If the file name you choose is already in use you will be asked to confirm that you wish to overwrite the existing file.

EXPORT DXF
Create a DXF file of the active drawing in the same directory as the active drawing file. If the drawing has never been saved you will be asked to supply a file name. For more information on DXF files see the section: DXF files.

PRINT
Print the current drawing. This option is intended for A4 details. Available on toolbar.

RECENT FILES
The last ten files you worked on are shown here for easy access. Simply click on a file to open it.

EXIT
Finish using TAPE. You will be given the option to save any changed files.

8.4 Edit menu

Options available on the edit menu, as shown in Figure 8.3, are:

- Edit items
- Edit headings

- Cut
- Copy
- Paste
- Undo

- Select items
- Select region
- Select all
- Deselect all

- Move selected items
- Size selected items
- Rotate selected items

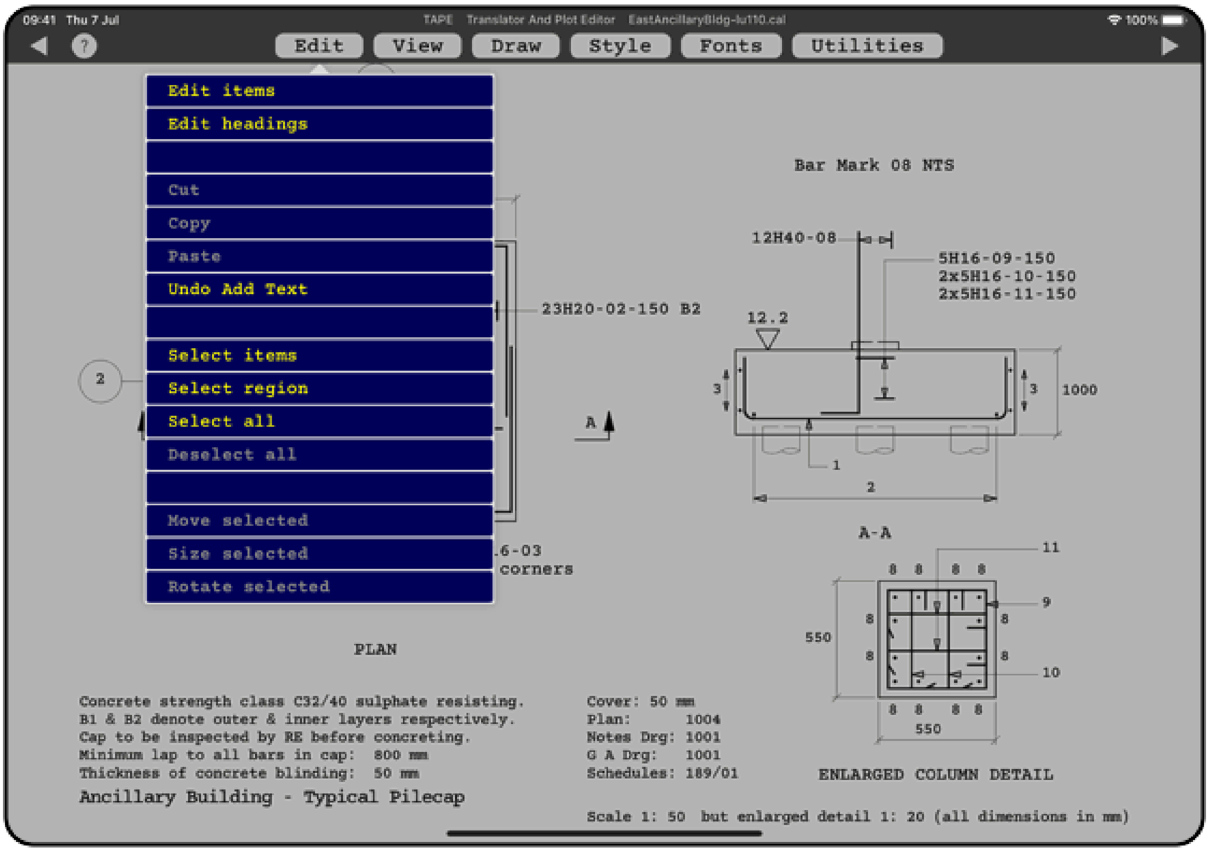


Figure 8.3: Edit menu options.

A greyed out menu item indicates that this option is not currently available, e.g. if nothing has been selected yet then the "Copy" option is not available as there is nothing to copy.

8.4.1 Edit menu → Edit items

This option can be used to change the dimensions and properties of an object. First select the object you wish to change, see Figure 8.4.

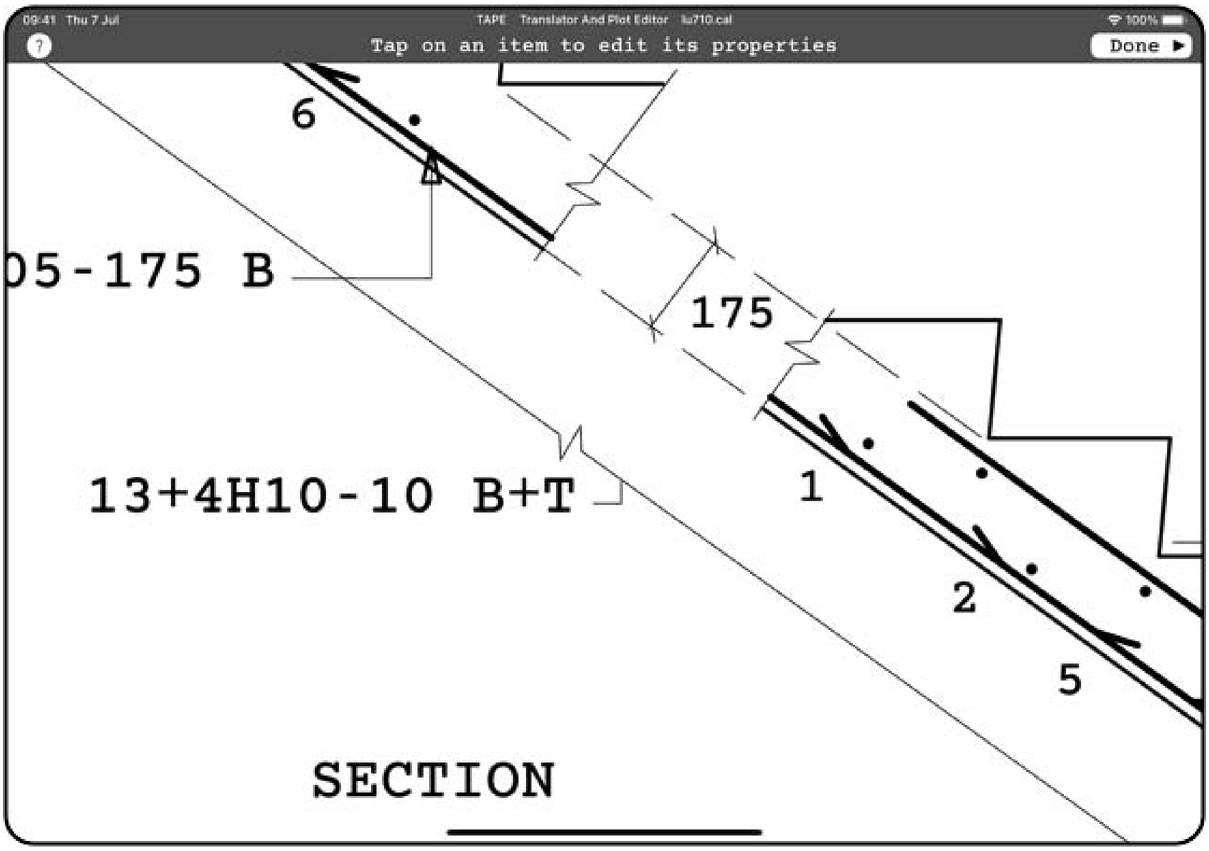


Figure 8.4: Edit items screen.

Selecting an object will launch the edit dialog, e.g. Figure 8.5. The edit dialog will show:

- Line - X and Y coordinates of the start and end points
- Dimension line - as line
- Circle - X and Y coordinates of the centre and the radius
- Box - X and Y coordinates of the diagonally opposite corners
- Text - X and Y coordinates of the start of the text along with the character width and height
- Polyline - X and Y coordinate of the selected vertex

The coordinates shown are in the current units (mm, in or ft). The values may be edited and will be applied if the "OK ▶" button is selected. Select the cancel button "◀" to leave the dialog without making changes.

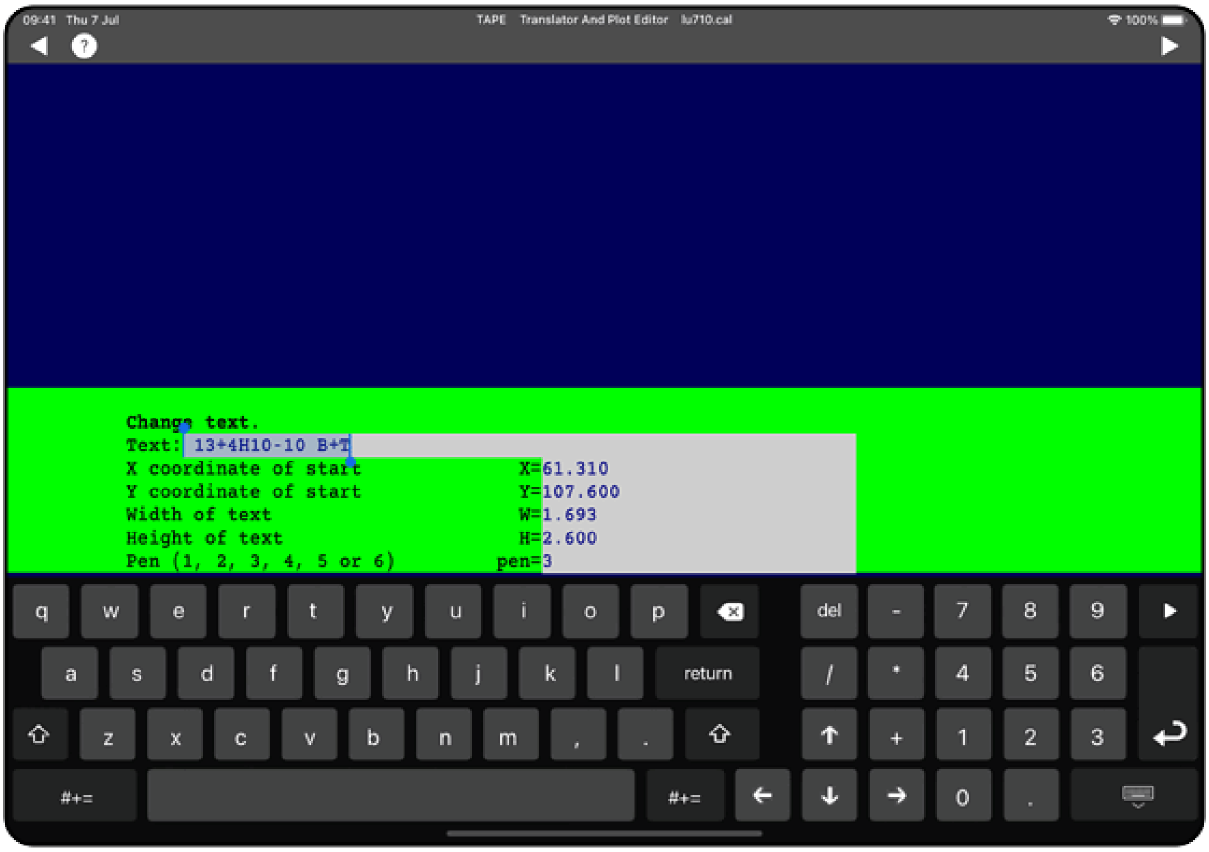


Figure 8.5: Edit items - change text screen.

8.4.2 Edit menu → Edit headings

Allows you to edit the standard page headings that appear when you print this drawing, as shown in Figure 8.6.

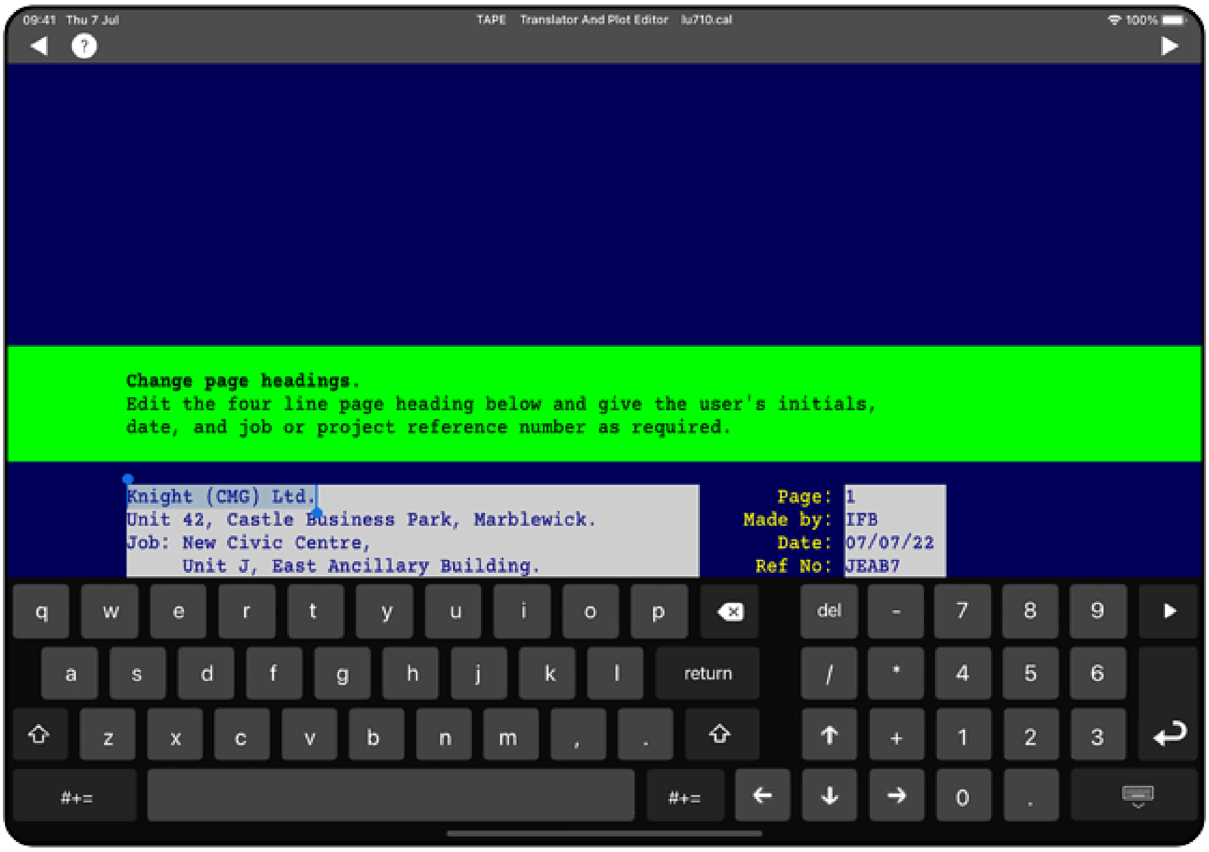


Figure 8.6: Editing the page headings.

8.4.3 Edit menu → Cut

Removes all selected objects from the drawing and copies them to TAPE's internal clipboard. Objects on the clipboard are available for pasting in this or another drawing until they are overwritten.
Available on toolbar.

8.4.4 Edit menu → Copy

Copies all selected objects from the drawing to TAPE's internal clipboard. Objects on the clipboard are available for pasting in this or another drawing until they are overwritten. Available on toolbar.

8.4.5 Edit menu → Paste

Allows pasting of objects held in TAPE's internal clipboard into the active drawing. Any currently selected objects will be deselected, and the objects in TAPE's internal clipboard will be pasted in the same location from where they were cut or copied. The pasted objects will be automatically set as selected, such that they are available for moving, sizing or rotating as required. The contents of the clipboard remain unchanged and may be pasted again. Available on toolbar.

8.4.6 Edit menu → Undo

The Undo menu option will undo the last operation used in generating the current view of the drawing. This option stores that last 100 operations. For a drawing that has just been loaded this buffer will initially contain the last 100 graphic objects added. The Undo buffer operates on a last in first out principle.

8.4.7 Edit menu → Select items

This allows items to be grouped for subsequent Edit menu options. Tapping/clicking on an item will highlight it. Tapping/clicking on the same item again will remove it from the selected group. You can continue to select items to add to the group. Any items in the group will remain selected until they are deselected. Available on toolbar.

8.4.8 Edit menu → Select region

As an extension to selecting individual items you can also select a group of items. Select the menu option Edit->Select region, this will automatically add a rectangular selection in middle of the screen. Drag the corners of the selection to the required locations then select "Done" to toggle the selection status of any items which intersect the rectangular region. All items that are partially within the box are treated as if they were picked individually. Any unselected items will be selected, and any selected items will be deselected.

By applying "Select region" multiple times you can for example select all the items around the outside of a cross section, by fist selecting the whole cross section then applying "Select region" again to deselect objects in the middle.

If you don't wish to add the region to the current selected items, select the "Cancel" menu button to return to the previous selection, if any.

Selecting a region is illustrated in Figure 8.7.

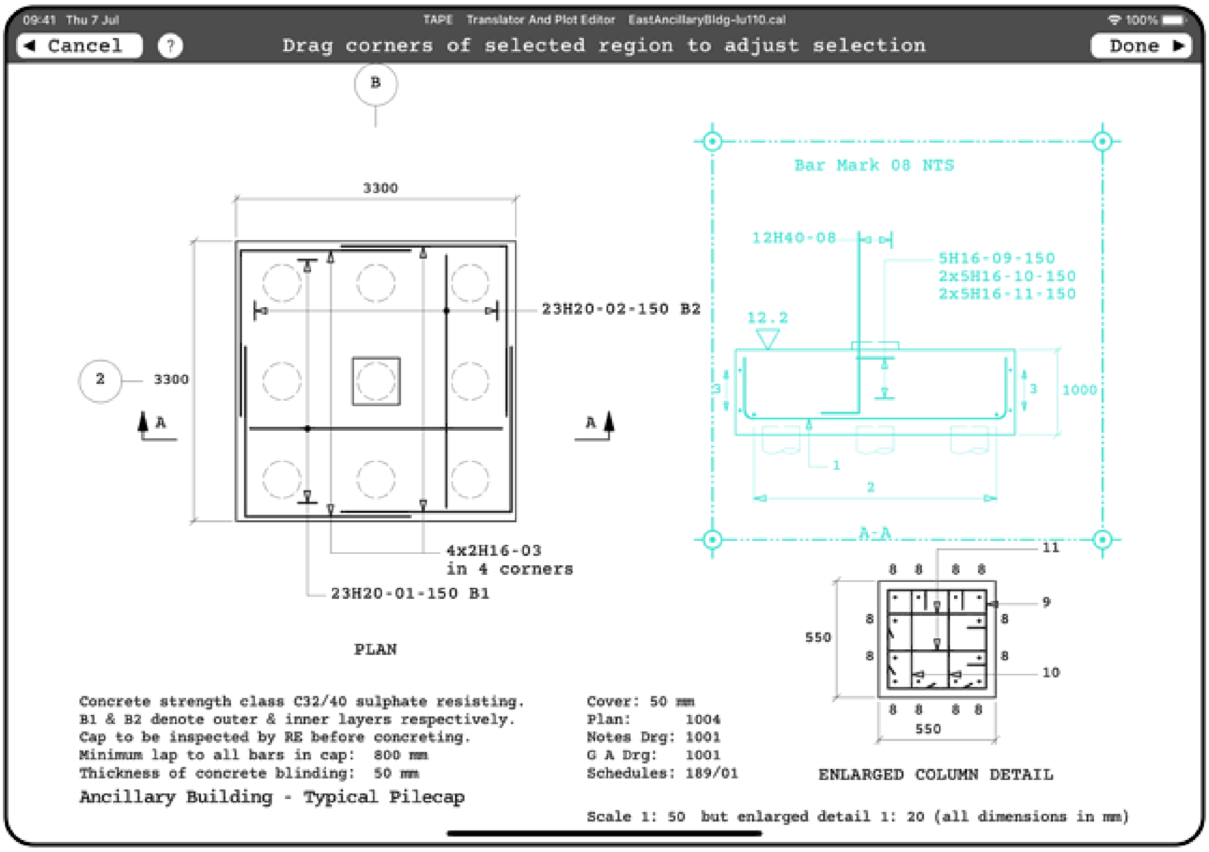


Figure 8.7: Selecting a region.

8.4.9 Edit menu → Select all

Select all objects in a drawing.

8.4.10 Edit menu → Deselect all

Cancels the selection of all selected objects in a drawing.

8.4.11 Edit menu → Move selected items

Select "Move selected items" mode to reposition the current selected objects, as shown in Figure 8.8. Touch a selected object and drag to move all the selected objects together to the required position. If your touch and drag position is not on a selected object, then you will just pan the screen as for a normal pan. You may also pinch to zoom in or out as required. These actions allow you to position the screen to the required position. Select "Done" when finished, or select "Cancel" to abandon the move and return the selected items to their initial positions.

The items remain selected after moving for subsequent operations if required, e.g. resizing, rotating, cutting and copying.

You can also set "Snap to grid" and/or "Orthogonal" modes to help in the movement of the selection if required.

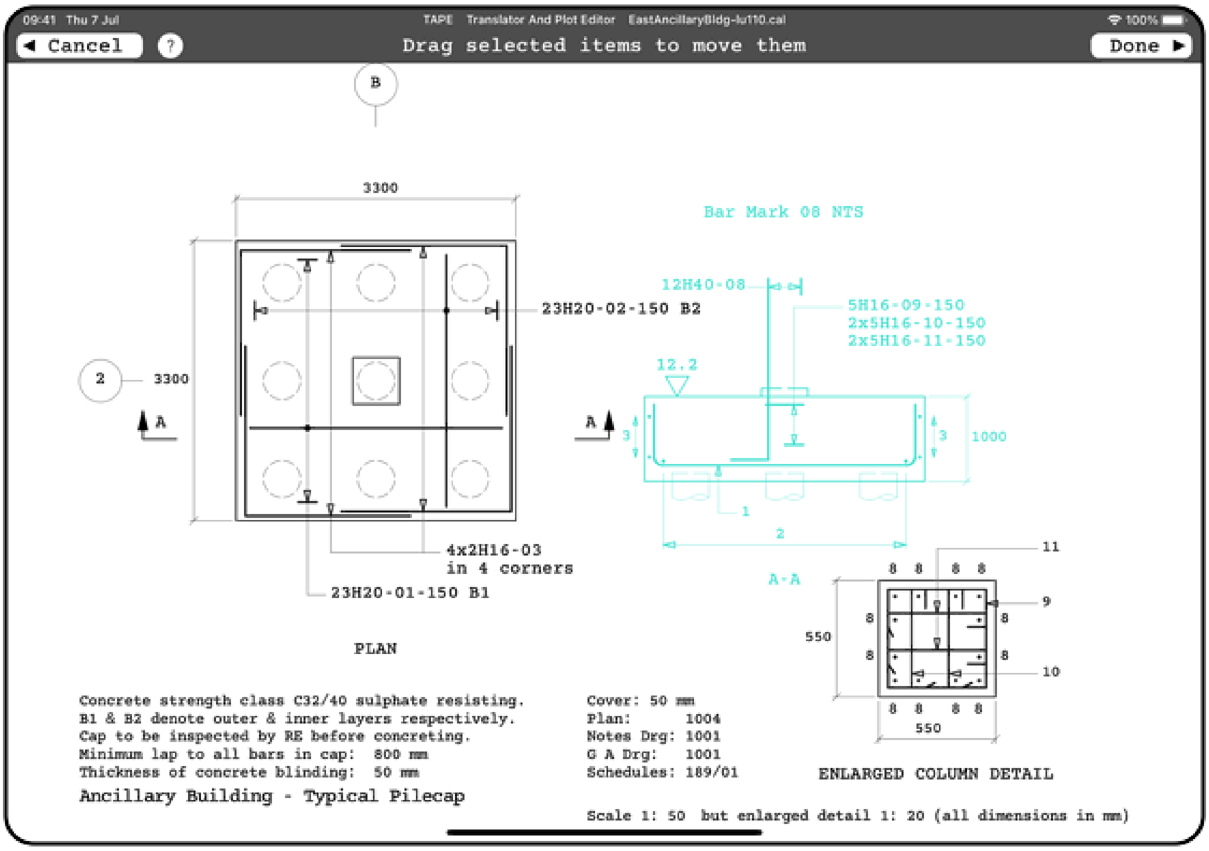


Figure 8.8: Moving a selection.

8.4.12 Edit menu → Size selected items

Allows local scaling of the currently selected objects. Enter the scale factor into the dialog as shown in Figure 8.9. The anchor point is defined by the bottom left corner of the bounding box of the selected items. The items remain selected after scaling for subsequent operations if required, e.g. moving, rotating, cutting and copying.

Figure 8.9: Sizing a selection.

8.4.13 Edit menu → Rotate selected items

Selected items may be rotated about an anchor point to a set rotation, enter the rotation required into the dialog as shown in Figure 8.10. The anchor point is defined by the bottom left corner of the bounding box of the selected items. Rotation is in degrees and is measured anti-clockwise from the horizontal X axis. The items remain selected after rotation for subsequent operations if required, e.g. moving, resizing, cutting and copying.

Rotation is applied to the opposite corners of the box. The resulting object is still a box of different size. Text remains horizontal but the anchor point is rotated. Circles remain circular as only the centre point is rotated. Figure 8.11 demonstrates the result of rotating the selection shown in Figure 8.7 through 45°.

Figure 8.10: Setting the rotation for a selection.

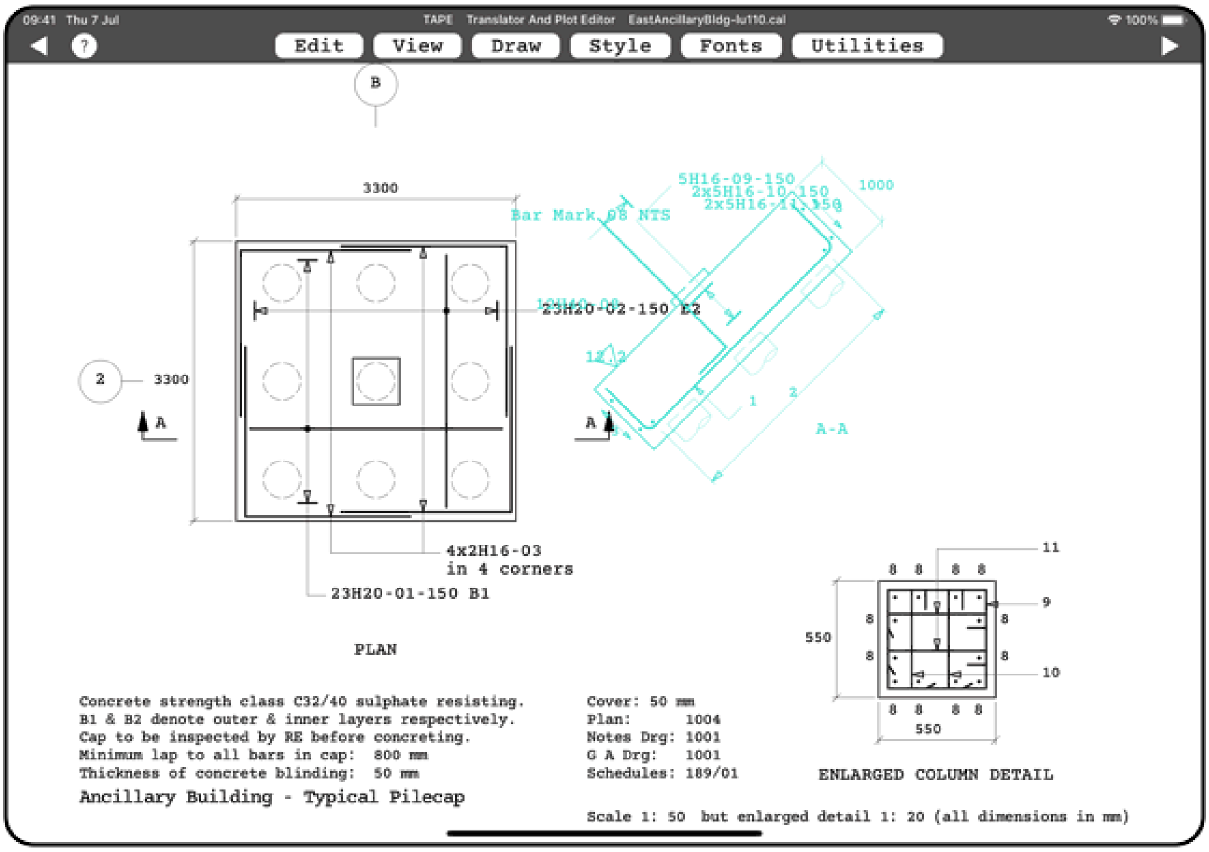


Figure 8.11: Selection after rotating 45°.

8.5 View menu

Options available on the view menu are:

- Zoom extents
- Zoom in
- Zoom out
- Pan up, down, left, right

8.5.1 View menu → Zoom extents

The view is zoomed to fit the entire drawing in the current window.

8.5.2 View menu → Zoom in

Windows:
Zoom in about the point at which the mouse is next clicked. You can keep clicking to zoom further. Available on toolbar.

iPad:
Tap this menu item to zoom in. Repeat as necessary.

You can also tap the '+' key on an external keyboard to zoom in.

Pinching apart with two fingers also zooms in, in the usual iPad manner.

8.5.3 View menu → Zoom out

Windows:
Zoom out about the point at which the mouse is next clicked. You can keep clicking to zoom further. Available on toolbar.

iPad:
Tap this menu item to zoom out. Repeat as necessary.

You can also tap the '-' key on an external keyboard to zoom out.

Pinching together with two fingers also zooms out in the usual iPad manner.

8.5.4 View menu → Pan up, down, left, right

The model may be panned up, down, left and right.

iPad:
Tap the relevant menu item to pan in that direction. Repeat as necessary.

You can also tap the arrow keys on an external keyboard to pan the view.

Dragging with one finger on the screen pans the view in the usual iPad manner.

8.6 Draw menu

Options available on the Draw menu, as shown in Figure 8.12, are:

- Add line
- Add dimension line
- Add polyline
- Add box
- Add circle
- Add text
- Delete items

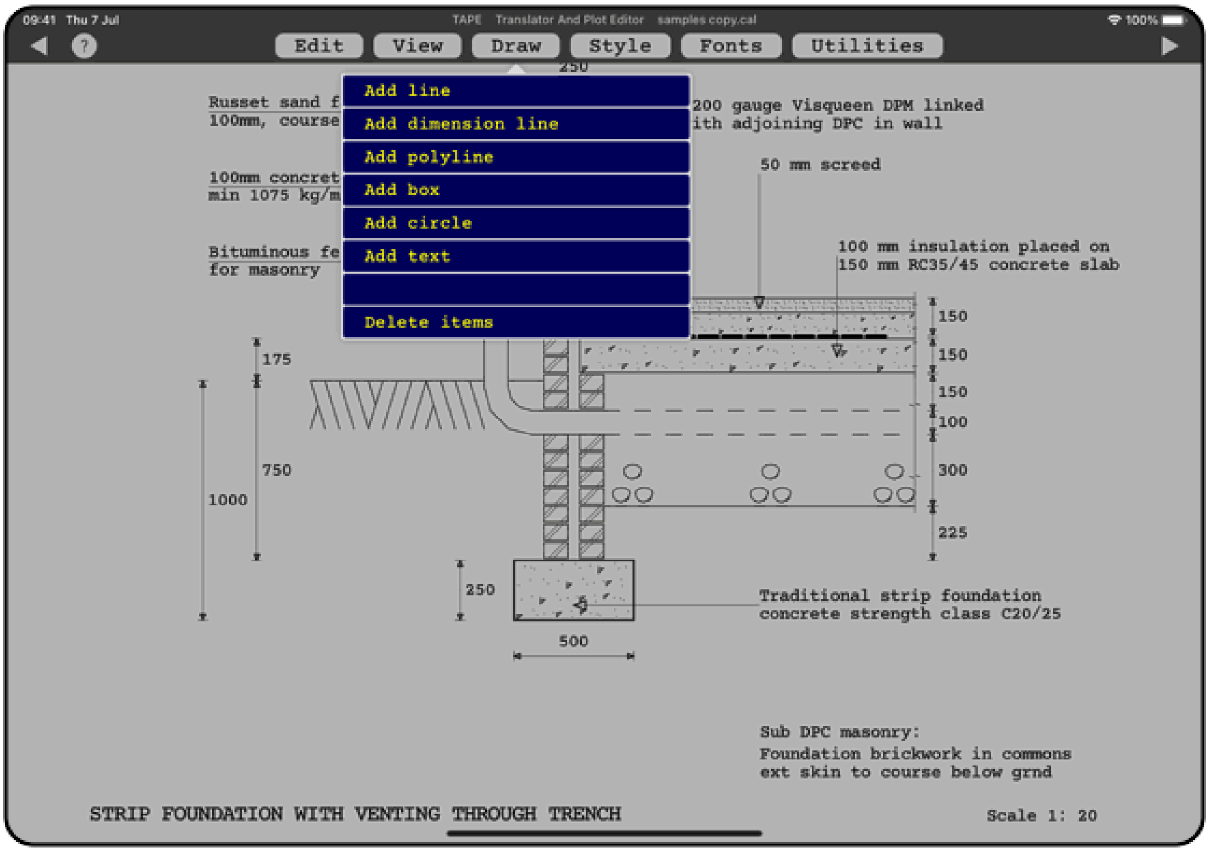


Figure 8.12: Draw menu screen.

8.6.1 Draw menu → Add line

A line is defined by a start and an end point.

iPad:
A line is added by first tapping at its start point. This point is displayed with a temporary target icon, as a check that it is in the desired position, similar to Add circle's Figure 8.13. Tap again to define the position of the end point. The defined line in the current pen and line style is added to the drawing. To cancel a line simply select the "Done" button.

You may pan and zoom using the iPad's drag and pinch gestures as required.

Obeys "Orthogonal" and "Snap to grid".

Windows:
It is added by clicking at its start point. As the cursor is moved the line is rubber-banded from its start point to the cursor. The end point

is defined by the position at which the mouse is clicked again. The defined line in the current pen and line style is added to the drawing. To cancel a line simply select Stop. The pen and/or line style can be changed during between adding lines. Obeys "Orthogonal" and "Snap to grid". Available on toolbar.

8.6.2 Draw menu → Add dimension line

A dimension line is defined by a start and an end point. A dimension line differs from a standard line in that start and end point of the line are drawn as standard sized arrows.

iPad:
A dimension line is added by first tapping at its start point. This point is displayed with a temporary target icon, as a check that it is in the desired position, similar to Add circle's Figure 8.13. Tap again to define the position of the end point. The defined dimension line in the current pen and line style is added to the drawing. To cancel a dimension line simply select the "Done" button.

You may pan and zoom using the iPad's drag and pinch gestures as required.

Obeys "Orthogonal" and "Snap to grid".

Windows:
A dimension line is added by clicking at its start point. As the cursor is moved the line is rubber-banded from its start point to the cursor. The end point is defined by the position at which the mouse is clicked again. The defined line in the current pen and line style is added to the drawing. To cancel a dimension line simply select Stop. The pen and/or line style can be changed during between adding lines. Obeys "Orthogonal" and "Snap to grid". Available on toolbar.

8.6.3 Draw menu → Add polyline

iPad:
A polyline is added by tapping at its start and subsequent vertex points. Once Polyline has been selected all lines are added to the current polyline until the "Done" button is selected. The previous vertex is displayed with a temporary target icon, as a check that it is in the desired position, similar to Add circle's Figure 8.13.

You may pan and zoom using the iPad's drag and pinch gestures as required.

Obeys "Orthogonal" and "Snap to grid" for all vertices of the polyline.

Windows:
A polyline is added by clicking at its start and subsequent vertex points. Rubber-banding is used as described in Lines above. Once Polyline has been selected all lines are added to the current polyline until the polyline option is selected again or the right mouse button clicked. Obeys "Orthogonal" and "Snap to grid" for all vertices of the polyline. Available on toolbar.

8.6.4 Draw menu → Add box

A box is defined by two diagonally opposite corners.

iPad:
A box is added by first tapping at one corner. This corner is displayed with a temporary target icon, as a check that it is in the desired position, similar to Add circle's Figure 8.13.

Tap again to define the opposite corner.

You may keep adding boxes in this way, to cancel adding boxes select the "Done" button.

You may pan and zoom using the iPad's drag and pinch gestures as required.

Obeys "Snap to grid" for the corners of the box.

Windows:
It is added by clicking at one corner. As the cursor is moved the box is rubber- banded from its start point to the cursor. The second corner is defined by the position at which the mouse is clicked again. The defined box in the current pen and line style is added to the drawing. To cancel a box select Stop. The pen and/or line style can be changed during between adding boxes. Obeys "Snap to grid" for the corners of the box. Available on toolbar.

8.6.5 Draw menu → Add circle

A circle is defined by its centre and a point on the circumference.

iPad:
A circle is added by first tapping at its centre. This will lead to Figure 8.13. The centre point is displayed with a temporary target icon, as a check that it is in the desired position.

Tap again to define the final radius, as show in Figure 8.14.

You may keep adding circles in this way, to cancel adding circles select the "Done" button.

You may pan and zoom using the iPad's drag and pinch gestures as required.

Obeys "Snap to grid" for the centre point are circumference point of the circle.

Figure 8.15 demonstrates the final calculations with the circle added.

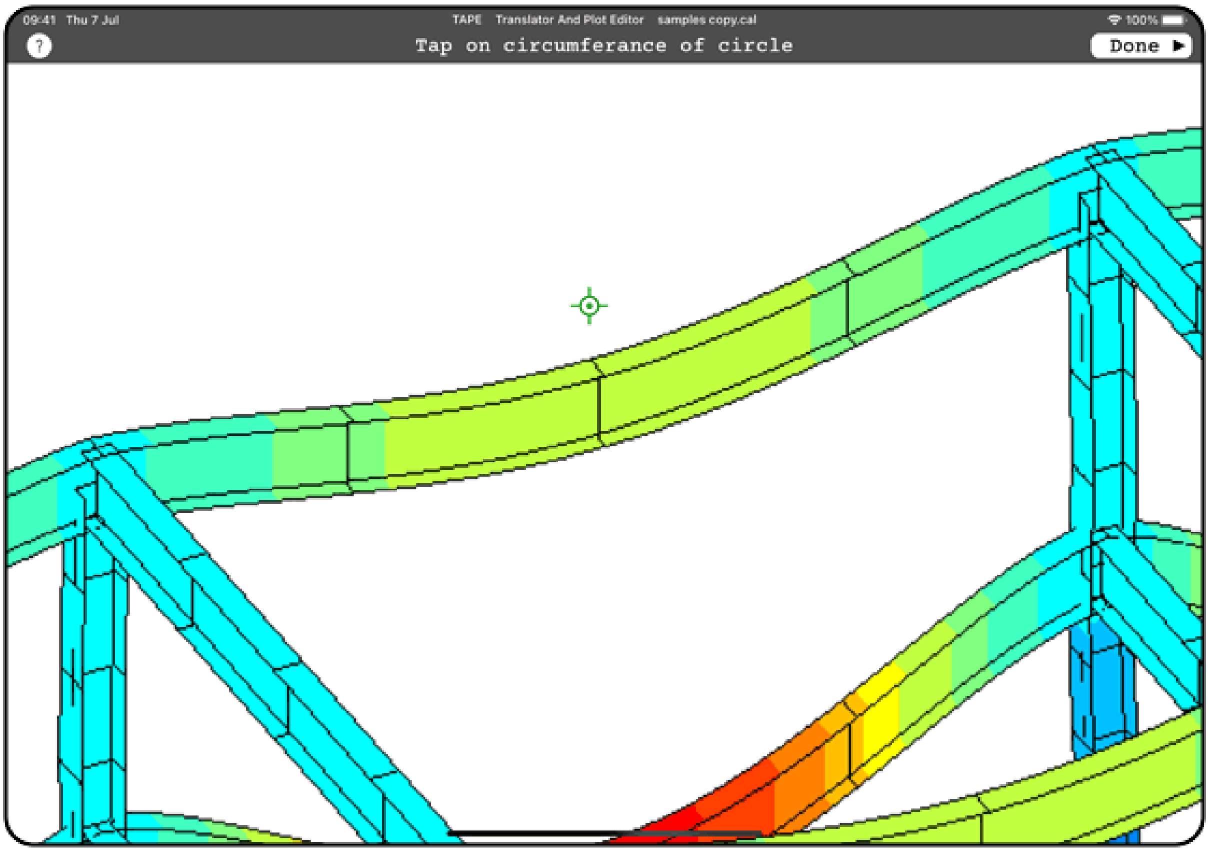


Figure 8.13: Add circle, centre point set.

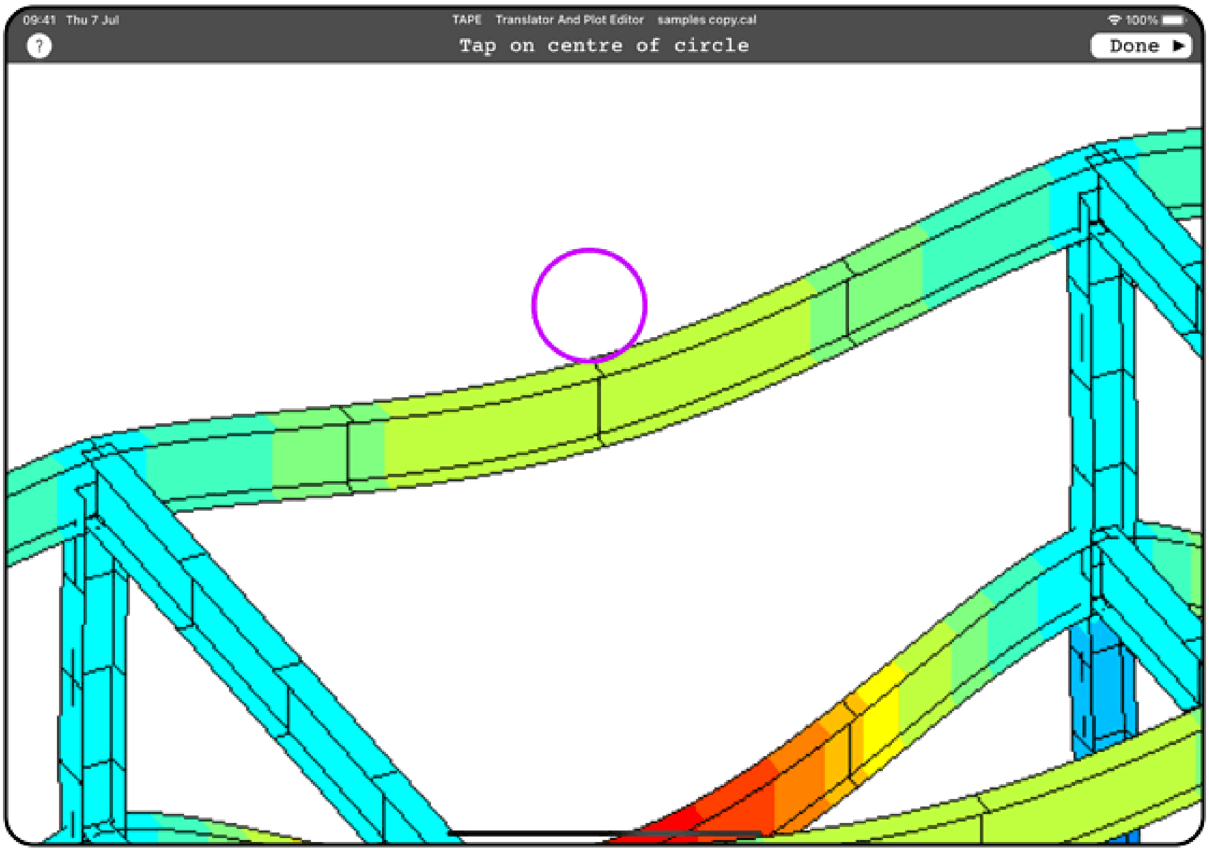


Figure 8.14: Add circle, circumference point set.

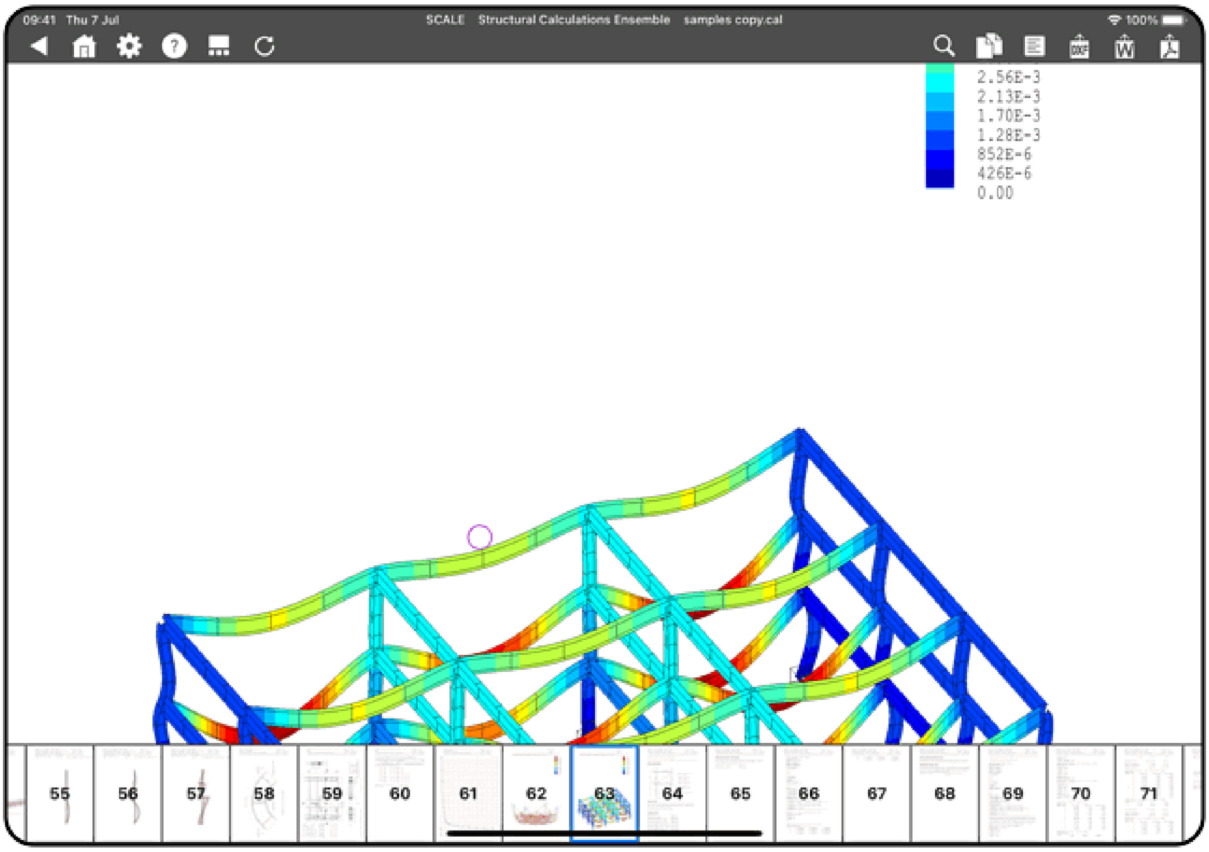


Figure 8.15: End calcs pdf display showing added circle.

Windows: A circle is added by first clicking at its centre. As the cursor is moved the circle is rubber-banded from its centre to the cursor. The final radius is defined by the position at which the mouse is clicked again. To cancel a circle select Stop. Obeys "Snap to grid" for the centre point of the circle. Available on toolbar.

8.6.6 Draw menu → Add text

Text is added by tapping/clicking at its start point (bottom left of the character). It is added in the current pen and text font, but always with a solid line style. When the start position has been selected a one line edit box appears. Enter the text to be added. Click the OK button to add the text, or Cancel to abandon. An empty text string is treated as if Cancel had been selected. Obeys "Snap to grid" with text starting at the nearest grid corner. Available on toolbar.

8.6.7 Draw menu → Delete items

Removes single items from the drawing. Tap/click on an item to remove it. The item is removed from the drawing and placed in the Undo buffer. Consider this tool as the equivalent of a traditional eraser. Available on toolbar.

8.7 Style menu

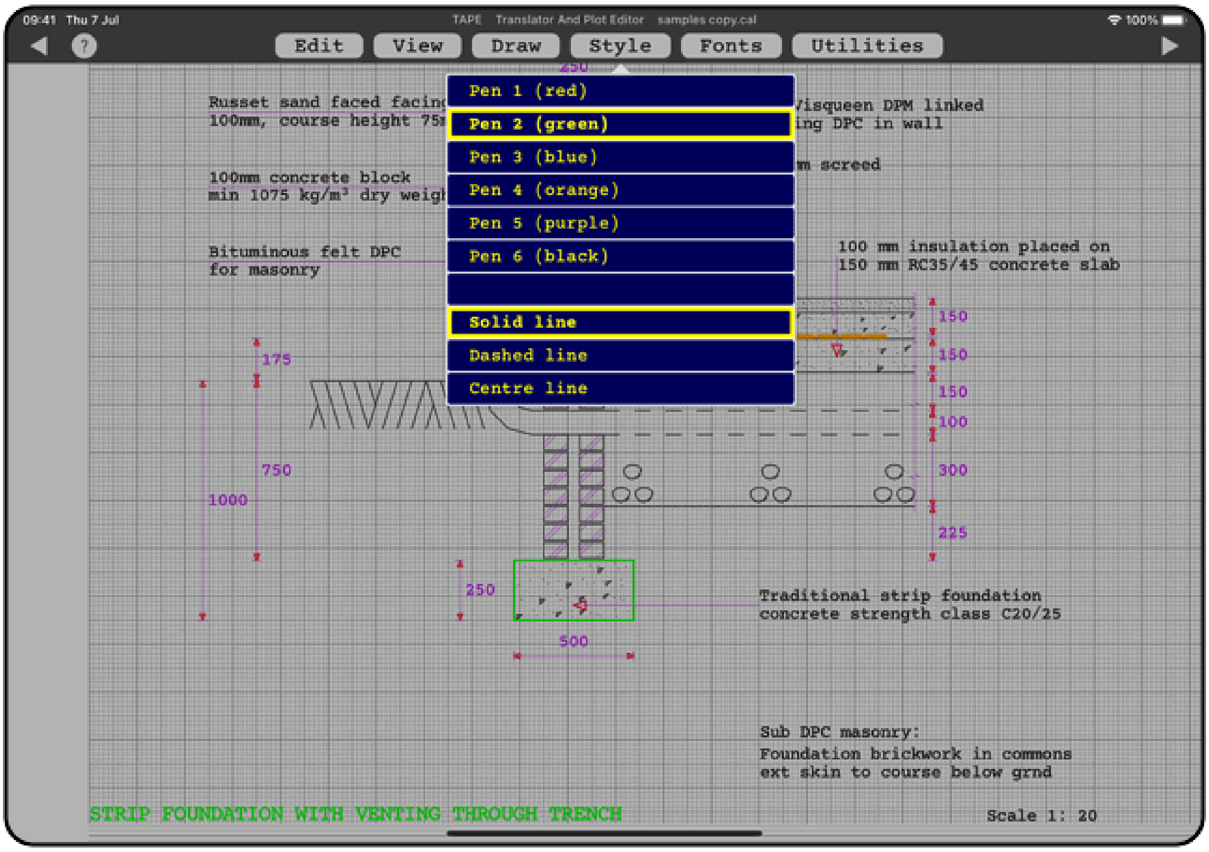


Figure 8.16: Style menu.

PEN 1 to 6
Use this pen for subsequent items.

LINE STYLE
Use the selected style for subsequent items
- Solid - Use solid line style.
- Dashed - Use dashed line style.
- Centre - Use dashed-dotted line style.

8.8 Fonts menu

Use the selected font for all subsequent text.

8.9 Utilities menu

Options available on the Utilities menu, as shown in Figure 8.17, are:

- Orthogonal
- Snap to grid
- Show grid
- Set grid spacing
- Show border
- Show coloured pens
- Align grid X
- Align grid Y
- Align grid XY
- Align grid reset

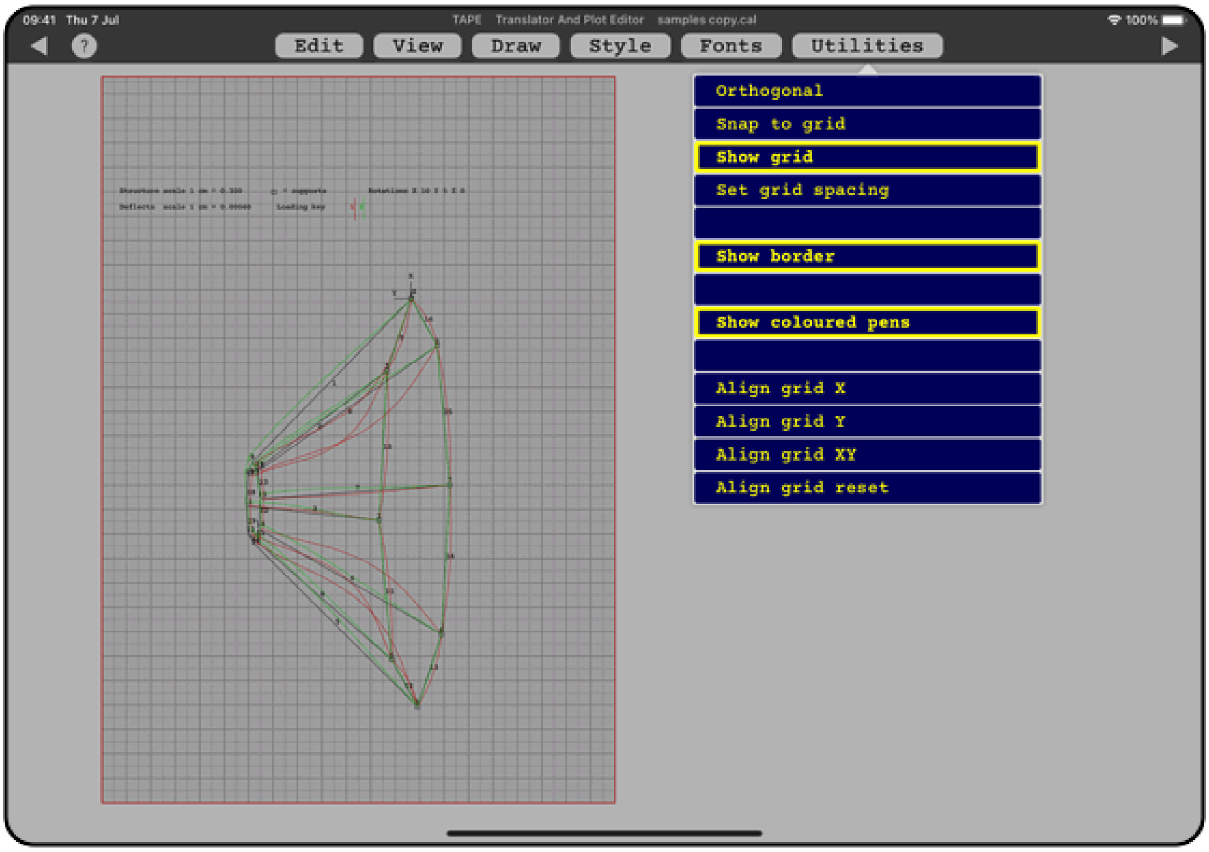


Figure 8.17: Utilities menu.

8.9.1 Utilities menu → Orthogonal

Toggle for orthogonal drawing operations. Any new lines added will be constrained to run vertically or horizontally only.

Can be combined with "Snap to grid".

8.9.2 Utilities menu → Snap to grid

Toggle to adjust cursor clicks or taps to the nearest grid location.

Can be combined with "Orthogonal" drawing operations.

See also "Set grid spacing".

8.9.3 Utilities menu → Show grid

Toggles grid display on and off.

8.9.4 Utilities menu → Set grid spacing

The current grid spacings in X and Y directions are shown in the current units. Simply change the spacings and select OK to change the grid spacings, as shown in Figure 8.18. The grid is used if "Snap to grid" is turned on in the Utilities menu. Setting the grid spacings to very small spacings will make the drawing very difficult to interpret when the Show grid toggle is on. In many cases it is better to retain the grid size to a sensible setting and align the grid to an object of interest using the "Align grid" options in the Utilities menu.

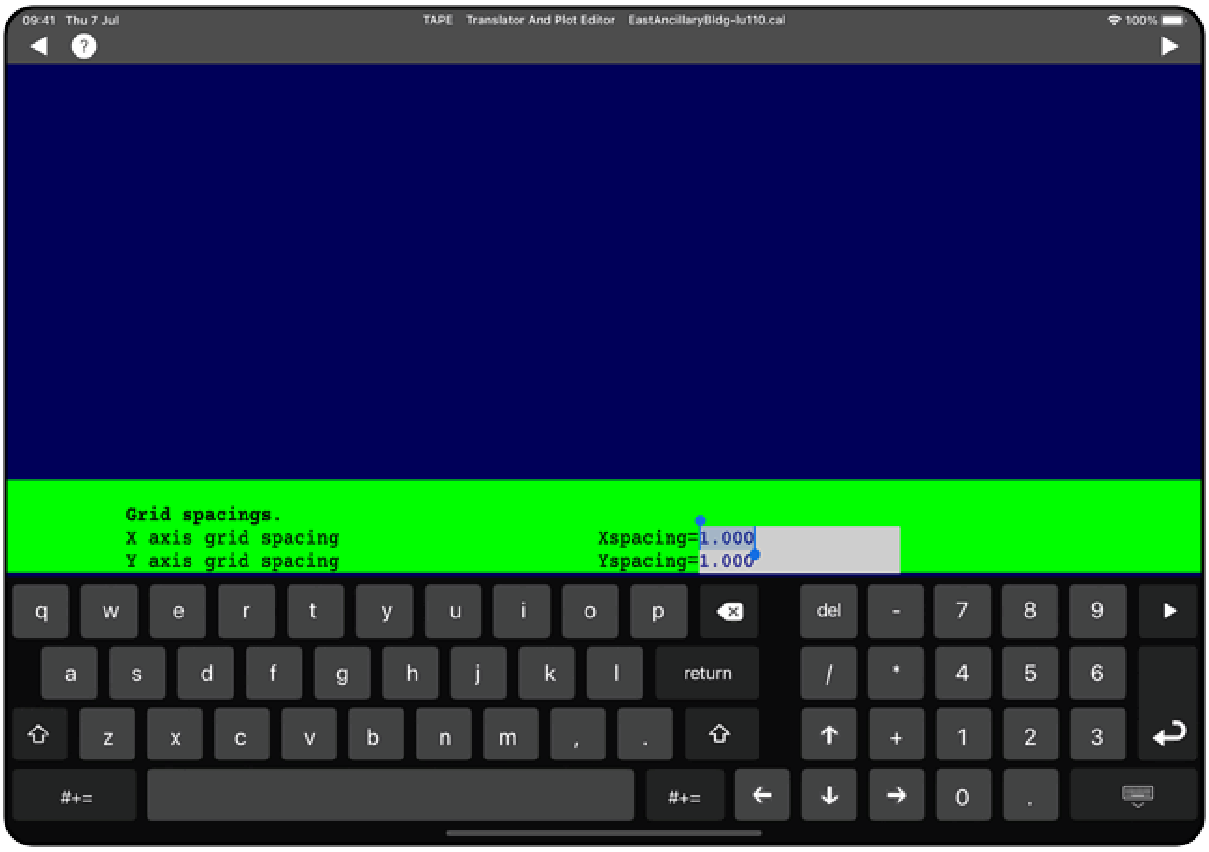


Figure 8.18: Setting the grid spacing.

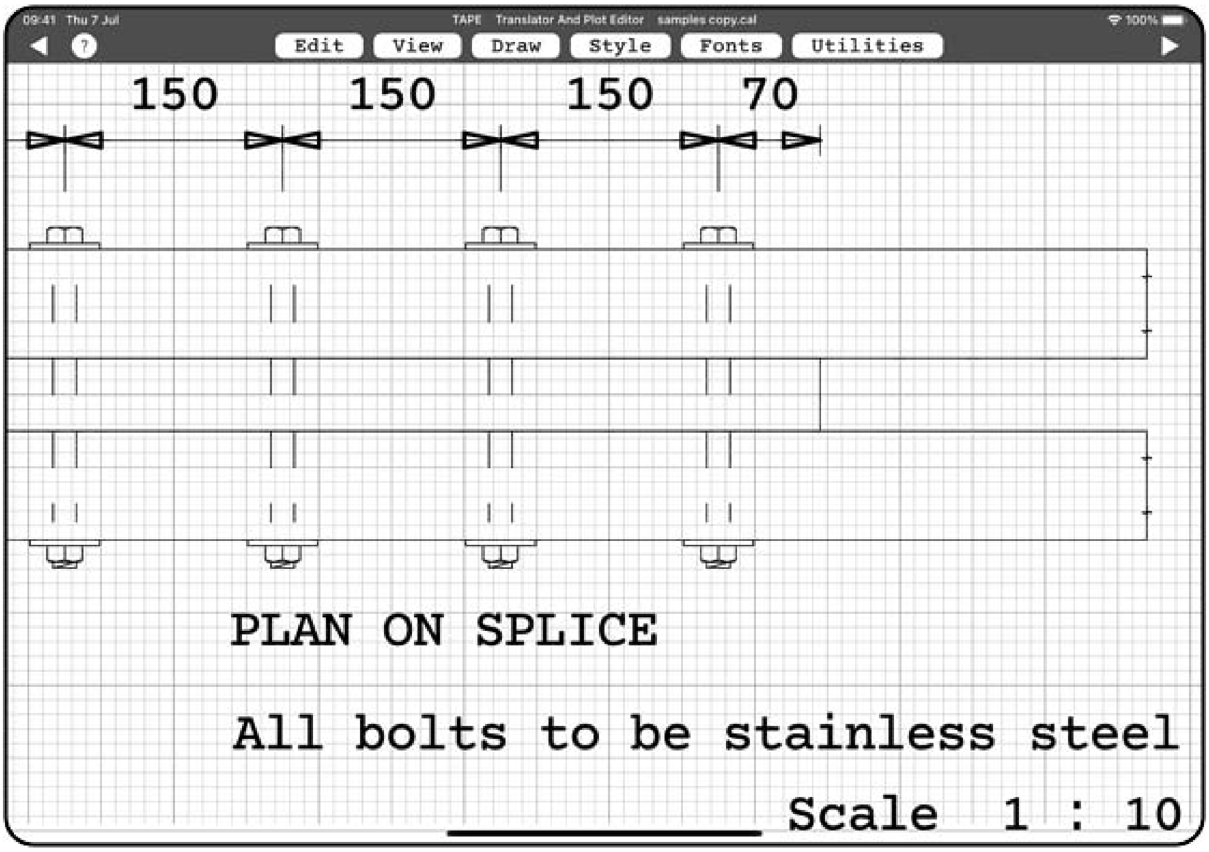


Figure 8.19: Grid spacing of 1mm.

Figure 8.19 shows a zoomed in plot with the default 1mm grid. As a check of the SPADE plot, the 150mm bolt spacing at a scale of 1:10 should measure 15mm on the plot, it can be seen that the bolt spacing does indeed measure 15 x 1mm grid squares.

8.9.5 Utilities menu → Show border

Shows the edge of the current model size, typically the A4 page boundary. See Figure 8.17 for a coloured NL-PLOT, zoomed out, with grid and border shown. The page headings are not drawn directly in TAPE, but can be edited using the "Edit headings" menu option.

8.9.6 Utilities menu → Show coloured pens

Toggles the display of coloured pens between black and white and coloured.

This setting does not make any changes to the drawing, however it does make it easy to see which pens are being used for different parts of the drawing, and consequently which layers different objects will go to if the drawing is saved as DXF, as pens 1 to 6 will go to DXF layers PEN1 to PEN6, see DXF files for further info.

Figure 8.20 illustrates a drawing with coloured pens set, to highlight the different layers present in a SPADE detail.

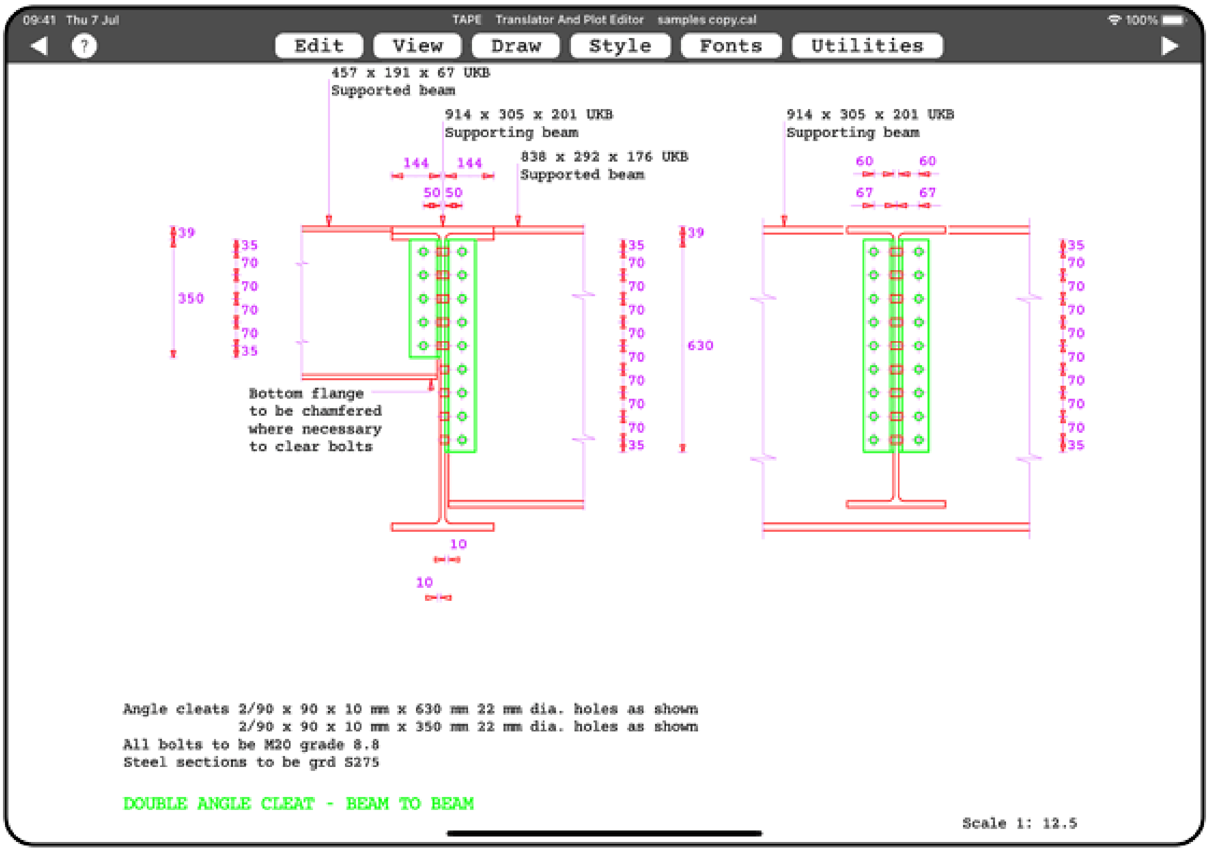


Figure 8.20: Showing coloured pens to illustrate drawing layers.

8.9.7 Utilities menu → Align grid X

Aligns the X grid to the nearest edge on a chosen object.

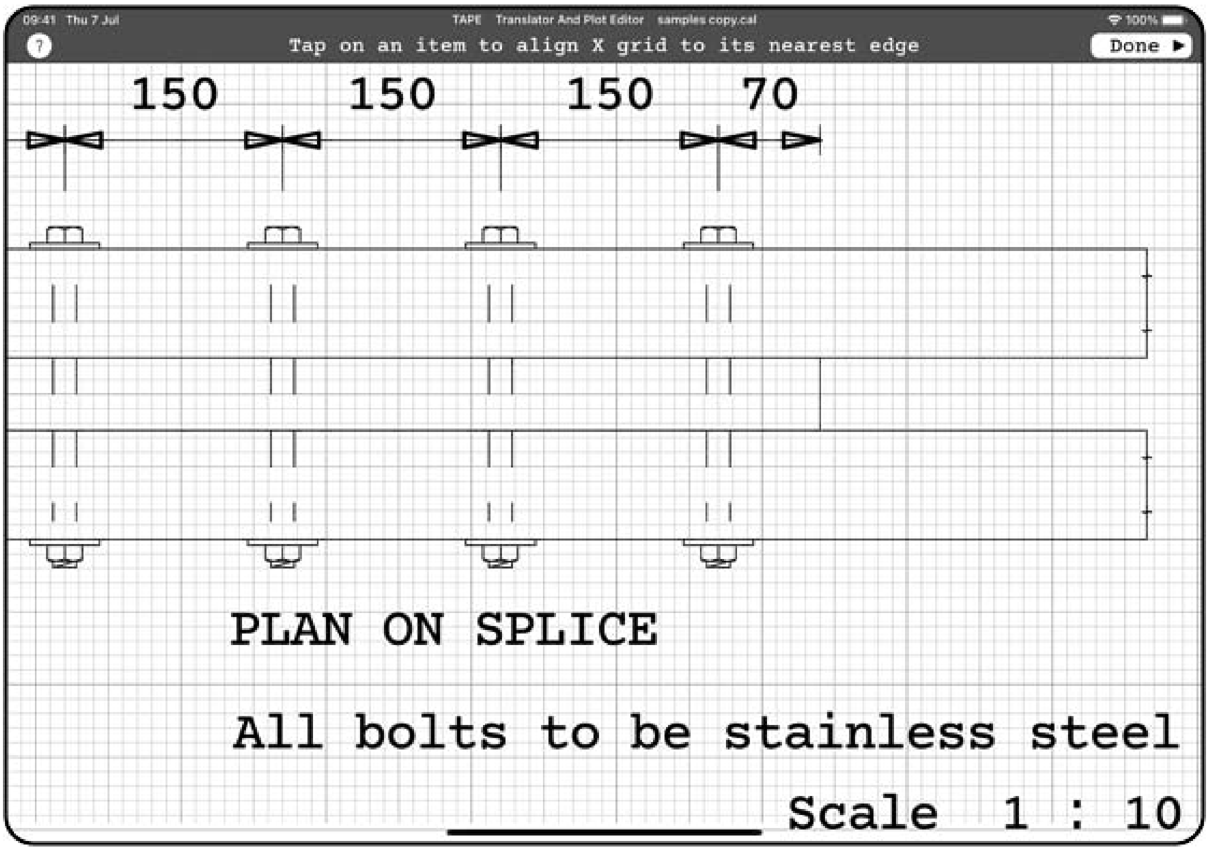


Figure 8.21: Aligning the grid to an object.

As an example to illustrate "Align grid", Figure 8.19 shows bolts through a splice plate, however the centrelines of the bolts do not line up with the grid spacing. By selecting the option "Align grid X" and then tapping on one of the centrelines, this will then move the grid to align with the item that was tapped. As shown in Figure 8.21, the centrelines now lie on the grid, making it easier to check the scaling for the 70mm and 150mm spacings. If "Snap to grid" is set, then any new lines will align with the new grid positions.

Select "Done ►" when you have selected the desired grid alignment, to re-align with the drawing border, select the option "Align grid reset".

Re-aligning the grid in this way does not have any effect on the location of any existing objects in the drawing.

8.9.8 Utilities menu → Align grid Y

Aligns the Y grid to the nearest edge on a chosen object, see description in previous section.

8.9.9 Utilities menu → Align grid XY

Aligns the X and Y grid to the nearest edge on a chosen object, see description in previous section.

8.9.10 Utilities menu → Align grid reset

Resets the grid to align with the drawing border.

8.10 DXF files

To allow you to import details into another CAD system for storage or to add additional details, TAPE allows the drawing to be written in AutoCad's Drawing eXchange File format (DXF).

This feature is not directly accessed from TAPE, but instead from the end calcs screen. When viewing the final calcs for SCALE, LUCID, SPADE and NL-STRESS runs, or viewing an existing calcs file with SCALE option 15, if the currently viewed page contains a diagram, then tapping the DXF button on the right of the toolbar will automatically launch the popup asking where to save the DXF file that will be automatically created by TAPE.

The DXF file produced may be imported into AutoCAD and AutoSketch using their DXFIN or Read DXF commands. Preferably the drawing should be imported into a new drawing to retain its LAYER properties. It can then be turned into a BLOCK and transferred to other drawings if required. Importing a DXF file to an existing drawing will only do a partial transfer of entities only.

The style of the original file is retained as far as possible through the use of LAYERs.

AUTOCAD LAYERS
Early versions of AutoCAD worked on the principle that a LAYER supported a line type and colour. To retain the pen information a

layer is set up for each of the six pens used by TAPE. The layers are named, not surprisingly, PEN1 to PEN6. These layers can be configured in AutoCAD to match your own preferences for colours, linestyles and plotter pen sizes. Items from TAPE are put on the layer corresponding to the pen with which they were drawn.

AUTOCAD COLOURS
Colours are set in the DXF file BYLAYER. The colour numbers start at one and simply increment as each new LAYER is generated up to a maximum of 6 (the number of pens supported by TAPE).

AUTOCAD LINE TYPES
Only the default AutoCAD "Solid line" is used. All patterned lines are generated as a series of shorter solid lines.

AUTOCAD TEXT
Text is converted to AutoCAD TEXT entities with STANDARD style. Text is scaled if the Scale text option is checked in the Configure Drawing options. As far as possible character sizes are matched to those used in the HPGL file. When producing DXF files of a project drawings and text are not scaled as the component details may be of widely differing scales.

AUTOCAD SCALE
Drawings imported to AutoCAD can be scaled using its SCALE option (remembering to change any references to scale on the drawing). TAPE drawings are converted to real world coordinates using the scale active at the time of writing the DXF file if the configuration check box DXF 1:1 is checked. If AutoCAD has been configured to work in the same units as TAPE (mm, in or ft) then entities in the drawing are shown as full-size. Projects are always written without scaling to avoid problems with differing scales of the component parts.

AUTOCAD FONTS
TAPE will produce an AutoCAD font to match any of those used in details. If a DXF file is produced from a detail with the DXF 1:1 check box checked the font sizes will be scaled only if the Scale text box is checked. If the DXF 1:1 check box is unchecked (or a project DXF file is produced) then both the text and drawing will be not be scaled.

8.11 Toolbar (Windows only)

When enabled the toolbar provides short cuts to some of the more common menu options.

If you move the mouse cursor over a button, the button will become highlighted, and a tooltip will appear below the button containing a brief description of what will happen if you click the button. The highlight will disappear when you move the mouse off the button. The tooltip will disappear by itself after a period of time, or if you move the mouse off the button.

start a new drawing
open an existing drawing
save drawing

print drawing
print preview

cut selected items to the clipboard
copy selected items to the clipboard
paste the clipboard contents into the drawing

edit items
draw lines
draw boxes
draw circles
draw text
draw polylines
delete objects
select objects

toggle orthogonal status
toggle snap to grid status
toggle drawing of grid status

zoom in on drawing about point at which mouse is next clicked
zoom out on drawing about point at which mouse is next clicked

abandon current drawing activity.

8.12 Navigation Bar (Windows only)

At the bottom of the Tape window is a navigation bar containing buttons for the frequently used editing options: "Edit Item", "Add Line", "Add Text", "Delete", "Preview" and "Continue" as described below.

The buttons may be activated by clicking on them with the mouse.

The buttons may also be launched by using a keyboard shortcut for the relevant highlighted letter, for example pressing Alt+E will simulate one mouse click on the "Edit Item" button.

EDIT ITEM

This option can be used to change the dimensions and properties of an object. Pick the object you wish to change.

ADD LINE

Add lines defined by a start and end point.

ADD TEXT

Add text defined by a point and the entered text.

DELETE

Delete a single item by clicking on it.

PREVIEW

This launches the Print Preview window to show exactly what the drawing will look like when printed.

CONTINUE

This closes the current editing screen and returns to the previous operation, any changes made to the drawing are saved.

9. Brief description of every SCALE, LUCID, SPADE, NL-STRESS proforma

When a proforma is selected to be run, a brief summary is displayed to the user. The summary presents the options that are available within the proforma.

This help chapter brings together all the summaries for every proforma. This makes it easier for the user to search through for a specific keyword or topic.

9.1 SCALE - Mathematics and Miscellaneous

sc001# BM&SF diagrams for structure previously analysed by NL-STRESS

sc001 Drawing bending moment and shear force diagrams. This is possible only if that structure has already been analysed by NL-STRESS. The diagrams define envelopes; a worst positive and worst negative result at each point along the member. The lines of the diagram are composed of digits. Digit 1 signifies that this value arises from loading case 1, digit 2 that the value arises from loading case 2 etc. (for load cases 10-35 the letters A-Z are used and for load cases 36 onwards the letters a- z). Values of maximum and minimum deflection, dimensions and properties of the member, and maximum and minimum axial loads are tabulated beneath the diagrams.

sc015 Display a text file, also edit, append, save to pdf and docx

Option 15: Display, edit, append, save to pdf and docx.
This option can be used to add notes to a calculations file without needing to run through a proforma again, or to change saved proforma values before re-running a proforma.
The following file types can be edited with this option:

- .dat SCALE page headings file, or NL-STRESS data file.
- .cal SCALE calculations file.
- .stk stack file containing all the variables used in a proforma.
- .res NL-STRESS results file.
- .plt NL-STRESS hpgl plot file.

sc018 Add a company logo to each pdf calculation page

Option 18: Add a company logo to each calculation page.
Use this option to add a logo to each calculation page, typically this will be a banner image aligned with the page heading, but you can place the image anywhere on the page. The image must be a jpg, jpeg or png.

sc020 Pages with heading only

sc020 Pages with heading only, creates headed plain pages for contents, sketches, hand calculations, notes etc. Useful for appending into collections of calcs using the FILE option.

sc021 Invoice for structural work

sc021 Invoice for structural work. This option is intended for use with preprinted stationery and therefore omits the page heading normally associated with SCALE. As with all other proformas, it may be edited using any text editor and thereby make it more suitable for the office concerned.

sc022 First page of calculations

sc022 First page of calculations. This option enables the engineer to enter the design team, project summary, construction method, design philosophy etc. collectively comprising the first page/s of a set of SCALE calculations.

sc025 Job information sheet

sc025 Job information sheet. This option lists typical headings of a job information sheet for the engineer to word-process or edit any section or all of the document, to produce a finished job information sheet.

sc027 Solution of triangles

sc027 Solution of triangles. This option, which is based on Trigonometry & Kempe's Engineers Year Book solves triangles:

- given three sides
- given two sides and included angle
- given two sides and non included angle
- given two angles and one included side.

sc028 Solution of simultaneous equations

sc028 Simultaneous linear equations. This option solves:

- two simultaneous equations having two unknowns
- three simultaneous equations having three unknowns
- four simultaneous equations having four unknowns.

sc029 Solution of quadratic, cubic & quartic equations

sc029 Solution of quadratic, cubic and quartic equations. This option which is based on 'Advanced Mathematics for Technical Students' by Geary, Lowry & Haden Part 1 1959 deals with the solution of:

- quadratic equations
- cubic equations using Graeffe's method
- quartic equations using Graeffe's method
- cubic, quartic and higher powered by trial and error.

sc030 Coordinate geometry

sc030 Coordinate geometry. This option solves the following:

- setting out parabolic arc
- setting out circular arc
- equation of line through x1,y1 and x2,y2 and optionally distance of point x3,y3 from the line
- equation of plane through x1,y1,z1 x2,y2,z2 and x3,y3,z3 and optionally distance of point x4,y4,z4 from the plane
- intersection of line joining (xa1,ya1) & (xa2,ya2) with line joining (xb1,yb1) & (xb2,yb2)
- rotation of a point in space about the X, Y and Z axes.

sc031 Differentiation of standard forms

sc031 Differentiation of standard forms. This option differentiates the following standard forms: x^n, e^(k.x), sin(a.x+b), cos(a.x+b), loge(a.x+b), tan(x), cot(x), sin^-1(x/a), cos^-1(x/a), tan^-1(x/a), sinh(x), cosh(x), tanh(x).

sc032 Differentiation of a general function

sc032 Differentiation of a general function. This option finds the first and second derivatives of a general function.

sc033 Integration of standard forms

sc033 Integration of standard forms. This option integrates the following standard forms between lower and upper limits: x^n, 1/x, e^x, sin(x), cos(x), tan(x), cot(x), sec(x), cosec(x), sec^2(x), cosec^2(x), sec(x)tan(x), cosec(x)cot(x), sinh(x), cosh(x), 1/(x^2+a^2), 1/(x^2-a^2), 1/(a^2-x^2), 1/(a^2-x^2)^0.5, 1/(x^2+a^2)^0.5, 1/(x^2-a^2)^0.5.

sc034 Integration of a general function

sc034 Integration of a general function. This option integrates a general function between lower and upper limits using Simpson's rule.

sc035 Mensuration of plane areas and solids

sc035 Mensuration of plane areas and solids. This option computes lengths, areas and volumes for: rectangle, parallelogram, trapezium, circle, segment of circle, ellipse, solid cylinder, hollow cylinder, cone and sphere.

sc036 Statistics and quality control

sc036 Statistics and quality control. This option computes: mean value, variance, standard deviation and coefficient of variation for a set of concrete tests cubes based on 'Concrete Technology, Volume 1' by D.F. Orchard.

sc037 Latent roots

sc037 Latent roots or characteristic values or eigenvalues. In a number of problems a set of linear simultaneous equations of the form K.v = lambda.v occurs, where K is a square matrix, v a column matrix and lambda a number. It can be shown that these equations have non zero solutions if, and only if, the matrix of coefficients is singular. That is det | K – lambda.I | = 0. This equation is called the characteristic equation of matrix K, and is of fundamental importance in the application of matrices. The roots of this equation in lambda are called the latent roots of the matrix K, this option will compute them for simple cases.

sc038 Centre of gravity

sc038 Centre of gravity, or centroid of several weights. This option will compute the centroid for a set of weights applied at known x,y,z coordinates.

sc039 Matrix inversion

sc039 Matrix arithmetic. This option will perform: matrix addition, matrix multiplication, and matrix inversion for square matrices (square symmetric for matrix inversion).

sc041 Automated running of multiple instances of option 42

sc041 Automated running of option 42 for multiple PARAMETER tables for multiple proformas.

sc042 Verifying the correctness of SCALE proformas & NL-STRESS models

sc042 How to make a SCALE proforma or NL-STRESS Verified Model check its own logic. Models which contain a PARAMETER table or have an external parameter table, may have their logic checked by running them under sc042 and responding to the prompts. To see if a SCALE proforma contains an internal PARAMETER table, include the /P switch following the option number. External parameter tables are held in a file of the same name as the proforma but with the extension .prm All NL-STRESS Verified Models contain a PARAMETER table and also include a self check.

sc044 For testing only, automated running of all examples

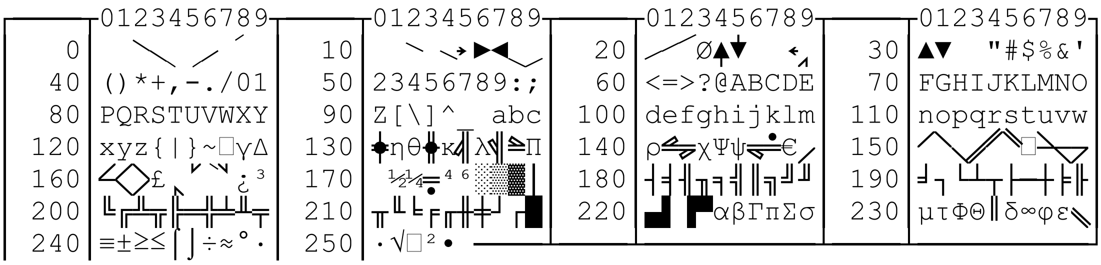
Option 44 is provided for checking that the SCALE, LUCID, SPADE, NL-STRESS, NL-PLOT and SCP programs, proformas and data files are all functioning correctly, and to demonstrate their capabilities.
Once started, option 44 will automatically generate sample output calculations, detailing drawings, analysis runs and plots of results for over 4000 runs of SCALE, LUCID, SPADE and NL-STRESS and collate all output into one hyperlinked pdf file.

sc048 Description of every SCALE/LUCID/SPADE/NL-STRESS proforma

sc048 Listing of every SCALE, LUCID, SPADE, and NL-STRESS proforma, with a brief description for each one.

sc050 Greek alphabet and SCALE character set

sc050 Greek alphabet, SCALE character set, and sample figures as used in SCALE, LUCID, SPADE and NL-STRESS.



sc051 Conversion between Imperial and metric and vice versa

sc051 Conversion of units from Imperial to metric & vice versa. This option converts: length, area, volume, first moment of area, second moment of area, force, UDL, pressure, stress, and weight/unit volume to Imperial, metric and SI units as required.

sc052 Compound interest for mortgage and loan payments

sc052 Compound interest. This option computes:

- the principal or amount A at the end of n years of an initial sum s lent at p % per annum, or
- the monthly repayment, m, on a mortgage loan of s at p percent compound interest over n years, or
- the rate of interest being charged on a loan s to be repaid at m per month over n years, or
- the principal or amount A at the end of n weeks of an initial sum s lent at p % per week.

sc053 Expressing a set of data points as a polynomial

sc053 Expressing a set of data points as a polynomial.
This proforma derives a polynomial expression for fitting a set of coordinates. The engineer may input the order of the polynomial, or let the procedure choose the order which gives the best fit. The data points are then plotted with the polynomial for comparison.

sc055 Cross referencing of numerical variables in proforma files

sc055 Cross referencing of numeric variables for proforma files
This option produces a listing showing each 'variable' followed by each line in which the 'variable' occurs. The listing is useful when checking a proforma.

sc070 Simple test example

sc070 Simple test example. I-section properties - given section depth, breadth, web & flange thicknesses, computes cross-sectional and shear areas, inertias about principal axes and torsion constant.

9.2 SCALE - Reinforced Concrete Design to Eurocode 2 and BS8110

sc072 Rectangular beam - flexure only

sc072 Rectangular beam - computes areas of tension & compression steel to BS8110 or Eurocode 2 assuming a rectangular concrete stress-block and limiting the depth to the neutral axis. The spacing of the chosen reinforcement is then checked.

sc073 Rectangular beam - flexure, span/depth, curtailment, laps

sc073 Rectangular beam - computes areas of tension & compression steel to BS8110 or Eurocode 2 assuming a rectangular concrete stress-block and limiting the depth to the neutral axis. Contains options to determine to design link reinforcement to resist shearing forces, and to check for excessive deflection by means of the limiting span/effective-depth ratio.

sc074 Tee beam - flexure only

sc074 Tee beam - computes areas of tension & compression steel to BS8110 or Eurocode 2 assuming a rectangular concrete stress-block and limiting the depth to the neutral axis. Contains options to determine to design link reinforcement to resist shearing forces, and to check for excessive deflection by means of the limiting span/effective-depth ratio.

sc075 Tee beam - flexure, span/depth check

sc075 Tee beam - computes areas of tension & compression steel to BS8110 or Eurocode 2 assuming a rectangular concrete stress-block and limiting the depth to the neutral axis. Contains options to determine bending-moment values corresponding to bar curtailment points, to evaluate bar lengths needed for anchorage bond and laps, to design link reinforcement to resist shearing forces, and to check for excessive deflection by means of the limiting span/effective-depth ratio.

sc076 Design of walls

sc076 Design of RC walls. Deals with the design to BS8110 or EC6 of concrete walls that contain some reinforcement (unreinforced walls are classified as masonry and are covered by the requirements of BS5628 or Eurocode 6). The option is based on information given in Table 170 of the current Reinforced Concrete Designers' Handbook.

sc077 Wall to wall intersection

sc077 Wall to wall intersections. Deals with the design of the reinforcement to resist bending at wall-to-wall intersections where the applied moment tends to 'open' the corner. The option is based on information given in Table 173 of the current (10th) edition of the RCD Handbook, which was originally developed from data given in 'Reinforcement detailing of frame corner joints' by P.S.Balint and H.P.J.Taylor (C&CA publication 42.462. 1972) and an article by Dr.F.A.Noor, which appeared in 'Concrete' in July 1977. Calculations are also in accordance with BS EN 1992-1-1.

sc078 Rectangular and Tee beam - shear only

sc078 Shear in rectangular & Tee beams to BS8110 with av enhancement or Eurocode 2. The procedure adopted is to input first the number of legs transversely and then the link diameter, from which the longitudinal spacing is calculated. Options are available to alter the size, number and spacing.

sc080 Solid slab - flexure, tension steel only, span/d /m width

sc080 Solid R.C. slabs reinforced in tension only to BS8110 or Eurocode 2. Deals with the design of solid reinforced concrete slabs reinforced with tension steel only and with any redistribution limited to 10%. Checks are included to ensure compliance with limiting span/effective-depth and minimum steel requirements. Calculations are in accordance with BS8110 and Eurocode 2.

sc081 Solid slab - flexure, tension & comp steel, span/d /m width

sc081 Flexure in solid slabs, design per metre width. The required areas of tension steel (As) and compression steel (As' or As2) are determined using the design formulae in BS8110 or Eurocode 2 assuming a rectangular concrete stress-block and limiting the depth to the neutral axis.

sc082 Solid slab - flexure, tens & comp steel, span/d given width

sc082 Flexure in solid slabs designed for a given width. The required areas of tension steel (As) and compression steel (As' or As2) are determined using the design formulae in BS8110 or Eurocode 2 assuming a rectangular concrete stress-block and limiting the depth to the neutral axis.

sc083 Solid slab - shear only

sc083 Shear in zone of solid slab of given width (no enhancement)to BS8110 or Eurocode 2. The procedure adopted is to input first the number of legs transversely and then the link diameter, from which the longitudinal spacing is calculated. Options are available to alter the size, number and spacing of links.

sc084 Slab rib - shear only

sc084 Shear in rib of ribbed slab (no enhancement). This option, which is based on the requirements of BS8110 and was written by Professor Bill Cranston, calculates the number and size of links required (if any) to resist the shearing force in the rib of a ribbed slab. The requirements are similar to those for shear in the rib of a flanged beam, except that no nominal links are needed when the actual shear stress v is less the critical value vc and the maximum permitted spacing of any links is now equal to the effective depth. Calculations are also in accordance with BS EN 1992-1-1.

sc085 Two way slabs - moments only

sc085 Moments in two-way slabs under UDL. This option calculates the bending moments in the middle strips of solid reinforced concrete slabs subjected to uniform load in accordance with the provisions of Clauses 3.5.3 of Part 1 of BS8110. The appropriate coefficients are read from Table 3.15 or, in the case of slabs simply supported at all four edges, calculated from Equations 12 and 13 of the Code. Moments are calculated over a width of middle strip of three-quarters of the slab width for slabs with torsional restraint, and over the full width for the case when the slab is simply supported. Calculations are also in accordance with BS EN 1992-1-1.

sc086 Two way slabs - with design

sc086 Design of two-way slabs supporting uniform load. Calculates the moments in the middle strips of solid slabs carrying uniform loads in accordance with the requirements of Clauses 3.5.3 of BS8110. The appropriate coefficients are read from Table 3.15 or, in the case of slabs that are freely supported at all four edges, calculated from Equations 12 and 13. Moments are calculated for the middle strips, and the sections are designed as singly-reinforced slabs. Finally, the minimum area of steel is calculated. Calculations are also in accordance with BS EN 1992-1-1.

sc087 Flat slabs - simplified method

sc087 Moments in flat slabs - simplified method - no drop panels. The simplified method of calculating moments described in Clauses 3.7.2 to 3.7.4 is only applicable if lateral stability does not depend on the slab-column connections and where are least 3 rows of panels of approx. equal span are provided in the direction being considered. The total support and span moments are calculated first from Table 3.13 with the support moments in column and middle strips being derived in accordance with the 75% and 25% figures from Table 3.20. Calculations are also in accordance with BS EN 1992-1-1.

sc088 Slabs - punching shear

sc088 Punching shear calculations to clause 3.7.7 of BS8110. The basic procedure employed is to determine the total number of legs of shear reinforcement required in each zone, starting with Zone 1. You can choose how many legs to provide at the first steel perimeter, and the remainder are assigned to the next perimeter out. Calculations for successive zones follow, knowing the number of inner perimeter legs. When a zone is found where only nominal steel is needed, the outer steel perimeter is allocated a proportion of the minimum steel area. Calculations are also in accordance with BS EN 1992-1-1.

sc089 Annular column

sc089 Annular R.C. column section to BS8110 or Eurocode 2. The analysis is based on a set of design tables derived from charts that are similar to, and based on the same principles as, those provided in the Joint IStructE/ICE 'Manual for the design of R.C. building structures'. An explanation of the derivation of the formulae used to prepare such charts is given in 'Examples of the design of buildings to BS8110' (4th Ed.) by Reynolds and Steedman.

sc090 Stocky column - biaxial bending

sc090 Short braced column subjected to biaxial bending. Determines the reinforcement required in a rectangular section to resist a given combination of bending about both major axes, together with axial load. The required proportion of steel using BS8110 is read from sets of tables based on the relevant design charts. In the Eurocode version iterative formulae are used to determine the area of reinforcing steel required. Calculations are also in accordance with BS EN 1992-1-1.

sc091 Slender rectangular column - uniaxial bending

sc091 Slender rectangular column subjected to uniaxial bending. Determines the reinforcement required in a rectangular section to resist a given combination of bending about an axis, together with axial load. The required proportion of steel using BS8110 is read from sets of tables based on the relevant design charts. In the Eurocode version iterative formulae are used to determine the area of reinforcing steel required. Calculations are also in accordance with BS EN 1992-1-1.

sc092 Slender rectangular column - biaxial bending

sc092 Slender rectangular column subjected to biaxial bending. Determines the reinforcement required in a rectangular section to resist a given combination of bending about both major axes, together with axial load. The required proportion of steel using BS8110 is read from sets of tables based on the relevant design charts. In the Eurocode version iterative formulae are used to determine the area of reinforcing steel required. Calculations are also in accordance with BS EN 1992-1-1.

sc094 Circular column - design for ultimate

sc094 Ultimate-resistance design of circular section to BS8110. Determines the reinforcement required in a circular section of given dimensions and material properties to resist a specified design applied moment and axial load. The trial-and-adjustment procedure operates automatically with only minimal information being displayed on-screen for monitoring purposes. Output is provided when a solution has been obtained or when a further decision is needed. The full calculations for the final successive trial are printed, with details of the reinforcement selected. Calculations are also in accordance with BS EN 1992-1-1.

sc095 Circular column - cracking

sc095 Analysis of circular section including crack width calculation to BS8110. Employs an iterative procedure to obtain the response of a given section which is subjected to increasing curvatures. At each curvature the strain at the outer edge is adjusted until the required ratio of N/M is obtained. When the value obtained for M exceeds the prescribed value, the curvature and edge strain corresponding to the required value of M are interpolated. Check calculations for the strain profile that is thus defined are then undertaken, and finally the maximum crack width is calculated. Calculations are also in accordance with BS EN 1992-1-1.

sc096 Circular column - cracking with tension stiffening

sc096 Analysis of circular section including crack-width calculation, with tension stiffening to BS8110 or Eurocode 2. Employs an iterative procedure to obtain the response of a given section which is subjected to increasing curvatures (with loads that are proportionate to the prescribed N/M ratio). At each curvature the strain at the outer edge is adjusted until the required ratio of N/M is obtained. When the value obtained for M exceeds the prescribed value, the curvature and edge strain corresponding to the required value of M are interpolated. Check calculations for the strain profile that is thus defined are then undertaken, and finally the maximum crack width is calculated.

sc097 Rectangular column uniaxial bending - rigorous procedure

sc097 Ultimate-resistance design of rectangular section for uniaxial bending and thrust to BS8110. The analysis begins with a percentage of 0.4%, and iterates to find the capacity of the section under either an ultimate strain of 0.0035 in the concrete, or 0.02 in the tension steel, with the required ratio of N/M. If N and M to be resisted are less than those calculated, these results are presented; otherwise the analysis is repeated with a steel proportion of 6%. If the target values of N and M fall between these limiting values a trial percentage of steel is interpolated, and the capacity of the resulting section determined. Calculations are also in accordance with BS EN 1992-1-1.

sc098 Rectangular column biaxial bending - rigorous procedure

sc098 Design of rectangular section for biaxial bending to BS8110. Determines the minimum size of bar theoretically necessary to reinforce a given section to resist a specified axial force plus bending about both axes, assuming that the stress-strain relationship in the concrete is that in Figure 2.1 of BS8110. The trial-and-adjustment process employed operates in a hidden mode, printout appearing only when a solution has been found or when the user must make further decisions (e.g. to increase the concrete grade or the size of the section). Calculations are also in accordance with BS EN 1992-1-1.

sc099 Circular column in accordance with IStructE/ICE manual

sc099 Circular R.C. column design to BS8110. Enables the rapid design of any circular column, either short or slender, when subjected to any combination of axial thrust and bending. The area of reinforcement required is found by using a set of look-up tables derived from the formulae given in 'Examples of the Design of Reinforced Concrete Buildings' by Reynolds and Steedman, which are similar to those used to prepare the charts in the IStructE/ICE 'Manual for the Design of Reinforced Concrete Building Structures'. For slender columns, a cycling procedure is used to determine the appropriate moment-reduction factor K. Calculations are also in accordance with BS EN 1992-1-1.

sc100 Simply supported rectangular beam/slab with general loading

sc100 Rectangular simply supported reinforced concrete beam/slab to BS8110 or Eurocode 2. Factors dead and imposed loads and computes BM's, SF's and service deflections for a simply supported beam subjected to any number of UDL's, triangular loads and point loads. Computes tension and compression steel details at the position of maximum moment. Computes shear reinforcement details, and optionally computes the deflection limits for the reinforcement selected. addition the BS8110 version computes anchorage and lap lengths.

sc101 RC beam on elastic piles subj. to train of moving point loads

sc101 RC beam on elastic piles subjected to train of moving loads. The geometry, material properties and loading for the beam are input interactively i.e. by question and answer. A text file of data is written and saved in the file sc101.dat; the program NL-STRESS is invoked to analyse the data and produce a results file having the name sc101.res. The structure is displayed on the screen; the structure is analysed; bending moments, shear forces & displacements are computed and plotted as required. The form of the NL-STRESS data is similar to that for the well-known program called STRESS; reference 'STRESS: A User's Manual' M.I.T. Press 1964.

sc102# Rectangular beam/slab section

sc102 Rectangular reinforced concrete beam/slab section to BS8110 or Eurocode 2. Designs rectangular beam or slab sections of normal-weight concrete. Computes tension and compression steel details, shear reinforcement details and deflection limits.
In addition the B8110 version calculates anchorage and lap lengths and for beams of depth greater than 750 mm, side reinforcement is provided in accordance with the requirements of clause 3.12.11.2.6.

sc103 Rectangular beam/slab section - checking aid

sc103 Rectangular beam/slab section design chart to BS8110 or Eurocode 2. RC design charts are particularly suitable for checking. On a real job there may be perhaps a dozen different beam sizes, and perhaps half a dozen different column sizes. It is suggested that charts be produced for each size of beam & column for the material strengths for the job; and that the charts be bound into the job calculations.

sc104# Biaxially bent stocky column

sc104 Design of rectangular biaxially bent stocky column. Calculations for reinforcement are in accordance with BS 8110 Clause 3.8.4.5 and IStructE 'Manual for the design of reinforced concrete building structures'. Alternatively design to Eurocode 2 may be considered. In both versions checks on slenderness and eccentricity of axial load are considered and then computes the equivalent moment about one axis and designs reinforcement accordingly.

sc105 Rectangular column section - checking aid

sc105 Rectangular column section design chart to BS8110 or Eurocode 2. RC design charts are particularly suitable for checking. On a real job there maybe perhaps a dozen different beam sizes, and perhaps half a dozen different column sizes. It is suggested that charts be produced for each size of beam & column for the material strengths for the job; and that the charts be bound into the job calculations.

sc106# Flanged beam section design to IStructE manual

sc106 Design of R.C. flanged beam to IStructE Manual requirements. Deals with the design of flanged sections strictly in accordance with the requirements of the IStructE/ICE 'Manual for the Design of Reinforced Concrete Building Structures', utilising the simplified rectangular concrete stress-block described in Clauses 4.5.5 of that document. Calculations are also in accordance with BS EN 1992-1-1.

sc107# Flanged beam section design to Clause 3.4.4.4

sc107 Design of flanged beam section to BS8110 or Eurocode 2. Deals with the design of flanged sections strictly in accordance with the requirements of BS8110 or Eurocode 2. Computes tension and compression reinforcement in the BS8110 version, tension reinforcement only in the Eurocode version. Computes shear reinforcement and checks deflection limits.

sc108 Torsion steel for rectangular section

sc108 Torsion on reinforced concrete section. For a given torque, shear, and area of steel required for bending, this option computes torsional steel requirements in accordance with BS8110: Part 2 Clause 2.4.4. The torsion steel comprises closed links and symmetrically spaced longitudinal steel, both of which must be added to the reinforcement required for bending and shear. Calculations are also in accordance with BS EN 1992-1-1.

sc109 Ground-supported concrete slabs for industrial buildings

sc109 Ground-supported concrete slabs for industrial buildings. This option designs ground-supported concrete floors for industrial buildings. The design method employed was first described in the 'Design of floors on ground' by J.W.E. Chandler (Cement and Concrete Association Technical Report 550: 1982). For further information see BCA Interim Technical Note 11, and the 1988 Concrete Society report 'Concrete Industrial Ground Floors'.

sc110 Pad footing with uniaxial bending, including section design

sc110 Design of pad footing. Deals with the design of an isolated reinforced concrete footing to BS8110 or EC2. The eccentricity of load relative to the centroid of the base is first found, and a check made that the line of action lies within the base. Design is carried out first in the direction in which the moment can be applied. Design in the other direction is then undertaken assuming that this steel is positioned above the first layer. Where shear steel is needed an option is provided to allow the slab depth to be increased otherwise a message is printed. Finally a check is undertaken for punching shear.

sc111 Deep beams - Kong's method

sc111 Design of deep reinforced concrete beams. Deals with the design of deep beams using the simplified formulae originally presented by F.K.Kong, P.J.Robins and G.R.Sharp in 'The Structural Engineer' in April 1975 (Vol.53, No.4, pp.173-80). The method and formulae in this option are as adapted in Table 148 of the 10th edition of the 'RCD Handbook'. Basic conditions that must be satisfied: a) loads are static and are applied to top of beam only; b) Only uniform loads and symmetrically-arranged twin point loads may be considered; c) only one opening (of limited size) or two symmetrically-arranged openings may be provided in the web of the beam.

sc112 Modular-ratio design of reinforcement for circular column

sc112 Modular-ratio theory: steel required in circular column. Calculates the reinforcement required in a circular column section subjected to any combination of axial thrust and bending. The area of steel needed is determined using the formulae developed in 'Reinforced Concrete Design' by G. P. Manning (Longmans: 3rd edition, 1966. Pages 84 to 88). This analysis assumes that the individual reinforcing bars can be represented without significant error by a continuous ring of steel having the same total area.

sc113 Modular-ratio calculation of stresses for circular column

sc113 Modular-ratio theory: stresses in circular column. Calculates the resulting stresses in a circular column section due to any combination of axial thrust and bending. The maximum stresses are determined using the formulae developed in 'Reinforced Concrete Design' by G. P. Manning (Longmans: 3rd edition, 1966. Pages 84 to 88). This analysis assumes that the individual reinforcing bars can be represented without significant error by a continuous ring of steel having the same total area.

sc117 Simply supported R.C. staircase

sc117 Simply supported staircase. Deals with bending in simply supported staircase flights, designed for a given width, with checks on minimum steel and span/effective-depth ratio. The required areas of tension steel (As) are determined using the design formulae in Clause 3.4.4.4 of BS8110, i.e. assuming a rectangular concrete stress-block and limiting the depth to the neutral axis to 0.5*d. When determining the allowable clear distance between the bars, fs is determined from equation (8) in Clause 3.4.6.5. Calculations are also in accordance with BS EN 1992-1-1.

sc118 Concrete nibs

sc118 Design of concrete nibs to BS8110 or EC2. Load may be considered to act: a) at outer edge of bearing pad (if provided); b) at beginning of chamfer (if provided); c) at outer edge of nib. The procedure computes diameter and spacing of bars and bond length requirements. A check is made on the shear resistance.

sc119 Fire resistance

sc119 Fire resistance to BS8110: Part 2: 1985. Computes the minimum dimensions and cover thicknesses required to meet the more detailed fire resistance requirements set out in Part 2 of BS8110 (1985). The option is based on information given in Table 81 of the current RCD Handbook.

9.3 SCALE - Reinforced & Prestressed Concrete Design to Eurocode 2, BS5400/DOT

sc122 General reinforced concrete section - design

sc122 Design of a general reinforced concrete section.
Computes the resistance moment and shear capacity at the Ultimate Limit State (ULS) and crack widths and stress values at the Serviceability Limit State (SLS). Design calculations are in accordance with BS 5400-4:1990 as implemented by Standard BD 44/15. Calculations are also in accordance with EN 1991-1-1 and EN 1992-2. The section is divided into a number of cross-sections which may be trapezoidal. The choice of cross sections will reflect the geometry of the section. Tension and compression steel may be included in the section.

sc123 General reinforced concrete section - assessment

sc123 Assessment of general reinforced concrete section.
Computes the resistance moment and shear capacity at the Ultimate Limit State (ULS) and crack widths and stress values at the Serviceability Limit State (SLS). Assessment calculations are in accordance with Departmental Standards BD 44/15 and CS 454.
The section may be divided into a number of cross-sections which may be trapezoidal. The choice of cross—sections will reflect the geometry of the section. Tension and compression steel may be included in the section. EC assessment currently uses BS assessment.

sc124 Early thermal cracking

sc124 Computes the area of distribution reinforcement required for the control of early thermal cracking of reinforced and prestressed post-tensioned concrete members in accordance with Standard BD 28/87 'Early Thermal Cracking in Concrete'. As EN 1992-1-1 and EN 1992-2 do not provide guidance on early thermal cracking the proforma adopts guidance in EN 1992-3 for liquid retaining and containing structures.

sc125 Half joint

sc125 Assessment of reinforced concrete half-joints at the Serviceability Limit State (SLS) in accordance with Advice Note BA 39/93. Computes strains and crack widths. The Advice Note is applicable to upper and lower reinforced concrete half-joints. It may also be applied to pre-tensioned and post-tensioned prestressed half joints which for this purpose can be considered as reinforced concrete elements. For pre-tensioned members, the prestressing force and tendons should be ignored but for post-tensioned members the prestress should be considered as an external force acting on the joint.
EC assessment currently uses BS assessment.

sc126 Torsion in reinforced concrete - design

sc126 Design for torsion in reinforced concrete sections. Computes the values of torsion stresses and considers the requirements for torsion reinforcement at the Ultimate Limit State (ULS) in box, rectangular, T, L and I sections. Design calculations are in accordance with BS5400: Part 4: 1990 as implemented by Standard BD 44/15. Calculations are also in accordance with EN 1992-1-1 Clause 6.3 and "The Concrete Centre" publication entitled "Concise EC2 for Bridges".

sc127 Torsion in reinforced concrete - assessment

sc127 Assessment for torsion in reinforced concrete sections. Computes values of torsion stresses and considers the requirements for torsion reinforcement at the Ultimate Limit State (ULS) in box, rectangular, T, L and I sections. Assessment calculations are in accordance with Appendix A of Standard BD 44/15. EC assessment currently uses BS assessment.

sc128 Temperature effects

sc128 Differential temperature effects in concrete bridge decks. Computes the load effects due to temperature differences in Group 4 deck construction including concrete slabs, concrete deck on concrete beams or box girders. Calculations are in accordance with the composite version of BS 5400-2:2006 and Departmental Standard CS 454 entitled "Assessment of highway bridges and Structures". Calculations are also in accordance with EN 1992-2 and EN 1991-1-5.

sc130 HA loading to BS 5400-2:2006

sc130 Design values for Type HA uniformly distributed lane loading and knife edge loading. Design calculations are in accordance with the composite version of BS 5400: Part 2: 2006 entitled "Steel, concrete and composite bridges". Calculations are also in accordance with EN 1991-2 and NA to EN 1991-2 entitled "EC1: Actions on structures Part 2: Traffic loads on bridges".

sc131 HA loading assessment to CS 454

sc131 Assessment values for Type HA uniformly distributed lane loading, knife edge loading and Load Reduction Factors K. Assessment calculations are in accordance with Standard CS 454 "Assessment of Highway Bridges & Structures". For EC assessment use BS assessment.

sc132 ULS flexure in prestressed members - design

sc132 Design for flexure in a prestressed concrete section. Computes the resistance moment at the Ultimate Limit State (ULS) in accordance with BS5400: Part 4: 1990 as implemented by Standard BD 44/15. Two methods of analysis are available. Method 1 - the resistance moment of rectangular beams and slabs and flanged beams using the approximate method from Equation 27 of BS 5400: Part 4: 1990. Method 2 - the rigorous approach described in the first part of Cl. 6.3.3.1 of BS 5400: Part 4: 1990 using the stress-strain curve for concrete in Figure 1. Calculations are also in accordance with BS EN 1991-1-1 and BS EN 1992-2.

sc133 ULS flexure in prestressed members - assessment

sc133 Assessment for flexure in a prestressed concrete section. Computes the resistance moment at the Ultimate Limit State (ULS) in accordance with Appendix A of Standard BD 44/15 and reference is made to BS 5400-4:1990 as implemented by Standards BD 44/15 and CS 454. Two methods are available. Method 1 - the resistance moment of rectangular beams and slabs and flanged beams using the approx method from Eq. 27 of Appendix A of Standard BD 44/15. Method 2 - the rigorous approach described in Cl. 6.3.3.1 of Appendix A of Standard BD 44/15 using the stress-strain curve for concrete in Fig 1. EC assessment currently uses BS assessment.

sc134 ULS shear in prestressed members - design

sc134 Design for shear in a prestressed concrete section. Computes the shear capacity for Class 1 and Class 2 members at the Ultimate Limit State (ULS) in accordance with BS5400: Part 4: 1990 as implemented by Standard BD 44/15. Calculations for Class 3 members are also included. Calculations are also in accordance with EN 1992-1-1.

sc135 ULS shear in prestressed members - assessment

sc135 Assessment for shear in a prestressed concrete section. Computes the shear capacity at the Ultimate Limit State (ULS) in accordance with Appendix A of Standard BD 44/15 and reference is made to BS 5400: Part 4: 1990 as implemented by Departmental Standards BD 44/15 and CS 454. EC assessment currently uses BS assessment.

sc114 TY beam with in-situ infill

sc114 Precast prestressed concrete TY beam with infill concrete. Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc115 TY beam with in-situ top slab

sc115 Precast prestressed concrete TY beam with concrete top slab. Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc116 TYE beam with in-situ top slab

sc116 Precast prestressed concrete TYE beam with concrete top slab. Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc165 TYE beam with in-situ infill

sc165 Precast prestressed concrete TYE beam with infill concrete. Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc138 SY beams with insitu top slab

sc138 Precast prestressed concrete SY beam with concrete top slab. Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc139 YE beams with insitu top slab

sc139 Precast prestressed concrete YE beam with concrete top slab. Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc140 Inverted T beam

sc140 Precast prestressed concrete T beam with infill concrete. Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc141 M beam

sc141 Precast prestressed concrete M beam with concrete top slab. Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc142 UM beam

sc142 Precast prestressed concrete UM beam with concrete top slab. Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc143 Y beam

sc143 Precast prestressed concrete Y beam with concrete top slab. Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc144 U beam

sc144 Precast prestressed concrete U beam with concrete top slab. Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc145 Wide box beam

sc145 Precast prestressed concrete wide box beam. Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span.

sc146 Box beam

sc146 Precast prestressed concrete box beam.
Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc148 General precast prestressed bridge beam

sc148 Non standard precast prestressed concrete bridge beam.
Computes stresses at the Serviceability Limit State (SLS) to BS5400: Part 4: 1990 as implemented by Standard BD 44/15 for a single span simply supported member. Results are produced at cross-sections along the beam defined by a distance from the mid-span. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc149 Concrete mix design

sc149 Design of normal concrete mixes. Deals with the design of normal concrete mixes based on Building research establishment report. See 'Design of normal concrete mixes' published by DOE. Given required characteristic strength, age, margin, proportion defectives, standard deviation, percentage air content and other parameters, designs mix for ordinary portland, sulphate-resisting, rapid-hardening portland cement, using crushed or uncrushed aggregate.

sc151 Rectangular column - section interaction - design

sc151 Design of a Rectangular column subject to axial load and uniaxial bending. The calculation is in accordance with BS 5400: Part 4: 1990 and the limiting bending moment/axial force relationship is produced in diagrammatic and tabular form. The method used is that described in Clause 5.5.3.2 of BS 5400: Part 4: 1990 with the stresses in the concrete in compression derived from the stress-strain curve in Figure 1. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc152 Rectangular column - section interaction - assessment

sc152 Assessment of a Rectangular column subject to axial load and uniaxial bending. The calculation is in accordance with Appendix A of Standard BD 44/15 and the limiting bending moment/axial force relationship is produced in diagrammatic and tabular form. The method used is that described in Clause 5.5.3.2 of Appendix A of BD 44/15 with the stresses in the concrete in compression derived from the stress-strain curve in Figure 1. EC assessment currently uses BS assessment.

sc153 Circular column - section interaction - design

sc153 Design of a Circular column subject to axial load and uniaxial bending. The calculation is in accordance with BS 5400: Part 4: 1990 and the limiting bending moment/axial force relationship is produced in diagrammatic and tabular form. The method used is that described in Clause 5.5.3.2 of BS 5400: Part 4: 1990 with the stresses in the concrete in compression derived from the stress-strain curve in Figure 1. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc154 Circular column - section interaction - assessment

sc154 Assessment of a Circular column subject to axial load and uniaxial bending. The calculation is in accordance with Appendix A of Standard BD 44/15 and the limiting bending moment/axial force relationship is produced in diagrammatic and tabular form. The method used is that described in Clause 5.5.3.2 of Appendix A of BD 44/15 with the stresses in the concrete in compression derived from the stress-strain curve in Figure 1. EC assessment currently uses BS assessment.

sc155 General column - section interaction - design

sc155 Design of a General column section subject to axial load and uniaxial bending. The calculation is in accordance with BS 5400: Part 4: 1990 and the limiting bending moment/axial force relationship is produced in diagrammatic and tabular form. The method used is that described in Clause 5.5.3.2 of BS 5400: Part 4: 1990 with the stresses in the concrete in compression derived from the stress-strain curve in Figure 1. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc156 General column - section interaction - assessment

sc156 Assessment of a General column section subject to axial load and uniaxial bending. The calculation is in accordance with BS 5400: Part 4: 1990 and the limiting bending moment/axial force relationship is produced in diagrammatic and tabular form. The method used is that described in Clause 5.5.3.2 of BS 5400: Part 4: 1990 with the stresses in the concrete in compression derived from the stress-strain curve in Figure 1. EC assessment currently uses BS assessment.

sc157 Rectangular column - biaxial bending - design

sc157 Design of a Rectangular column subject to axial load and biaxial bending. The calculation is in accordance with BS 5400: Part 4: 1990 and the limiting bending moment/axial force relationship is produced in diagrammatic and tabular form. The method used is that described in Clause 5.5.3.2 of BS 5400: Part 4: 1990 with the stresses in the concrete in compression derived from the stress-strain curve in Figure 1. Calculations are also in accordance with BS EN 1992-2 and NA to BS EN 1992-2.

sc158 Rectangular column - biaxial bending - assessment

sc158 Assessment of a Rectangular column subject to axial load and biaxial bending. The calculation is in accordance with Appendix A of Standard BD 44/15 and the limiting bending moment/axial force relationship is produced in diagrammatic and tabular form. The method used is that described in Clause 5.5.3.2 of Appendix A of BD 44/15 with the stresses in the concrete in compression derived from the stress-strain curve in Figure 1. EC assessment currently uses BS assessment.

9.4 SCALE - Reinforced Concrete to Eurocode 2, BS8007, BS8666, Section Analysis

sc160 Wall/slab section

sc160 Rectangular wall/slab section to BS8007 & BS8110 or EC2. This option checks whether a given section subjected mainly to bending (but possibly to some axial force as well), meets both the ultimate limit-state requirements of BS8110 or EC2 and the cracking limit-state and temperature-cracking requirements of BS8007 or EC2 Part 3. To control restrained shrinkage and thermal movement cracking, tension and compression reinforcement must be provided as near to the surfaces of the section as is consistent with concrete cover requirements. In order to ascertain equilibrium conditions due to service load, an iteration procedure is used.

sc162 Beam section

sc162 Rectangular beam section to BS8007 & BS8110 or EC2. This option checks whether a given section subjected mainly to bending (but possibly to some axial force as well), meets both the ultimate limit-state requirements of BS8110 or EC2 and the cracking limit-state and temperature-cracking requirements of BS8007 or EC2 Part 3. To control restrained shrinkage and thermal movement cracking, tension and compression reinforcement must be provided as near to the surfaces of the section as is consistent with concrete cover requirements. In order to ascertain equilibrium conditions due to service load, an iteration procedure is used.

sc164 Two way spanning wall slab/plate with hydrostatic pressure

sc164 Two way spanning wall panel with hydrostatic pressure. Based on PCA 'Rectangular Concrete Tanks', this option computes edge and span bending moments and shears for panels with various edge conditions and sizes subjected to hydrostatic pressure from a fluid of any density.

sc189 Designated concrete to BS8500-1 conforms with BS EN 206-1

sc189 Designated and Designed concrete to BS8500-1 conforms to BS EN 206-1. The following options are offered:

- buried concrete
- general applications e.g. kerb bedding etc.
- ground bearing floors
- non-buried concrete.

sc190 Bar scheduling to BS8666:2005

sc190 Bar bending schedule. This option produces a bar schedule for a maximum of 200 barmarks. The schedule takes the form of a table having headings given below.

This schedule complies with the requirements of BS8666:2005.

Member	Bar mark	Type and size	No. of mbr	No. of bar	Total no.	Lngth ofbar (mm)	Shape code	A (mm)	B (mm)	C (mm)	D (mm)	E/R (mm)	Rev ltr

sc191 (Withdrawn) Bar scheduling to BS4466

sc191 Bar bending schedule. This option produces a bar schedule for a maximum of 200 barmarks. The schedule takes the form of a table having headings given below.

This schedule complies with the requirements of BS4466: 1989.

Member	Bar mark	Type and size	No. of mbr	No. of bar	Total no.	Length of bar (mm)	Shape code	A (mm)	B (mm)	C (mm)	D (mm)	E/R (mm)

sc192 Properties of transformed sections

sc192 Properties of transformed sections. This option calculates the sectional properties of a rectangular or a flanged reinforced concrete section using the formulae set out in Table 136 of the 10th Edition of the 'Reinforced Concrete Designer's Handbook by Reynolds and Steedman. The properties of a transformed reinforced concrete section are determined on the assumption that the contribution of the tension reinforcement is equal to the area of steel multiplied by the modular ratio, and that of the tension steel equals the area of the reinforcement times (modular ratio - 1). The concrete is assumed to be ineffective in tension throughout

sc193 Biaxial bending of rectangular section

sc193 Elastic biaxial bending of a rectangular reinforced concrete section. Computes the concrete and reinforcement stresses in a short rectangular reinforced concrete section subjected to any combination of bending moments and axial load. All materials are assumed to behave elastically in compression, but only the steel reinforcement can carry tension. Stability, torsion and shear are not considered. The calculations are not specific to any Code or Standard.

sc194 Biaxial bending of circular section

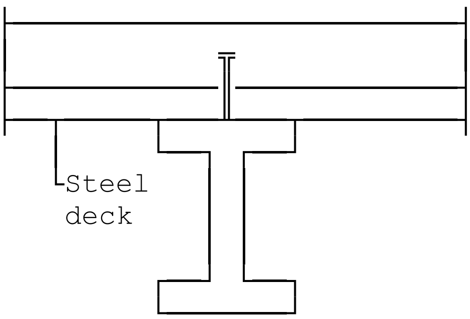
sc194 Elastic biaxial bending of a circular reinforced concrete section. Computes the concrete and reinforcement stresses in a short circular reinforced concrete section subjected to any combination of bending moments and axial load. All materials are assumed to behave elastically in compression, but only the steel reinforcement can carry tension. Stability, torsion and shear are not considered. The calculations are not specific to any Code or Standard.

sc195 Biaxial bending of user defined section

sc195 Elastic biaxial bending of a user defined reinforced concrete section. Computes the concrete and reinforcement stresses in a short user defined reinforced concrete section subjected to any combination of bending moments and axial load. All materials are assumed to behave elastically in compression, but only the steel reinforcement can carry tension. Stability, torsion and shear are not considered. The calculations are not specific to any Code or Standard.

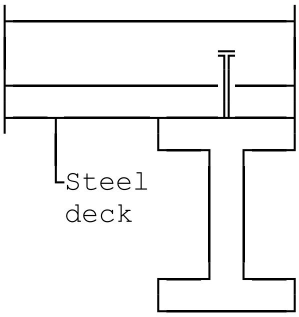
9.5 SCALE - Composite Construction to Eurocode 4, BS5950 and BS5400

sc210 Internal beam with UDL to BS5950 and EC4



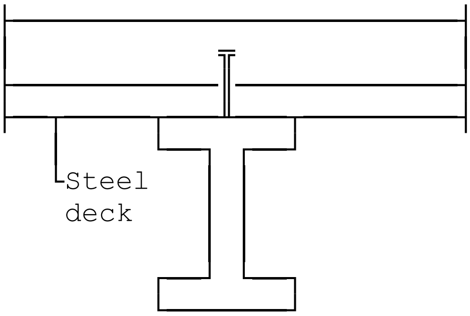
sc210 Composite steel deck floor internal beam with U.D. loading. Calculations in accordance with BS 5950 Part 3 Section 3.1 'Code of practice for design of simple and continuous composite beams'- July 1990 and with 'BS 5950-1: 2000' for construction stage design. Calculations are also in accordance with EC4 Part 1-1.

sc212 Edge beam with UDL to BS5950 and EC4



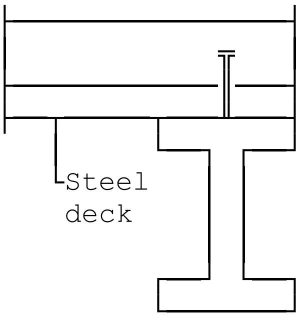
sc212 Composite steel deck floor edge beam with U.D. loading. Calculations in accordance with BS 5950 Part 3 Section 3.1 'Code of practice for design of simple and continuous composite beams'- July 1990 and with 'BS 5950-1: 2000' for construction stage design. Calculations are also in accordance with EC4 Part 1-1.

sc214 Internal beam with secondary beams to BS5950 and EC4



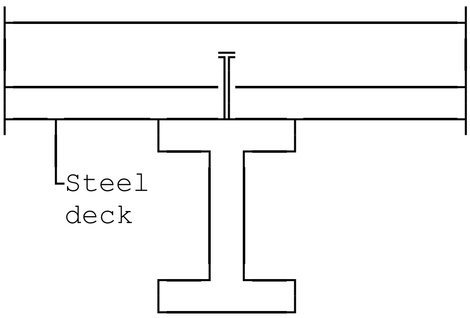
sc214 Composite steel deck floor internal beam with secondary beams. Calculations in accordance with BS 5950 Part 3 Section 3.1 'Code of practice for design of simple and continuous composite beams'- July 1990 and with 'BS 5950-1 2000' for construction stage design. Calculations are also in accordance with EC4 Part 1-1.

sc216 Edge beam with secondary beam to BS5950 and EC4



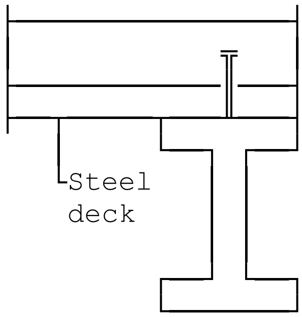
sc216 Composite steel deck floor edge beam with secondary beams. Calculations in accordance with BS 5950 Part 3 Section 3.1 'Code of practice for design of simple and continuous composite beams'- July 1990 and with 'BS 5950-1: 2000' for construction stage design. Calculations are also in accordance with EC4 Part 1-1.

sc218 Internal beam with secondary beams at 1/3 pts to BS5950 & EC4



sc218 Composite steel deck floor internal beam with secondary beams at third points. Calculations to BS 5950 Part 3 Section 3.1 'Code of practice for design of simple and continuous composite beams'- July 1990 and with 'BS 5950-1: 2000' for construction stage design. Calculations are also in accordance with EC4 Part 1-1.

sc220 Edge beam with secondary beams at third points to BS5950 & EC4

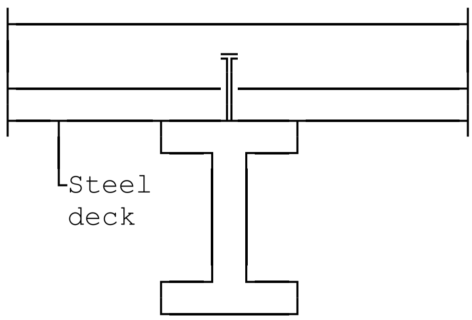


sc220 Composite steel deck floor edge beam with secondary beams at third points. Calculations to BS 5950 Part 3 Section 3.1 'Code of practice for design of simple and continuous composite beams'- July 1990 and with 'BS 5950-1: 2000' for construction stage design. Calculations are also in accordance with EC4 Part 1-1.

sc221 Lateral restraint from steel decking (construction phase) EC4

sc221 Lateral restraint from steel decking (construction phase). The calculations are in accordance with EC3 Parts 1-1, 1-3 and SCI P360 entitled 'Stability of Steel Beams and Columns'. The objective is to investigate whether during the construction phase (prior to concrete hardening), the steel decking offers sufficient lateral restraint to the beams such that the beams may be considered as fully restraint against lateral torsional buckling.

sc222 Section properties to BS5950 and EC4



sc222 Composite steel deck floor - section properties at serviceability and ultimate limit states. Calcs to BS 5950 Part 3 Section 3.1 'Code of practice for design of simple and continuous composite beams'- July 1990 and with 'BS 5950-1: 2000' for construction stage design. Calculations are also in accordance with EC4 Part 1-1.

sc224 Shear connector design to BS5950 and EC4

sc224 Composite beam shear stud design. Deals with the requirements for transferring shear forces between the concrete and steel flanges in a simply supported composite beam of steel section: UB, UC, RSJ, Channel or RHS. Calculations are in accordance with BS5950 Part 3 Section 3.1 'Code of practice for design of simple and continuous composite beams' and also in accordance with EC4 Part 1-1.

sc225 Temperature effects in composite bridge decks - BS5400 & D.Tp.

sc225 Differential temperature effects in composite bridge decks. Produces calculations for the load effects due to temperature differences in concrete deck on steel box, truss or plate girders. Based on Departmental Standard CS 454, see also "Composite Universal Beam Simply Supported Span" by G.F.J.Nash. Calculations are also in accordance with BS EN 1994-2 and BS EN 1991-1-5.

sc226 Differential shrinkage in composite bridge decks - BS5400 & D.Tp.

sc226 Shrinkage modified by creep in composite bridge decks. Produces calculations for shrinkage effects modified by creep in a concrete deck on steel box, truss or plate girders. Based on BS 5400-5:1979 and Standard BD 16/82 and Amendment No.1. See also Sheet 11 of 'Design Guide for Simply Supported Composite Bridges' by D.C.Iles. Calculations are also in accordance with EN 1994-2 and EN 1992-1-1.

9.6 SCALE - Timber Design to Eurocode 5 and BS5268

sc250# Rectangular section with axial load & bending

sc250 Rectangular timber member subject to axial load and bending. Computes axial, bending and shear stresses and substitutes in interaction equation in accordance with BS 5268-2:2002 or BS EN 1995-1-1:2004 (Eurocode 5).

sc251# Rectangular Glulam column/tie with axial load

sc251 Rectangular Glulam member with horizontal laminations subject to axial load and bending. Computes axial, bending and shear stresses and substitutes in interaction equation in accordance with BS 5268-2:2002 or BS EN 1995-1-1:2004 (Eurocode 5).

sc252 Domestic floor joist

sc252 Domestic floor joist. Calculations follow the domestic floor joists example by V C Johnson in TRADA Design Aid DA1 in accordance with BS 5268-2:2002 or BS EN 1995-1-1:2004 (Eurocode 5).

sc253 Biaxial bending of timber section

sc253 Biaxial bending of timber section to BS 5268-2:2002 or BS EN 1995-1-1:2004 (Eurocode 5). This option checks the adequacy of a timber section resisting bending about the major and minor axes. Such a situation is common in traditional roof construction where canted purlins support rafters and no collar or ceiling ties exist. If a mid height collar tie or a ceiling tie is present with a symmetrical dual pitch roof then biaxial bending of the purlin does not occur. For further discussion on this subject refer to "Structural timber design and technology" by C.J. Mettem (TRADA Publication).

sc254 Simply supported beam with general loading

sc254 Simply supported timber beam with udl & point loads. Computes applied axial, bending and shear stresses and deflections for a simply supported timber beam and solves the interaction equation in accordance with BS 5268-2:2002 or BS EN 1995-1-1:2004 (Eurocode 5).

sc255 Simply supported Glulam beam with general loading

sc255 Simply supported Glulam beam with udl & point loads. Computes applied axial, bending and shear stresses and deflections for a simply supported Glulam beam and solves the interaction equation in accordance with BS 5268-2:2002 or BS EN 1995-1-1:2004 (Eurocode 5).

sc256 Simply supported flitched beam with general loading

sc256 Flitched timber beam with udl & point loads. This option is for the design of the following types of flitched beams in accordance with BS5268-2:2002 and BS449 1990 or BS EN 1995-1-1:2004 and BS EN 1993-1-8:2005 (Eurocodes 5 and 3).

- steel plates to top & bottom faces of timber
- one steel web plate with a timber member on each side
- steel plates to each side face of timber section
- two steel web plates and three timber members.

sc257 Spaced timber column

sc257 Spaced rectangular section with axial load and bending. This option is for the design to BS 5268: Part 2: 2002 or BS EN 1995-1-1:2004 of a rectangular timber composed of two or more equal shafts spaced apart by end and intermediate packing pieces. Member is subject to axial load and bending.

sc258 Ply web and box beam

sc258 Ply box beams with udl & point loads. This option is for the design of ply web and box beams to BS5268-2:2002 or EC5. Ply webs may be fixed to top and bottom chords of box beams using:

- webs glued to timber chords with adhesion by nails or mechanical clamps - all vertical splices in webs to be bridged by ply splice plates
- webs nailed to timber chords - vertical splices to be staggered each side of the beam.

sc259 Stressed skin timber floor joist

sc259 Stressed skin joists. This deals with the design of a stressed skin floor joist to BS5268-2:2002 or EC5. Both uniformly distributed and point loads can be applied to the floor and the types of floor considered are:

- glued stressed skin floor with top and bottom ply sheathing
- glued stressed skin floor joist with top ply sheathing only.

sc260 Howe truss

sc260 Howe Truss analysis. This option uses coefficients from the Timber Designers Manual by Ozelton & Baird to compute forces in the members of a Howe Truss (N girder with end diagonals in compression). Trusses may be 4, 6, or 8 panel. Loading may consist of rafter node point loading and/or ceiling node point loading. The central deflection can be calculated based on applied loading.

sc262 Pratt truss

sc262 Pratt Truss analysis. This option uses coefficients from the Timber Designers Manual by Ozelton & Baird to compute forces in the members of a Pratt Truss (N girder with end diagonals in tension). Trusses may be 4, 6, or 8 panel. Loading may consist of rafter node point loading and/or ceiling node point loading. The central deflection can be calculated based on applied loading.

sc266 Fink truss

sc266 Fink Truss analysis. This option uses coefficients from the Timber Designers Manual by Ozelton & Baird to compute forces in the members of a Fink Truss. Trusses may be 4 or 8 panel. For 4 panel trusses loading may consist of:

- rafter point loading and/or ceiling node point loading — truss joist is assumed to be 3 equal bays
- rafter point loading only — truss end bays are equal. (This condition also applies to 8 panel trusses.)

The central deflection can be calculated based on applied loading.

sc267 Racking resistance of timber stud panels

sc267 Racking resistance of timber stud panels to BS5268-2:2002 or EC5. This deals with the design racking resistance of timber framed wall panels. Racking calculations of timber framed walls are made to establish the resistance of buildings to wind loading. The recommendations of allowable racking load are based on either ply or OSB sheathing fixed to stud framework by nailing.

sc268 Fire design of timber beam

sc268 Fire design of timber beam with udl & point loads. This deals with the structural design of a timber member for fire resistance in accordance with BS5268-4 or EC5. Fire resistance of a member is the period of time which the member is required to support the design load without failure whilst subjected to the fire. The method assumes that a depth of charring occurs around the exposed face of the section, and the corners of the beam become rounded. The residual section is checked for stability strength and for beams, deflection is limited to Span/20.

sc269 Residential floor vibration check to EC5

sc269 Floor vibrations to EC5. Deals with floor vibrations which need to be controlled to an acceptable level and provides a design method for dwellings. The following floor systems are offered for selection:

- solid timber
- glued thin webbed joists
- mechanically jointed floor trusses
- glulam beams
- stressed skin timber floor joist
- user defined bending stiffness.

sc271 Nailed joint

sc271 Nailed timber joint connection. This deals with the design of a nailed timber joint to BS 5268-2:2002 or BS EN 1995-1-1:2004 for joints of the following types:

- timber-to-timber joint without pre-drilled holes
- timber-to-timber joint with pre-drilled holes
- steel plate-to-timber joint without pre-drilled holes
- plywood-to-timber joint without pre-drilled holes
- particleboard-to-timber joint without pre-drilled holes.

sc272 Screwed joint

sc272 Screwed timber joint connection. This deals with the design of a screwed timber joint to BS 5268-2:2002 or BS EN 1995-1-1:2004 for joints of the following types:

- timber-to-timber joint
- steel plate-to-timber joint
- plywood-to-timber joint
- timber joint with screws in withdrawal.

sc273 Bolted joint

sc273 Bolted timber joint connection. This deals with the design of a bolted timber joint to BS 5268-2:2002 or BS EN 1995-1-1:2004 for joints of the following types:

- timber-to-timber joint
- steel plate-to-timber joint.

sc274 Spliced timber joint

sc274 Bolted timber connection with shear load to BS 5268-2:2002 or BS EN 1995-1-1:2004. The following joint types are considered:

- two bolts along length or across width overlapped timber
- three to five bolts along length or across width overlapped timber
- four or five bolt overlapped timber
- two bolts along length or across width symmetrical joint with inline splice plate
- three bolts along length or across width symmetrical joint with inline splice plate
- four or five bolt symmetrical joint with inline splice plate.

sc275 Toothed plate connector

sc275 Tooth plate timber connection. Deals with the design of tooth-plate timber connections to BS 5268-2:2002 or EC5 with connectors to BS 1579 or BS EN 912:2000. Connectors may be round or square and double or single sided.

sc280 Rafter or sloping beam

sc280 Rafter or sloping beam. Computes axial loads, bending moments and shear forces for a rafter pinned or on rollers at its lower level with a horizontal prop at its higher level subjected to various loads.

sc284 Couple roof

sc284 Couple roof. Computes axial loads, bending moments shear forces, deflections and support reactions for a couple roof (two rafters pinned at the apex and at the supports) subjected to various loads.

sc286 Collar-tie roof

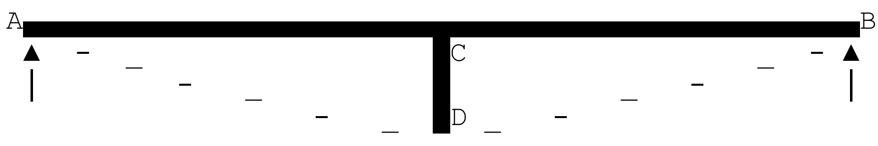
sc286 Collar tie roof. Computes axial loads, bending moments, shear forces, deflections and support reactions for a couple roof (two rafters pinned at the apex with a tie forming an A frame, one support pinned the other on rollers) subjected to various loads.

sc288 Lean-to roof

sc288 Lean-to. Computes axial loads, bending moments, shear forces, deflections and support reactions for a lean-to pinned at foot, eaves and apex.

sc290 Trussed beam

sc290 Trussed beam. Computes axial loads, bending moments, shear forces, and stresses for a trussed beam in accordance with 'Theory of Structures' by A Morley, published Longmans. In the trussed beam shown below AB is a continuous timber beam; CD is a strut braced to the beam ends A and B by tie-rods AD and DB.



sc292 Braced sheds (for attic and other frames, see NL-STRESS below)

sc292 Braced sheds and braced portals. Computes axial loads, bending moments, shear forces, and reactions for braced sheds and braced portals in accordance with 'Theory of Structures' by A Morley, published Longmans.

9.7 SCALE - Steel Design to Eurocode 3 and BS5400/DOT

sc356 List of proformas

sc356 Lists the options available in SCALE for BS5400: Part 3: 2000 and DTp Departmental Standard BD 13/06. This option also lists all bridge proformas available to EN 1993-2 and highlights the relevant Clauses, Tables ans/or other BS EN Standards.

sc357 Design objectives and partial safety factors

sc357 Design objectives and partial safety factors. This option is in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges" and describes the design objectives of Part 3 and the partial safety factors to be used. This option is also in accordance with BS EN 1993-2 and NA to BS EN 1993-2.

sc358 Steel grades to BS EN 10 025 & BS4360

sc358 Steel grades. This option lists the steel grades available in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with BS EN 1993-2 and NA to BS EN 1993-2.

sc359 Nominal yield stress

sc359 Nominal yield stresses. This option produces a table of nominal yield stresses in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with BS EN 1993-2 and NA to BS EN 1993-2.

sc360 Notch toughness

sc360 Notch toughness. This option checks for compliance with notch toughness requirements in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". It does not include calculations for stress concentrations in accordance with Clause 6.5.6. This option is also in accordance with EN 1993-1-10 and NA to EN 1993-1-10.

sc361 Stress analysis; allowance for shear lag

sc361 Stress analysis - allowance for shear lag. This option calculates the effective breadth ratio in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with BS EN 1993-2 and NA to BS EN 1993-2.

sc362 Shape limitations for flanges

sc362 Shape limitations for flanges - check for compact section. This option checks flanges for compliance with the shape limitations of Clause 9.3.2 and compact section to Clause 9.3.7.3.1. to be in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with BS EN 1993-2 and NA to BS EN 1993-2.

sc363 Openings in webs and compression flanges

sc363 Openings in webs and compression flanges. This option outlines the requirements from Clause 9.3.3. in accordance with BS 5400: Part 3: 1982 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with EN 1993-2, NA to EN 1993-2 and SCI publication P355.

sc364 Shape limitations for flat stiffeners

sc364 Shape limitations for flat stiffeners. This option checks for compliance with the shape limitations of Clause 9.3.4.1.2 in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with BS EN 1993-1-5 and NA to BS EN 1993-1-5.

sc365 Shape limitations for bulb flat stiffeners

sc365 Shape limitations for bulb flat stiffeners. This option checks for compliance with the shape limitations of Clause 9.3.4.1.3 in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". The EC design option states that the rules of EN 1993-1-5 are considered to be consrvative and bulb flat stiffeners are unlikely to comply. The user is referred to alternative stiffener options instead.

sc366 Shape limitations for angle stiffeners

sc366 Shape limitations for angle stiffeners. This option checks for compliance with the shape limitations of Clause 9.3.4.1.4 in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with BS EN 1993-1-5 and NA to BS EN 1993-1-5.

sc367 Shape limitations for tee stiffeners

sc367 Shape limitations for tee stiffeners. This option checks for compliance with the shape limitations of Clause 9.3.4.1.5 in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with BS EN 1993-1-5 and NA to BS EN 1993-1-5.

sc368 Shape limitations for closed stiffeners

sc368 Shape limitations for closed stiffeners. This option checks for compliance with the shape limitations of Clause 9.3.4.2. in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with EN 1993-1-5 and NA to EN 1993-1-5.

sc369 Shape limitations for flanges curved in elevation

sc369 Shape limitations for flanges curved in elevation. This option checks for compliance with the shape limitations of Clause 9.3.5. in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with BS EN 1993-1-5 and NA to BS EN 1993-1-5.

sc370 Shape limitations for circular hollow sections

sc370 Shape limitations for circular hollow sections. This option checks for compliance with the shape limitations of Clause 9.3.6. in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with EN 1993-2 and NA to EN 1993-2.

sc371 Effective section

sc371 Effective section. This option calculates the effective section to Clause 9.4. in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with BS EN 1993-2 and NA to BS EN 1993-2.

sc372 Evaluation of stresses

sc372 Evaluation of stresses. This option covers the evaluation of stresses to Clause 9.5. in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with BS EN 1993-2 and NA to BS EN 1993-2.

sc373 Effective length for lateral torsional buckling

sc373 Effective length for lateral torsional buckling. This option calculates the effective length for lateral torsional buckling to Clause 9.6. in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with BS EN 1993-2 and NA to BS EN 1993-2.

sc374 Slenderness

sc374 Slenderness. This option calculates the slenderness parameter lambdaLT to Clause 9.7 in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with BS EN 1993-2 and NA to BS EN 1993-2.

sc381 Bearing stiffeners at end support

sc381 Bearing stiffeners at end support in accordance with BS EN 1993-2:2006 "EC3 Design of Steel Structures - Part 2: Steel bridges", the NA to BS EN 1993-2:2006 and BS EN 1993-1-5 section 9.

sc382 Transverse web stiffeners other than at supports

sc382 Transverse web stiffeners other than at supports. Deals with transverse web stiffeners other than at supports for webs without longitudinal stiffeners to Clause 9.13 in accordance with BS 5400: Part 3: 2000 and Departmental Standard BD 13/06 "Design of Steel Bridges". This option is also in accordance with BS EN 1993-2, NA to EN 1993-2 and BS EN 1993-1-5 Section 9.

9.8 SCALE - Steel Design to Eurocode 3 and BS5950-1:2000

sc339 Structural robustness of steel framed buildings

sc339 Structural robustness of steel framed buildings. The building is assumed to be a hot-rolled steel framed building. The Eurocode strategies for structural robustness and designing for the avoidance of disproportionate collapse as required by the UK Building Regulations will be considered by this proforma.

sc340 Bolts in tension and shear

sc340 Bolts in tension and shear. Deals with the design of bolts in tension and shear following the treatment in 'Manual on Connections for Beam and Column Construction, Conforming with the requirements of BS 449 Part 2:1969', published by BCSA.

sc350 Steel guardrailing & Stainless Steel CHS rail/balustrade design

sc350 Steel guardrailing & SS rail/balustrade design. Deals with the design of guardrailing to limit to permissible values the following: the bending stress in the rail; the horizontal deflection of the rail between posts; the bending stress in the post; the lateral deflection of the vertical post. Loading to BS 6399:Part 1:1984:Table 4.

sc351 Balluster base plate - either concentric or not with baseplate

sc351 Balustrading Design to BS5950 or BS EN 1993-1-1:2005.
Two options are available for the base plate:

- baluster base plate concentric with baluster
- baluster base plate non-concentric with baluster.

sc355 Steel runway beam

sc355 Steel runway beam. Deals with the design of UBs, UCs and RSJs used as runway beams in accordance with BS 2853:1957 and BS449. The method of support affects the way in which the longitudinal bending stresses and the transverse bending stresses in the bottom flange of the beam are checked for acting in combination. Three cases of support to the ends of runway beams are considered viz: simply supported on padstones or built into walls; cantilevered; simply supported by means of bolt hangers. Runway beam calculations are also in accordance with BS EN 1993-6:2007 and BS EN 1993-1-1:2005.

sc384 Stainless steel circular hollow section design

sc384 Stainless steel circular hollow section design. Deals with the design of stainless steel CHS sections in accordance with BS5950-1:2000 and SCI publication "Structural Design of Stainless Steel". Calculations are also in accordance with SCI publication entitled "Design Manual For Structural Stainless Steel" 3rd Edition dated 18th April 2006.

sc385 Stainless steel hollow section design

sc385 Stainless steel rectangular hollow section design. Deals with the design of stainless steel RHS sections in accordance with BS5950-1:2000 and SCI publication "Structural Design of Stainless Steel". Calculations are also in accordance with SCI publication entitled "Design Manual For Structural Stainless Steel" 3rd Edition dated 18th April 2006.

sc386 Simply supported stainless steel beam (CHS, SHS, RHS, Channel)

sc386 Simply supported stainless steel beam. Deals with the design of simply supported stainless steel beams in accordance with BS5950-1:2000 and SCI publication "Structural Design of Stainless Steel". Calculations are also in accordance with SCI publication entitled "Design Manual For Structural Stainless Steel" 3rd Edition dated 18th April 2006.

sc387 Axially loaded stainless steel column (CHS, SHS, RHS, Channel)

sc387 Axially loaded stainless steel column. Deals with the design of axially loaded stainless steel columns in accordance with BS5950-1:2000 and SCI publication "Structural Design of Stainless Steel". Calculations are also in accordance with SCI publication entitled "Design Manual For Structural Stainless Steel" 3rd Edition dated 18th April 2006.

sc388 Axially loaded stainless steel angle

sc388 Axially loaded stainless steel angle. Deals with the design of axially loaded stainless steel angles in accordance with BS5950-1:2000 and SCI publication "Structural Design of Stainless Steel". Calculations are also in accordance with SCI publication entitled "Design Manual For Structural Stainless Steel" 3rd Edition dated 18th April 2006.

sc390# Concrete filled square and rectangular hollow section column

sc390 Concrete filled SHS columns. Deals with the design of concrete filled square and rectangular hollow section columns based on "British Steel Tubes Division Booklet TD 296 - SHS Design Manual for Concrete Filled Columns". Calculations are also in accordance with the "Design Guide for Concrete Filled SHS Columns" by Y.C.Wang published in 2009, EC3 Part 1-1 and EC4 Part 1-1.

sc391# Concrete filled circular hollow section column

sc391 Concrete filled CHS columns. Deals with the design of concrete filled circular hollow section columns based on "British Steel Tubes Division Booklet TD 296 - SHS Design Manual for Concrete Filled Columns". Calculations are also in accordance with the "Design Guide for Concrete Filled SHS Columns" by Y.C.Wang published in 2009, EC3 Part 1-1 and EC4 Part 1-1.

sc392 Fire design of concrete filled structural hollow section column

sc392 Concrete filled SHS columns - fire design. Deals with the design for fire of concrete filled structural hollow section columns based on "British Steel Tubes Division Booklet TD 296 - SHS Design Manual for Concrete Filled Columns". Calculations are also in accordance with the "Design Guide for Concrete Filled SHS Columns" by Y.C.Wang published in 2009 , EC3 Part 1-1 and EC4 Part 1-1.

sc396 Moment connection between RHS column and column base

sc396 Moment Connections - Column Base RHS Sections. Deals with the design of unstiffened slab bases for the support of square and rectangular hollow section columns subjected to axial compressive load and bending moment about the major axis, based on BS5950-1:2000 and "Joints in Steel Construction - Moment Connections" published by The Steel Construction Institute. Calculations are also in accordance with EC3 Part 1-1, EC3 Part 1-8 and publication entitled "Joints in Steel Construction: Simple Joints to EC3" by SCI.

sc407 Simple design, builder's arithmetic, general check, safe loads

sc407 Simple design. This option carries out 3 functions:
(1) Application of builders' arithmetic to the sizing of structural steel sections. It is intended as a guide to sizing only, it does not carry out all the necessary checks demanded by BS5950-1:2000
(2) A general check on any form of steel section
(3) Produces safe load tables for any UB, UC or RSJ subject to:
axially loading when acting as a stanchion
or lateral torsional buckling when acting as a beam.
Calculations are also in accordance with BS EN 1993-1-1:2005.

sc408 SS beam (UB, UC, Joist, or Channel) general loading

sc408 Simply supported UB, UC, RSJ, or Channel. Deals with the design of rolled steel sections to BS5950: Part 1: 2000 and follows the procedure used in Example 2 in the 'Steelwork Design Guide'. Calculations are also in accordance with BS EN 1993-1-1:2005.

sc409 UB, UC or RSJ with bottom flange plate

sc409 Simply supported UB, UC, or RSJ strengthened by flange plate. Deals with the design of rolled steel sections strengthened by a bottom flange plate to BS5950-1:2000 and follows the procedure used in Example 2 in the 'Steelwork Design Guide'. Calculations are also in accordance with BS EN 1993-1-1:2005.

sc410# I beam section design

sc410 I section beam design. Assumes the member being considered is part of a continuous structure and design moments & forces have been calculated prior to use. The design aspects covered are: cross-section classification; cross-sectional resistances; lateral torsional buckling resistance between supports or intermediate restraints and interaction checks under combined bending and axial load. Calculations are in accordance with BS5950-1:2000 and Eurocode 3 and the relevant SCI publications. Calculations are also in accordance with BS EN 1993-1-1:2005.

sc411# Portal frame rafter design

sc411 Design of portal frame rafters. Deals with the design of restrained members or portions of members with an unrestrained compression flange between effective torsional restraints to both flanges in accordance with Appendix G of BS5950-1:2000. Provided that the moment capacity of the section is not exceeded this option allows the positioning of the torsional restraints to be considered which influences the buckling capacity and load carrying ability of the section. Calculations are also in accordance with the requirements of BS EN 1993-1-1:2005 Clause 6.3.5 and Annex BB.3.

sc412# RHS beam section design

sc412 RHS section beam design. Assumes the member being considered is part of a continuous structure and design moments & forces have been calculated prior to use. The design aspects covered are: cross-section classification; cross-sectional resistances; lateral torsional buckling resistance between supports or intermediate restraints and interaction checks under combined bending and axial load. Calculations are in accordance with BS5950-1:2000 and Eurocode 3 and the relevant SCI publications.

sc414# CHS beam and column section design

sc414 CHS section beam or column design. Assumes the member being considered is part of a continuous structure and design moments & forces have been calculated prior to use. The design aspects covered are: cross-section classification; cross-sectional resistances; lateral torsional buckling resistance between supports or intermediate restraints and interaction checks under combined bending and axial load. Calculations are in accordance with BS5950-1:2000 and Eurocode 3 and the relevant SCI publications.

sc415 Checking aid for SS UB's with various loadings & restraints

sc415 Checking aid for 80 Universal Beams having various lengths and restraints. Analysis is for a simply supported I-section subjected to various combinations of distributed loads in the Z & Y directions and axial compression in the X direction, for 0,1,3 intermediate restraints i.e. zero restraint, or one central restraint or restraints at quarter points. When the EN 1993-1-1 requirements are more onerous than the plastic analysis requirements of NL-STRESS and vice-versa then the more onerous requirements are satisfied.

sc416 Design of RHS Slimflor edge beam

sc416 Slim Floor Design - RHS Edge Beams. Deals with the design of square and rectangular section edge beams based on BS5950-1:2000 & SCI publication "Design of RHS Slimflor Edge Beams". Calculations are also in accordance with BS EN 1993-1-1:2005 and SCI publication entitled "Design of RHS Slimflor Edge Beams".

sc418 Slim floor design using precast units

sc418 Slim floor design. Deals with the design of slim floors in accordance with BS5950-1:2000 and 'Slim Floor Design & Construction' published by SCI. Calculations are also in accordance with SCI P385 and BS EN 1993-1-1:2005.
Two types of slim floor construction are considered:

- Non-composite beams, the void surrounding the beam is filled to half the depth of the UC with grout which does not provide any support to the compression elements
- Semi-composite beams, the void surrounding the beam is filled with structural concrete which provides additional support to the compression elements.

sc419 Slim floor design using deep decking

sc419 Slim floor construction using deep decking. Deals with the design of slim floors in accordance with BS5950-1:2000 & 'Slim Floor Construction Using Deep decking' published by SCI. Calculations are also in accordance with SCI P385 and BS EN 1993-1-1:2005.
Assumptions:

- no restraint is provided to the compression flange during the construction stage
- in the construction stage the beam is assumed to be torsionally fixed and warping free at the supports
- for the imposed loading condition the beam is assumed to be laterally restrained
- the steel deck is propped by a central line of props.

sc421 Base plate subject to load and moment

sc421 Moment Connections - Column Base. Designs an unstiffened column slab base subjected to axial load and bending moment in accordance with BS5950-1:2000. Calculations are in accordance with 'Joints in Steel Construction Moment Connections' published by The Steel Construction Institute. Calculations are also in accordance with EC3 Parts 1-1, 1-8 and publication entitled "Joints in Steel Construction: Simple Joints to EC3" by SCI.

sc422 Extended bolted end plate connection beam to column

sc422 Bolted End Plate Connection - Beam to Column. Designs an extended end plate subjected to bending moment, shear and axial force. Calculations are in accordance with BS 5950-1:2000 and the BSI publication entitled 'Joints in Steel Construction - Moment Connections'. Calculations are also in accordance with EC3 Parts 1-1, 1-5, 1-8 and follow the procedure in the SCI publication entitled "Joints in Steel Construction: Moment Connections".

sc423 Flush bolted end plate connection beam to column

sc423 Bolted Flush End Plate Connection - Beam to Column. Designs a flush end plate or flush end plate with or without a mini haunch subjected to bending moment, shear and axial force. Calculations are in accordance with BS 5950-1:2000 and the BSI publication entitled 'Joints in Steel Construction - Moment Connections'. Calculations are also in accordance with EC3 Parts 1-1, 1-5 and 1-8.

sc424 Beam splice - Extended end plate connection

sc424 Extended bolted end plate beam splice. Designs a one or two way extended end plate beam splice subjected to bending moment, shear and axial force. Calculations are in accordance with BS 5950-1 and the BSI publication entitled 'Joints in Steel Construction - Moment Connections'. Calculations are also in accordance with EC3 Parts 1-1, 1-5 and 1-8 and follow the procedure in the SCI publication entitled "Joints in Steel Construction: Moment Connections".

sc425 Beam splice - Flush end plate connection

sc425 Flush beam splice. Designs a bolted end plate beam splice one or two-way extended subjected to bending moment, shear and axial force. Calculations are in accordance with BS 5950-1:2000 & BSI publication entitled 'Joints in Steel Construction - Moment Connections'. Calculations are also in accordance with EC3 Parts 1-1, 1-5 and 1-8 and follow the procedure in the SCI publication entitled "Joints in Steel Construction: Moment Connections".

sc426 Welded beam to column connection

sc426 Welded beam to column connection. Designs a welded beam to column connection subjected to bending moment, shear and axial force. Calculations are in accordance with BS 5950-1:2000 & BSI publication entitled 'Joints in Steel Construction - Moment Connections'. Calculations are also in accordance with EC3 Parts 1-1, 1-5 and 1-8 and follow the procedure in the SCI publication entitled "Joints in Steel Construction: Moment Connections".

sc427 Flush beam splice with end plates and single flange plate

sc427 Flush beam splice with end plates and single flange plate. The design is in accordance with BS5950-1:2000 and generally follows the procedure given 'Joints in Steel Construction - Moment Connections' published by The Steel Construction Institute, together with recommendations given by George Mathieson.
Calculations are also in accordance with EC3 Parts 1-1, 1-5 and 1-8 and follow the procedure in the SCI publication entitled "Joints in Steel Construction: Moment Connections".

sc428 Cantilevered beam

sc428 Steel cantilever. Deals with the design of cantilever beams in accordance with BS5950-1:2000 and the 'Steelwork Design Guide' published by the Steel Construction Institute. Calculations are also in accordance with EC3 Part 1-1.

sc429# Cantilevered beam - alternative linking with NL-STRESS

sc429 Cantilevered beam. Deals with the design of a cantilevered beam. Calculations are in accordance with BS5950-1:2000 and 'Steelwork Design Guide' published by the Steel Construction Institute. The values of bending moment, shear force and axial load about to be input should be the FACTORED values resulting from the loads on the structure multiplied by the relevant load factors given in Table 2 of BS5950-1:2000. Calculations are also in accordance with EC3 Part 1-1.

sc431 Beam (UB, UC or Channel) with biaxial bending

sc431 I section beam with biaxial bending to BS5950 and EC3.
The proforma assumes the member under consideration has already been analysed and the design moments and forces are readily available for input into the proforma. The design aspects covered are namely: cross-section classification, cross-sectional resistances, design buckling resistance, lateral torsional buckling and interaction checks under combined bending and axial load.

sc432 Fully restrained SS I beam with UDL and point load/s

sc432 Steel fully restrained simply supported I beam with UDL & point loads. The design aspects covered are: calculation of design values of actions; cross-section classification; cross-sectional resistances and calculation of the vertical deflection. Calculations are in accordance with BS5950-1:2000 and Eurocode 3 & the relevant SCI publications. Calculations are also in accordance with EC3 Part 1-1 and NCCI sheets provided by Access Steel.

sc434 SS I Beam with UDL - restrained at ends only

sc434 Steel simply supported I beam with UDL restrained at ends only. The design aspects covered are: calculation of design values of actions; cross-section classification; cross-sectional resistances;lateral torsional buckling resistance between supports and calculation of the vertical deflection. Calculations are in accordance with BS5950-1:2000 and Eurocode 3 & the relevant SCI publications. Calculations are also in accordance with EC3 Part 1-1 and NCCI sheets provided by Access Steel.

sc436 SS I Beam with UDL & point loads restrained or unrestrained

sc436 Steel simply supported I beam restrained at load points. The design aspects covered are: calculation of design values of actions; cross-section classification; cross-sectional resistances;lateral torsional buckling resistance between intermediate restraints and calculation of the vertical deflection. Calculations are in accordance with BS5950-1:2000 and Eurocode 3 & the relevant SCI publications. Calculations are also in accordance with EC3 Part 1-1 and NCCI sheets provided by Access Steel.

sc437 SHS/RHS/CHS beam restrained at ends and loads

sc437 Simply supported SHS beam restrained at load points or fully restrained. The design aspects covered are: calculation of design values of actions; cross-section classification; cross-sectional resistances;lateral torsional buckling resistance between supports or intermediate restraints and calculation of the vertical deflection. Calculations are in accordance with BS5950-1:2000 and Eurocode 3 & the relevant SCI publications. Calculations are also in accordance with EC3 Part 1-1 and NCCI sheets provided by Access Steel.

sc438 Purlin on sloping roof

sc438 Purlin on sloping roof. As an alternative to the proprietary cold-formed sections used on large projects, channels are popular for purlins on smaller projects. This option follows Example 6 in 'Structural Steelwork Design' by L.J.Morris and D.R.Plum and is in accordance with BS5950-1:2000. Calculations are also in accordance with EC3 Part 1-1.

sc439 Side rail

sc439 Side rail. As an alternative to the proprietary cold-formed sections used on large projects, angles are popular for side rails on smaller projects. This option follows Example 7 in 'Structural Steelwork Design' by L.J.Morris and D.R.Plum and is in accordance with BS5950-1:2000. Calculations are also in accordance with EC3 Part 1.

sc440# Column (UB, UC or Channel) with biaxial bending

sc440 I section column bending about major axis. Assumes the member being considered is part of a continuous structure and design moments & forces have been calculated prior to use. The design aspects covered are: cross-section classification; cross-sectional resistances; lateral torsional buckling resistance between supports or intermediate restraints and interaction checks under combined bending and axial load. Calculations are in accordance with BS5950-1:2000 and Eurocode 3 and the relevant SCI publications.

sc441# Cased I section column

sc441 Cased Column. Deals with the design of a cased column in accordance with BS5950-1:2000 sections 4.7, 4.8 and 4.14. See also Example 8.7.2 in 'Structural Steelwork - Design to Limit State Theory' by T.J. MacGinley & T.C. Ang. Calculations are also in accordance with EC3 Part 1-1 and EC4 Part 1-1.

sc442# H Section column with biaxial bending

sc442 H section column bending about minor axis. Assumes the member being considered is part of a continuous structure and design moments & forces have been calculated prior to use. The design aspects covered are: cross-section classification; cross-sectional resistances; lateral torsional buckling resistance between supports or intermediate restraints and interaction checks under combined bending and axial load. Calculations are in accordance with BS5950-1:2000 and Eurocode 3 and the relevant SCI publications.

sc443 Effective length of compression members

sc443 Effective length to actual length ratios for rigid frames Computes the effective length for in-plane buckling of a column in a continuous structure with moment-resisting joints in accordance with BS5950 Annex E Effective lengths of compression members. Calculations are also in accordance with NCCI SN008a-EN-EU.

sc444# SHS and RHS column section design (axial load and bending)

sc444 RHS section column design. Assumes the member being considered is part of a continuous structure and design moments & forces have been calculated prior to use. The design aspects covered are: cross-section classification; cross-sectional resistances; lateral torsional buckling resistance between supports or intermediate restraints and interaction checks under combined bending and axial load. Calculations are in accordance with BS5950-1:2000 and Eurocode 3 and the relevant SCI publications.

sc445# Plastic design of stanchions

sc445 Plastic design of stanchions. Deals with the plastic design of I sections based on "Steelwork Design Guide to BS 5950 Vol 2" Example 13 and BS 5950-1:2000. Plastic design may be used where loading is predominantly static and fatigue will not be a design criterion. Calculations are also in accordance with EC3 Part 1-1, Clause 6.3.5 and Annex BB.3.

sc446 Single angle - section design

sc446 Axially loaded members formed from single angle. Deals with the design of axially loaded members formed from single angle in accordance with BS5950-1:2000 Clause 4.7.10 with reference to Example 14 of Steelwork Design Guide to BS5950. Calculations are also in accordance with EC3 Part 1-1.

sc447 Design of "I" beam curved on plan

sc447 Design of "I" beam curved on plan. This option is based on BS5950-1: 2000 & SCI publication "Design of Curved Steel"
Section types available include: Universal beam, Universal column, Rolled steel joist, any other I or H section.
The design is based on elastic design methods, and although this option does not interface with frame analysis results, it relies on an analysis of the beam having been carried out. Calculations are also in accordance with EC3 Part 1-1.

sc448 Design of SHS beam curved on plan

sc448 Design of SHS beam curved on plan. This option is based on BS5950-1: 2000 & SCI publication "Design of Curved Steel"
Section types available include: Hot Finished SHS, Hot Finished RHS, Cold Formed SHS, Cold Formed RHS. The design is based on elastic design methods, and although this option does not interface with frame analysis results, it relies on an analysis of the beam having been carried out. Calculations are also in accordance with EC3 Part 1-1.

sc449 Design of CHS beam curved on plan

sc449 Design of CHS beam curved on plan. This option is based on BS5950-1: 2000 & SCI publication "Design of Curved Steel" The design is based on elastic design methods, and although this option does not interface with frame analysis results, it relies on an analysis of the beam having been carried out. Calculations are also in accordance with EC3 Part 1-1.

sc450 Pin ended I, H, or channel section column

sc450 Axially loaded open section column. The design aspects covered are: cross-section classification; cross-section resistance and buckling resistance with or without intermediate restraint. Calculations are in accordance with BS5950-1:2000 and Eurocode 3 & the relevant SCI publications.

sc451 SHS, RHS and CHS column section design (axial load only)

sc451 Axially loaded steel hollow section column. The design aspects covered are: cross-section classification; cross-section resistance and buckling resistance with or without intermediate restraint. Calculations are in accordance with BS5950-1:2000 and Eurocode 3 & the relevant SCI publications.

sc452 I column in simple construction

sc452 I section column in 'simple' construction. The design aspects covered are: cross-section classification and the simple interaction criteria for combined axial compression and bi-axial bending. Calculations in accordance with Access Steel document SN048 for Eurocode 3 and Clause 4.7.7 of BS 5950-1-2000

sc453 SHS, RHS and CHS column in simple construction

sc453 SHS column in simply multi-storey construction. The design aspects covered are: cross-section classification and the simple interaction criteria for combined axial compression and bi-axial bending. Calculations are in accordance with Access Steel document SN048 for Eurocode 3 and Clause 4.7.7 of BS 5950-1-2000.

sc454 Channel bending about minor axis

sc454 Channel bending about minor axis. Deals with the design of a simply supported steel channel bending about its minor axis in accordance with BS5950-1:2000. The channel may have a tapered or parallel flange. This option factors dead and imposed loads and computes BM's, SF's and service deflections for a simply supported channel section used as a beam and subjected to any number of UDL's, triangular loads and point loads applied about the minor axis. Calculations are also in accordance with EC3 Part 1-1 and NCCI sheets provided by Access Steel.

sc455 Steel stair with flat plate stringers

sc455 Staircase with flat plate stringers to EC3. The proforma assumes the staircase stringers are simply supported for the evaluation of design bending moment, design shear and deflection under serviceability loads. As an alternative to the proprietary sections, flat plate stringers are popular for steel staircases and hence the reason for this proforma. The staircase treads are assumed to provide lateral restraint to the flat plate stringer compressive zone, which ensures bending about the major axis.

sc456 Laced or battened strut

sc456 Laced or battened strut. Deals with the design of a laced or battened strut in accordance with BS5950-1:2000 and makes reference to Example 10 in 'Steelwork Design Guide to BS5950' published by The Steel Construction Institute. Each member to be battened may be a UB, UC, tapered or parallel flange channel, or any other I or H section, or any other channel section. Calculations are also in accordance with EC3 Part 1-1.

sc457# Design of SHS section curved on elevation

sc457 Design of SHS beam curved on elevation. This option is based on BS5950-1: 2000 & SCI publication "Design of Curved Steel" Section types available include: Hot Finished SHS, Hot Finished RHS, Cold Formed SHS, Cold Formed RHS. Calculations are also in accordance with EC3 Part 1-1.

sc458# Design of CHS section curved on elevation

sc458 Design of CHS beam curved on elevation. This option is based on BS5950-1: 2000 & SCI publication "Design of Curved Steel". Calculations are also in accordance with EC3 Part 1-1.

sc459# Design of "I" beam curved in the vertical plane

sc459 Design of "I" beam curved in the vertical plane. This option is based on BS5950-1: 2000 & SCI publication "Design of Curved Steel" This option is based on elastic design methods and assumes:

- primarily it will be interfaced with plane frame analysis
- the member being considered is either rigidly connected to its supports or is part of a continuous member
- maximum moment and maximum shear force co-exist as is usual in continuous construction
- the web bearing and buckling checks may be for positions along the member length or at end supports.

Calculations are also in accordance with EC3 Part 1-1.

sc460 Portal frame design - plastic analysis

sc460 Portal frame design - plastic analysis.
For BS5950 "simple" plastic design method - plastic hinges form near the portal ridge and in the column at the underside of the haunch, element sizing is checked.
For EC3: three analyses will be carried out by this proforma:

- Initial sway analysis to determine elastic critical load factor, acr Optional base stiffness as 10% of column stiffness (SCI pub P397)
- Elastic analysis, check in-plane stability 'amplified moment method'
- Horizontal deflections check at SLS. Option to take base stiffness as 20% of the column stiffness (SCI pub P397) will be offered.

sc462 Portal frame in boundary conditions (fire)

sc462 Portal Frames in boundary conditions (fire). Deals with the design of portal frames subject to boundary conditions using the recommended procedure set out in The Steel Construction Institute publication "The behaviour of steel portal frames in boundary conditions" (2nd Edition), makes reference to Example 2; checks suitability of base plate arrangement. Calculations are also in accordance with EC3 Part 1-1.

sc463 Channel subject to eccentric loading

sc463 Channel subject to eccentric loading. Deals with the design of a channel unrestrained along its length subjected to eccentric loading in accordance with BS5950-1:2000 and SCI publication 'Beams subject to combined bending and torsion'. The end conditions of the beam influence greatly the torsional stresses along the member. This option considers only "torsion fixed, warping free" i.e. the end of the member is prevented from twisting (the beam is "pinned" at its ends), but is allowed to warp freely. Calculations are also in accordance with EC3 Part 1-1 and SCI publication entitled 'Design of Steel Beams in Torsion'.

sc464 Beam subject to eccentric loading

sc464 Beam subject to eccentric loading. Deals with the design of a steel beam unrestrained along its length subjected to eccentric loading in accordance with BS5950-1:2000 and SCI publication 'Beams subject to combined bending and torsion'. The end conditions of the beam influence greatly the torsional stresses along the member. This option considers only "torsion fixed, warping free" i.e. the end of the member is prevented from twisting (the beam is "pinned" at its ends), but is allowed to warp freely. Calculations are also in accordance with EC3 Part 1-1 and SCI publication entitled 'Design of Steel Beams in Torsion'.

sc465 RHS beam - eccentric loading

sc465 RHS beam subject to eccentric loading. Deals with the design of a RHS beam unrestrained along its length subjected to eccentric loading in accordance with BS5950-1:2000 and SCI publication 'Beams subject to combined bending and torsion'. The end conditions of the beam influence greatly the torsional stresses along the member. This option considers only "torsion fixed, warping free" i.e. the end of the member is prevented from twisting (the beam is "pinned" at its ends), but is allowed to warp freely. Calculations are also in accordance with EC3 Part 1-1 and SCI publication entitled 'Design of Steel Beams in Torsion'.

sc466 Steel beam with openings in the web

sc466 Steel beam with opening in the web. Deals with the design of a steel beam with openings in the web in accordance with BS5950-1:2000 & 'Design for openings in the web of composite beams' a CIRIA/SCI joint publication. The method assumes: the beam has previously been checked for shear, bending, lateral torsional buckling, bearing and deflection; no opening is located closer than 2D or 10% of the span to the support; openings are not less than the beam depth D, apart; point loads are not applied at less than D from the side of the adjacent opening; where stiffeners are required their minimum anchorage length beyond the edge of the opening is 150 mm. Calculations are also in accordance with EC3 Part 1-1 and SCI publication P355.

sc467 Cellular and castellated beams

sc467 Cellular and castellated beams. Deals with the design of cellular and castellated beams in accordance with BS5950-1:2000 and 'Design of Composite and Non-Composite Cellular Beams' By SCI. Calculations are also in accordance with EC3 Part 1-1 and SCI publication P355.

sc468 Beams formed from single angle

sc468 Beams formed from single angle. Deals with the design of a rolled steel angle acting as a beam in accordance with BS5950-1:2000. Calculations are also in accordance with EC3 Part 1-1 and NCCI sheets provided by 'Access Steel'.

sc470 Stiffened plate girder

sc470 Stiffened plate girder. Deals with the design of a stiffened plate girder, fully restrained throughout its length, in accordance with BS5950-1:2000. The design method follows Example 6 in 'Steelwork Design Guide to BS5950' published by The Steel Construction Institute. Calculations are also in accordance with EC3 Part 1-1.

sc472 Design of Structural T-sections

sc472 Design of structural Tee sections. Deals with the design of elements formed from structural Tee's, based on Clause 4.10 of BS5950. Calculations are also in accordance with EC3 Part 1-1.

sc474 Gantry girder

sc474 Gantry girder. Deals with the design of a simply supported gantry crane rail according to the method given in Example 15 of 'Steelwork Design Guide' published by SCI and BS5950-1:2000. The girder is assumed to be a rolled universal section. Gantry girders are intended to carry cranes of loading class defined by BS2573: Part 1. Calculations are also in accordance with EC3 Part 1-1 and Part 6.

sc476 Wind moment design of unbraced frames

sc476 Wind-moment design of unbraced frames. Deals with the design of unbraced frames in accordance with BS5950-1:2000 and SCI publication 'Wind-Moment Design for Unbraced Frames'. Wind loads are in accordance with CP3:Ch V:Part 2. Calculations are also in accordance with EC3 Part 1-1 and SCI-P-263 publication entitled 'Wind-moment Design of Low Rise Frames'.

sc477 Axially loaded steel lattice connections

sc477 Axially loaded steel lattice connections. Designs a KN - gap and overlap, or TYX joint. Calculations are in accordance with BS5950-1:2000 & ENV 1993-1-1-ANNEX K using the procedures of 'Design of SHS Welded Joints' published by British Steel Tubes & Pipes. Calculations are also in accordance with EC3 Part 1-1 and TATA Steel welded joints publication entitled 'Design of Welded Joints'.

sc478 Chord and bracing connections with moments incl. knee joints

sc478 Chord and bracing connections with moments. Axially loaded steel lattice connections. Designs CHS chord and bracings for T- Y- X joints or K- N- joints with gap; RHS chord and bracing for T- Y- X- joint; or welded knee joint in RHS. Calculations are in accordance with BS5950-1:2000 & ENV 1993-1-1-ANNEX K using the procedures of 'Design of SHS Welded Joints' published by British Steel Tubes & Pipes. Calculations are also in accordance with EC3 Part 1-1 and TATA Steel welded joints publication entitled 'Design of Welded Joints'.

sc479 Hollow section chords w. gusset plates/universal section brace

sc479 Hollow section chords with gusset plates/universal section. Designs a hollow section chord with gusset plate, or a hollow section chord with universal section brace, for joint types: X - joint with brace angle of 90 deg. or T - joint with brace angle of 90 degrees. Calculations are in accordance with BS5950-1:2000 & ENV 1993-1-1-ANNEX K using the procedures of 'Design of SHS Welded Joints' published by British Steel Tubes & Pipes.
Calculations are also in accordance with EC3 Part 1-1 and TATA Steel welded joints publication entitled 'Design of Welded Joints'.

sc480 Angle cleat connection

sc480 Angle cleat connection. Deals with the design of a double angle cleat connection in accordance with 'Steelwork Design Guide to BS5950-1:2000 published by The Steel Construction Institute. Analysis of the connection follows the treatment in the BCSA publication 'Manual on Connections Volume 1 - Joints in simple construction' amended to comply with the requirements of BS5950-1:2000.
Calculations also follow the treatment in the SCI publication P358 entitled 'Joints in Steel Construction: Simple Joints to EC3' and are in accordance with EC3 Part 1-8.

sc481 Fin plate connection

sc481 Fin plate connection. Deals with the design of a fin plate connection in accordance with BS5950-1:2000 and following the recommended design procedure checks given in the BCSA/SCI publication 'Joints in Simple Construction' Second Edition.
Calculations are in accordance with EC3 Part 1-1 and the analysis follows the recommended design procedure checks given in the BCSA/SCI publication P358 entitled 'Joints in Steel Construction: Simple Joints to EC3'.

sc482 Angle seat connection

sc482 Seating bracket. Deals with the design of a seating bracket, welded or bolted to the supporting member, in accordance with BS5950-1:2000 and BCSA publication 'Joints in simple construction'.
Calculations are in accordance with EC3 Part 1-8.

sc483 Shelf angle supporting slab

sc483 Shelf angle supporting precast units or solid slab to BS5950 and EC3. Deals with the design of a shelf angle continuous along the length of the member, welded to the web of the supporting member using hit and miss fillet welds. The latter is assumed to be a UB or UC section.

sc484 Flexible end plate

sc484 Flexible end plate connection. Deals with the design of a flexible end plate connection in accordance with the requirements of BS5950-1:2000 and the BCSA publication 'Manual on Connections Volume 1 - Joints in simple construction Second Edition'.
Calculations also follow the treatment in the BCSA/SCI publication entitled 'Joints in Steel Construction: Simple Joints to EC3' and EC3 Part 1-1.

sc485 Full-depth end plate connection

sc484 Full-depth end plate connection. Deals with the design of a full-depth end plate connection in accordance with BS EN 1993-1.
The analysis of the connection follows the treatment in the BCSA/SCI publication 'Joints in Steel Construction: Simple Joints to EC3'.

sc486 Flush end plate haunched connection

sc486 Flush end plate - haunched connection. Deals with the design of a flush end plate - haunched connection in accordance with BS 5950-1: 2000 and SCI publication 'Joints in Steel Construction - Moment Connections'. Calculations are also in accordance with EC3 Part 1-1, Part 1-5 and Part 1-8.

sc487 Extended end plate haunched connection

sc487 Extended end plate - haunched connection. Deals with the design of an extended end plate connection in accordance with BS 5950 and SCI publication 'Joints in Steel Construction - Moment Connections'.
The design assumes: the tension produced by the moment is transmitted through the bolts adjacent to the tension flange; the centre of rotation is taken as being about the bottom edge of the compression flange; the load distribution is such that the top two rows of bolts reach their design tensile strength; the shear is transmitted through the bottom row of bolts; the factored moment/forces at the end of the rafter act on a cross-section normal to the axis of the basic rafter.
Calculations are also in accordance with EC3 Part 1-1, 1-5 and 1-8.

sc488 Axially loaded column base for I sections

sc488 Slab base subject to axial load and moment. Deals with the design of a slab base subject to axial load and moment in accordance with BS5950-1:2000 using the effective area method. Load may be compressive and/or uplift. Calculations are also in accordance with EC3 Part 1-1, Part 1-5 and Part 1-8.

sc489 Axially loaded column base for SHS, RHS and CHS sections

sc489 Slab base subject to axial load - hollow sections. Deals with the design of a slab base for hollow sections in accordance with BS5950 for the following conditions: axially loaded SHS, RHS and CHS sections; load may be compressive and/or uplift. Calculations are also in accordance with EC3 Part 1-1, Part 1-5 and Part 1-8.

sc490 Column splice

sc490 Column splice. Deals with the design of a bolted column splice in accordance with BS5950-1:2000, and EC3 Part 1-1, Part 1-5 and Part 1-8. Dealing with factored loads and nominal moments induced by simple beam eccentricity in accordance with Subclauses 4.7.6 and 4.7.7 of BS595-1:2000 and referring to Example 10.3.2 in 'Manual on Connections — Vol 1' published by BSCA. The connection can be bearing type where the loads are transferred directly in bearing or non-bearing (where the loads are transferred via the bolts and the splice plates). The column splice is just above floor level (about 500 mm above) thus moment due to strut action is insignificant.

sc491 Bolted end column splice

sc491 Extended bolted end plate column splice. Calculations are in accordance with BS5950-1:2000 and follow the procedure given 'Joints in Steel Construction - Moment Connections' published by The Steel Construction Institute. Calculations are also in accordance with EC3 Part 1-1, Part 1-5 and Part 1-8.

sc492 Beam splice

sc492 Beam splice. Deals with the design of a beam splice plate subjected to an applied moment and shear force in accordance with BS 5950-1:2000 & 'Guide to BS5950 — Vol 2' published by SCI. The method assumes: the flange plates resist the full bending moment and axial loading; the web splice resists the vertical shear and the torsional moment induced by the eccentricity of this loading on the bolt groups on each side of the joint; the splice uses HSFB bolts to avoid producing bolt slip and consequent additional beam deflection. Calculations are also in accordance with EC3 Part 1-1, 1-5 and 1-8.

sc493 Universal section chord with hollow section brace

sc493 Universal section chord with hollow section brace. Designs X - subject to axial load and moment; T- Y- subject to load and moment; K- N- gap joint subject to axial load only; K- N- lap joint subject to axial load only. Calculations are in accordance with BS5950-1:2000 & ENV 1993-1-1-ANNEX K using the procedure of Section 5.4 of 'Design of SHS Welded Joints' published by British Steel Tubes & Pipes. Calculations are also in accordance with EC3 Part 1-1 and TATA Steel publication entitled 'Design of Welded Joints'.

sc494 Portal apex connection

sc494 Apex end plate - haunched connection. Deals with the design of an apex end plate - haunched connection in accordance with BS 5950-1: 2000 and SCI publication 'Joints in Steel Construction - Moment Connections'. Calculations are also in accordance with EC3 Part 1-1, Part 1-5 and Part 1-8.

sc495 Cast-in plate to connect structural steel beam to concrete wall

sc495 Design of cast-in steel plates for connecting structural steel beams to concrete walls or other elements (e.g. support brackets for services), in accordance with SCI publication P416 "The Design of Cast-In Plates", EC2 Patt 1-1, EC2 Part 1-8 and EC3 Part 1-1. Nominally pinned connections in simple construction for the building design are assumed.
Reference to the 'Green Book' SCI publication P358 which sets out the rules for simple joints between steel elements is made in the proforma.

sc496 Load bearing stiffener design

sc496 Load bearing stiffener. Deals with the design of a load bearing stiffener in accordance with BS5950-1:2000 Clause 4.5. This option can be used to check if a load bearing stiffener is required and/or to check the load bearing stiffener. The stiffeners may be either intermediate or at an end support. Calculations are also in accordance with EC3 Part 1-1, Part 1-5 and Part 1-8.

sc497 Section properties of haunched section

sc497 Properties of haunched section. Computes section properties of a Universal Beam or Column section with a haunch. Parent section and haunch are assumed to be of the same steel grade. Calculations relating to the welding of the haunch to the parent section are not included.

sc498 Bolts subject to eccentric loading

sc498 Bolts subject to eccentric loading. Deals with the design of a bolt group subject to eccentric loading in accordance with BS 5950-1: 2000 and Guide to BS 5950 Vol.2 by SCI, and refers to 'Structural Steelwork - Design to Limit State Theory' by MacGinley and Ang. Eccentric loads applied to the bolt group may be either: applied in the plane of the fasteners (the bolt group is subject to shear and torsion); or applied out of the plane of the fasteners (the bolt group is subject to shear and tension). Calculations are also in accordance with EC3 Part 1-1.

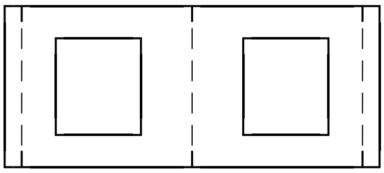
sc499 Stability checks for notched "I" beam to EC3

sc499 Stability checks for notched "I" beam to EC3. Deals with the design of a flexible end plate connection. Analysis of the connection follows the treatment in the BCSA/SCI publication entitled 'Joints in Steel Construction: Simple Joints to EC3' and in accordance with EC3 Part 1-1.

9.9 SCALE - Masonry Design to Eurocode 6, BS5628 and MEXE

sc503 Cantilever wall spanning horizontally

sc503 Wall design to BS5628 and EC6



Elevation on wall

Wall section between windows is considered as a cantilever spanning horizontally. The window is assumed to be spanning horizontally transferring wind loading to the end of the cantilever. Between the windows is assumed to be a wall, a column or a wind post secured to floors.

sc505 Wall panel

sc505 Laterally loaded wall panels. Deals with the design of laterally loaded masonry wall panels in accordance with BS5628 Table 8 or EC6 Annex E and follows the procedure adopted in Andrew Orton's book entitled "Structural Design of Masonry".

sc506 Panel with opening

sc506 Masonry wall panel with opening to BS5628 or EC6. When irregular shapes of panels or those with substantial openings, are to be designed it will often be possible to divide them into sub-panels, which can then be calculated based on BS5628 Table 8 or EC6 Annex E.
Andrew Orton's book entitled "Structural design of masonry" Appendix A, gives further guidance including methods for converting the line loads at sub-panel edges (from other sub-panels spanning onto them) into equivalent UDL's.

sc510 Single leaf wall

sc510 Masonry single leaf wall spanning vertically. Deals with the design of a single leaf wall subjected to vertical load and horizontal wind load. The calculations are in accordance with BS5628 or EC6. It is assumed there is no uplift and the roof or floor construction provides lateral resistance to lateral movement.

sc511 Single leaf wall by effective eccentricities

sc511 Masonry single leaf wall by effective eccentricity. Deals with the lateral resistance of a vertically loaded single leaf masonry wall by arching. The calculations are in accordance with BS5628-1:2005 or EC6 Part 1-1. There is assumed to be no uplift at the top and resistance to lateral movement is provided at both the top and bottom of the panel.

sc512 Lateral resistance of wall by arching

sc512 Masonry single leaf wall by vertical arching. Deals with the design of a single leaf masonry wall by effective eccentricity. The calculations are in accordance with BS5628-1:2005 or EC6 Part 1-1. The form of this calculation is in accordance with BS5628 clause 36.8 or EC6 Part 1-1 clause 6.3.2.

sc515 Column

sc515 Masonry column. Deals with the design of a masonry column constructed of bricks or blocks. The calculations are in accordance with BS5628-1:2005 or EC6 Part 1-1.

sc519 Bed joint reinforcement for laterally loaded masonry walls

sc519 Bed joint reinforcement to laterally loaded wall panels. Deals with the design of bed joint reinforcement in accordance with BS5628-2:2005 or EC6 Part 1-1. The design is based on Example 10 of "Handbook to BS5628" by The Brick development Association.

sc520 Freestanding wall

sc520 Freestanding masonry wall. Deals with the design of a freestanding masonry wall. The calculations are in accordance with BS5628-1:2005 or EC6 Part 1-1.

sc521 Brick beam

sc521 Masonry beam. Deals with the design of a simply supported singly reinforced rectangular brick beam in accordance with BS5628-2:2005 or EC6 Part 1-1 and Part 2.

sc525 Bearing

sc525 Masonry bearing. Deals with the design of a masonry bearing. The calculations are in accordance with BS5628-1:2005 or EC6 Part 1-1.

sc528 Simplified sub-frame analysis to EC6

sc528 Simplified sub-frame analysis to EC6. This option deals with the evaluation of moments at the top and bottom of vertically loaded walls in accordance with Annex C of BS EN 1996-1-1:2005.

sc529 Internal masonry wall

sc529 Internal masonry wall. Deals with the design of an internal masonry wall. Calculations are in accordance with BS5628 or EC6. The following factors, which affect stability, should be provided for in the design: accommodation for movement; presence and position of openings; the likelihood of exceptional lateral loading arising from the nature of the use of the building; chasing.

sc530 External cavity wall

sc530 External cavity wall. Deals with the design of an external cavity wall constructed of brick or block. The calculations are in accordance with BS5628 or EC6.

sc532 External cavity wall - by vertical arching

sc532 External cavity wall by vertical arching. Deals with the design of an external cavity wall constructed of bricks or blocks. The calculations are in accordance with BS5628-1:2005 or EC6 Part 1-1.

sc535 Shear wall

sc535 Masonry shear wall. Deals with the design of a masonry shear wall constructed with bricks or blocks. The calculations are in accordance with BS5628-1:2005 or EC6 Part 1-1.

sc536 Fin wall

sc536 Masonry fin wall building. For a full treatment of the design of fin and diaphragm walls, see the 'Structural Masonry Designers' Manual' by Curtin, Shaw, Beck & Bray. Deals with the design of a single storey building having loadbearing brick fin walls with panels between of cavity construction; the roof plate provides a prop at top of the fin walls, and transmits wind forces to the flank and end walls. The calculations are in accordance with BS5628-1:2005 or EC6 Part 1-1 and Part 3.

sc537 Reinforced brick retaining wall

sc537 Reinforced masonry retaining wall. Deals with the design of a reinforced masonry retaining wall in accordance with BS5628-2:2005 or EC6 Part 1-1 and Part 2.

sc538 Reinforced brick column

sc538 Reinforced masonry column. Deals with the design of a reinforced masonry column in accordance with BS5628-2:2005 or EC6 Part 1-1. An additional option to design a reinforced hollow concrete block column is also offered when EC6 is selected.

sc540 Load distribution to shear walls

sc540 Distribution of wind load to concrete or masonry shear walls. Wind forces are distributed to the shear walls by the 'slab-over' acting as a stiff horizontal diaphragm (the slab must make good connections with the walls). The method is described in 'Structural Design of Masonry' by Andrew Orton.

sc541 Support angles

sc541 Masonry support angles. Deals with the design of masonry support angles in accordance with 'Design of Masonry Support Angles' by J.R.Veale, Steel Construction Today. 1988, 2, pp 81-88.

sc543 Pippard/MEXE analysis for unit applied load

sc543 Pippard/MEXE analysis for masonry arch bridges. This option analyses a masonry arch as a two-pinned structure separately under dead and live loads. A unit width of the arch is modelled as a plane frame. The method should only be used when there is well compacted fill between spandrels. The results from the frame analysis will be the stresses from the axial load and bending moment for each load case. The analysis is continued by calculating the allowable load in terms of a single axle and then the allowable multiple axle loads from CS 454 Appendix E Figures E.5 and E.6. The capacity in terms of gross vehicle weights is obtained from CS 454 Appendix E Table E.3.

sc546 Pippard/MEXE analysis: C&U, AW, user defined bogies

sc546 Pippard/MEXE analysis for masonry arch bridges C&U, AW and user defined single axles and bogies. This option analyses a masonry arch as a two-pinned structure separately under dead and live loads. A unit width of the arch is modelled as a plane frame. The method should only be used when there is well compacted fill between spandrels. The results from the frame analysis will be the stresses from the axial load and bending moment for each load case.

sc547 Pippard/MEXE analysis for user defined loading

sc547 Pippard/MEXE analysis for masonry arch bridges under user defined loading. This option analyses a masonry arch as a two-pinned structure separately under dead and live loads. A unit width of the arch is modelled as a plane frame. The method should only be used when there is well compacted fill between spandrels. The results from the frame analysis will be the stresses from the axial load and bending moment for each load case.

sc548 Assessment of masonry arch bridges by modified MEXE

sc548 Assessment of masonry arch bridges by the modified MEXE method. Deals with the assessment of the strength of the arch barrel in accordance with Departmental Standard CS 454. The modified MEXE method may be used to estimate the carrying capacity of arches spanning up to 18 metres, but for spans over 12 metres it becomes increasingly conservative compared to other methods. The method should not be used where the arch is flat or appreciably deformed. When the depth of fill at the crown is greater than the thickness of the arch barrel the results should be confirmed using an alternative method.

sc550 Mechanism analysis of arches

sc550 Mechanism analysis of masonry arch. This option for the masonry arch problem was contributed by Professor Michael Horne based on that due to Harvey published in The Structural Engineer 1st March 1988. The geometry of the intrados, loading and hinge positions of an assumed mechanism is used to solve for horizontal reaction, depth of arch, and moments at supports. After the depth of arch has been computed, a check is made to ensure that the line of thrust lies within the intrados and extrados. If it does, then the mechanism is valid, if not then the analysis must be repeated with new hinge positions at sections of greatest violation.

sc552 Masonry dams

sc552 Masonry dams. Deals with the analysis of masonry dams; checks the profile of a dam to ensure that the line of thrust comes within the middle third. Stresses at the back and front of the dam are also computed. For a full treatment of the traditional theory of masonry dams, see 'The Analysis of Engineering Structures' by Pippard and Baker.

sc554 Masonry gravity wall

sc554 Masonry gravity wall. Deals with the design of a masonry gravity wall. This option is in accordance with BS5628-1:2005 being the Code of practice for the use of masonry ⁄ Part 1: Structural use of unreinforced masonry.

9.10 SCALE - Drainage and Surveying

sc560 Drain & sewer pipe sizing

sc560 Drain & sewer pipe sizing. Deals with the sizing of pipes for: soil sewage, surface water, combined sewer. There are many publications dealing with the provision of drainage, the most important being: BS 8301 Building Drainage; BS 6367 Drainage of roofs and paved areas; BS 8005 Sewerage; Sewers for adoption; Tables for the hydraulic design of pipes and sewers. There are also many formulae for the hydraulic design of pipes and sewers viz: Chezy, Crimp and Bruges, Manning, and Colebrook and White. The favoured formula for the design of pipes is that of Colebrook & White (used by Hydraulics Research) and used in this option.

sc561 Gutters and rain water pipes

sc561 Roof drainage gutters. Deals with the drainage of flat, single pitched and double pitched roofs based on BS EN 12056-3:2000. The method of design assumes that: the gutter slope is not more than 1 in 350; the gutter has a uniform cross sectional shape; the outlets are large enough to allow the gutter to discharge freely; the distance between a stop end and an outlet is less than 50 times the upstream water depth, or the distance between two outlets is less than 100 times the maximum water depth.

sc565 Road alignment vertical curve

sc565 Road alignment vertical curve. Deals with the design of a vertical curve for road alignment using a simple parabolic curve. The method is based on 'Highways' second edition by C A O'Flaherty. 'Roads and Traffic in Urban Areas' by IHT and DTp.

sc566 Road alignment horizontal curves

sc566 Road alignment horizontal transition & circular curves. This option produces setting out information for the theodolite and tape method based on 'Surveying' by A.Bannister and S.Raymond. The horizontal curve may consist of: circular curve only; circular curve and entry and exit transitions; wholly transitional curve.

sc569 Areas and volumes for cuttings and embankments

sc569 Areas and volumes for cuttings and embankments. This option computes areas of cross sections of cuttings and embankments, and sections part in cut and part in fill. The formulae are given in 'Surveying' by Bannister & Raymond, 1959. This option also computes volumes by the Prismoidal Formula.

sc570 Volumes from spot levels

sc570 Volumes from spot levels. Having located the outline of a structure on the ground, the engineer divides up the area into squares or rectangles, marking the corner points. Levels are taken at each corner point, and by subtracting from these the corner formation levels, the average depth of each rectangle can be found and the volume of soil thereby computed. This option computes the volume of soil from the spot levels and formation level based on the method in 'Surveying' by Bannister & Raymond.

sc571 Simpson's rule for volumes

sc571 Volumes by prismoidal formula. Accurate computation of volumes from cross sectional areas - equally spaced - requires the use of Simpson's rule for volumes. The number of sections must be odd; the reader is referred to A Goldstein's letter to The Structural Engineer 20-8-96 on the need for sufficient cross sections. Using just the average of the end areas times the distance between can give erroneous results. Use SCALE option 650 for computing the areas A(1), A(2) .... A(n); then use this option for computing the volume based on Simpson's rule for volumes, see 'Surveying' by Bannister & Raymond.

sc580 Setting out piles

sc580 Setting out of piles. This option evaluates the setting out angles of piles, given theodolite station and pile position coordinates. Up to twenty stations may be given, and five hundred piles.

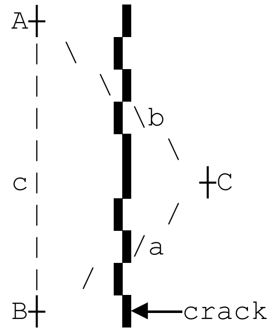
sc581 Analytical geometry based on the COGO approach

sc581 Co-ordinate geometry. This option can be used in the solution of geometrical problems and is based on the IBM COGO Programming System. The engineer defines point numbers and associated X and Y co-ordinates which are stored in a co-ordinate table and these values are available for generating the X and Y co-ordinates of further points. The maximum number of points is 500.

sc582 Traverse survey - open or closed

sc582 Traverse survey. This option tabulates station coordinates for open or closed traverse surveys based on 'Surveying' by Bannister & Raymond. If the traverse is open, all angles are measured clockwise, taking a backsight on the previous station and a foresight on the next station. If the traverse is closed, it must be carried out in an anti-clockwise direction, all internal angles being measured clockwise, taking a backsight on the previous station and a foresight on the next station. For a closed survey, corrections are applied.

sc590 Crack movements - by tell-tales



sc590 Crack movement by tell-tales.

Tell-tales are located at positions A, B & C. Tell tales A & B are approximately parallel to the crack denoted thus: ▌. The dimensions a, b & c are measured by micrometer at various time intervals.

This option computes the movements normal and parallel to the crack.

9.11 SCALE - Analysis by Traditional Methods

sc601 Table of deflections for checking beams

sc601 Table for checking beam deflections. This option tabulates deflections for a range of spans, and a range of inertias. The table is intended for checking purposes; deflections are tabulated directly for simply supported, propped cantilever and fully fixed supports for loads applied as central point loads, as triangularly distributed, and as uniformly distributed. The table may extend to perhaps ten or twenty pages but it has the advantage that it is quickly 'looked up' without the engineer having to think whether to divide by 10^8 or multiply by it. To save paper, cases where the deflections would be too large, or quite small, are suppressed.

sc602 Shears, moments, etc. for beams

sc602 Shears, moments and deflections of beams. This option computes shears, moments and deflections for simply supported beams, cantilevers and propped cantilevers, built-in beams, and a two span beam one of which is a cantilever the other of which is the 'tie-down' span. Shear, moment and deflection formulas for beams are generally taken from 'Formulas for Stress and Strain' by Roark, published by McGraw- Hill with additions from the 'Steel Designer's Manual'.

sc603 Beams under combined axial and transverse loading

sc603 Beams under combined axial and transverse loading. Computes bending moments and deflections on single span beams with a wide range of support conditions and axial and transverse loadings. This option makes use of formulas for beams under combined axial and transverse loading, taken from 'Formulas for Stress and Strain' by Roark, published by McGraw-Hill.

sc604 Continuous beam - moment distribution

sc604 Continuous beam - moment distribution. Deals with the analysis of continuous beams, with or without cantilevers, subjected to point loads, uniformly distributed loads and partial distributed loads, using moment distribution. The method of analysis assumes that the supports do not settle.

sc605 Cantilever beam with varying section properties

sc605 Cantilever beam with varying section properties.
This proforma computes end deflection and rotation for a cantilever with varying section properties, when subjected to an end point load.

sc606 Simply supported beam

sc606 Simply supported beam. This option factors dead and live loads and computes BM's, SF's and service deflections for a simply supported beam subjected to any number of UDL's, triangular loads and point loads. The load factors may be set to unity if designing to a permissible stress Code.

sc608 Cranked or dogleg beam

sc608 Cranked or dogleg beams. Deals with the analysis of two types of popular beam which are cranked in elevation and have pinned ends. This option computes bending moments, shear forces and support reactions for various different loading conditions.

sc610 Influence lines

sc610 Influence lines. This procedure computes influence lines for beams from 1 to 8 spans; inertia may vary from span to span. Influence lines may be computed for reaction, shear & moment; theory was contributed by Donald Alcock. The procedure sets up and reduces a stiffness matrix in which each beam element contributes the submatrices. The matrix is banded 4 elements wide. For each influence line, forces are applied to the appropriate element. The deflected form of the beam is the influence line by the Muller-Breslau principle. Areas in each span are computed by Simpson's rule.

sc611 Pigeaud's tables for concentrated loads on slabs

sc611 Pigeaud's tables for multiple concentrated loads. Deals with the calculation of bending moments in slabs spanning either one way or both ways using coefficients developed by M. Pigeaud. Pigeaud's coefficients originally appeared in 'Annales des Ponts et Chaussees, Memoires', 1929, and the tabulated values used here are interpolated from graphs in the 'Reinforced Concrete Designer's Handbook'. Note with freely-supported spans, that when the load covers the entire slab, the moments obtained when using this method are substantially (i.e. up to 30%) less than those given by the corresponding Code coefficients for 2-way slabs.

sc612 Rectangular culverts

sc612 Design and moments in box culverts. This option calculates the moments and shears in roof, walls and floor of rectangular box culverts due to the following types of loading: central concentrated load on roof; uniform load on roof; self-weight of walls; earth (i.e. triangular) pressure on walls; surcharge (i.e. uniform) pressure on walls; internal hydrostatic pressure.

sc614 Circular plate with UDL

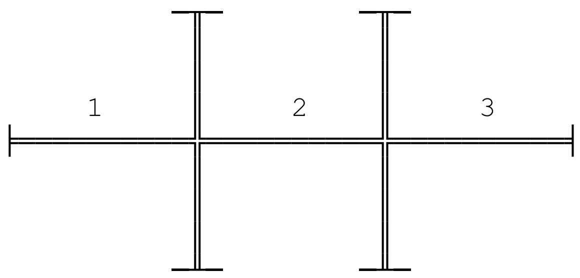
sc614 Circular plate. This option computes stress and deflection at any point on a circular plate in accordance with Formulas for Stress & Strain, by Roark.

sc615 Rectangular plates

sc615 Rectangular plate. This option computes deflections, and bending moments for rectangular plates in accordance with "Theory of Plates & Shells" by Timoshenko & Woinowsky-Krieger. Timoshenko's tables are for a Poisson's ratio of 0.3 which is appropriate to steel; the Poisson's ratio for concrete is usually taken as 0.2, nevertheless this option may be used to check that design moments - computed in the analysis - are in the right field.

sc620 Subframe analysis

sc620 Subframe analysis.



Computes bending moments and shear forces for the subframe shown subjected to point loads, UDL's and partial UDL's on beams 1 to 3.

sc621 Plane truss analysis - joint by joint

sc621 Plane truss analysis for triangulated frameworks. This option facilitates the manual method for the resolution of forces at joints of triangulated frameworks having pin-ended members. Each joint may have up to two unknown forces (resolving vertically and horizontally solves for two unknowns) and four known forces. Start at a support and then proceed to an adjacent joint having no more than two unknown forces.

sc622 Plane truss analysis

sc622 Plane truss analysis. This option determines forces in the members of a plane truss by the method of joints. For details of this method, see Chapter 6 of 'Computer Methods in Solid Mechanics' by Gennaro. Joint numbers are assigned in the same sequence that an engineer would follow in the manual solution of trusses. Thus joints are numbered consecutively such that only two unknown forces occur at each joint, all other forces having been previously determined from the equilibrium analysis of the joints with lower numbers.

sc623 Multi storey and multi bay frames

sc623 Multi-storey & multi-bay frames by moment distribution. This option follows that given in 'Computer Methods in Solid Mechanics' by Gennaro. The effects of axial and shear deformations are ignored, as is traditional in moment distribution. Loading data comprises: horizontal loads applied at each floor level, vertical point loads and full or partial UDL's applied to beams.

sc624 Multi storey frames

sc624 Multi-storey frames by Naylor's Method. In an article entitled 'Side Sway in Symmetrical Building Frames', published in the Structural Engineer of April 1950, N Naylor presented a modification of moment distribution for the analysis of multi—storey frames. This option uses Naylor's method for 1 to 4 storey frames having one bay.

sc626 Vierendeel girders

sc626 Vierendeel girders by Naylor's Method. In an article entitled 'Side Sway in Symmetrical Building Frames', published in the Structural Engineer of April 1950, N Naylor presented a modification of moment distribution for the analysis of frames. This option uses Naylor's method for 2 to 4 bay Vierendeel girders.

sc628 Shear legs

sc628 Shear legs. This option computes forces in a set of shear legs based on 'Theory of structures', by A Morley.

sc632 Rectangular portal

sc632 Rectangular pinned base portal. This option computes moments and reactions on a rectangular pinned base portal subjected to various loadings. The method is based on Kleinlogel.

sc634 Ridged portal

sc634 Ridged pinned base portal. This option computes moments and reactions on a ridged pinned base portal subjected to various loadings. The method is based on Kleinlogel.

sc635 Two bay ridged portal

sc635 Two bay ridged pinned base portal. This option computes moments and reactions on a two bay ridged pinned base portal subjected to various loadings. The method is based on Kleinlogel.

sc636 Couple roof

sc636 Couple roof. This option computes moments and reactions on a couple roof subjected to various loadings. The method is based on Kleinlogel.

sc638 Circular arch

sc638 Circular Arch. This option computes moments and reactions on a circular arch subjected to various loadings. The method is based on Roark - Fourth edition.

sc639 Helical stair

sc639 Helical stair with fixed supports. This option computes moments and reactions for a helical stair with fixed supports. The method is based on 'RC Designers Handbook' 10th ed. sec. 25.4.2 and the following assumptions are made: the loading on the stair is uniform and symmetrical; the ratio of shear modulus G to elastic modulus E is 0.42; the torsional constant C is taken as one-half of St.Venant value J. For further information refer to 'Charts for the Design of Helical Stairs' by Santathadaporn and Cusens (Concrete & Constructional Engineering, February 1966).

sc640 Deep beams

sc640 Beams of relatively great depth. In beams of small span/depth ratio, the shear stresses are likely to be high, and the deflection due to shear must be added to the deflection due to bending; this option computes the shear deflection. For extremely short beams (span/depth ratios < 3), the assumption of linear stress distribution is no longer valid. Actual extreme fibre stresses, and horizontal shear stresses are significantly higher than those computed by the simple theory. Roark in 'Formulas for stress & strain' gives a table of values by which stresses from the simple theory must be multiplied, and these values are used in this option.

sc642 Roof trusses

sc642 Roof trusses. This option computes forces in the members of roof trusses having from 2 to 5 spans as detailed in the Dorman Long Handbook 1964.

sc643 Section design for UB's UC's SHS's RHS's & CHS's

sc643 Section design for factored axial load in combination with bi-axial bending for: Universal Beams, Universal Columns, Square Hollow Sections, Rectangular & Circular Hollow Sections. Tables for all these sections are provided and may be displayed or printed.

sc649 Plastic section properties

sc649 Plastic section properties. This option computes plastic section properties based on formulas given in the 'NL-STRESS Reference Manual' for: solid rectangle, hollow rectangle, circle, ellipse, hollow circular section, hollow ellipse, solid octagon, hollow octagon, I section, T section.

sc650 Section properties - general

sc650 Section properties. This option computes elastic section properties based on formulas given in 'Formulas for stress and strain' by Roark and Young for: channel, T section, Z section, I section, RHS and angle. This option also computes section properties for any shaped section using Donald Alcock's method and for this case the section is defined by coordinates of corner points taken in anti-clockwise order round the section; the cross section is kept to the left of the edge running from a previous point to the next point; the section being closed when the original point is specified again.

sc651 Section properties with weightings

sc651 Properties of a plane section - with weightings. A weighting is applied to each item to take account of different elastic moduli. The geometrical information may be input in one of three ways: the item is defined by co-ordinates of corner points taken in anti-clockwise order around the item; the item being closed when the co-ordinates of the first point input are re-specified; if the item is a rectangle, circle or ellipse, the dimensions and location of centroid and item weighting are input; the second moments of area and product second moment of area about the item centroid together with the location of the centroid and item weighting are input.

sc652 Section properties - plate girder with welded connections

sc652 Section properties of a plate girder with welded connections. This option computes the gross and net elastic and plastic section properties of a plate girder with welded connections between the web and flanges.

sc653 Section properties - plate girder with angle connections

sc653 Section properties of a plate girder with angle connections. This option computes the gross and net elastic and plastic section properties of a plate girder with angle connections between the web and flanges.

sc659 Influence surfaces of elastic plates - Pucher's charts

sc659 Influence surfaces of elastic plates. This option computes the the bending moments resulting from a defined set of loading in accordance with 'Influence surfaces of elastic plates' by Adolf Pucher. Bending moments at the following locations may be calculated:

- cantilever: restrained edge, centre, free edge
- plate strip with two supported edges: centre, quarter points
- plate strip with two restrained edges: centre, quarter points restrained edge.

sc660 Yield-lines

sc660 Yield line analysis. This option computes ultimate moments on a wide range of slab shapes and edge restraint conditions based on 'Yield line formulae for slabs' by Johansen, and 'Ultimate Load Analysis of Reinforced Concrete Structures' by L.L.Jones.

sc662 Suspension bridges

sc662 Suspension bridges. This option applies the so called elastic theory of suspension bridges as given in 'The analysis of engineering structures' by Pippard & Baker to the analysis of:

- the hanging cable
- suspension bridge with three-pinned stiffening girder
- suspension bridge with two-pinned stiffening girder.

sc663 Suspension cable analysis

sc663 Suspension cable analysis. The method of analysis deals with the two dimensional cable problem in which only horizontal and vertical loads are considered, and these act in the plane of the cable. A Newton iteration procedure is used to satisfy 'equilibrium' and 'compatibility'.

sc664 Dynamical behaviour

sc664 Dynamical behaviour of structures. This option computes the natural frequencies for beams, single and multi-storey frames using: Raleigh's formula, Dunkerley's empirical formula & other well known formula given by GB Warburton in 'The dynamical behaviour of structures'.

sc666 Elastic stability

sc666 Elastic stability of bars and plates. This option computes the elastic critical loads and stresses for bars and plates with various support conditions, based on 'Formulas for Stress and Strain' by Roark.

9.12 SCALE - Pre and Post Processors for Structural Analysis

sc670 HB Vehicle - load sharing

sc670 NL-STRESS - vehicle load sharing. This option generates a file of loading data for a vehicle in various positions on a bridge deck. The file of loading data can then be combined with the data file containing the geometry and material properties of the deck and the combined file submitted to the NL-STRESS program for analysis. Before this option can be run, a data file ending in the extension .dat must be prepared containing structure parameters, joint coordinates and member incidences.

sc672 Post-processing NL-STRESS results

sc672 Post-processing NL-STRESS results. An NL-STRESS analysis produces two files, both of which have the same name as the data file but having the following filename extensions:

- .res (standing for RESults) giving the results in text format
- .arr (standing for ARRays) giving the results and the data in binary format.

The results file may be incorporated into a spreadsheet directly, or edited and then incorporated. The arrays file may be converted from binary format to text format, as described in this option, and then incorporated into a spreadsheet, or used directly as also described.

sc674 Derives moments and principal stresses from plate deflections

sc674 Derivation of moments & stresses from deflections of plates. Given material properties for a plate and nine deflections on a rectangular grid, this option computes principal stresses and angles.

sc677 Elimination of expressions from an NL-STRESS data file

sc677 Elimination of expressions from NL-STRESS data file. This option converts an NL-STRESS data file containing expressions or assignments into one in which the expressions or assignments are replaced by their numerical values. The file is converted, and a backup of the original file is saved with a .bak extension.
NOTE: If there are two or more spaces following the variable or expression, then SCALE simply overprints. This means either leave just one space between data items or - if you are preparing your data in columns - leave sufficient room in each field to print the computed value.

sc678 Guyed masts

sc678 Guyed mast. The analysis of structures which comprise members which are completely flexible together with members which have some stiffness e.g. guyed masts may be carried out using this option. Although special programs have been written for guyed masts, many ignore mast shortening and buckling effects (which often have a major influence on mast stiffness), furthermore this option offers more flexibility viz: the pretension, section properties, direction, ground level and loading may be different for every guy. Following an interactive dialogue with the engineer, this option prepares a data file for a mast, and then invokes NL-STRESS for analysis of the mast.

sc679 Circular tanks

sc679 Analysis of circular tanks. The computer analysis of circular tanks (usually concrete) requires the tank to be modelled by a number of elements joining joints. This option accepts the tank radius, height etc. and computes and lists Joint Coordinates etc. for inclusion in a file of structural data for analysis. The base of the tank is modelled by 'rings' of members connected by 'spokes'. The innermost ring may be thought of as the hub to which the spokes are connected. The walls are similarly modelled by a number of horizontal rings connected by vertical members.

sc680 Joint coordinates of circular members

sc680 Polar to Cartesian coordinate conversion etc. The analysis of circular members such as spiral staircases, requires that the radius of the stair, angle of the tread, and riser (riser is zero for bow girder) be converted to Cartesian coordinates. This option accepts the radius, initial angle for the start of a spiral, riser, angle between and number of treads, initial joint number and increment in joint numbering and computes and lists a table of Joint Coordinates for inclusion in a file of structural data for analysis by NL-STRESS. Alternatively the option will compute the coordinates of a circle in space, rotated about the Y, Z and X axes in order.

sc682 Beta angle for space frame members

sc682 Beta angle for space frame members. Space structures usually have the webs of members normal to the ground. However when an I-section, for example, is used on the side of an 'Eiffled' tower, it is necessary to work out the angle BETA between the vertical plane through the member and the plane through the web of the I-section. This option accepts the end coordinates xj,yj, zj and xk,yk,zk of the member and the coordinates of a third point lying in the plane through the web of the member (just another point on the side of the tower), and from these computes the angle BETA to be used in the structural analysis.

sc684 Member stiffness matrices

sc684 Member stiffness matrices and fixed end moments.
This option computes member stiffness matrices for plane trusses, plane frames, plane grids, space trusses and space frames. Optionally fixed end moments may be computed for plane & space frames, and grids. The component terms of the member stiffness matrices may be 'posted' to the structure stiffness matrix and option 39 used to invert the stiffness matrix and thereby provide a manual and checkable analysis method for simple frames.

sc686 Loss of stiffness due to partial plasticity

sc686 Elastic-plastic analysis of compression members. For triangulated frames, in which the structural action of the members lies essentially in their resistance to axial forces, unit shape factor analysis (in which plastic deformation is assumed to be confined to discrete plastic 'hinges') may give results which are not sufficiently reliable. This option gives a method enabling NL-STRESS to deal with compression members in the plastic analysis of triangulated frames by introducing imperfections into the members of the frame such that the imperfections model requirements of structural codes.

sc687 Equilibrium & compatibility check: plane frame & truss members

sc687 Equilibrium and compatibility check of plane frame and plane truss members. The engineer provides the member end forces and moments and properties, this option then carries out the check. This simple check assumes that the member/segment to be checked is not subjected to loading within the length of the member/segment. As an alternative to typing in the data, the data may be extracted from the last NL-STRESS analysis, by running SCALE option 1 in batch mode, i.e. respond 1/b to the Option prompt. This causes SCALE to produce tables containing the required data to be written to the files summary1.res to summary6.res, which can be picked up subsequently by option 687.

sc688 Equilibrium & compatibility check: plane grid members

sc688 Equilibrium and compatibility check of plane grid members. The engineer provides the member end forces and moments and properties, and this option carries out the check. This simple check assumes that the member/segment to be checked is not subjected to loading within the length of the member/segment. As an alternative to typing in the data, the data may be extracted from the last NL-STRESS analysis, by running SCALE option 1 in batch mode, i.e. respond 1/b to the Option prompt. This causes SCALE to produce tables containing the required data to be written to the files summary1.res to summary6.res, which can be picked up subsequently by this option 688.

sc689 Equilibrium & compatibility check: space frame & truss members

sc689 Equilibrium and compatibility check of space frame and space truss members. The engineer provides the member end forces and moments and properties, this option then carries out the check. This simple check assumes that the member/segment to be checked is not subjected to loading within the length of the member/segment. As an alternative to typing in the data, the data may be extracted from the last NL-STRESS analysis, by running SCALE option 1 in batch mode, i.e. respond 1/b to the Option prompt. This causes SCALE to produce tables containing the required data to be written to the files summary1.res to summary6.res, which can be picked up subsequently by option 689.

sc690 Support displacements to model for guy pretension

sc690 Support displacements to model for guy pretension. NL-STRESS permits the effects of finite displacements to be taken into account in the analysis when METHOD SWAY is selected; NL-STRESS can therefore be used to analyse structures with attached guys. The foot of guys are normally pretensioned to some value while the guy is subject only to its own self weight. The pretension is best modelled by applying JOINT DISPLACEMENTS at the foot of each guy and this option provides the engineer with values of joint displacements to be used to model the guy pretension.

sc692 Modelling of guys for analysis by stiffness method

sc692 Guy loading to model for artificial guy stiffness. The analysis of guyed structures by NL-STRESS requires three runs as follows: a first run with cables artificially increased by a factor 's' with corresponding reduction in elastic modulus to keep the product EA unchanged; a second run to compensate for the change in cable diameter by applying a fictitious uniform transverse loading to the cable together with compensating loads at each end of the cable; a third run using predicted final transverse loads computed from first two runs to give true modelling of structure with flexible guys. This option gives values to be used for fictitious loads for both second and third runs.

sc694 Loads on continuous beams - used in sheet pile design

sc694 Continuous beams for temporary works. In sheet pile design the loading is fixed, but support positions are varied to obtain economy in the design. This option accepts loading information and support positions separately, and writes a file of data for analysis by NL-STRESS.

9.13 SCALE - Loadings

sc701 Snow loading

sc701 Roof and snow loading to to BS6399-3:1988 or
BS EN 1991-1-3:2003. This option derives the snow loading on roofs for the following roof types:

- flat or pitched roofs
- mono or multi-pitch roofs
- single span or multi-span roofs.

Curved roof profiles are excluded from this option.

sc702 Wind loads to BS6399-2:1997 and Eurocode 1

sc702 Wind loads to BS6399-2:1997 or BS EN 1991-1-4:2005. This option will allow the user to check:

- dynamic wind pressure qs
- external pressure coefficients Cpe for walls
- external pressure coefficients Cpe for pitched roofs of rectangular buildings
- external pressure coefficients Cpe for flat roofs of rectangular buildings.

sc704 Lateral earth pressure from surcharge

sc704 Lateral earth pressure due to surcharges. This option computes the lateral earth pressure at various depths due to point loads, line loads and strip loads applied to the surface, in accordance with analysis of lateral earth pressure for surcharges by theory of elasticity; ref: 'Foundation Analysis & Design' by Joseph E. Bowles.

sc705 Pressures from retained materials

sc705 Pressures from retained materials. This option deals with the calculation of pressures on vertical or inclined surfaces due to dry or submerged granular or cohesive materials, based strictly on information given in Tables 17 and 18 of the RCD Handbook 10th edition.

sc706 Moments and forces in hopper bottoms

sc706 Bending moments and shearing forces in hopper bottoms. Deals with the calculation of the bending moments and direct forces acting on inclined hopper bottoms. This option is based on information given in Table 186 of the RCD Handbook 10th edition, to which reference should be made for further information.

sc707 Moments and forces in cylindrical containers

sc707 Analysis of cylindrical containers with wall monolithic with base. Deals with the calculation of the maximum bending moments and direct forces acting on a cylindrical container, where the bottom of the wall is monolithic with the base, based on information given in Table 184 of the current RCD Handbook. The original analysis of this condition was published by Dr. Reissner in Germany, and explanatory articles by H.Carpenter first appeared in the UK in the magazine 'Concrete & Constructional Engineering in April 1927 and June 1929. In 'Thin Shells - Computing and Theory', Dr J.E.Gibson develops a similar analysis and his formulae are also evaluated in this option.

sc710 Loading on strip footing

sc710 Loads on strip footing. This option computes: characteristic dead and live and reduced live loads, and factored dead and live and reduced live loads, at roof level and each floor level down to the strip footing level, summarising the loads in a table.

sc711 Column & foundation loads

sc711 Column and foundation loads. This option computes: characteristic dead and live and reduced live loads, and factored dead and live and reduced live loads, at roof level and each floor level, down the column to the foundation level, summarising the loads in a table.

sc715 Unit loads

sc715 Unit loads. This options computes unit loads for: sloping roofs including plaster/plasterboard; ceilings to roof spaces including plaster/plasterboard; external dormer sides; flat timber roofs; existing timber floors; new timber floors including plaster/plasterboard; brick and block cavity walls with plaster/render; solid brick walls with plaster/render; solid block walls with plaster/render; existing timber stud partitions; new timber stud partitions; insitu or precast concrete slabs; concrete walls; staircases.

sc716 Loading on domestic lintel

sc716 Loading on domestic lintel. Computes loading based on BS 5977: Part 1: 1981 assuming: all the weight of the masonry within the load triangle is carried as a load on the lintel; any point or distributed loads applied to the masonry within the load triangle are dispersed at 45 degrees and carried by the lintel; any point or distributed loads applied to the masonry within the interaction zone are reduced by 50%, dispersed at 45 degrees and carried by the lintel; no openings within the load triangle; if the height of masonry above the lintel is less than 0.6 times the clear span then all loads are taken directly by the lintel. Calculations are also in accordance with EC6 Manual published by the IStructE and PD 6697:2010.

sc717 Unit loads for steel frames

sc717 Unit loads for steel frames. This option computes unit loads for: single storey portal frame structures; lattice girder structures; roof truss structures; plaster/plasterboard ceiling or a false ceiling and services load to underside of truss or girder. Loadings are taken from BS 648: 1964: Schedule of Weights of Building Materials including AMD 105 and AMD 344 and British Steel General Steel Commercial Office publication 'Single Storey Buildings'.

sc718 Steel beam with welded bottom flange

sc718 Steel beam with welded bottom flange plate. This option computes the centroid and eccentricity of loads carried by a beam with an extended bottom flange plate. Four cases are considered each with different load combinations for the engineer to ascertain the optimum design case when later checking the chosen section using option 464.

sc720 Weighting of items comprising several parts

sc720 Volume and weight of parts. This option computes volume & weight for an item having one or more parts (e.g. flange, web etc).

sc725 Factored load summary sheet

sc725 Unfactored and factored load sheet. This option allows the user to input unfactored values for dead and imposed loads and the computed factored combination loads will be for chosen partial safety factors. Loadings may be: uniformly distributed, concentrated, or area loads.

sc726 Eurocode load combinations

sc726 Eurocode load combinations. Deals with the combination of actions by means of expressions 6.10, 6.10a and 6.10b in BS EN 1990:2002. The proforma offers guidance on combination of actions (load combinations) for the following cases:

- beam with one variable action
- beam with two independent of each other variable actions
- multi-storey steel braced frame (simple construction)
- multi-storey steel moment frame (continuous construction)

9.14 SCALE - Foundations

sc731 Notes on pilecaps

sc731 Pilecap design - general notes. This option gives general notes applicable to options sc732 to sc749 for the design of pilecaps.

sc732 Pilecap for 2 piles - truss analogy

sc732 Pilecap design for two piles: truss analogy. Clause 3.11.4.1 of BS8110 specifies that pile caps may be designed either by bending theory or truss analogy. In this option, the latter method of analysis has been employed and the tensile forces acting between pile heads determined by statics. Calculations are also in accordance with EC2 Part 1-1.

sc734 Pilecap for 4 piles - truss analogy

sc734 Pilecap design for four piles: truss analogy. Clause 3.11.4.1 of BS8110 specifies that pile caps may be designed either by bending theory or truss analogy. In this option, the latter method of analysis has been employed and the tensile forces acting between pile heads determined by statics. Calculations are also in accordance with EC2 Part 1-1.

sc735 Pilecap for 5 piles - truss analogy

sc735 Pilecap design for five piles: truss analogy. Clause 3.11.4.1 of BS8110 specifies that pile caps may be designed either by bending theory or truss analogy. In this option, the latter method of analysis has been employed and it is assumed that the central pile in the group (i.e. that located directly beneath the column) supports one-fifth of the total load on the system. Calculations are also in accordance with EC2 Part 1-1.

sc742 Pilecap for 2 piles - bending theory

sc742 Pilecap design for two piles: bending theory. Clause 3.11.4.1 of BS8110 specifies that pile caps may be designed either by bending theory or truss analogy. In this option, the former method of analysis has been employed but, because of the high depth/span ratio it is probable that the pilecap will tend to behave more as a truss than as a beam, and so all of the main bars are turned up into the sides of the pilecap to provide a full bond length beyond the critical plane. Calculations are also in accordance with EC2 Part 1-1.

sc743 Pilecap for 3 piles - bending theory

sc743 Pilecap design for three piles: bending theory. Clause 3.11.4.1 of BS8110 specifies that pile caps may be designed either by bending theory or truss analogy. In this option, the former method of analysis has been employed but, because of the high depth/span ratio it is probable that the pilecap will tend to behave more as a truss than as a beam, and so all of the main bars are turned up into the sides of the pilecap to provide a full bond length beyond the critical plane. Calculations are also in accordance with EC2 Part 1-1.

sc744 Pilecap for 4 piles - bending theory

sc744 Pilecap design for four piles: bending theory. Clause 3.11.4.1 of BS8110 specifies that pile caps may be designed either by bending theory or truss analogy. In this option, the former method of analysis has been employed but, because of the high depth/span ratio it is probable that the pilecap will tend to behave more as a truss than as a beam, and so all of the main bars are turned up into the sides of the pilecap to provide a full bond length beyond the critical plane. Calculations are also in accordance with EC2 Part 1-1.

sc745 Pilecap for 5 piles - bending theory

sc745 Pilecap design for five piles: bending theory. Clause 3.11.4.1 of BS8110 specifies that pile caps may be designed either by bending theory or truss analogy. In this option, the former method of analysis has been employed and it is assumed that the central pile in the group (i.e. that located directly beneath the column) supports one-fifth of the total load on the system. Because of the high depth/span ratio, it is probable that the cap will tend to behave more as a truss than as a beam, and so all the main bars are turned up into the sides of the cap to provide full bond length beyond the critical plane.
Calculations are also in accordance with EC2 Part 1-1.

sc746 Pilecap for 6 piles - bending theory

sc746 Design of pilecap for six piles: bending theory. Clause 3.11.4.1 of BS8110 specifies that pile caps may be designed either by bending theory or by truss analogy, and in this option the former method has been employed. However, because of the high depth/span ratio, it is probable that the cap will tend to behave more as a truss than a beam and so all the main bars are turned up into the sides of the cap to provide a full bond length beyond the critical plane. It is assumed that each pile supports one-sixth of the column load.
Calculations are also in accordance with EC2 Part 1-1.

sc747 Pilecap for 7 piles - bending theory

sc747 Design of pilecap for seven piles: bending theory. Clause 3.11.4.1 of BS8110 specifies that pile caps may be designed either by bending theory or by truss analogy, and in this option the former method has been employed. However, because of the high depth/span ratio, it is probable that the cap will tend to behave more as a truss than a beam and so all the main bars are turned up into the sides of the cap to provide a full bond length beyond the critical plane. It is assumed that each pile carries one-seventh of the column load.
Calculations are also in accordance with EC2 Part 1-1.

sc748 Pilecap for 8 piles - bending theory

sc748 Design of pilecap for eight piles: bending theory. Clause 3.11.4.1 of BS8110 specifies that pile caps may be designed either by bending theory or by truss analogy, and in this option the former method has been employed. However, because of the high depth/span ratio, it is probable that the cap will tend to behave more as a truss than a beam and so all the main bars are turned up into the sides of the cap to provide a full bond length beyond the critical plane. It is assumed that each pile supports one-eighth of the column load.
Calculations are also in accordance with EC2 Part 1-1.

sc749 Pilecap for 9 piles - bending theory

sc749 Design of pilecap for nine piles: bending theory. Clause 3.11.4.1 of BS8110 specifies that pile caps may be designed either by bending theory or by truss analogy, and in this option the former method has been employed. However, because of the high depth/span ratio, it is probable that the cap will tend to behave more as a truss than a beam and so all the main bars are turned up into the sides of the cap to provide a full bond length beyond the critical plane. It is assumed that each pile supports one-ninth of the column load. Calculations are also in accordance with EC2 Part 1-1.

sc750 Concrete retaining wall

sc750 Design of concrete retaining wall. This option calculates the basic pressures, moments and forces in a concrete cantilever retaining wall and the surrounding soil. The soil pressure may be calculated using either the Rankine or the Coulomb theory. The Rankine method assumes that the resultant pressure acts parallel to the soil surface (but the theory is invalid if the surface slope is negative); the Coulomb method permits the angle of friction between the soil and the wall surface to be set by the engineer. With both theories, the engineer may choose whether or not to take account of the stabilising force provided by the vertical component of the pressure (if any).

sc751 Active pressure on retaining wall to BS8002:1994 or Eurocode 7

sc751 Active pressure on a retaining wall to BS8002:1994 or Eurocode 7 using Coulomb's Wedge Theory for cohesive and cohesionless soils. The theory assumes: a neutral or ineffective zone within which there is no adhesion or friction along the back of the wall or along the plane of rupture. The theory takes into account cohesion along the failure plane and along the plane of the wall.

sc752 Stability calculations for a retaining wall

sc752 Stability calculations for a retaining wall. This option calculates the factors of safety for a reinforced concrete retaining wall subjected to soil pressures such as evaluated by option sc751.

sc754 Biaxial bending on pad base

sc754 Biaxial bending on rectangular base. This option calculates the bearing pressures beneath a rectangular base that is subjected to biaxial bending combined with axial load. This option is to a large extent based on formulae given on page 525 of Part 2 of the 1971 Edition of 'Beton-Kalander', published by Wilhelm Ernst und Sohn.

sc755 Pad base

sc755 Pad base with overturning moment. This option computes the pressures beneath a concrete pad base subjected to vertical and horizontal forces and an overturning moment. The method follows current practice, evaluating the self weight of the base and combining this with the applied vertical load. This total vertical load is then combined with the overturning moment to find the centroid of the vertical load, which is then checked for being within or without the middle third. Finally the factor of safety against overturning is computed.

sc756 Preliminary sizing of trapezoidal combined base

sc756 Preliminary design of trapezoidal base. Deals with the preliminary sizing of trapezoidal bases to support pairs of columns. There are three main options. The first determines the optimum base dimensions such that the pressure beneath the base is near-uniform and safe. The second finds the critical pressures, moments and shears for a base of specified dimensions. The third allows the engineer to specify all but one of the base dimensions and the program will then attempt to find a suitable value such that the conditions described in the first option are met.

sc757 Preliminary sizing of T-shaped combined base

sc757 Preliminary calculations for T-shaped combined base. Deals with the preliminary sizing of T-shaped combined bases to support pairs of columns. There are three main options. The first determines the optimum base dimensions such that the pressure beneath the base is near-uniform and safe. The second finds the critical pressures, moments and shears for a base of specified dimensions. The third allows the engineer to specify all but one of the base dimensions and the program will then attempt to find a suitable value such that the conditions described in the first option are met.

sc758 Preliminary sizing of rectangular combined base

sc758 Preliminary calculations for rectangular combined bases. Deals with the sizing of rectangular bases carrying pairs of columns, positioned eccentrically so as to reduce the variation of pressure beneath the base to a minimum. There are three main options. The first determines the optimum base dimensions such that the pressure beneath the base is near-uniform and safe. The second finds the critical pressures, moments and shears for a base of specified dimensions. The third allows the engineer to specify all but one of the base dimensions and the program will then attempt to find a suitable value such that the conditions described in the first option are met.

sc759 Preliminary design of tied separate bases

sc759 Preliminary design of tied bases. Deals with preliminary calculations for the design of pairs of bases supporting a trestle or similar structure, where each pair of bases is linked by a connecting beam.

sc760 Combined base

sc760 Analysis of combined footing. This option calculates the maximum soil pressures, shear forces and bending moments beneath a symmetrical rectangular or trapezoidal base of constant thickness supporting two vertical concentrated loads acting on the centre-line of the base.

sc761 Traditional raft foundation design

sc761 Raft and strip foundations. Raft foundations were traditionally designed as strip foundations in two directions. The stages in the calculations are: compute the centroid of the loads; compute the pressures at each end of the raft/strip assuming the raft/strip is completely rigid thereby defining a linear pressure distribution under the raft giving the pressure at each load; at each load position compute the shears and moments due to the effects of the upward soil pressure and the downward loads. The traditional approach has largely been replaced by a grid analysis modelling the soil as a number of springs; see option 561.

sc764 Beam on elastic foundation with load train - Hetenyi

sc764 Ground beam on elastic foundation subjected to load train. This option uses the formulae for deflection, slope, bending moment and shearing force for a concentrated force at an arbitrary point, as given by M.Hetenyi in 'Beams on elastic foundations' published by University of Michigan Press. This option allows up to 50 point loads in the load train, and the train to be stepped any number of increments to the right.

sc765 Beam on elastic foundation

sc765 Beam on elastic foundation. Deals with the beam on elastic foundation by applying 'Formulas for stress & strain' by Roark. The formulas assume a very long beam is supported continuously along its length by a surface that may be regarded elastic, in the sense that the surface may be considered to deflect in proportion to the intensity of the applied pressure. Three conditions of loading are available: a point load; a uniformly distributed load over a given length; and end point load and couple. By superposition, the results for the three cases may be applied to a variety of loading conditions.

sc767 Active thrust on wall: cohesionless soil w. horizontal surface

sc767 Active thrust on wall - cohesionless soil with horizontal surface. Computes total active thrust and distance of line of action of thrust below top of wall for: water table above bottom of wall
water table below bottom of wall.
Based on Example 6.1 of Soil Mechanics by RF Craig, 6th Edition, published Spon 1997.

sc768 Active & passive pressures on sheet pile wall - with two soils

sc768 Active and passive pressures on sheet pile wall - with two soils. Soil 1 is above water table, soil 2 is below.
Based on Example 6.2 of Soil Mechanics by RF Craig, 6th Edition, published Spon 1997.

sc769 Active wall pressure (Rankine) retaining soil slope

sc769 Active wall pressure (Rankine) retaining soil slope.
Computes the active wall pressure and total active thrust on a wall retaining a soil slope.
Based on Example 6.3 of Soil Mechanics by RF Craig, 6th Edition, published Spon 1997.

sc770 Slip circle analysis - Taylor's stability numbers

sc770 Slip Circle Analysis. This option computes factor of safety of a soil slope using Taylor's stability numbers.

sc771 Slip circle analysis - Swedish method

sc771 Slip circle analysis by the Swedish (or strip) method. Deals with slope stability analysis by the Swedish method (see 'The Mechanics of Engineering Soils' by Capper & Cassie). This option computes the intersection points of the slip circle and the boundary lines of the strata and surfaces of the soil. It then generates the weight and moment of the mass of soil and resistance to rotation. Dividing the resistance by the moment gives the factor of safety.

sc772 Coefficient of volume compressibility and compression index

sc772 Coefficient of volume compressibility, compression index, and preconsolidation pressure. Accepts pressures and dial gauge readings from the oedometer test and determines the coefficient of volume compressibility at required stress increments. The preconsolidation pressure is found by a numerical procedure based on Casagrande's method. The numerical procedure avoids the need to plot void ratio against the effective pressure to a LOG scale.
Based on Example 7.1 of Soil Mechanics by RF Craig, 6th Edition, published Spon 1997.

sc773 One dimensional consolidation beneath rectangular raft

sc773 One dimensional consolidation beneath rectangular raft. It is assumed that consolidation is approximately one-dimensional e.g. a clay layer can drain into the soil above and below the layer.
Based on Example 7.2 of Soil Mechanics by RF Craig, 6th Edition, published Spon 1997.

sc775 Safe bearing pressures

sc775 Safe bearing pressures. This option uses Terzaghi's, Meyerhof's, and Skempton's bearing capacity factors to determine the soil safe bearing capacity taking into account the removal of overburden pressure when calculating the net ultimate bearing capacity. This option assumes the soil type is constant.

sc776 Structural design of bituminous roads

sc776 Flexible pavement layers to TRRL Report 1132. This is the most appropriate design document for general pavement design. If you are aware of the current "normal" traffic flows, type of traffic and CBR value of the subgrade "1132" will guide you through the pavement design for that road.

sc777 Flexible pavement design to HD26/06

sc777 Flexible pavement design to HD26/06 and IAN 73/06 Rev1 (HD25). The proforma deals with the design of the upper pavement which entails the surface course, binder course and base to HD 26/06 and pavement foundation which entails the Subbase and Capping over the Subgrade to HD25. Two approaches are offered for calculating the overall thickness of the pavement foundation namely Restricted Foundation Design (Classes 1, 2 & 3) or Performance Foundation Design (Classes 1, 2, 3 & 4). Interim Advice Note IAN 73/06 Rev 1 (2009) is utilised by this proforma as currently HD25 is in DRAFT.

sc780 Soil pressures due to stacks

sc780 Soil pressures due to coal and other stacks. It is sometimes necessary to pile in the vicinity of coal and other stacks. The stack produces a vertical and horizontal stress and accompanying shear stress at chosen depth and distance from the stack. The computed soil stresses refer to those due to the stack alone, and must be combined with those due to the soil itself.

sc781 Elastic stresses within foundation material

sc781 Elastic stresses within foundation material.
Computes vertical stress, radial stress, tangential stress and shear stress due to a point load applied at the surface. Computes vertical stress due to general loading applied at the surface.

sc782 Elastic stresses resulting from UDL on rectangular area

sc782 Elastic stresses resulting from UDL on rectangular area.
Computes elastic stresses at any position beneath a raft which applies a uniformly distributed load the soil beneath.
Based on Example 5.1 of Soil Mechanics by RF Craig, 6th Edition, published Spon 1997.

sc786 Vertical loads on straight bored piles in clay

sc786 Vertical loads on straight bored piles in clay. This option computes: pile capacity, cut off level and pile length to carry a vertical load in ground having layers of known soil properties. The method is due to Wlodek Borzyslawski.

sc787 Large diameter belled piles in clay

sc787 Large diameter belled piles in clay. This option computes: pile capacity, cut off level and pile length to carry a vertical load in ground having layers of known soil properties. The method is due to Wlodek Borzyslawski.

sc788 Pile loads from a crane base

sc788 Pile loads from a crane base. Deals with the design of a four pile pilecap supporting the crane base. This option will calculate the vertical loads on piles. In assessing the total safe working load on piles the following additional loads need to be considered by the designer: self weight of pilecap; horizontal loads on piles.

sc789 Pile group analysis

sc789 Pile group analysis. This option follows the principles set out in Microcomputer Applications in Structural Engineering by Mosley and Spencer.

sc790 Loads on piles in groups

sc790 Loads on piles in groups. This option computes: loads on group of vertical piles; loads on group of piles raking in the loading plane based on Reinforced Concrete Designers Handbook 1957 Edition. This option is intended for the analysis of large groups of relatively-flexible piles carrying a rigid cap. It is not intended for analysing small systems since the analysis makes the simplifying assumption that each pile is hinged at head and toe. A detailed description and the derivation of the formulae used is given by Chas Reynolds in the journal 'Concrete and Constructional Engineering', December 1952.

sc791 Pile types and characteristics

sc791 Pile types and characteristics. This option allows the engineer to list pile types and characteristics including usage, for any or all of the following: small diameter rotary bored; large diameter rotary bored; small diameter continuous flight augered; large diameter continuous flight augered; small diameter percussive bored; large diameter percussive bored; excavated barrettes; hollow steel section; solid steel section; hollow cylinder concrete; Franki type; Vibroform; closed ended steel section; West shell type; closed ended hollow steel section; precast concrete; rotary bored; percussive bored; continuous flight augered; jacked preformed; driven steel cased.

sc792 Sheet retaining structures

sc792 Sheet retaining structures. This option analyses sheet retaining structures which rely on ground penetration alone or in combination with a single tie to provide rotational stability, for example: steel sheet piles, contiguous bored piles or diaphragm walls. Earth pressure coefficients are calculated using the Muller-Breslau Equations. Walls are analysed using the Free Earth Support Method. Pressures and moments are calculated at fixed intervals down from the top of the wall so that the intervals coincide with change in soil strata and water levels.

sc793 Laterally loaded piles

sc793 Laterally loaded piles. This option analyses single piles subjected to lateral loading, and considers pile stability only (not deformation) and computes the lateral resistance due to earth pressures using Brom's method. The maximum bending moment and its level, and the level of the bottom of the cage are computed. Slightly different maximum moments might be expected from a deformation/soil reaction analysis, but the method used has proved safe in a great many cases. Pressures and moments are calculated at fixed intervals down from the top of the pile so that the intervals coincide with change in soil strata and water levels.

sc794 Sheet pile wall - Das

sc794 Sheet-pile wall. Deals with the design a sheet-pile wall based on methods by Das in 'Principles of Foundation Engineering'. The soil being retained above the dredge line is assumed to be cohesionless (i.e. sandy). Either sand (cohesionless) or clay (cohesive) soil may be selected below dredge line level, but it is assumed that the same type of soil occurs on both sides of the sheet-piling. A single line of soil anchors near the top of the piling (i.e above water-table level) may be incorporated. The analysis generally ignores the flexibility of the piling and the reduction in bending moments that occur as a result.

sc796 Embedded depths of holding down bolts

sc796 Holding down bolts. Deals with the design of holding down bolts. It is assumed that: bolts are arranged symmetrically about the major axis; the failure mode of bolts pulled from a concrete foundation is based on a conical pull-out, such that the surface area is 4.44D^2, where D is the depth of embedment of the bolts; holding down bolts should be anchored into the foundation by a washer plate or other load distributing member embedded in the concrete; bolts have been previously checked for combined shear and tension.

9.15 SCALE - Steel Design to BS449 including Amendment 8,1989

sc300# I beam

sc300 I section beam design. Deals with the design of UB & UC sections used as beams. Calculations are in accordance with BS449: Part 2: 1969.

sc305# I column

sc305 I section column design. Deals with the design of UB & UC sections used as columns. Calculations are in accordance with BS449: Part 2: 1969.

sc310# RHS beam

sc310 RHS beam design. Deals with the design of rectangular hollow section beams in accordance with DESIGN IN SHS to BS449 published by BSC Tubes Division.

sc315# RHS column

sc315 RHS column design. Deals with the design of rectangular hollow section columns in accordance with DESIGN IN SHS to BS449 published by BSC Tubes Division.

sc320 Angle cleat

sc320 Angle cleat connection. Deals with the design of an angle cleat connection following the treatment in 'Manual on Connections for Beam and Column Construction, Conforming with the requirements of BS 449 Part 2:1969', published by BCSA.

sc325 Angle seat

sc325 Angle seat connection. Deals with the design of an angle seat connection following the treatment in 'Manual on Connections for Beam and Column Construction, Conforming with the requirements of BS 449 Part 2:1969', published by BCSA.

sc330 Flexible end plate

sc330 Flexible end plate connection. Deals with the design of a flexible end plate connection following the treatment in 'Manual on Connections for Beam and Column Construction, Conforming with the requirements of BS 449 Part 2:1969', published by BCSA.

sc335 Extended end plate

sc335 Extended end plate moment connection. Deals with the design of an extended end plate moment connection following the treatment in 'Manual on Connections for Beam and Column Construction, Conforming with the requirements of BS 449 Part 2:1969', published by BCSA.

sc340 Bolts in tension and shear

sc340 Bolts in tension and shear. Deals with the design of bolts in tension and shear following the treatment in 'Manual on Connections for Beam and Column Construction, Conforming with the requirements of BS 449 Part 2:1969', published by BCSA.

sc345 Column base

sc345 Slab base with optional bending moment. Deals with the design of a column base with optional bending moment following the treatment in 'Manual on Connections for Beam and Column Construction, Conforming with the requirements of BS 449 Part 2:1969', published by BCSA.

sc350 Steel guardrailing & Stainless Steel CHS rail/balustrade design

sc350 Steel guardrailing & SS rail/balustrade design. Deals with the design of guardrailing to limit to permissible values the following: the bending stress in the rail; the horizontal deflection of the rail between posts; the bending stress in the post; the lateral deflection of the vertical post. Loading to BS 6399:Part 1:1984:Table 4.

sc355 Steel runway beam

sc355 Steel runway beam. Deals with the design of UBs, UCs and RSJs used as runway beams in accordance with BS 2853:1957 and BS449. The method of support affects the way in which the longitudinal bending stresses and the transverse bending stresses in the bottom flange of the beam are checked for acting in combination. Three cases of support to the ends of runway beams are considered viz: simply supported on padstones or built into walls; cantilevered; simply supported by means of bolt hangers. Runway beam calculations are also in accordance with BS EN 1993-6:2007 and BS EN 1993-1-1:2005.

9.16 LUCID - Reinforced Concrete Detailing System

lu110 Pile caps

lu110 Pile caps. This option will detail pile caps both in plan and section, together with a detail of the connection to the column which may take the form of starter bars, a pocket, or holding down bolts. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu120 Square and rectangular reinforced and mass concrete bases

lu120 Square and rectangular reinforced and mass concrete bases. This option will detail foundations both in plan and section, together with a detail of the connection to the column which may take the form of starter bars, a pocket, or holding down bolts. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu130 Isolated, internal and edge strip footings

lu130 Isolated, internal and edge strip footings. This option will detail strip footings both internal and external combined with a ground slab. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu210 Free standing cantilever retaining walls

lu210 Free standing cantilever retaining walls. This option will detail free standing retaining walls both in elevation and section, together with a base key if required. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu220 Propped retaining walls

lu220 Propped retaining walls. This option will detail propped retaining walls both in elevation and section. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu310 Culverts and subways

lu310 Culverts and subways. This option will detail box culverts and subways in section. The details permit the culvert cross-section to be square or rectangular. Two further details show on plan the diagrammatic arrangement of reinforcement at curves and skew ends. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu410 Simply supported single panel slabs

lu410 Simply supported single panel slabs. This option will detail solid simply supported single panel slabs which span between line supports such as beams or walls. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu420 One way spanning slabs

lu420 One way spanning slabs. This option will detail solid one-way spanning continuous panel slabs which span between line supports such as beams or walls. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu430 Two way spanning slabs

lu430 Two way spanning slabs. This option will detail solid two-way spanning continuous panel slabs which span between line supports such as beams. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu440 Flat slabs

lu440 Flat slabs. This option will detail solid flat slabs which are supported by columns. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu450 Flat slabs - shear reinforcement at columns

lu450 Flat slabs - shear reinforcement at columns. This option will detail shear reinforcement at columns supporting solid flat slabs. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu460 Holes and chairs for top reinforcement

lu460 Holes and chairs for top reinforcement. This option offers two methods of trimming holes in solid slabs and two alternative ways of fixing chairs for supporting top slab reinforcement. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu510 Square, rectangular and circular columns

lu510 Square, rectangular and circular columns. The details provided cover columns of square, rectangular or circular cross-section. The reinforcement is detailed on a schematic elevation together with a cross-section at mid height. If desired a second cross-section near the top of the column may be provided when the column size reduces above the upper floor level. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu610 Walls

lu610 Walls. The details provided cover short, medium or long panels. In all cases the vertical steel is either stopped off or carried on upwards as starters for the next lift. The medium range includes panels with door openings. Reinforcement details are shown in elevation and in section. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu710 In-situ staircases

lu710 In-situ staircases. This option will detail in-situ staircase flights and landings. Each staircase flight is spanning principally between line supports at its top and bottom and is detailed in section only. A key plan can be provided on the drawings to assist in specifying the orientation and location of the staircase. Reinforcement details are shown in elevation and in section. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu810 Simply supported and continuous beams

lu810 Simply supported and continuous beams. This option will detail simply supported and continuous beams with up to two layers of steel in both the bottom of the span and the top over a support. The shear reinforcement provided consists of vertical stirrups with from 2 to 6 legs at a section and these may be arranged in up to 3 different zones along the beam. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu820 Cantilever beams

lu820 Cantilever beams. This option will detail cantilever beams with up to two layers of steel in both the bottom of the span and the top over a support. The shear reinforcement provided consists of vertical stirrups with from 2 to 6 legs at a section and these may be arranged in up to 3 different zones along the beam. The user can run option 910 automatically if required, to append a bar bending schedule to the calculations file.

lu910 Bar schedule

lu910 Bar bending schedule. This option produces a bar schedule for a maximum of 200 barmarks. The schedule takes the form of a table having headings given below.

This schedule complies with the requirements of BS8666:2005.

Member	Bar mark	Type and size	No. of mbr	No. of bar	Total no.	Lngth ofbar (mm)	Shape code	A (mm)	B (mm)	C (mm)	D (mm)	E/R (mm)	Rev ltr

9.17 SPADE - Structural Parts And Details Ensemble

sp202 Timber beam to column connection

sp202 Timber connections. This option will detail beam to column connections of the following types:

- U-plate
- concealed fixing
- single plate
- T-plates on both sides.

sp250 Timber spliced joints - various

sp250 Timber beam splice. This option will detail spliced joints of the following types:

- timber or steel splice on one side
- timber or steel splice on two sides.

sp252 Timber joist bearings on steel beams

sp252 Timber joists bearings on steel beams. This option will detail timber joists bearings on steel beams of the following types: continuous timber supported on top flange; timber on one side supported on top flange; timbers overlapping supported on top flange; timber on both bottom flanges; timber on one bottom flange.

sp265 Timber fan truss - general arrangement

sp265 Fan truss general arrangement. This option will detail fan trusses having four or six top bays. With each of the following options the basic truss setting out details will be shown on the elevation:

- full outline of truss members with member offset dimensions
- partial outline of truss members with member offset dimensions
- centre line of truss members only.

sp266 Timber fink truss - general arrangement

sp266 Fink truss general arrangement. This option will detail fink trusses having 'equal bay joists' or 'four equal top panels'. With each of the following options the basic truss setting out details will be shown on the elevation:

- full outline of truss members with member offset dimensions
- partial outline of truss members with member offset dimensions
- centre line of truss members only.

sp267 Timber howe truss - general arrangement

sp267 Howe truss general arrangement. This option will detail Howe trusses having four top bays. With each of the following options the basic truss setting out details will be shown on the elevation:

- full outline of truss members with member offset dimensions
- partial outline of truss members with member offset dimensions
- centre line of truss members only.

sp268 Timber pratt truss - general arrangement

sp268 Pratt truss general arrangement. This option will detail Pratt trusses having four equal top and bottom bays. With each of the following options the basic truss setting out details will be shown on the elevation:

- full outline of truss members with member offset dimensions
- partial outline of truss members with member offset dimensions
- centre line of truss members only.

sp269 Timber queen post truss - general arrangement

sp269 Queen post truss general arrangement. This option will detail Queen post trusses. With each of the following options the basic truss setting out details will be shown on the elevation:

- full outline of truss members with member offset dimensions
- partial outline of truss members with member offset dimensions
- centre line of truss members only.

sp310 Portal eaves haunched connection

sp310 Portal eaves haunched connection - flush top. This option will detail portal eaves haunched connections which have a flush top.

sp320 Portal apex haunched connection

sp320 Portal apex haunched connection. This option will detail a portal apex haunched connection. This option assumes that the haunch is cut from the same section as used for the rafter.

sp410 Flexible end plate connection: beam to beam

sp410 Flexible end plate connection - beam to beam. This option will detail flexible end plate beam to beam connections of the following types:

- one beam supported by another beam
- two beam supported by another beam.

sp412 Flexible end plate connection: beam to column

sp412 Flexible end plate connection - beam to column. This option will detail flexible end plate beam to column connections of the following types:

- one beam connected to column flange
- one beam connected to column web
- two beams connected to column web.

sp416 Extended end plate connection: beam to column

sp416 Extended end plate connection - beam to column. This option will detail extended end plate beam to column connections.

sp420 Double angle cleat: beam to beam connection

sp420 Double angle cleat - beam to beam connection. This option will detail double angle cleat beam to beam connections of the following types:

- one beam supported by another beam
- two beams supported by another beam.

sp422 Double angle cleat: beam to column connection

sp422 Double angle cleat - beam to column connection. This option details one or two beams connected to the supporting column. The principal axes of all members lie on a common vertical plane. If two beams are to be connected to the column then the upper flange levels must be the same.

sp430 Angle seat: beam to column connection

sp430 Angle seat - beam to column connection. This option will detail angle seat beam to column connections of the following types:

- one beam connected to column flange
- one beam connected to column web
- two beams connected to column web.

sp440 Fin plate connections: beam to beam

sp440 Fin plate connection - beam to beam. This option details one or two beams connected to a supporting beam by fin plates. If there are two beams to be connected, the principal axes must lie on a common vertical plane. The upper flange levels of all beams must be common.

sp442 Fin plate connections: beam to column

sp442 Fin plate connection - beam to column. This option details one or two beams connected to a supporting column by fin plates. If there are two beams to be connected, the principal axes must lie on a common vertical plane. The upper flange levels of beams must be common.

sp450 Moment connections: Tongue plate connection - beam to column

sp450 Tongue plate connection - beam to column. This option will detail a beam to column connection using a tongue plate.

sp452 Moment connections: Direct welded connection - beam to column

sp452 Direct welded connection - beam to column. This option details one or two beams directly welded to a supporting column. The principal axes of all members must lie on a common vertical plane. If two beams are to be connected then the upper flange levels must be common.

sp454 Moment connections: Tee connections - beam to column

sp454 Tee connection - beam to column. This option will detail a beam to column connection using Tees at top and bottom of the beam.

sp458 Moment connections: Beam stub connection

sp458 Beam stub connection. This option will detail a beam stub connection.

sp460 Beam over beam connection

sp460 Beam over beam connection. This option will detail a beam over another beam connection.

sp461 End plate connection - beam to beam

sp461 End plate connection - beam to beam. This option will detail a beam with an end plate connected to a supporting column.

sp462 Beam over column connection

sp462 Beam over column connection. This option will detail a beam over a column with an end plate at the head of the column.

sp463 Crank beam over column connection

sp463 Crank beam over column connection. This option will detail a crank beam over a column with plates at the head of the column.

sp464 Diagrammatic elevation of box frame

sp464 Diagrammatic elevation of box frame. This option details a box frame in elevation. Use this proforma in conjunction with SPADE proformas 462 and 463 to produce details A & B.

sp490 Splices: Beam splice

sp490 Beam splice. This option will detail flange and web connecting plates for a beam splice.

sp492 Splices: Column splice

sp492 Column splice. This option will detail flange and web connecting plates for a column splice.

sp510 Column base plate

sp510 Column base plate. This option will detail a column base plate and holding down bolts.

sp512 Column base - welded gusset column base plate

sp512 Welded gusset column base plate. This option will detail a welded gusset column base plate and holding down bolts.

sp540 Shop detail - non skew beam - drilled

sp540 Shop details - non skew beam - drilled. This option will produce a shop detail for a non skew beam with groups of holes along its length and ends, and optionally top and bottom end notches.

sp550 Shop detail - column - drilled

sp550 Column shop details (drilled). The proforma will draw shop details for UB and UC sections. Columns could extend over 1 or 2 storeys provided even number of bolts at each level for flange bolt holes and web bolt holes are used.

sp580 Beam and column fire casing: timber framing - up to 1 hour

sp580 Fire encasement (timber framing - up to 1 hour). This option will detail beam and column fire casings with timber framing for up to a 1 hour fire resistance.

sp582 Beam and column fire casing: steel angle framing - up to 2 hrs

sp582 Fire encasement (steel angle framing - up to 2 hours). This option will detail beam and column fire casings with steel angle framing for up to 2 hours fire resistance.

sp590 Location drawing: single bay portal frame

sp590 Location drawing - single bay portal frame. This option will produce a location drawing for a double pitched portal frame, showing drawing reference numbers for eaves, apex and base plate connections.

sp592 Location drawing: multi-storey frame

sp592 Location drawing - multi-storey frame. This option will draw an elevation for a multi-storey frame, showing section sizes for beams and columns, together with other reference information.

sp605 Raft foundation: wide toe

sp605 Raft Foundation - wide toe. This option will detail a wide toe foundation with reference to Figure 5 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp608 Raft foundation: deep edge beam

sp608 Raft foundation - deep edge beam. This option will detail a deep edge beam to a raft foundation with reference to 'Structural design of masonry' by Andrew Orton Figure B2.11.

sp610 Raft foundation: plain edge detail

sp610 Raft foundation - plain edge detail. This option will detail a plain edge to a raft foundation with reference to Figure 6 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp612 Raft foundation: plain internal wall support

sp612 Raft foundation - plain internal wall support detail. This option will detail a plain internal wall support with reference to Figure 6 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp613 Bored pile foundation to resist uplift

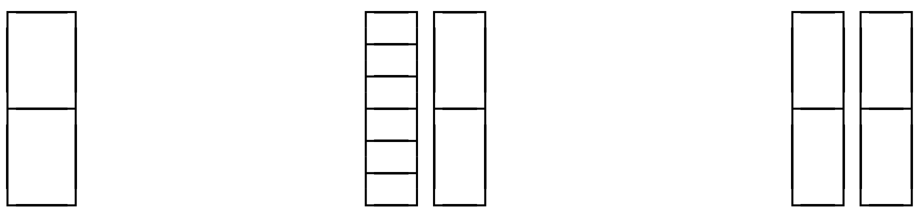
sp613 Bored pile foundation to resist uplift. This option will detail a bored pile foundation to resist uplift in swelling clay conditions with reference to Figure 11 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp614 Bored pile foundation

sp614 Bored pile foundation. This option will detail a bored pile foundation with reference to Figure 8 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp615 Masonry walls

sp615 Masonry walls. This option will detail masonry walls of the following types:



solid block   brick/block with cavity   block with cavity

sp616 Trench fill foundation

sp616 Trench fill foundation. This option will detail a trench fill foundation with reference to 'Structural design of masonry' by Andrew Orton Figure B2.7.

sp618 Traditional strip footing with concrete floor

sp618 Traditional strip foundation with concrete floor. This option will detail a traditional strip foundation with reference to Figure 1(a) of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp619 Traditional strip footing with suspended timber floor

sp619 Traditional strip foundation with suspended timber floor. This option will detail a traditional strip foundation with reference to Figure 1(b) of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp620 Wide strip foundation

sp620 Wide strip foundation. This option will detail a wide strip foundation with reference to Figure 2 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp622 Pad and stem foundation for loose fill

sp622 Pad and stem foundation for loose fill. This option will detail pad and stem foundations for houses sited on loose fill or soft compressible soils with reference to Figure 10 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp624 Precast driven segmental pile foundation

sp624 Precast driven segmental pile foundations for loose fill soft soils. This option will detail precast driven segmental pile foundations for houses sited on loose fill, soft compressible soils and open heavy clay with reference to Figure 9 of 'Foundations for low-rise buildings' by MJ Tomlinson, R Driscoll & JB Burland, published in 'The Structural Engineer' June 1978.

sp630 Granular layer beneath slab venting through trench

sp630 Granular layer beneath slab venting through trench. This option will detail a granular layer beneath slab venting through trench foundation with reference to Figure 6(a) of 'Foundations for low-rise buildings' by RMC Driscoll, MS Crilly & AP Butcher, published in 'The Structural Engineer' June 1996.

sp632 Granular layer beneath slab venting through trench with riser

sp632 Granular layer beneath slab venting through trench with riser. This option will detail a granular layer beneath slab venting through trench with riser foundation with reference to Figure 6(b) of 'Foundations for low-rise buildings' by RMC Driscoll, MS Crilly & AP Butcher, published in 'The Structural Engineer' June 1996.

sp634 Granular layer beneath slab venting through slotted pipe&riser

sp634 Granular layer beneath slab venting through slotted pipe with riser. This option will detail a granular layer beneath slab venting through slotted pipe with riser foundation with reference to Figure 6(c) of 'Foundations for low-rise buildings' by RMC Driscoll, MS Crilly & AP Butcher, published in 'The Structural Engineer' June 1996.

sp801 Graph plotting

sp801 Graph plot program. This option will accept X and Y coordinates defining a function and plot the shape of the function.

9.18 NL-STRESS - Run NL-STRESS, the plot program and the GUI directly

sc677 Elimination of expressions from an NL-STRESS data file

sc677 Elimination of expressions from NL-STRESS data file.
This option converts an NL-STRESS data file containing expressions or assignments into one in which the expressions or assignments are replaced by their numerical values. The file is converted, and a backup of the original file is saved with a .bak extension.
NOTE: If there are two or more spaces following the variable or expression, then SCALE simply overprints. This means either leave just one space between data items or - if you are preparing your data in columns - leave sufficient room in each field to print the computed value.

9.19 NL-STRESS - SPACE FRAME proformas

sc800 Circular concrete tank, internal fluid pressure, on springs

```
║                              ║
║ No. of base & wall rings.    ║
║ Dia. of 'tyre' & of hub.     ║
║                              ║
╚══════════════════════════════╝
├───dim────┼──dhb──┼───dim────┤
├────────────dty──────────────┤
```

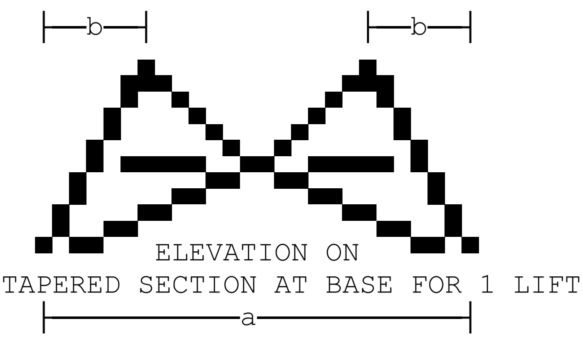
sc800 Circular concrete tank with internal fluid pressure supported on springs to model for soil stiffness.
Proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc801 Multi-storey frame, nx/nz bays along X/Z axes, ny storeys up Y

```
 1 Y out  2        3  ──X   4 ─┬─
                               zs
 5         6        7       8 ─┼─
                               zs
 9        10       11      12 ─┼─
                               zs
13        14       15      16 ─┴─
 ├───xs───┼───xs───┼───xs───┤
 Z      Foundation plan
```

sc801 Multi-storey frame having nx bays along X axis, nz bays along Z axis & ny storeys up along Y axis.
Proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc802 Lattice tower, tapered lower section, optional straight upper



sc802 Lattice tower with tapered lower section & optional straight upper section.
Proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc803 Cantilever stair analysed as a space frame

sc803 Cantilever stair analysed as a space frame.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc804 Ring beam supported by vertical Tee columns

sc804 Ring beam analysed as a space frame. Ring beam supported by vertical Tee columns to show method for setting the 'beta' angle. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc805 Curved semicircular balcony member

sc805 Curved semicircular balcony member analysed as a space frame as per SCI publication P281 "Design of Curved Steel" Design Example 6 Job No. BCC 842. The curved member will be divided into 10 equal, straight elements and a udl will be applied vertically. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc806 Space structure (square on square)

sc806 Space structure - square on square
There is a variety of configurations for space structures, see 'Practical Design of Space Structures' by RG Taylor, page 803 of Third International Conference on Space Structures, edited by Dr H Nooshin, Elsevier Applied Science Publishers. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc807 Shear (flexural) centre eccentricity example

sc807 Shear (flexural) centre eccentricity example - Analysis of Structural Member Systems by Jerome JJ Connor. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc808 Conical roof, ring beam at apex & at wall with optional columns

sc808 Conical roof using ring beam at apex and ring beam at wall with optional columns beneath. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc809 Orange segment roof truss

sc809 Orange segment roof truss, these are manufactured from circular hollow sections and are becoming popular. Essentially they are formed from two flat trusses with a circular top chord and a straight common bottom chord.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc810 Spiral stair in reinforced concrete

sc810 Spiral stair in reinforced concrete formed from an initial straight flight, then curved, then a final straight.
Modern spiral stairs are formed by a single curved reinforced concrete plate (i.e. the flight is modelled as a single member).
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc811 Temporary works column outriggers for all bolted erection

sc811 Temporary works column outriggers for erection using an all-bolted construction. The proforma provides an alternative method to guys method using bolted construction where the column base plate is bolted to the column with angle cleats, bolted outriggers can temporarily support any column until it is connected into the frame.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc812 Dynamical behaviour of 3D multi-storey frame

sc812 Dynamical behaviour of 3D multi-storey frame.
The proforma calculates the natural frequency of multi-storey frames having 'nx' & 'nz' spans in the X & Z directions and 'ny' storeys.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc820 3D Portal frame structure (2-10 bays) with optional haunches

sc820 3D Portal frame structure (2-10 bays) with optional haunches.
Mono pitch, rectangular and gable roof frames could be considered with several vertical bracing options. End frames could either be moment frames or frame posts and simply supported rafters. The geometry, material properties and loading for the frame are input interactively.
The proforma offers options for: wind loading assessment & unit loads assessment. Overhead travelling crane and/or mezzanine floor are also available user options. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces and deflected shape.

9.20 NL-STRESS - PLANE FRAME proformas

sc850 Frame analysis by NL-STRESS

sc850 Plane frame analysis. The proforma creates and runs an NL-STRESS data file, by a process of question and answer and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc854 Sway stability of braced frames

sc854 Sway stability of braced frames. The proforma considers several braced frames for analysis taking into account second order effects using the "amplified moments method". The amplification factor is evaluated in STAGE 1 and then used in STAGE 2 to amplify the wind and equivalent horizontal forces. The space frame column bases and all other frame joints are assumed to be pinned throughout.

sc855 Portal frame - 1-4 bays, optional haunches/single ridge

sc855 Portal frame - 1-4 bays, optional haunches/single ridge.
The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces and deflected shape.

sc856 Cantilever or built-in beam or rafter

sc856 Cantilever or built-in beam or rafter. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc857 Mansard beam, lean-to, cranked, dogleg

sc857 Lean-to, cranked, dogleg or Mansard beams. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of bending moments, shear forces, and deflected shape.

sc858 Gangnail type of roof truss

sc858 Gang nail type of roof truss. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc859 Lattice girder

sc859 Lattice girder. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc860 Lattice portal - N or Pratt, Howe or Warren

sc860 Lattice portal - N or Pratt, Howe or Warren. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc861 Attic room roof truss

sc861 Attic room roof truss. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc862 Collar-tie & collar-and-tie roof truss

sc862 Collar-tie and collar-and-tie roof truss. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc863 Couple/couple-close truss

sc863 Couple/couple close truss. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc864 Fink roof truss

sc864 Fink roof truss. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc865 King post roof truss

sc865 King post roof truss. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc866 Queen post roof truss

sc866 Queen post roof truss. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc867 Mansard roof or Mansard portal

sc867 Mansard roof. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc868 Bents, trestles and pipe racks

sc868 Bents, trestles & pipe racks. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc869 Box culvert

sc869 Box culvert. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc870 Continuous beam with train of moving point loads

sc870 Continuous beam with train of moving point loads. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc871 Coupled shear wall

sc871 Coupled shear wall. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc872 Circular and parabolic arch

sc872 Circular arch. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc873 Multi-storey multi-bay

sc873 Multi-storey multi-bay. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc874 Continuous beam with optional cantilever/s

sc874 Continuous beam with general loading. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc875 Sub-frame - 2 to 10 bays

sc875 Sub-frame - 2 to 10 bays. The geometry, material properties and loading for the frame are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc876 Simply supported beam including shear deflection

sc876 Simply supported beam including shear deflection, carrying uniformly distributed loading. Results compare the matrix stiffness method with those using Chebyshev Polynomials; CPs via J.M. Rolfe; The Structural Engineer 07.12.2004.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc877 Continuous beam with pattern loadings

sc877 Continuous beam with pattern loadings. Continuous beam of 'nm' spans, with characteristic uniform, concentrated, linear, dead and imposed loads. Results compare support bending moments by the stiffness method with those by moment distribution.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc878 Ground beam on an elastic foundation or on elastic piles

sc878 Ground beam on an elastic foundation/piles, subjected to train of moving loads; checking of results against Hetenyi's Classical Solution, University of Michigan Press; and by builders' arithmetic.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc879 Influence lines for continuous beams

sc879 Influence lines for continuous beams. Influence lines for reaction, shear and moment for a continuous beam having 'nsp' spans. Results compare the matrix stiffness method with that of Muller-Breslau.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc880 Two rafters with tie / post & tie

sc880 Two rafters with a tie or tie & post forming a roof, subjected to udl on plan on rafters and tie and wind loads normal to the rafters. Stiffness method checked by the 'method of joints'.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc881 Roof truss, three segment rafters, Pratt internals

sc881 Roof truss, three segment rafters (Pratt/Howe/Fink/Warren internals). Ridged roof truss, with 3 segment rafters, subjected to udl on plan on rafters and tie, wind loads, and vertical loads on joints. Stiffness method checked by the 'method of joints'.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc882 Pipe tree having 2/4/6 branches

sc882 Pipe tree having two, three or six horizontal branches, including checks for: compatibility and equilibrium for each member, overall equilibrium, and check that strain energy equals external work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc883 One/two/three storey bent having vertical/raking piles

sc883 One two or three storey bent having vertical/raking piles.
The proforma deals with vertical or raking columns including checks for: compatibility and equilibrium for each member, overall equilibrium, and check that strain energy equals external work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc884 Bent, or rectangular portal frame

sc884 Bent or rectangular portal frame. Rectangular portal frame, or bent, with fixed feet, subjected to vertical and sway loads including a check of results against the column analogy method devised by Prof. Hardy Cross.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc885 Rigid pile cap with several piles

sc885 Pile cap with several piles. Pile cap subjected to vertical and sway loads, including comparison of results with the classical method in Reinforced Concrete Designers' Handbook, 5th Edition.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc886 Gable frame with inclined legs

sc886 Gable frame with inclined legs, subjected to wind normal to members, udl on plan and joint loads, including checks for: compatibility, local and overall equilibrium and that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc887 Portal frame with skew corners

sc887 Portal frame with skewed corners, subjected to wind normal to members, udl on plan and joint loads, including checks for: compatibility, local and overall equilibrium and that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc888 Trapezoidal frame

sc888 Trapezoidal frame, subjected to wind normal to members, udl on plan and joint loads, including checks for: compatibility, local and overall equilibrium and that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc889 Vierendeel girder and vierendeel frame (end verticals only)

sc889 Vierendeel girder & Vierendeel frame (with end verticals only) with udl's and point loads including checks for: compatibility and equilibrium for each member, overall equilibrium, and check that strain energy equals external work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc890 Vierendeel roof frame

sc890 Vierendeel roof frame. Pitched Vierendeel roof with udl, point & wind loads including checks for: compatibility and equilibrium for each member, overall equilibrium, and check that strain energy equals external work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc891 Multi-storey frame, equilibrium & compatibility self-check

sc891 Multi-storey frame with equilibrium & compatibility self-check. Multi-storey frame having 'nb' bays & 'ns' storeys subjected to udl & vertical & horizontal point loads including checks for: compatibility, local & overall equilibrium, & that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc892 Pierced shear walls

sc892 Pierced shear walls. Coupled/pierced shear walls subjected to horizontal wind load. Results compare the matrix stiffness method with: Analysis & Design of Pierced Shear Walls by D Magnus, Concrete Publications, 1968.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc893 Outrigged and braced outrigged frame

sc893 Outrigged frame (braced or un-braced) carrying point load on member 4 with pinned support at joint 1 and horizontal prop at joint 3 including a check of results using Castigliano's Theorem.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc894 Curved beam with in-plane loading

sc894 Curved beam with in-plane loading.
Curved beams are often used for their aesthetic appeal and for their structural efficiency when subjected to in-plane loading.
This option features trigonometric functions, and uses a label and conditional GOTO to generate a curved beam from 'nseg' straight segments. Comparison with figure 10.15 in 'Structures' by R.Bhatt, published by Longman.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc895 Bridge abutment, vertical piles and optional raking piles

sc895 Bridge abutment supported by vertical piles and optionally raking piles. The pile centres into media will be taken equal to len (i.e. the length of wall between pile centres into the media=len).
To convert line loads to point loads, multiply the line load on top of the wall by the pile centres into the media e.g. point load at joint 5 = v*len.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

9.21 NL-STRESS - PLANE GRID proformas

sc920 Plane grid analysis

sc920 Plane grid analysis. Deals with the analysis of simple plane grids. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc921 Foundation raft

sc921 Analysis of foundation rafts. Deals with the computer analysis of a foundation raft which requires the raft be modelled by a number of members joining joints which sit on springs. This foundation model is frequently referred to as a Winkler foundation after the man who introduced the concept of subgrade reaction in 1867. The number of pseudo-beams chosen, and the frequency of spring supports have a significant bearing on the solution.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading,
bending moments, shear forces, and deflected shape.

sc922 Suspended slabs/bridges

sc922 Analysis of suspended slabs and bridge decks. Deals with the computer analysis of suspended slabs and bridge decks. The method used is described in 'Recommendations on the use of grillage analysis for slab and pseudo-slab bridge decks' published by C&CA/CIRIA. The engineer's attention is also drawn to C&CA research report No.21 titled 'The use of grillage analogy for the analysis of slab and pseudo-slab bridge decks' by R.West.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading,
bending moments, shear forces, and deflected shape.

sc923 Beam curved on plan

sc923 Beam curved in plan. The geometry, material properties and loading for the beam are input interactively i.e. by question and answer. The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading,
bending moments, shear forces, and deflected shape.

sc924 Cantilever or propped-cantilever on plan

sc924 Cantilever or propped-cantilever on plan. Cantilever/propped grillage beam subjected to uniformly distributed loading and end vertical load, including checks for: compatibility, local and overall equilibrium, & that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading,
bending moments, shear forces, and deflected shape.

sc925 Circular arc cantilever on plan

sc925 Circular arc cantilever on plan. Circular-arc cantilever subjected to concentrated & distributed loads; checking of results against classical theory of Pippard & Baker,
The Analysis of Engineering Structures (3rd Edition).
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading,
bending moments, shear forces, and deflected shape.

sc926 Circular arc bow girder on plan

sc926 Circular-arc bow girder on plan, subjected to concentrated and distributed loads; checking of results against classical theory of Pippard & Baker, The Analysis of Engineering Structures (3rd Edition).
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc927 Grillage of beams, classical check

sc927 Grillage of beams, classical check. Grillage of girders and stiffeners subjected to concentrated and distributed loads; checking of results against Modern Formulas for Statics and Dynamics by Pilkey & Chang, pub. McGraw-Hill.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc928 Grillage of beams, modern check

sc928 Grillage of beams, modern check. Grillage of girders and stiffeners subjected to point loads at intersections and distributed loads including checks for: compatibility, local and overall equilibrium, and that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc929 Ground slab subjected to loading from racks and/or fork lifts

sc929 Ground slab subjected to loading from racks and/or fork lift trucks at various positions. High concentrated loads from racks on a warehouse floor compress the supporting soil beneath the racks causing sagging moments beneath the racks and hogging moments between the racks i.e. in the aisles. This mix of sagging and hogging moments means that reinforcement should be the same in the top and bottom of the slab if cracking is to be avoided.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

9.22 NL-STRESS - PLANE TRUSS proformas

sc940 Pratt through truss

sc940 Pratt through truss. N/Pratt through lattice truss, optional height taper, end diagonals in compression, subjected to udl on top and bottom chords, shared to joints. Stiffness method checked by the 'method of joints'.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc941 Pratt deck truss

sc941 Pratt deck truss. N/Pratt deck lattice truss, optional height taper, end diagonals in tension, subjected to udl on top and bottom chords, shared to joints, and sway loads. Stiffness method checked by the 'method of joints'.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc942 Howe through truss

sc942 Howe through truss with optional height taper, end diagonals in compression, subjected to udl on top and bottom chords, shared to joints. Stiffness method checked by the 'method of joints'.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc943 Howe deck truss

sc943 Howe deck truss with optional height taper, end diagonals in compression, subjected to udl on top and bottom chords, shared to joints. Stiffness method checked by the 'method of joints'.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc944 Warren through truss

sc944 Warren through truss with constant height, end diagonals in compression, subjected to udl on top and bottom chords, shared to joints. Stiffness method checked by the 'method of joints'.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc945 Warren through truss with verticals

sc945 Warren through with verticals, optional height taper, end diagonals in compression, subjected to udl on top and bottom chords, shared to joints. Stiffness method checked by the 'method of joints'.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc946 Warren deck truss

sc946 Warren deck lattice truss with optional height taper, end diagonals in tension, subjected to udl on top and bottom chords, shared to joints. Stiffness method checked by the 'method of joints'.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc947 Warren deck with verticals

sc947 Warren deck lattice truss with verticals, optional height taper, end diagonals in tension, subjected to udl on top and bottom chords, shared to joints. Stiffness method checked by the 'method of joints'.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

9.23 NL-STRESS - Plastic analysis

sc960 Plastic analysis of cantilever

sc960 Plastic analysis of cantilever I-beam subjected to uniformly distributed loading & end point load including checks for: compatibility, local & overall equilibrium, & that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc961 Plastic analysis of propped cantilever

sc961 Plastic analysis of propped cantilever I-beam subjected to udl and point loading, including checks for: compatibility, local & overall equilibrium, & that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc962 Plastic analysis of continuous beam

sc962 Plastic analysis of continuous I-beam subjected to uniformly distributed loading & point loads, including checks for: compatibility, local & overall equilibrium, & that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc963 Plastic analysis of rectangular portal

sc963 Plastic analysis of rectangular portal frame I-beam section subjected to udl & point loads, including checks for: compatibility, local and overall equilibrium, & that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc964 Plastic analysis of ridged portal

sc964 Plastic analysis of single bay ridged portal frame I-beam section subjected to udl & point loads, including checks for: compatibility, local & overall equilibrium, & that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc965 Plastic analysis of multi-bay ridged portal

sc965 Plastic analysis of multi-bay ridged portal. Single/multi-bay portal frame/s with haunches having 'nb' bays subjected to vertical and sway, including checks for: compatibility, local & overall equilibrium & that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc966 Plastic analysis of multi-storey frame

sc966 Plastic analysis of multi-storey frame subjected to udl & vertical & horizontal point loads, including checks for: compatibility, local & overall equilibrium, & that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

9.24 NL-STRESS - Stability

sc980 Cantilever beam with large displacements

sc980 Cantilever with large displacements subjected to uniformly distributed loading & end vertical loads, including checks for: compatibility, local & overall equilibrium, & that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc981 Stability of columns with various supports

sc981 Stability of columns with various supports. Sway and within-member stability of columns subjected to axial load, for various support conditions. Buckling load for various support conditions compared to Formulas for Stress & Strain by Roark.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc982 Stability of circular ring/pipe

sc982 Stability of circular ring/pipe. Within-member stability of circular ring or pipe subjected to uniform external radial pressure. Buckling load compared to that given by Roark in Formulas for Stress & Strain, Fourth Edition.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc983 Stability of cantilever with udl & end load

sc983 Stability of cantilever with udl & end load. Cantilever beam subjected to uniformly distributed loading & end point loads. Stability analysis checked by audit of strain energy & external work, equilibrium and compatibility by classical theory.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc984 Multi-storey frame using non-linear elastic analysis

sc984 Multi-storey frame using non-linear elastic analysis, subjected to udl & vertical & horizontal point loads including checks for: compatibility, local & overall equilibrium, & that strain energy equals work done.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc985 Hanging cable with flexible platform

sc985 Hanging cable with flexible platform. Cable of negligible weight carries by suspension rods a flexible platform with a udl. Compared to The Analysis of Engineering Structures by Pippard & Baker, Third Edition, 1957.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc986 Suspension bridge with three pinned stiffening girder

sc986 Suspension bridge with three-pinned stiffening girder. Results compare the matrix stiffness method with The Analysis of Engineering Structures, Pippard & Baker, Third Edition, 1957.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

sc987 Suspension bridge with two pinned stiffening girder

sc987 Suspension bridge with two-pinned stiffening girder. Results compare the matrix stiffness method with The Analysis of Engineering Structures, Pippard & Baker, Third Edition, 1957.
The proforma creates and runs an NL-STRESS data file, and then presents the results and plots of applied loading, bending moments, shear forces, and deflected shape.

10. PRAXIS Reference Manual

10.1 PRAXIS Reference Manual

PRAXIS is a simple 'notation' particularly suited to making structural engineers' calculations comprising formulae and table look up. This manual describes in detail the facilities available in PRAXIS for composing proformas and running them on the computer.

The use of the word 'notation' rather than 'language' is deliberate as simplicity has been achieved by making the program input, workings, and format of the output, one readable text file - given a filename ending with the extension .PRO (standing for PROgram or PROforma). The language in which the programs/proformas are written is English interspersed with the PRAXIS notation, which may include for example the programming structures: IF-ELSE-ENDIF, REPEAT-UNTIL-ENDREPEAT, DEFINE-ENDDEFINE.

There are many programs written to decide upon dimensions of beams, areas of reinforcement, location of bars in a cross section, curtailment of bars, and so on. Most such programs print schedules or plot working drawings or both; few go on to print calculations which justify the numbers in the schedules or the dimensions on the drawings. So the designer must use pen and calculator to draw up a set of calculations in a form suitable for the Authority to check. PRAXIS has been devised to solve this problem by reproducing "proforma" calculations specified by the designer in a readable and checkable format.

Codes of practice specify formulae for computing values. But where values are not derivable by formulae, look-up tables are provided. Such tables may be copied from the code of practice and stored in the proforma for reference during a calculation. PRAXIS has a facility for looking up entries in stored tables; interpolating those that would fall somewhere between the precise values tabulated.

10.2 Illustrations

An example of calculations produced by PRAXIS is shown in Figure 10.1. To produce a set of calculations PRAXIS needs a "job data file" as illustrated in Figure 10.2 and a "proforma file" as illustrated in figure 10.3. The rest of this section takes the reader through the calculations illustrated in figures 1 to 3 and introduces important concepts by example. The proforma file includes special characters e.g. suppress the output of lines.

```
Knight (CMG) Ltd.                                          Page:  1
Unit 42, Castle Business Park, Marblewick.              Made by:  IFB
Job: New Civic Centre,                                     Date:  07.07.15
     Unit J, East Ancillary Building.                    Ref No:  JEAB7
_______________________________________________________________________

Location: Bent A1-3

Natural frequency of multi-storey frames
________________________________________

For frames in which the mass can be assumed to be lumped at floor
levels, and where compression of columns and rotation of column heads
may be considered negligible in comparison to sway effects, the period
of the fundamental mode is given by:

             _______________
            | sigma(mi.xi^2)       Raleigh's formula
  T = 2.PI  | _____________
           \| g.sigma(mi.xi)

Where:  T is the period of the fundamental mode

        mi and xi are the i'th mass and its displacement for
        the set of masses m1,m2,... ,mn and corresponding
        static displacements under gravity forces
        x1,x2,... ,xn

Number of levels                  n=2
Level i=1
Lumped mass at this level         M(1)=24.91
Corresponding displacement        X(1)=0.017502
Running total of m.x product      sigmx=sigmx+M(i)*X(i)
                                       =0+24.91*0.017502
                                       =0.43597
Running total of m.x^2            sigmx2=sigmx2+M(i)*X(i)^2
                                        =0+24.91*0.017502^2
                                        =0.0076304
Level i=2
Lumped mass at this level         M(2)=23.216
Corresponding displacement        X(2)=0.079553
Running total of m.x product      sigmx=sigmx+M(i)*X(i)
                                       =0.43597+23.216*0.079553
                                       =2.2829
Running total of m.x^2            sigmx2=sigmx2+M(i)*X(i)^2
                                        =0.0076304+23.216*0.079553^2
                                        =0.15456
Period                            T=2*PI*SQR(sigmx2/(32.2*sigmx))
                                   =2*3.1416*SQR(0.15456/(32.2*2.2829))
                                   =0.2881 sec
Natural frequency                 f=1/T=1/0.2881=3.471 Hz
```

Figure 10.1 - Calculations

```
TITLE Knight (CMG) Ltd.
TITLE Unit 42, Castle Business Park, Marblewick.
TITLE Job: New Civic Centre,
TITLE      Unit J, East Ancillary Building.
MADEBY IFB
DATE 07.07.15
REFNO JEAB7
```

Figure 10.2 - Job data file

```
START
Natural frequency of multi-storey frames
________________________________________

For frames in which the mass can be assumed to be lumped at floor
levels, and where compression of columns and rotation of column heads
may be considered negligible in comparison to sway effects, the period
of the fundamental mode is given by:

              _______________
             | sigma(mi.xi^2)        Raleigh's formula
  T = 2.PI   | --------------
            \| g.sigma(mi.xi)

Where:  T is the period of the fundamental mode

        mi and xi are the i'th mass and its displacement for
        the set of masses m1,m2,... ,mn and corresponding
        static displacements under gravity forces
        x1,x2,... ,xn

Number of levels                  +n=????
IF n>50
Number of levels exceeds 50.
STOP
ENDIF
! Initialise variables            +i=0
!                                 +sigmx=0
!                                 +sigmx2=0
REPEAT
!                                 +i=i+1
Level i= +i
Lumped mass at this level         +M(i)=????
Corresponding displacement        +X(i)=????
Running total of m.x product      +sigmx=sigmx+M(i)*X(i)
Running total of m.x^2            +sigmx2=sigmx2+M(i)*X(i)^2
UNTIL i=n
ENDREPEAT
Period                            +T=2*PI*SQR(sigmx2/(9.81*sigmx)) sec
Natural frequency                 +f=1/T Hz
STOP
FINISH
```

Figure 10.3 - Proforma file

10.2.1 Essential steps in running PRAXIS

The first step in using PRAXIS is to decide how to identify each calculation page (firm's name, job or contract reference number, date etc.) and type in the filename of the "job data file" e.g. C702.DAT followed by the job details. A listing of a typical job data file is reproduced in Figure 10.2.

The second step is to compose a calculation such as that shown in Figure 10.3. This is typed at the keyboard of the computer to create a "proforma file". If there is already a suitable proforma calculation on the disk then this step is unnecessary. The proforma file is typed using any text editor or word-processor in "non-document mode" i.e. as a plain text file without special control characters such as Escape sequences for font selection or printer control.

The third step is to run PRAXIS, nominating both a job data file and proforma file. From these two files PRAXIS creates a third file, the "calculations file", on the disk.

The fourth step is to print the calculations file. The printout comprises a well laid-out set of calculations, each page numbered and headed as illustrated in Figure 10.1.

10.2.2 What PRAXIS is designed to do

Comparison of the calculation file with the proforma file (Figure 10.1 with Figure 10.3) demonstrates what the program is designed to do. It is designed to copy the proforma file to the calculations file line by line, prompting and accepting values to replace ???? in the proforma file, resolving equations and printing numerical results on the way, suppressing information not required to be printed.

In the simplest case a line is copied from the calculation file without alteration; for example the line beginning "Where: T is the period...". An exclamation mark is not copied, nor is the remainder of that line; for example the line beginning "! Initialise variables..." is not copied at all because everything follows the exclamation mark. Suppression does not mean the line is ignored; in the above example the assignment +i=0 is still carried out.

Plus signs introduce names of "variables". A variable may be thought of as a named box which contains a number. As each variable is copied to the results file the number in the box is substituted for the name of the box. Compare the line beginning "Natural frequency" in the proforma file with the line beginning "Natural frequency" in the calculations file. There are more examples of substitution in Figure 10.1.

The equals sign in +f=1/T causes the result of 1/T (3.471 in this example) to be put into the variable named f. This is called "assignment" and is a fundamental concept.

The line containing 'IF n>50' in Figure 10.3 is not copied to the results file. (IF is a "keyword". Lines beginning with keywords - which are always in capital letters - are not copied but used to control what the program does next.) IF introduces a "condition". In this example the condition is IF n>50. According to the outcome of n>50 the program decides whether to copy the lines between IF and ENDIF.

To find the outcome of n>50 the program compares the number stored in the variable named n with 50. If the number in n is greater than the value 50 (in other words the condition is true) then the program copies the line beginning "Number of levels exceeds..."; otherwise the line beginning "Level i=..."

REPEAT in the proforma signifies that lines between REPEAT and ENDREPEAT are to be copied over and over again until some condition is met. In Figure 10.3 the condition is UNTIL i=n; in other words until the number stored in variable i is equal to that in n. When (or if) this condition is found to be true the program stops repeating and goes on to deal with the line following ENDREPEAT.

In Figure 10.3 the last line says FINISH, which marks the end of the proforma file. It also tells the program to stop work.

Pages of calculations are numbered automatically and have the same title block in which names, titles and dates are copied from the job data file.

PRAXIS has many facilities not demonstrated in the examples above but all are defined in subsequent sections.

10.3 Definitions

In the previous section phrases such as "name of a variable" are used and examples given; for example f and T are "names of variables". But could U235 or As of A's be used as the name of a variable too. This section answers such questions by defining essential terms - such as "numbers" and "names" - more thoroughly than in section 10.2.

10.3.1 Numbers

Numbers may be typed in the conventional way. The following examples need no further explanation:

```
25   25.00   -9.25   +0.2500   0.000
```

Numbers may also be typed in exponent form. The letter E says "Times ten to the power of...". For example the first example below means five million; the second means a five-millionth:

```
5E6    5E-6
```

The number before the E may be in any of the forms shown as examples above; the value after the E must be a whole number and may be preceded by a plus or minus sign.

Numbers (other than zero) may not be given if smaller in absolute magnitude than 1.0E-40 or -1.0E-40. Numbers may not be given if greater in absolute magnitude than 1.0E+40 or -1.0E+40.

The form of each number in the results is determined by the program. In general, whole numbers are printed without a decimal point, numbers with a fractional part to five significant figures, very large and small numbers are printed in E-form with the number after E always a multiple of three.

10.3.2 Names

Examples of names (more correctly "symbolic names") are:

```
f'c   fy   d'   d   As   A's   U235
```

Symbolic names are used to name variables and procedures.

A symbolic name starts with a letter. It may be one character long or longer. Characters after the initial letter may be letters, digits, apostrophes, or a mixture of these. A capital letter is distinct from its corresponding lower-case letter; thus A's and A'S are distinct symbolic names, so are X and x.

Any number of characters may be used to compose a symbolic name but the program ignores those after the sixth; Epsilon6 and Epsilon7 would be treated as the same symbolic name, Epsilo.

10.3.3 Keywords

Keywords are names. They are used in the proforma file to control what the program should do when the keyword is encountered. STOP is a self-evident example.

The keywords of PRAXIS are: ABS, ACS, AND, APR, ASC, ASN, ATN, BATCH, CMD, COS, CSH, DATE, DE0-DE3, DEFINE, DEG, DFR, ELSE, ENDDEFINE, ENDIF, ENDREPEAT, EXIST, EXP, FILE, FINISH, GOTO, HEADING, IF, INT, LINE, LOG, MADEBY, MMI, MULTI, NRESP, OFFICE, OPTION, OR, PI, RAD, REFNO, REPEAT, RTA?, SGN, SIN, SKIP, SNH, SQR, STOP, STORE, STRUCTURE, TABLE, TAN, THEN, TNH, TRACE, UNTIL, VER, WIN.

A name invented for a variable should not be the same as any keyword. For example a variable may not be named ELSE. However, a variable may be named ELSETS or Else or just else. Names are "the same" only if their first six characters match precisely; REPEATER is considered "the same" as REPEAT. Although names invented for a variable may exceed six characters in length, they must not exceed twelve.

Several keywords have special behaviour: PI is the name of a variable which, at the start of the program, is automatically made to contain 3.14159265358979324. PI is intended for use in assignments such as: +area=PI*radius^2

VER is the name of a variable which holds the value 2 for SCALE version 4 on Windows XP onwards, and the value 5 for SCALE version 5.

MULTI is the name of a variable which initially holds the value 1. MULTI may also be reset within a proforma to pass information to SCALE. For example if reset to 2 (by +MULTI=2 in a proforma) at the time WIN is read, then it closes the .CAL file before invoking the program which follows WIN or the command which follows CMD. MULTI is always the third variable on the stack (after PI & VER) thus it is simple for SCALE to test its value for modification of the behaviour of any functions. Setting MULTI=3 causes the current line to be displayed together with any error message (proforma SC677 gives an example). In response to 'Start page number' # may be given to suppress the page heading. This may also be done by setting +MULTI=4 within a proforma.

HEADING is the name of a variable which holds the value 1 if the .DAT file was a page heading file, or zero if the file was an NL-STRESS data file. This variable is used in SC560-SC600.

LINE is the name of a variable which holds the current line number on the current output page.

NRESP is the name of a variable which holds the number of responses passed from NL-STRESS to SCALE.

OFFICE is the name of a variable which holds the office/licence number of the user.

OPTION is the number of the option (or proforma) being run.

10.3.4 Functions

Examples of functions are:

INT(a+b)   SIN(2*PI+x)   EXP(x)

A function is a keyword followed immediately by an expression in brackets. The expression in brackets is called the "argument" of the function.

There may be no spaces anywhere in a function.

When a function is encountered its argument is evaluated and transformed to return a single value in place of the function; for example INT(2*3.4) returns 6 which is the integral part of the argument 6.8. Because expressions may contain functions it is possible to have functions of functions; thus SIN(RAD(30)) returns 0.5 because RAD(30) returns 0.5236 - the number of radians in 30 degrees - then SIN(0.5236) returns 0.5.

The keywords of all available functions are listed below together with an explanation of what each function returns.

First the arithmetic functions:

ABS Absolute value. ABS(2.5) and ABS(-2.5) both return 2.5, ABS(0) returns 0

APR Approximate match to unity. APR(.99) returns 0.99, APR(.999999) returns 1. This function is for particular use in comparing two values say a & b thus: IF APR(a/b)=1 ...

INT Integral part by truncation of the absolute value. INT(2.9) returns 2, INT(-2.9) returns -2, INT(0) returns 0

DE0 DEcimal rounding to 0 decimal places. DE0(2.9) returns 3, DE0(2.3) returns 2, DE0(-2.9) returns -3, DE0(-2.3) returns -2.

DE1 DEcimal rounding to 1 decimal places. DE1(2.95) returns 3.0, DE1(2.35) returns 2.4, DE1(-2.35) returns -2.3.
DE2-DE3 similar to above for rounding to 2-3 decimal places.

DFR Decimal FRaction. DFR(3.235) returns 0.235, DFR(3) returns 0, DFR(-6.2) returns -0.2.

SGN Signum. Returns 1 if the argument is positive, -1 if negative, 0 if zero. SGN(0.01) returns 1, SGN(-270) returns -1.

LOG Natural (base e) logarithm. LOG(1.0) returns 0, LOG(2.718282) returns 1. LOG(0) or LOG(-1) provokes an error message. To convert between LOGe & LOG10 use: LOG10(e) =1/LOGe(10) =0.4342945 thus LOG10(2) =LOGe(2)*0.4342945 =0.69315*0.4342945 =0.30103

EXP Natural antilogarithm (e to the power of ...). EXP(0) returns 1, EXP(1) returns 2.718282, EXP(-1) returns 0.3678794 (i.e. 1/e)
To convert between EXP & ANTilog10 reverse above for LOG:
thus ALG10(0.30103) =EXP(0.30103/0.4342945) =EXP(0.69315) =2.

SQR Square root. SQR(16) returns 4, SQR(0) returns 0, SQR(-16) provokes an error message.

Next the trigonometric functions:

DEG The argument is an angle in radians; the function returns the value of the angle in degrees. DEG(PI) returns 180, DEG(-1) returns -57.29578, DEG(0) returns 0

RAD The argument is an angle in degrees; the function returns the value of the angle in radians. RAD(180) returns 3.141593, RAD(57.29578) returns 1

SIN The sine of an angle measured in radians. SIN(-PI/6) returns -0.5, SIN(0) returns 0

ASN Arcsine; "The angle whose sine is..." ASN(-0.5) returns -.5235988, ASN(0) returns 0

COS The cosine of an angle measured in radians, COS(-PI/6) returns 0.8660254, COS(0) returns 1, COS(PI) returns -1

ACS Arccosine; "The angle whose cosine is..." ACS(1) returns 0, ACS(-1) returns 3.141593

TAN The tangent of an angle measured in radians. TAN(0) returns 0, TAN(PI/4) returns 1

ATN Arctangent; "The angle whose tangent is..." ATN(0) returns 0, ATN(1E20) returns 1.5708 (very nearly PI/2), ATN(-1) returns -.7853982

Next the hyperbolic functions:

SNH Sinh; the hyperbolic sine of argument x, or (e^x-e^-x)/2

CSH Cosh; the hyperbolic cosine of argument x, or (e^x+e^-x)/2

TNH Tanh; the hyperbolic tangent of argument x, or SNH(x)/CSH(x)

On occasion it is necessary to assign a sequence of subscripted variables e.g. +a(12)=3.2 +a(13)=b +a(14)=-5.7

As an alternative to the above PRAXIS has a VEC function, VEC is short for VECtor, e.g. +a12=VEC(3.2,b,-5.7). In a strict mathematical sense a set of value is not necessarily a vector, but in a programming sense the term vector is used to describe a one dimensional array. For the above example, a(12) is assigned the first value =3.2, a(13) the second =b, a(14) the third =-5.7. Each data item within the VEC() function, must be a single non-subscripted variable, or a single number prefixed with an optional minus sign. Negative decimal numbers less than 1, should have a leading zero before the decimal point e.g. b1=VEC(127,-0.45,tot).

Generally the VEC function is used as a replacement for a TABLE of values when interpolation is not required. When the VEC function is to be printed in the calculations, a minor rearrangement takes place to make interpretation more intuitive e.g. +a(1)=VEC(1,2,3,4,5,6)

```
                              is printed as  a(1)...=(1,2,3,4,5,6)
```

where the 3 dot ellipsis implies a(2),a(3) etc.

The maximum permissible number of data items within the brackets is 25. The data items must be separated by commas and there must be no spaces between the brackets. Avoid a leading zero in front of a number e.g. avoid -023.6; but provide a leading digit in front of a decimal point e.g. avoid -.236. Examples follow:

```
                                          Line in proforma┐
+pro1=VEC(0.8,0,7,79,81356.2,0.82,10E39,a,0.00123,0.015E6)┘
 pro1...=(0.8,0,7,79,81356.2,0.82,10E39,a,0.00123,0.015E6)┐
```

```
                                      Line in calculations┘
                                                 Line in proforma┐
+pro11=VEC(-0.8,0,-7,-79,-81356.2,-0.82,-10E39,a,-0.00123,-0.015E6)┘
 pro11...=(-0.8,0,-7,-79,-81356.2,-0.82,-10E39,a,-0.00123,-0.015E6)┐
                                             Line in calculations┘
```

For regularly repeating values it is permissible to add a multiplier after the closing bracket e.g. +a12=VEC(3.2,b,-5.7)*2 which causes the assignments to be continued for a second time thus a(15)=3.2, a(16)=b, a(17)=-5.7. As a further example, assuming the variable b=200 then a(1)=VEC(24345)*b would assign a(1) thru a(200) with the value 24345.

VEC may be used for regularly repeating values which are incremented each time around. VEC(v1,v2,...vl)/n says repeat the values v1,v2... 'n' times incrementing the values by vl each time around, e.g. a1=VEC(1,1)/5 will assign a1=1 a2=2 a3=3 a4=4 a5=5.
As a further example: +asta=4 +ainc=3.5 +anum=11 followed by +a1=VEC(asta,ainc)/anum will generate: a1=4 a2=7.5 a3=11 a4=14.5 a5=18 a6=21.5 a7=25 a8=28.5 a9=32 a10=35.5 a11=39. The VEC function is for assigning 2 or more variables, thus a1=VEC(3) will be faulted.

Another example: a set of 37 bending moments may be established by 3 parameters e.g. the maximum bending moment +mbm=240 kNm, the number of lines +nol=37, the bending moment increment +bmi=-mbm/(nol-1), followed by +BM1=VEC(mbm,bmi)/nol would generate bending moments:

  240,233.33,226.67,220,213.33,206.67,200,193.33,186.67,180,173.33,
  166.67,160,153.33,146.67,140,133.33,126.67,120,113.33,106.67,100,
  93.333,86.667,80,73.333,66.667,60,53.333,46.667,40,33.333,26.667,
  20,13.333,6.6667,0 on a falling scale

or the maximum bending moment +mbm=240 kNm, the number of lines +nol=37, the bending moment increment +bmi=mbm/(nol-1), followed by +BM1=VEC(0,bmi)/nol would generate bending moments:

  0,6.6667,13.333,20,26.667,33.333,40,46.667,53.333,60,66.667,
  73.333,80,86.667,93.333,100,106.67,113.33,120,126.67,133.33,
  140,146.67,153.33,160,166.67,173.33,180,186.67,193.33,200,
  206.67,213.33,220,226.67,233.33,240 on a rising scale.

String expressions provide another function capability. Supposing a particular calculation requires the frequent computation of double the current value of a variable 'a' plus half the current value of a variable 'b'. This may be achieved by setting a string expression

e.g. +$20=+2*a+b/2 which may be used in any subsequent expression

e.g. +$20 which is replaced by +2*a+b/2 and then evaluated, or used in an assignment

e.g. +x= +$20 which after substitution for $20 becomes

+x=+2*a+b/2 and the value of 'x' computed. For an assignment the plus sign in front of the 2 may be omitted.

In the above example the value of x is computed as a function of the current values of the variables a and b. It is sometimes desirable to have a function call in which the relationship between the variables x and a and b in the above example is expressed parametrically. This may also be achieved by defining the function as a procedure within the DEFINE-ENDDEFINE program structure and giving 'arguments' when invoking the procedure as described in 10.6.11. The example given above could be recast as an invocation:

double a b x

```
where the procedure double is:   DEFINE double
                                 ! +dou3=2*dou1+dou2/2
                                 ENDDEFINE
```

```
As described in section 10.6.11 the value 'a' is passed to 'dou1'
                                          'b' ........... 'dou2'
and the computed value for 'dou3' is returned as the value 'x'.
```

Right adjustment using the RTA keyword (see 10.6.28) is a formatting device rather than a function, and therefore not included here.

10.3.5 Expressions

Expressions define what arithmetic the program is required to do; they are used in assignments and conditions. Examples of expressions isolated from this context are:

```
+x   +26.5   +-RAD(alpha)   +2-26.5  +(x-26.5)*SIN(RAD(alpha))
```

An expression may comprise a single variable, single number, single function. But in general an expression comprises variables, numbers and functions bound together with operators and brackets.

There may be no spaces anywhere in an expression.

The operators are:

```
^        exponentiate: highest precedence

* and /  multiply and divide: of equal and middle precedence

+ and -  add and subtract: of equal and lowest precedence.
```

In the absence of brackets, operators are applied in order of precedence; from left to right where there is no relative precedence.

Expressions are evaluated as one would expect from the rules of algebra. Thus 4*3^2 means 36 not 144; 4+3/2 means 5.5 not 3.5. Overriding with brackets, (4*3)^2 means 144 and (4+3)/2 means 3.5. The brackets may be nested to any reasonable depth. A simple example of nesting is 2^(3*(1+2)) which reduces to 2^9 or 512.

Note that +6/2*2 evaluates to 6; but +6/2/2 evaluates to 1.5.

10.3.6 Conditions

Conditions are used to determine what the program must do next; they are introduced by the keywords IF, UNTIL. Examples of conditions isolated from this context are:

```
x<a   x+12.5>=RAD(alpha)   x=2   $(12)=FRED
```

A condition comprises a pair of expressions separated by any of the following comparators:

```
=   equal to
<>  not equal to
>   greater than
>=  greater than or equal to
<   less than
```

<= less than or equal to

There may be no spaces anywhere in a condition.

When a condition is encountered the two expressions are evaluated and the results compared. Comparison yields the result 'true' or 'false'. For example with x<a the program compares the numbers stored in variables x and a. If the number in x is the smaller of the two then x<a is 'true', otherwise false. Only the comparators = and <> may be used with string comparisons.

A compound condition comprises several conditions separated by the keywords AND or OR e.g. a>b AND c<d OR i>20. The compound condition returns a result 'true' or 'false' and must follow the keywords IF or UNTIL. String comparisons may be included in compound conditions. Rules for evaluation are:

- each of two conditions separated by AND must return a result 'true' for the pair of conditions to be 'true' else the pair of conditions has a result 'false'
- if either of two conditions separated by OR returns a result 'true' then the pair of conditions has a result 'true' else the pair of conditions has a result 'false'.

```
When a=1, b=2, c=3 then:  a>b AND a+b<>c OR b<2 OR b<3  returns 'true',
                  b>a AND a+b=c OR b=2 OR c=3 AND a+b<3 returns 'false'.
```

Consider the following numerical examples for which: a=5 b=2 c=4 always. Brackets may be optionally included thus:
IF (a)=(5) AND (b)=2 OR (c)=(4) ...
but brackets must not be used thus:
IF (a=5 AND b=2) OR (c=4)... are neither supported nor needed with compound conditions.

```
IF a=5 AND b=2 OR c=4   will obviously return 'true'.
IF a=4 AND b=2 OR c=4   will also return 'true' because OR c=4
                        returns true even though the first condition
                        i.e. a=4 AND b=2, returns false.
```

If the engineer meant:
IF a=4 AND b=2 OR a=4 AND c=4 then 'false' would be returned as expected. Be warned that the following left and right columns can produce different results.

```
IF a=3
IF b=4                         IF a=3 AND b=4
! +h=0                         ! +h=0
ELSE                           ELSE
! +h=1                         ! +h=1
ENDIF                          ENDIF
ENDIF
```

For the right hand column, h will always be set (to either 0 or 1) whereas for the left hand column, h will only be set if a=3. The user should therefore take special care when using compound conditions with an IF-ENDIF structure which contains an embedded ELSE.

Lines before the START of a proforma are read, actioned and discarded; such lines are considered as 'external' to the proforma which starts with the keyword START and ends with the keyword FINISH. Similarly lines read from an external file (named following the # symbol) are read, actioned and discarded and are also 'external' to the proforma. For such 'external' lines, the structured IF given in the above examples can not be used as the single line treatment will not work. To get around this problem, the single line IF, as described below, may be used as an 'external' line (and also as 'internal' line between the START and FINISH); when used as an 'external' line only the first of the two forms may be used.

The single line IF takes one of two forms:

```
IF condition [ AND|OR condition ] THEN assignment/s [ENDIF]

IF condition [ AND|OR condition ] GOTO label
```

```
e.g.  IF a<b AND a>c THEN a=20 b=12
      IF a<b AND a>c THEN $100=N.B. Hogging moment at midspan.
      IF a<b AND a>c GOTO 100
```

Note that brackets are not necessary (nor permitted) as by the rules of logic the AND acts as a bracket around Boolean expressions on either side of the AND.

A word of warning when using the single line IF with string comparisons such as: IF $100=s THEN a=b
The match for the string $100 will be the string 's THEN a=b', thus if the intention was to assign 'a' the value of 'b' when $100 contained 's', then use:

```
IF $100=s
! +a=b
ENDIF
```

Please note that it is not permissible to include anything other than assignments following a 'THEN'; unrecognised text or a subroutine call will be faulted. Several numerical assignment or a single string assignment may follow a 'THEN'; but a mix of numerical and string assignments will be faulted.

For the single line IF, the closing ENDIF is optional; the single line IF closes after the last assignment on the line (limit 80 chars) or the label after the GOTO. GOTO may also be used on its own on a line e.g. GOTO 100. In either case the label following GOTO must be a integer number in the range 1-32000, or an expression preceded by a plus sign which evaluates to an integer. The destination for the jump i.e. :100 should be on its own on a line with the : starting in the first character position, if anything follows the :100 it will be ignored.

```
IF As=0 GOTO 100
Area of compression steel required +As
Dia of compression steel required  +dia
:100
```

or alternatively:

```
IF As=0
GOTO 100
ENDIF
Area of compression steel required +As
Dia of compression steel required  +dia
:100
```

When looping is implemented by a conditional GOTO then - to stop looping indefinitely - PRAXIS checks that each line in a proforma is not read more than 128000 times, faulting when this limit is exceeded.

Experienced programmers will wonder why GOTO which is a non-structured programming device has been included in PRAXIS which is a fully structured notation. Its inclusion is to simplify the transcription of old BASIC and FORTRAN programs into PRAXIS. PRAXIS checks that the jump is legal, e.g. the following would be faulted as the jump attempts to go into the structure starting 'IF fcu<20' (were the label to follow the ENDIF then the jump would be OK).

```
IF As=0 GOTO 100
Area of compression steel required +As
Dia of compression steel required  +dia
IF fcu<20
:100
Low strength concrete has been used ....
ENDIF
```

IF EXIST returns 'true' or 'false' depending on whether the following filename exists e.g.

```
IF EXIST \sand\sc410.pro
! +m=1
ELSE
! +m=0
ENDIF
```

returns the value 'true' if the file \sand\sc410.pro exists, else returns the value 'false'. Thus if the file exists, the above example sets the variable m=1, else m=0. PRAXIS generally places a limit of 80 characters on the length of the pathname, but this limit would be reduced in the above example by the fact that every line in PRAXIS must not exceed a length of 80 characters.

10.3.7 Subscripted variables

A variable name may be subscripted e.g. fs(23)   F'(j-3)   bc(i,j)

For a singly subscripted variable, as in the first two examples above, there is no need to declare size in a dimension statement. The subscript may be an integer or single variable having a value in the range 0 to 999, or a single variable with addition/subtraction of an integer value as in the 2nd example above. In all cases the subscript must evaluate to an integer. A subscripted variable may have no more than three characters in its name (the part before the opening bracket) and there may be no spaces within or between the name or subscript.

The last example shows a doubly subscripted variable bc(i,j), for which there is a need to declare its size. Doubly subscripted arrays are assumed to be square thus only one dimension needs to be set. The dimension is set by assigning it to the array name e.g. bc=3, before the first use of the array in doubly subscripted form. Functionality is important in programming; Praxis allows subscripted variables to be used in: non-subscripted form e.g. bc6
singly subscripted form bc(a-7)
doubly subscripted form bc(i,j).
As mentioned above, it is necessary to declare the dimension of an array before its first use in doubly subscripted form. Praxis stores its elements left to right, top to bottom, thus if bc=3 then the array bc(,) contains:

$$\begin{bmatrix} bc(1,1) & bc(1,2) & bc(1,3) \\ bc(2,1) & bc(2,2) & bc(2,3) \\ bc(3,1) & bc(3,2) & bc(3,3) \end{bmatrix}$$

It follows that bc6=bc(6)=bc(2,3). One use of such functionality is that a doubly subscripted array may be assigned on a single line e.g.
bc=3 bc1=VEC(1,0,0,0,1,0,0,0,1)
which would set up a unity matrix bc(,)=

$$\begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \end{bmatrix}$$

└ 0 0 1 ┘

without the need for doubly nested loops.

As just a single dimension (for the number of columns) is set for doubly subscripted variables, the number of rows in the array may be less than, equal to, or greater than the number of columns. The array bc() above has three columns, and three rows. The width (number of columns) may be re-dimensioned as required, e.g. if bc=6 is assigned then the array bc(,) is referenced:

```
┌bc(1,1) bc(1,2) bc(1,3) bc(1,4) bc(1,5) bc(1,6)┐
└bc(2,1) bc(2,2) bc(2,3) bc(2,4) bc(2,5) bc(2,6)┘
```

i.e. two rows when previously dimensioned =3, have now been put on a single row; the order - as stated before - is always left to right, top to bottom, and has not been changed, only the referencing. In the above, the use of the symbolic name bc=6 to assign a width for the array bc() is simple and elegant, care must be taken to avoid assigning out of range values e.g. bc=-1E39. As an example in sc9240.pro the assignments zrr(i)=-1E39 zrr=zrr(i) caused a bug; the bug was corrected by using zRR(i)=-1E39 zrr=zRR(i); never set the array size by assigning the size to an element within the array i.e. avoid zrr=zrr(i).

N.B. PRAXIS treats a subscripted variable e.g. fs(23) as a non-subscripted variable by omitting the brackets i.e. fs23. Although the variable 't' will never be the same as the subscripted variable 't(n)' whatever the value of n; the variable 't12' would be identical to 't(n)' if n=12. To avoid such problems, PRAXIS swaps the first digit to the left of the '(' bracket with the characters "#%&:;_|~ in order with the digits 0 to 9; thus in the example above, PRAXIS would replace AB1() with the subscripted variable name AB#() for use while the proforma is being processed but restore the AB1() when the value is displayed or printed. The replacement character would only be visible if the stack file were viewed or edited.

When the subscript needs to be more complicated than the simple forms described above, assign the expression to a variable beforehand and then use that variable name as the subscript.

For those who are unfamiliar with double subscripts, section 10.6.11 gives an example of matrix arithmetic, when more than one subscript is required, but using just a single subscript.

10.3.8 Units

The following units are recognised by PRAXIS and will appear after the data entry field for single line prompts and multiline EDIT /W prompts:

%	'	"	/°C	/m	°	°C	με
με/°C	cm	cm2	cm3	cm4	cm6	hours	kg
kg/m	kg/m2	kg/m3	kg/mm2	km	km/h	kN	kN/m
kN/m2	kN/m3	kN/mm2	kNm	kNm/m	kNm/r	kNm2	l/s
l/s/m2	m	m/kN	m/s	m/s2	m/s3	m2	m2/MN
m3	m4	mg/l	min	mm	mm/m	mm2	mm2/m
mm2/mm	mm3	mm4	mm6	months	N	N/mm2	rd/Nmm
tonnes	years						

10.4. Titles

Each page of calculations has a title block as illustrated in figure 1. The information arranged in the title block (firm's name, date etc.) is taken from the job data file where it is introduced by descriptive keywords (STRUCTURE, DATE etc.). This section describes the composition of the title block.

10.4.1 Arrangement of titles

The keywords to introduce titles and names for the title block are: STRUCTURE, MADEBY, DATE, REFNO. The order in which the keywords appear in the job data file is not significant. The program always arranges the titles as shown in Figure 10.4. Wherever a keyword is omitted the corresponding section of the title block appears blank, except for the date.

The numbers and texts introduced by MADEBY, DATE and REFNO are arranged down the right of the title block after the legends:

```
1st line following STRUCTURE or TITLE                  Page:
2nd line following STRUCTURE or TITLE               Made by:
3rd line following STRUCTURE or TITLE                  Date:
4th line following STRUCTURE or TITLE                Ref no:
```

Figure 10.4

When keyword MADEBY, DATE or REFNO is omitted, the text after the corresponding legend is left blank as illustrated in Figure 10.4.

Any line following the keyword STRUCTURE which starts with the keyword NUL is suppressed from the page heading. If one or more lines are so suppressed then the horizontal line beneath the page heading is omitted, thus if the 2nd, 3rd & 4th line in Figure 10.4 all started with NUL then only the first line would be shown in the heading. NULL works in a similar fashion to NUL except that it causes a blank line to be printed in place of any heading line.

To replace the four line page heading with four blank lines, respond with a hash (#) to the 'Start page number' prompt; the hash will cause the contents of the page heading file to be ignored.

10.4.2 Length of titles and texts

Titles introduced by STRUCTURE should be no longer than 50 characters, those longer are truncated. The first of these is the character following the space which is obligatory after the keyword.

Texts introduced by MADEBY, DATE and REFNO should be no longer than 8 characters each, those longer are truncated. The eight characters following MADEBY give sufficient room for both the initials of the producer and those of the checker.

10.4.3 Page numbering

If the user gives a starting page number (e.g. 132) it will be printed following the word Page in the top right corner of the first page of calculations, subsequent pages being numbered 133, 134 etc.

Page numbering may commence with an UPPER CASE alpha prefix. If the user gives a starting page number such as C3/12 it will be printed following the word Page in the top right corner of the first page of the calculations, subsequent pages being numbered C3/13, C3/14 etc.

If the user gives the starting page number as 0 (zero) this will be treated as an instruction to omit all page numbering following the word Page on all pages of the results. To facilitate page numbering, a number starting from 1 for each particular run will be printed at the end of the line immediately below the heading.

If the user gives a starting page number without an integer suffix (for example CJA alone) then this will be printed following the word Page in the top right corner of the first page of the calculations and on subsequent pages. A number starting from 1 will be printed at the end of the line immediately below the heading.

10.5 Tables

The examples in 10.5.1 -10.5.3 show a table of data which could be used in a proforma file, a reference to that table (a "look-up") to obtain the value beta, and the outcome of looking up a particular value in the calculations file.

The example illustrates a table with 3 rows and 4 columns of data, but tables with a single row or column may also be included in the proforma file.

The example in 10.5.3 illustrates a reference to a table in which the arguments match both the column and row headings precisely. But a precise match is not necessary, provided that the reference lies somewhere within the range of table headings. In such cases the value to be looked up is established by linear interpolation.

The rest of this section explains the mechanism of storing tables at the start of the proforma file and referring to such tables in the proforma proper which follows the keyword START.

10.5.1 Storing and removing a table

Tables are generally stored at the start of the proforma file before the appearance of the keyword START.

The keyword for introducing data for a table is STORE. This keyword is followed by three numbers separated by spaces. An example is:

```
STORE 25 3  4
```

The three numbers signify in order:

- table number (TABLE 25 in the example below) may be any positive integer or decimal number, or single non-subscripted variable when the STORE command comes after the command START
- number of rows in the table (3 in this example)
- number of columns in the table (4 in this example).

The second line of data gives the heading for the columns. The third & subsequent lines of data are rows of the table. Each row starts with a row heading. After the row heading come tabulated numbers for that row. Items are separated from each other by spaces. For tables with too many columns to fit on a line it is permissible to continue the elements on subsequent lines providing the above implied order is maintained (i.e. the numbers come in order left to right top to bottom including column and row headers).

Column and row headers are numbers rather than names. This is to make interpolation possible.

An example of a 3 by 4 table is:

```
STORE 25 3 4
        25    30    40   100
0.25  0.28  0.28  0.28  0.28
0.50  0.40  0.44  0.44  0.44
1.00  0.52  0.56  0.60  0.60
```

Notice there are 4 column headings followed by rows of 1+4 items. The extra number is the "row heading".

A table with a solitary row has no row heading. An example with 6 columns is:

```
STORE 26 1 6
  15    20    25    30    40   100
2.30  2.68  3.00  3.28  3.80  3.80
```

A table with a solitary column has no column heading. An example with 3 rows is :

```
STORE 23 3 1
1    0.9
2    0.8
3    0.6
```

The limit to the number of table elements which may be included in a proforma is 32,000. Any table which has been stored may be removed from memory to free up space for other tables. To remove a table from memory, use the STORE command as before but give the table number prefixed by a minus, e.g. to remove table 25:

```
STORE -25 3 4
```

The numbers of rows and columns following the table number is ignored when a table is being removed from memory, the number of elements to be removed being taken from memory.

10.5.2 Overriding a table

Tables need not be numbered from 1. For example an external file (10.5.6) could be used to store TABLE 25, TABLE 26 and TABLE 23 (those used as examples above). If two tables are allocated the same table number the first is overridden by the second. It is possible to take advantage of this overriding e.g. having three different table number 30 the first for Grade 25, the second for Grade 30 and the third for Grade 40 and reading in the required table to override that already stored. The three tables must have the same structure (i.e. the number of rows and columns must be identical) otherwise PRAXIS reports an error.

10.5.3 Reference to a table

Tables are referred to from the calculation file. An example of a reference to TABLE 25 is:

```
+Su=TABLE(25,perc,Grade)
```

If variable perc contained 0.5, and variable Grade contained 25, then the above would cause a value of 0.40 to be "looked up" in TABLE 25 and assigned to variable Su (see example above). The value is found at the intersection of the row headed 0.5 and the column headed 25. On the results page the table reference would appear as:

```
Su=TABLE 25 for perc=0.5, Grade=25
  =0.40
```

An example of a reference to a table of a single row is:

```
Look up +Vs=TABLE(26,Grade) N/mm2
```

which would appear on the results page as:

```
Look up Vs=TABLE 26 for Grade=25
          =3.00 N/mm2
```

A reference to a table of a single column has precisely the same form as that above.

Every table reference has the form of an assignment to a variable as illustrated. If the table has both rows and columns the keyword TABLE takes three arguments:

- table number given as an integer or decimal number or as the name of a variable
- name of a variable or number which currently stores the row heading
- name of a variable or number which currently stores the column heading

If the table has a single row or single column the keyword TABLE takes only two arguments:

- table number given as an integer or decimal number or as the name of a variable
- name of a variable or number which currently stores the column heading or row heading.

10.5.4 Errors in table references

A table reference must have the form of an assignment to a variable. Thus the following reference would be treated as an error:

```
The value of Vs is: +TABLE(26,Grade) N/mm2 !Wrong!
```

(The correct way to make this reference is illustrated in 10.5.3 above)

It is a mistake to include spaces in a table reference - on either side of the equals sign, between TABLE and the left bracket, anywhere inside the brackets.

It is allowable, but bad practice, to present the argument (or arguments) following the number of the table as numbers rather than names of variables. For example:

```
+Su=TABLE(25,0.5,25)
```

The program then employs the standard names ROW and COLUMN when printing results. For example:

```
Su=TABLE 25 for ROW=0.5, COLUMN=25
  =0.40
```

ROW and COLUMN and not names of variables, just words added to clarify the results.

Only one table reference may be given on a line unless the line is not to be printed i.e. it starts with an ! or IF e.g.

```
! +x=TABLE(1,a(i),a(j)) +y=TABLE(2,b(i),b(j)) +c=2*b
IF a=2 THEN b=TABLE(a,2,2) c=TABLE(a,2,4) ENDIF
```

The trailing ENDIF in the line above may be omitted (see 10.3.6).

10.5.5 Interpolation from tables

The column or row heading specified in a table reference does not have to match any of those in the table itself. Where there is no precise match the program automatically interpolates a result from the table. An example of a table reference, assuming the variable named Grade contains 35, is:

```
+Vs=TABLE(26,Grade)
```

This would result in the following two lines on the results page:

```
Vs=TABLE 26 for Grade=35
  =3.54
```

Where the 3.54 is derived by linear interpolation between table headings 30 and 40. Table 26 is reproduced earlier.

An example of linear interpolation in two directions is given below. Variable perc is assumed to store 1.75.

```
+Su=TABLE(25,perc,Grade)
```

This would result in the following two lines on the calculations page:

```
Su=TABLE 25 for perc=0.75, Grade=35
  =0.51
```

as the reader may verify from Table 25 reproduced earlier.

The facility described above is for interpolation only; a table reference which would require extrapolation is treated as an error. However, the need for extrapolation may be avoided by adding extra rows or columns to the table.

10.5.6 External tables and procedures

Normally tables are placed at the beginning of a proforma before the command START. Alternatively tables may be placed in a separate file and the filename given following an @ sign. The @, which must be the first character in a line, tells the program to open the file whose name follows and read any tables found in it; e.g. @SC400.PRO tells the program to open the proforma file SC400.PRO and read any tables from it into memory.

This device permits commonly used tables such as section properties for UB's and UC's to be saved in a separate file and so avoid duplication by having to include them in the several proformas which use them. The separate file should not contain keywords such as START and FINISH but may contain assignments as described in section 10.6.15.

In a similar manner, regularly used procedures may be placed in external files and the name of the file given at the end of the proforma following the @ sign e.g. @SP001.PRO includes the procedures contained in the library SP001.PRO. Remember that procedures must be at the end of the proforma and a STOP is needed before the first DEFINE is found.

If an external procedure e.g. @KICKER is not found, it is ignored and deliberately not reported so that optionally KICKER may be supplied at a later date, or created by the proforma itself to give a pseudo neural ability.

Again in a similar manner, regularly used sections of code may be placed in files and the name of one of the files given on a line of the proforma following the # sign e.g. #H287.DTL includes the code contained in the LUCID overlay H287 (which details part of the support reinforcement) or #$(i) where the string variable $(i) contains the filename. Assuming $(i) contains 'H245.DTL' then PRAXIS will open the file H245.DTL - if it exists. The code held in a proforma so nominated may contain calls to subroutines but it must not contain any program structures (IF-ELSE-ENDIF, REPEAT-UNTIL-ENDREPEAT DEFINE-ENDDEFINE).

The maximum number of values which can be stored in tables held in memory is 32,000, this includes the row & column headers e.g. for a table having 315 columns and 100 rows, the number of values to be stored:

```
                    ┌────┬headers
            315*100 +315 +100  = 31915 values
```

which is just inside the memory limit of 32,000 currently set in PRAXIS.

Experienced programmers know about the power of tables for simplifying lengthy sections of logic. The program below is an example of reading tables from disk. In the example there are 10 Table 1's which are read in one at a time as required and dependent on 'sref'. Each table occupies the same space as that of the previous table. Of course all the tables have different values and headers, but they must have the same structure (i.e. the number of rows & columns must be the same).

```
START
Section reference (1-10)         +sref=????
Row    (1-231)                   +row=????
Column (1-25)                    +col=????
! Next line concatenates to filename hs1.pro, hs2.pro ....
! +$32000= hs +sref .pro
#$32000
Look up                          +val=TABLE(1,row,col)
FINISH
```

In the above example there are 10 Table 1's held in files hs1.pro to hs10.pro inclusive, invoked by preceding the filename with a # sign.

As discussed above, the filename following a # in the first position on a line of a proforma, is read into the current proforma and processed. The processing may be omitted entirely by setting the number of significant digits thus: DIGITS 1 (see 10.6.8); this accuracy is interpreted by PRAXIS as an instruction to send the lines read from an external file directly to the .CAL file. In a similar manner any help file (ending in the extension .hlp) is sent to the .CAL file without processing save for page control instructions following / and indenting the help file just one character position more than that read from the file.

To read in a .CAL or .RES (NL-STRESS results) file which has already been 'paged', use the following 4 lines:

```
DIGITS 1          ! To tell the program to ignore the text.
PAGELENGTH 2000   ! To suppress further page headings.
#NLKCMG.RES       ! To read in the file e.g. NLKCMG.RES.
DIGITS 5          ! To reset processing back to normal.
```

10.5.7 Table for special characters

SCALE version 4 only, removed from SCALE version 5: Table number 32000 is reserved for specifying an ASCII number and pixel positions for any special graphics characters (for display). Each character is an 8 by 14 array of pixels; numbered left to right and top to bottom. The example below sets a bar along the top of ASCII character 128, and a bar down the left hand side of ASCII character 130. Zero finishes the set of pixels, the maximum number of columns is 60 i.e. the maximum number of pixels that may be set for any character is 58 (as the first column holds the ASCII number, and the last must be zero).

```
STORE 32000  2  16
     1   2   3   4   5   6   7   8   9  10  11  12  13  14  15  16
1  128   1   2   3   4   5   6   7   8   0   0   0   0   0   0   0
2  130   1   9  17  25  33  41  49  57  65  73  81  89  97 105   0
```

10.6. Control

The example in Figure 10.3 illustrates the use of four question marks to act as a prompt to accept data interactively, an exclamation mark to suppress copying and the use of a plus sign to signify the presence of a variable. These symbols control the behaviour of the program. All control mechanisms - symbols and structures - are described in this section.

10.6.1 Exclamation mark

In general the action of the program is to copy each line of the calculation file to the results file. This does not apply to lines beginning with keywords; these are not copied unless the TRACE facility is in use (see 10.6.13). An exclamation mark in the proforma file suppresses copying of both the exclamation mark and of the remainder of that line.

The exclamation mark is useful for including notes and explanations in the calculation file when such notes should not be seen on the results page. It is also useful when an assignment is to be made but not set out in detail on the results page as would otherwise be the case. Use the minimum of hidden lines to avoid the possibility of PRAXIS producing a calculation which hides away an error.

The following example illustrates part of a calculation file in which all notes, and all details of the assignment, would be suppressed:

```
! Suppress details of table look-up
! +a=TABLE(25,perc,Grade)  Table shown earlier
                     Table(25) gives +a  !Nothing else disclosed
```

which would generate the following. Compare this with the example in 10.5.3

```
                     Table(25) gives 0.40
```

Note that in the above example there must be a space before the exclamation mark; were the mark to be appended to a word before, as in normal usage, then it would be displayed as normal! OK

An exclamation mark starting a line containing question marks (see 10.6.4) as a prompt for input data is displayed on the screen, but the users response is neither echoed to the screen, nor sent to the calculations. A similar line starting with a double exclamation mark causes PRAXIS to take any preset value from the stack without waiting for a response from the user. This feature is used in LU910.PRO when picking up bar schedule data written to LU910.STK by one of the detailing proformas.

For procedures which are called thousands of times and which contain many comments, the comments may be omitted from compilation by the methods described in 10.6.25. For example the line

```
! Suppress details of table look-up
```

would be ignored entirely if +KLBR=0 was set at the start of the proforma and the line commenced with left braces thus:

```
{ Suppress details of table look-up
```

10.6.2 Percentage sign

When PRAXIS finds a percentage sign in the first character position on a line in the proforma, the line is displayed on the screen but not copied to the calculations file. The percentage sign is useful for telling the user about the data required without the explanation being copied to the calculations file. The default background colour for such a line is green, to change it to the normal default colour i.e. blue, start the line with two percentage signs. Lines in LUCID and SPADE proformas which are not to be sent to the .CAL file for the main part and rather then displaying them as green they are displayed in the normal (default blue) screen colour. If the user wishes to show any such line with a green background, then use the double percentage; this action would result in the converse to that for SCALE.

10.6.3 Slash sign

When PRAXIS finds a slash sign in the first character position on a line in the proforma followed immediately by a number (for example /5) PRAXIS starts a new page unless there are 5 (in this example) or more lines available on the current page. The slash sign is used to prevent headings being separated from their first paragraph or diagrams being split between pages. The / sign on its own, is interpreted as an instruction to throw a page regardless of the number of lines remaining on the current page.

A double slash // as the first and second character on a line is interpreted as an instruction to clear the screen above where the line would be displayed. A double slash //n (where n is an integer number in the range 1 to 20) is interpreted as an instruction to clear 'n' lines from the screen above where the line would be displayed. The clearance is achieved by scrolling the screen down by 'n' lines.

To provide facilities for reading and printing NL-STRESS data files PRAXIS recognises lines which start */ as a page throw; thus when diagrams have been included in a data file the file may be printed from within SCALE avoiding the diagram being half on one page and half on another. It is permissible to use an exclamation mark in place of the asterisk.

10.6.4 Question marks

A line containing question marks is copied to the output but not immediately. The presence of one or more question marks (conventionally ????) causes that line to be displayed on the screen as a prompt to the user of PRAXIS. For example, in Figure 10.3 the line reading:

```
    Number of levels                  +n=????
```

is displayed on the screen - omitting the leading plus sign - leaving the user to type 2 (say) and press the Return key. The line is then copied to the output as though it had been given as:

```
    Number of levels                  n=2
```

in the proforma. This is what turns a generalised proforma into a particularised set of calculations.

Occasionally it is desirable to use a question mark within a proforma but not prompt for data, as in the following dialogue which has been taken from option SC561.

```
                                                           Alt+255
%                                                             |
%Are all the strips pointing in the X direction of equal width ?
%
!Answer ( 1=Yes, 0=No )           +eqx=????
```

In the above example, an extra Alt+255 has been included in front of the question mark to tell the program to ignore the question mark;

when the line is displayed the Alt+255 is removed. If the line had contained an expression e.g. '%Is strip +n in X direction?' then the Alt+255 should be present before the occurrence of the first expression and in this example has been place in front of 'strip'.

10.6.5 Plus sign

In general the plus sign tells the program that the word which follows is a symbolic name - the name of a variable. This, in turn, tells the program to copy to the results page not only the name of the variable but also the number stored in that variable.

There are five possible arrangements in which the program may copy the name and content of a variable. All five cases are illustrated in the following table. Explanations follow the table.

Case	As on proforma file	As on calcs page
1	+a	67.3
	+a*b	134.6
2	+b=12.5	b=12.5
3	+c=b/2	c=b/2=12.5/2=6.25
4	+d=2*c*SIN(RAD(a))	d=2*c*SIN(RAD(a))
		=2*6.25*SIN(RAD(67.3))
		=11.5317
5	+e=TABLE(26,Grade)	e=TABLE 26 for Grade=25
		=3.00

Case 1: individual variables, no equals sign. When copying an expression comprising variables to the calculations file the program simply substitutes the appropriate numerical value.

Case 2: Assignment of a single number to a variable. There is obviously no need for further substitution; the program just removes the plus sign. Only one assignment may be contained on a line, unless the line starts with an exclamation mark (see 10.6.1).

Case 3: Assignment of a simple expression to a variable. The assignment is copied to the calculations page and the values of the variables substituted followed by an equals sign and the value of the expression following on the same line if there is sufficient space. If there is insufficient space to substitute for the variables on the first line, see Case 4 below. If there is insufficient space to get the computed value on the same line, then it is printed on the next line. Only one assignment may be contained on a line unless the line starts with a '!'.

Case 4: Assignment of a general expression to a variable. The assignment is copied to the calculations page. On the next line the expression is reproduced with numbers substituted for names of variables. On the third line is printed the value of the expression. Equals signs are aligned. Arguments of functions are not individually resolved (e.g. the value of RAD(a) in case 4). To save paper, the program checks to see if there is sufficient space to print the second and third lines at the end of the first, and if so does (see Case 3 above). When there is insufficient space to print all of the second line on the second line, then as many lines as necessary are used - breaking the line at the arithmetical operators or brackets - and lining up the start of each printed line beneath the equals sign. Only one assignment may be contained on a line unless the line starts with a '!'.

When it is important to display a value it should be recorded explicitly; for example:

```
Angle in degrees is +a              ! as case 1
Distance +d=2*c*SIN(RAD(a))         ! as case 4
```

Case 5: Table reference. The use of tables is explained in section 10.5.

The plus sign signifies a variable when:

- it starts a line or is preceded by a space, and
- it is followed by a symbolic name or number

For a solitary variable (case 1 in table) there is a further requirement, failing which an error would be reported:

- the nominated variable must already contain a number

If a plus sign is preceded by a space but followed by something not a valid name (e.g. +bea<t) then an error is reported.

A plus sign with spaces on either side of it, or a plus sign incorporated in an item, is treated in the same way as any other symbol, digit or letter. For example:

Accept + or - values. Loading D+L. Grade 40+

In an expression (which, by definition, follows an equals sign) the plus sign signifies addition - or a positive quantity - in the conventional way, For example:

```
+a=c+d+e+f
```

10.6.6 Greater than and less than signs

When PRAXIS finds a greater than sign in the first character position on a line in the proforma, the position is saved as a reference to step back to when a less than sign is given in response to ???? prompts (see 10.6.4). The < causes control to transfer back to the >. If logical structures IF-ELSE-ENDIF, DEFINE-ENDDEFINE or REPEAT-UNTIL-ENDREPEAT are all closed at the point in the proforma where the > occurs, then operation is straight forward. If one or more logical structures are open at the point in the proforma where the > occurs, then PRAXIS checks that the levels of all logical structures are the same as those obtained at the place where the < was typed - warning if not so. This warning should not be ignored, and it is recommended that further < are issued to take the proforma back to the start. To avoid such a situation, if the > is used then the proforma should be constructed so that no data is input from within a procedure; and avoiding the input of data from within other program structures if at all possible.

When PRAXIS prompts for data, the button labelled 'Undo or <' has dual usage. Clicking the button or typing 'U' causes control to be transferred to the previous prompt, if there is one. This is achieved by the program rerunning the calculation, without prompting until it gets to the previous prompt if there is one.

```
START
>
! +D=200 +B=201 +t=10 +T=11
Depth of section                  +D=???? mm
>
Breadth of section                +B=???? mm
>
Web thickness                     +t=???? mm
>
Flange thickness                  +T=???? mm
FINISH
```

For the proforma given above, when the prompt +T=???? mm is offered, clicking Undo twice, or typing '<' twice, will return control to the prompt +B=???? mm. For long calculations, clicking Undo say 10 times to get back to revise the size of a Universal Beam, may take a minute. A well written proforma which contained a '>' on the line before the prompt for the size of the UB and no '>' between that and the point at which, say the unity factor exceeded 1, would return the engineer to the prompt for the size of the beam, by typing just one '<'.

Two greater than signs '>>' given in response to any prompt or as the first two characters on a line in the proforma, are interpreted by PRAXIS as fast forward; further prompts only being issued when data has neither been provided previously nor is available from the stack file. The '>>' may also be typed in response to the 'Option number' prompt or the 'Start page' prompt. Fast forward may be cancelled by by '><', thus PRAXIS provides a means of rushing through a section in a proforma and then returning to the normal question and answer mode. Three greater than signs '>>>' given as the first three characters on a line in the proforma, are interpreted by PRAXIS as fast forward but no screen display. This special fast forward may be cancelled by '><<' given as the first three characters on a subsequent line in the proforma, thus PRAXIS provides a means of rushing through a section in a proforma, without displaying it on the screen, and then returning to the normal question and answer mode. To stop a proforma (without prompting with ????), then '<<' at the start of a line in a proforma causes buttons 'Continue Display Escape' to be invoked; '<<//' behaves similarly but also clears the screen; '<<//n' where 'n' is an integer number in the range -15 to +15 causes buttons 'Continue Display Escape' to be invoked and the screen display scrolled down 'n' lines if negative, or up 'n' lines if positive. If scrolling is required and the '<<//n' is to be followed by a prompt line, then omit the <<.

As an example of the use of '<<//n' consider the following proforma:

```
START
+$375=REGISTRATION M/331
BLUE LINE 1 OF 2
BLUE LINE 2 OF 2
%GREEN LINE 1 OF 3
%GREEN LINE 2 OF 3
%GREEN LINE 3 OF 3
BEAM MARK: +$375
<<//-4
MORE TEXT LINE 1 OF 2
MORE TEXT LINE 2 OF 2
FINISH
```

On reaching the <<//-4 the screen displays:

```
REGISTRATION M/331
BLUE LINE 1 OF 2
BLUE LINE 2 OF 2
GREEN LINE 1 OF 3
GREEN LINE 2 OF 3
GREEN LINE 3 OF 3
BEAM MARK: REGISTRATION M/331
```

The << has caused the above 7 lines of text in the screen buffer to be displayed with the mouse cursor positioned on the Continue button. On clicking Continue the //-4 causes the above 7 lines to be scrolled down 4 lines (the minus sign means down; a positive or no-sign means up) and then the program runs to the end finally displaying:

```
REGISTRATION M/331
BLUE LINE 1 OF 2
BLUE LINE 2 OF 2
MORE TEXT LINE 1 OF 2
MORE TEXT LINE 2 OF 2
```

Three less than signs <<< invokes 'Press < and Return to revise, or Return to continue'

10.6.7 Full stop

When PRAXIS finds a full stop in the first character position on a line in the proforma, the full stop is deleted and the line slid to the left by one character. The purpose of full stop is to identify the line as one which must always be sent to the calculations file, even if a Summary has been requested (see 10.6.18).

10.6.8 Digits

The introductory example shows numerical results given to five significant figures. The user requiring more or less significant figures may set the number by:

```
DIGITS n
```

where n is an integer number in the range 0 to 15 (e.g. DIGITS 8) or a single non-subscripted variable in the same range which is preceded by a plus.

The DIGITS command may come anywhere in the proforma after the START. The DIGITS command may be included as often as necessary to control the printed number of significant figures; but in all occurrences it must be placed as the first word on its own line.

Whatever the number of digits specified, PRAXIS does all its arithmetic to 15+ decimal digits of precision, thus the DIGITS command only affects the printed calculation, not the arithmetical accuracy.

'DIGITS 1' has special usage when reading external files (see 10.5.6) where it causes lines to be sent to the output without doing any processing. When DIGITS 1 has been set, then lines containing Alt+255 followed immediately by a question mark do not cause a prompt, the Alt+255 stops the prompt, the DIGITS 1 stops the Alt+255 being filtered out.

'DIGITS 0' has special usage to clear the response stack.
Input data provided by the engineer in response to a prompt (e.g. a=????) for a numerical value, is not changed when the variable is subsequently reassigned. Thus when '4' is input in response to the prompt +a=????, it will assign the value of 4 to the variable 'a'; but if subsequently reassigned by +a=a*1000, and 'Changes' requested, the original input value of '4' will still be offered rather than the value of 4000. This device is to avoid the engineer being confused when 'Changes' are selected resulting in different data being offered to that previously input (see section 10.6.16 - Setting and resetting values for prompts). This device may be switched off globally by the command 'DIGITS 0'. SCALE options SC687.PRO to SC689.PRO give examples of the use of 'DIGITS 0'.

When DIGITS is set less than 5, e.g. DIGITS 4, it suppresses engineering notation e.g. 3.25E+8, and prints real numbers as decimal numbers. If 'a' were assigned the value 3.25E+8 with DIGITS set to 4, then 'a' would be printed as 325000000 in any expression or assignment.

10.6.9 Decisions

In many calculations the decision to evaluate one set of equations or another depends upon a particular outcome; for example whether a stress turns out to be permissible or not. The IF facility is designed to solve this problem.

IF is followed by a condition as defined in 10.3.6. For example:

```
IF s>0
```

On a line somewhere below IF must be:

```
ENDIF
```

to match the IF. Optionally, between IF and ENDIF, may be:

```
ELSE
```

When IF is met by the program, the associated condition is evaluated.
A compound condition comprising several conditions as described in section 10.3.6 may follow the IF e.g. IF a>b AND c>d OR i=5.
What the program does next depends on whether the result turns out the be 'true' or 'false'. The action also depends on the location of associated keywords ELSE and ENDIF.

The action of the program on meeting IF in the proforma file is described first for the case when ELSE is present:

- a true condition causes the program to deal only with the lines between IF and ELSE, ignoring those between ELSE and ENDIF

- a false condition causes the program to ignore the lines between IF and ELSE, dealing only with those between ELSE and ENDIF

The action of the program in the absence of ELSE is:

- a true condition causes the program to deal with the lines between IF and ENDIF

- a false condition causes the program to ignore the lines between IF and ENDIF

In all cases the program then goes on to deal with lines following the obligatory ENDIF.

Between the IF and ENDIF further IF-ELSE-ENDIFs may occur, this is called 'nesting'. In PRAXIS, IF-ELSE-ENDIF may be nested up to 50 levels. The TRACE command is useful for checking the levels as also is the /P switch which causes the Proforma to be sent to the output file such that it shows where the programming structures start and stop.

10.6.10 Repetition

The REPEAT facility makes it unnecessary to duplicate sets of lines in the calculation file when the only difference lies in the values of variables.

The structure is introduced by the line:

```
REPEAT
```

On a line somewhere below REPEAT must be the line:

```
ENDREPEAT
```

to match the REPEAT. Between REPEAT and ENDREPEAT must be the control word UNTIL followed by a single condition or a compound condition as defined in section 10.3.6, e.g.

```
UNTIL a>b
UNTIL a>b OR i=5
```

On meeting REPEAT the program takes note of the number of the line which follows REPEAT. This is the line to which the program must return on meeting ENDREPEAT. The program would "loop" indefinitely unless offered an escape by the condition after UNTIL. On meeting UNTIL the program evaluates the associated condition. If the condition proves to be true the program leaves the loop and deals with the line following ENDREPEAT.

Between the REPEAT and UNTIL, or between the UNTIL and ENDREPEAT, further REPEAT-UNTIL-ENDREPEATS may occur, this is called 'nesting'. In PRAXIS, REPEAT-UNTIL-ENDREPEAT may be nested up to 50 levels. The TRACE command is useful for checking the levels as also is the /P switch which causes the Proforma to be sent to the output file such that it shows where the programming structures start and stop.

To stop the program "looping" indefinitely, PRAXIS limits that the number of times each and every UNTIL is encountered to 32000, faulting with an error message when this limit is exceeded. When the nature of the calculation is such that looping more than 32000 times is required, use the label and conditional GOTO as described in section 10.3.6.

10.6.11 Procedures

The DEFINE procedure allows a sequence of lines of the proforma file to be named. Wherever this name subsequently appears it implies the nominated sequence of lines.

The facility is useful when it would otherwise be necessary to type a particular sequence of lines more than once. But its greatest use is to achieve clarity in the proforma file. In a complicated proforma it is difficult to appreciate the structure of nested REPEATS and IFs until long sequences of lines are reduced to single names by replacing them with names of procedures.

Definitions of procedures should all be placed at the end of the proforma file. A STOP command must be given on the line before the first occurrence of DEFINE.

A definition of a procedure is introduced by a line containing the control word DEFINE followed by a name to identify the procedure. For example:

```
DEFINE BEAMa
```

Beneath this line every line is deemed to be a line of the procedure until the following line appears:

```
ENDDEFINE
```

Wherever the given name subsequently appears - on its own line in the manner of a control word - the program behaves as though the lines of the procedure had been copied out in place of that name.

The name chosen to identify a procedure may not be a keyword, nor may it be the name of a variable. Every name in PRAXIS must be unique.

An example of a definition is:

```
DEFINE comment
This is a comment.
ENDDEFINE
```

An example of this procedure "invoked" from the calculation file is:

```
comment
+area=PI*radius^2
comment
+volume=area*height
comment
```

It is sometimes desirable to have a procedure call in which the relationship between the variables within the procedure is expressed parametrically. This may also be achieved by defining the procedure within the DEFINE-ENDDEFINE program structure and giving 'arguments' when invoking the procedure. As an example consider a function call modul (standing for modulus) where the modulus is the remainder after integer division of two numbers.

```
START
! +numerator=145.5 +denominator=10
```

```
modul numerator denominator remainder
Remainder is +remainder
STOP
DEFINE modul
! +mod3=mod1-mod2*INT(mod1/mod2)
ENDDEFINE
FINISH
```

The name of the procedure must have a minimum length of 4 characters (modul in the above example has 5), the first three of which are used to build symbolic names by the addition of a single digit suffix staring from 1 (mod1 to mod3 in the above example). Each of these symbolic names corresponds to the arguments taken in order thus:

```
'mod1' corresponds to 'numerator'
'mod2' .............. 'denominator'
'mod3' .............. 'remainder'.
```

There may be up to 9 arguments but only the last may have a 'returned value' and the last argument, if returned, should be a single variable. Of course the power of using arguments is the ability to invoke the same procedure or function with a variety of data without need to assign the data to the variables used in the procedure. Some typical invocations of 'modul':

```
modul 123.5 1 r          r returns .5
modul a*b c fract        fract returns 0.10375 if a=12.2, b=15.15
                         and c=8.
```

The INT function may be used to cycle for a special condition e.g. if it is required to set a value 'fac' =100 generally, but every sixth time in a loop, set to unity, proceed as follows:

```
set the base  +b=6
set the value +fac=100 arrange for the value 'a'
to cycle 1,2,3,4,5,6,1,2,3,4,5,6,1,2,3... and so on in a loop.
IF b=a-INT((a-1)/b)*b THEN fac=1 ENDIF
will set the value fac to: 100,100,100,100,100,1,100,100... etc.
```

One procedure may call (or invoke) another, this is called 'nesting'. In PRAXIS, the programming structures: IF-ELSE-ENDIF, REPEAT-UNTIL-ENDREPEAT, DEFINE-ENDDEFINE may be nested up to 200 levels. The TRACE command is useful for checking the levels as also is the /P switch which causes the Proforma to be sent to the output file such that it shows where the programming structures start and stop.

PRAXIS was not designed to replace other programming languages. (The PRAXIS notation provides facilities for the engineer to invoke Fortran or C programs for matrix operations, but providing paginated output through the Windows print manager.) PRAXIS was designed for the engineer who is able to use a simple editor (such as Notepad) and wants to write and use interactive programs (question and answer). For this reason, subscripted variables have been limited to two dimensions. For singly subscripted variables, a dimension statement is not required; see 10.3.7 for doubly subscripted variables. A variable A(n) may be used where n has the value 500. PRAXIS simply stores such a variable as A500, there is no requirement in PRAXIS that A1 to A499 even exist. If a proforma calculation needs 2 or more dimension, then calling a procedure with arguments as described above, can make matrix inversion and other such operations straightforward. The following simple proforma writes the element number (numbering from top left to bottom right) into an array A(20,20) and then reads the value in column 2 of row 15.

```
START
! +row=0
REPEAT
```

```
! +row=row+1 +col=0
REPEAT
! +col=col+1
arrayA 2 row col (row-1)*20+col
UNTIL col=20
ENDREPEAT
UNTIL row=20
ENDREPEAT
arrayA 1 15 2 value
value= +value
STOP
DEFINE arrayA ! 2D, width=20, 1=read, 2=write
! element number +n=(arr2-1)*20+arr3
IF arr1=1 THEN arr4=A(n)
IF arr1=2 THEN A(n)=arr4
ENDDEFINE
FINISH
```

10.6.12 STOP versus FINISH

The control word STOP, when encountered on the proforma file, makes the program stop work. There may be any number of STOPs in a calculation file. Here is an example of one:

```
IF compr>=0
The column load is +compr kN
ELSE
Something awry: negative column load.
STOP
ENDIF
```

The control word FINISH behaves as STOP if the program encounters it. However, FINISH is used to mark the very end of the proforma file. It makes itself the last effective line because the program would ignore any line beyond it.

SCALE is occasionally used to pipe data to NL-STRESS. The end of NL-STRESS data is normally FINISH, but if SCALE encounters the FINISH command it stops. To avoid the problem, when piping data to NL-STRESS, use the command FINISh at the end of the NL-STRESS data being piped. Avoid using FINISh at the end of SCALE, SCALE expects to see FINISH at the end of a proforma, not FINISh which can cause confusion. If FINISh is used at the end of a SCALE proforma, the .STK file will be seen to contain '+UNITS=5'.

During the development of a proforma, if when it is run the proforma goes AWOL, insert FINISH commands at approximately tenth points in the proforma. Test the proforma and if the first tenth is OK, remove the first FINISH command and carry out another test, and so on. If the removal of any FINISH command causes the proforma to go AWOL, the author will know which section contains the bug. Judicious use of the FINISH command can save development time.

10.6.13 Tracing a calculation

In generating the calculations file the program copies the proforma file, adding numerical calculations and suppressing certain lines on the way. The lines suppressed are:

- lines beginning with an exclamation mark and parts of lines from exclamation mark onwards

- lines beginning with keywords.

In a complicated calculation it may be difficult to see from the results file precisely how the program has dealt with nested IFs and REPEATs. To trace the behaviour of the program in such cases, it is only necessary to include the keyword TRACE on its own line, anywhere in the proforma, after the START.

When this keyword is encountered all subsequent lines beginning with keywords IF, ELSE, ENDIF, are copied as they are encountered together with their level e.g. if a 2 follows an IF then the IF opens an IF-ELSE-ENDIF structure within one already opened. The keyword TRACE encountered a second time switches off the trace, thus this keyword may be used to home-in on a problem. When TRACE is on, the screen displays keywords such as DEFINE, REPEAT... indented for clarity, but the output file (normally rust coloured - click Display to view) is not indented to improve its appearance.

Similarly REPEAT, UNTIL, ENDREPEAT, are copied as they are encountered together with their level e.g. if a 2 follows a REPEAT then the REPEAT opens a REPEAT-UNTIL-ENDREPEAT structure within one already opened. It will be seen from the trace of the REPEAT-UNTIL-ENDREPEAT structure that the pattern is REPEAT UNTIL ENDREPEAT UNTIL ENDREPEAT UNTIL ... The reason for this pattern is that one encountering an ENDREPEAT, control returns to the line following the REPEAT which opened the programming structure.

10.6.14 Page length

PRAXIS assumes an A4 page size. For installations where the stationery is of different length, the page length may be set by:

```
PAGELENGTH n m
```

where n is an integer number in the range 10-32000 giving the number of lines that can be printed per page, and m is an integer number giving the number of lines required to be printed per page. The pagelength command should come near the START, either before or after, and applies for the entire calculation.

A pagelength >=1000 is treated as an instruction to not page the calculation; a page length of 2000 is treated as an instruction to neither page nor indent the text; a page length of 3000 is treated as an instruction to add the page heading in HPGL but not indent the text; a page length of 4000 is similar in action to that for 2000, except the reserved strings $1-$9 are set up (see 10.6.21 & option 952), a page length of 5000 is similar to that for 4000 but is used for pre -processing NL-STRESS data file to remove expressions & assignments.

The above actions by the pagelength command only operate after the first line is sent to the .CAL file.

Occasionally, it is desirable to control pagination within the proforma itself, for an example of this please see vm452.pro.

10.6.15 EXTERNAL ASSIGNMENTS

The use of external files is described in section 10.5.6. External assignments such as +d=200 may also be placed in a separate file and the external file referenced in a proforma before the command START, by giving the filename preceded by @ (e.g. @SCALE.STA). As with external tables, the @ must be the first character in a line.

This device permits the reference to default data such as pagelength, section dimensions of profiled steel decking and also permits other programs to interface with PRAXIS.

10.6.16 Setting and resetting values for prompts

Any line which contains a prompt should not exceed 74 characters in length, if it does an error will be reported. If a variable is assigned in a proforma before the appearance of a prompt for the value of that variable e.g.

```
! +d=200
Depth of section     +d=????
```

then following the display of the prompt for Depth of section, the value 200 will be offered for acceptance by pressing Return alone, or rejection by typing another to replace the 200 before pressing Return. PRAXIS stores the response (200 in this example) separately from the current value of the variable 'd' so that if the user resets the value 'd' later on in the proforma e.g. +d=d-20, then when the proforma is run again the depth of section is still offered as 200 rather than the amended value of 180. Please note that if the variable name is used as an argument when calling a procedure and the value altered within the procedure, then for good reasons the value offered is the amended value, for example SPADE uses the procedure 'range' for limiting a value to a range, invoked by say:

```
range d 1000 2000 d
```

where the procedure 'range' checks that the current value of 'd' is within the range 1000 and 2000 and if outside makes d=1000 which would be offered in a subsequent prompt such as +d=????

There are occasions, for example when repeating a sequence, when it is desirable to prevent the offering of a variable value beneath a prompt. To reset a variable to 'off' give it a value 1E39 e.g.

```
! +d=1E39
Depth of section     +d=????
```

which will cause the Depth of section prompt to be displayed without a default value displayed beneath it. If d is subsequently assigned a new value then the new value will be offered as a default, thus in the example below the value of 300 will be offered. It is essential that the assignment '+d=300' immediately follows the '+d=1E39' as any assignments between these two will cause the wrong value for 'd' to be offered.

```
! +d=1E39 +d=300
```

```
Depth of section     +d=????
```

To add a variable to the stack so that its value can be tested, without altering its value if it has already a value, assign to it the pseudo value -1E39 e.g.

```
! +d=-1E39
```

If 'd' was already on the stack, then this assignment would be ignored, and the value held by 'd' would be that held before the assignment. If 'd' was not already on the stack, then the value held by 'd' would be 1E39, positive so that any prompt for 'd' would behave as above.

This numerical device has use in verifying the logic of proforma calculations using sc924.pro, see sc924.hlp for a discussion.

10.6.17 Special format operations

Embedded + signs are ignored as described in section 10.6.5. If a variable appears in a line of text with only one trailing space e.g.

```
The depth of the section ( +d ) is within permissible.
```

then the current value for +d replaces the +d in the line, with widening or reduction of the field width taking place dependent on the value and number of significant figures set with any DIGITS command e.g.

```
The depth of the section ( 200 ) is within permissible.
```

On some occasions e.g. tabular format, and when variable values are used with diagrams, it is desirable to print the current value of the variable without reformatting the line. This may be accomplished by giving two or more trailing spaces after the variable name. It is then up to the developer of the proforma to ensure that there is a sufficient field width to print the current value of the variable e.g.

```
 +a       +b         +c         +d         +e         +f
```

would print the values a to f left adjusted at the +, with a tab width of ten characters. If the command DIGITS 12 had been given, then overprinting could result.

During the course of a calculation it is sometimes helpful to remind the reader what the current value of a variable is e.g.

```
Depth of section     d= +d
```

Such a line does not assign a value to the variable d, but merely prints the line (assuming d has the value 200):

```
Depth of section     d= 200
```

Rather than leave the single space between the = and the 2, as would occur by the above rules for formatting, PRAXIS removes the single space, printing the line:

```
Depth of section     d=200
```

and thereby makes the line look the same as one in which an assignment had taken place. This closing up takes place if the character were =

as in the above example, or one of the characters + - * / ^ or).

When using PRAXIS for compiling plot files written in HPGL the above closing up is used when an expression follows LB (LB if HPGL for label) thus:

```
LB +a
```

would be sent to the plot file as

```
LB12.5
```

assuming the current value of the variable 'a' was 12.5.

Laptop computers are frequently set slow for scrolling text because of the screen retention associated with current technology. When the user is being prompted for input, PRAXIS uses scrolling to keep any picture on the screen but move it up one line. When a complete screen needs to be replaced, PRAXIS 'repaints' the screen from top down; this saves over 20 scrolls of the screen and therefore speeds up the display. The combination of scrolling and repainting is achieved by holding a screen buffer and repainting the screen when PRAXIS finds that it has a full buffer. Lines which contain the ???? prompt automatically cause the screen buffer to be flushed so that any help preceding the ???? prompt is shown on the screen. Some applications e.g. when LUCID or SPADE report how the drawing is proceeding, need statements such as 'Drawing plan' or 'Drawing section AA' to be displayed immediately. This is achieved by including the Alt+0255 character at the start of a line, or following the % character if the line starts with a % e.g. '% Drawing section AA' where the space between the % and the D contains the Alt+0255. The Alt+0255 character thereby gives the user control to scroll a line immediately rather than add it to the screen buffer.

10.6.18 Condensed print and summary options

In general a plus sign precedes an 'equation' in which there is a variable on the left and an expression on the right (see 10.6.5). For example:

```
+n=(az*B/2+az'*t/2)/AX
```

The following three lines are typical of what would be sent to the calculations file:

```
n=(az*B/2+az'*t/2)/AX
 =(2364.7*88.9/2+1737.2*8.6/2)/4101.9
 =27.446
```

A condensed print option may be invoked by changing the display mode when running SCALE from Normal to Condensed. This would cause the above example to be reduced to a single line:

```
n=(az*B/2+az'*t/2)/AX=27.446
```

To reduce the output even further, commence those lines to be output with a full stop (see 10.6.7) and select the Summary display mode. If the proforma does not include at least 5 lines which start with a full stop, and if a Summary is requested, then the request will be treated as if the Condensed print option had been given. This is to stop the user who has requested a summary, from being presented with a set of calculations without a summary (as the author of that proforma had considered that a summary was not appropriate).

10.6.19 Batch files

SCALE itself can be run as a batch file by including the switch /b followed by the option number following the filename in FIL.NAM. Thus if FIL.NAM contains "C702.DAT/b254" this will cause SCALE to run option 254 as a batch stream taking the page headings from C702.DAT and the data from SC254.STK.

In the example given above, proforma 254 will be run from beginning to end taking the responses from the stack file. When running in batch mode the keyword BATCH will be set to 1, otherwise this keyword in set to 0.

The keyword BATCH may also be used as a variable e.g.

```
IF BATCH=1
...
ENDIF
```

Generally proformas start with the prompt: +ans=???? to which the user responds with 1 to accept a standard set of responses (defaults); and then the engineer types in responses in reply to other prompts. At the end of a run, a stack (.STK) file - containing the engineer's responses. Rather than include '+ans=1' on the stack file which would cause the default values to be offered the next time the engineer ran the option, the program writes '+ans=0' to the stack file so that the data last provided by the engineer, is offered rather than the set of default values. When running in batch mode, where .STK files are copied for use in the next run, it is confusing when the current value of 'ans' is amended, thus when the program detects that it is being run in batch mode, the value of 'ans' is not set to zero.

10.6.20 Semi-colon

When PRAXIS finds a semi-colon in the first character position on a line in the proforma, the line is copied to the calculations file but not displayed on the screen. The semi-colon therefore permits HPGL (Hewlett Packard Graphics Language) instructions to be sent to the calculations file for subsequent plotting without displaying them on the screen - and in consequence - causing confusion.

It is permissible to include several Praxis statements on a single line provided that:

- each statement is separated by a semi-colon with one or more preceding blanks, or neither preceding nor following blank (if there is no preceding blank but there is a following blank, then the semi-colon is considered to be punctuation)
- none of the concatenated statements import data from an external file).

If errors are found, then the line reference in the error message is that corresponding to the expanded file.

10.6.21 Dollar sign

The dollar sign with an integer suffix - $43 in the first assignment below - is used as a variable name for a string of characters; thus the variable $43 holds the string of characters 'Electrodes to comply with BS639 grade E43'. The integer suffix must be in the range 1-32767 and may be subscripted e.g. $(a) in the second example below. Where a prompt for input does not specify a string variable name (e.g. 'Location: ????') then PRAXIS generates its own pseudo variable name by using the line number as the integer suffix. Thus if 'Location: ????' were on line 37, then PRAXIS would act as though it had read: 'Location: +$37=????'. Inspection of the .STK file will reveal the string variable names generated by PRAXIS. Examples follow:

```
+$43=Electrodes to comply with BS639 grade E43
+a=10 +b=5
+$(3*a+4+2*b)=Electrodes to comply with BS639 grade E43
```

Both the above string assignments accomplish two things:

- they store the string 'Electrodes to comply with BS639 grade E43' in the variables $43 & $44 respectively, thus permitting them to be saved in the .STK file
- when the prompt +$43=???? is displayed, the string stored in $43 is offered as a default response; similarly for +$(3*a+4+2*b)=???? the string stored in $44 is offered as a default response (assuming the variable 'a' contains 10 and 'b' contains 5).

The subscript for a string number may be of any complexity but it should evaluate to an integer number in the range 1-32767 whereas the subscript for a numeric variable in PRAXIS is limited to a single variable + or - a constant e.g. v(i+7).

```
String assignments fault if an '=' is found previously to the $ sign
thus faulting lines such as: ! +n=n+1 +$(n)= +n
                         or: ! +n=n+1 +$123= +n
 but allowing lines such as: ! +zvab= +$(....
 and allowing lines such as: ! +$11=FORCE-Y +$12=MOMENT-Z etc.
   also assignments such as: ! +$10=B C +K=2
        i.e. storing in $10:        B C +K=2
                         or: ! +$10=B   +K=2
        i.e. storing in $10:        B   +K=2
i.e. allowing a mixture of text & numerical assignments.
```

String assignments and comparisons have subtle features e.g.

```
! +$200=ASDFG
! +$201= +$200
IF $201= +$200
Never here
ENDIF
```

In this example the first line puts the characters ASDFG into $200, the second line substitutes the characters ASDFG for the +$200 string and then moves the characters to the left so that $201 is assigned ASDFG i.e. $200 and $201 match. The third line compares the contents of $201 with the characters ' +$200' and of course they do not match so the message 'Never here' will be displayed. Had the user intended to compare the contents of $201 with those of $200 then the third line should have read: IF $201=$200

Nine special strings $1 to $9 are preset on entry to the program. For implementations which interface with NL-STRESS, these contain member forces and properties as given in the SCALE Reference Manual; else the first 4 contain the page heading set by the STRUCTURE or TITLE command the next 4 contain: page number, MADEBY, DATE & REFNO and the last one contains the current data file name without its .DAT extension. Strings $1 to $4 may be reset within the proforma, e.g. SC964.PRO resets $1 to be used as a page heading dependent on the Manual being built.

Assignment to $31999 is used internally in association with the <<< command.

Assignment to special string numbers 32001-32100 causes the ASCII values of the characters in the string to be assigned to variables ASC(1:72). Thus after the assignment +$32001=ASDFG, ASC(1) would contain 65, ASC(2) would contain 83 and so on, with ASC(6:72) blank filled. When ASC() is printed, the characters rather than their ASCII value are printed. This facility permits substring operations to be performed e.g.

```
  ! +$32001=1234567890
  ! +ASC=5
  +ASC
```

where the first of the 3 lines above sets ASC(1)=65, ASC(2)=66 .... the second line fixes the length of ASC() to 5 characters (with a maximum permissible of 72 characters) and the third line prints 12345 i.e. the first 5 characters of the string $32001.

Assignment to special string number 32100 causes ASCII values excluding numerical digits 0-9 to be assigned to variables ASC(1:72). Thus

```
  ! +$32100=JEN7
```

would assign: ASC(1)=74, ASC(2)=69, ASC(3)=78, ASC(4:72)=32 i.e. blank.

Generally, the accuracy of results is independent of the number of DIGITS set for SCALE, LUCID, SPADE, CALCS. It will be clear however that rounding of printed results is an exception. For text strings assigned to $32001 to $32100, where maximum precision is required, it is sensible to set DIGITS 15 as used in proforma sc919.pro for sorting text or numbers.

It is permissible to directly assign values to say: ASC(31) etc. but before doing so, ASC() should be cleared by: +ASC(1)=VEC(32)*72 which will set blanks (ascii-32) in ASC(1) to ASC(72) inclusive. Failure to clear ASC(1) to ASC(72) before directly assigning their values will cause unpredictable behaviour. This is due to the assumption that ASC(1) to ASC(72) are sequentially located on the stack; e.g. if assignments: +ASC(37)=43 and +ASC(27)=40 then these ad hoc assignments will not result in ASC(1) to ASC(72) having sequential locations. See the start of proforma sc970.pro for the clearing of ASC(1) to ASC(2).

Assignment to special string numbers 32101-32172 causes the string to be right adjusted in a field width given by the last 2 digits. As an example, suppose the string $32108 held the characters JAMES then the line:

```
Checker for this procedure  +$32108
```

would be printed or displayed as:

```
Checker for this procedure     JAMES
```

Assignment to string numbers 32201-32272 causes the string to be truncated to a field width given by the last 2 digits. As an example, suppose the string $32204 contained the characters 3/16" then the line:

```
Fillet weld size +$32204
```

would be printed or displayed as:

```
Fillet weld size 3/16
```

PRAXIS has to work in many ways for SCALE, LUCID & SPADE proformas and to interface with NL-STRESS; in consequence string assignments have a number of subtle features e.g.

```
+$32220=+a mm
+$32220= +a mm
+$32220=  +a mm
```

the first of which assigns the characters +a mm to the string $32220; the second substitutes for the value of +a and then left shifts one space so that the string contains '5.25 mm' assuming the variable 'a' held the value 5.25; the third substitutes for the value of +a but does not left shift so that the string contains ' 5.25 mm'.
If the author wants values on the right hand side to be substituted before assignment to the string, then it is recommended that they use 2 spaces after the equals.

Special string: Assignment to $32766 causes any substrings (either alpha or numeric) to be extracted in exactly the same manner as for the DECODE command (see 10.6.32).

Special string: Assignment to $32767 is used for 'cleaning up'; any characters less than ASCII 32 or greater than ASCII 254 are replaced by blank.

One final special string: $32000, concatenates all text in any assignment. Thus in the example below, $32000 would contain 'h102.dtl' assuming $55=h10 and i=2. After concatenation the contents of $32000 may be copied to a further string as in the second line below; then tested as in the third and subsequent lines.

```
+$32000= +$55 +i .dtl
+$500= +$32000                (N.B. the gap causes the contents of
IF $500=h102.dtl              $32000 to be assigned to $500 and
File name matched OK          not just the characters '+$32000')
STOP
ENDIF
```

It is recommended that when several $ strings need to be concatenated, e.g. $11 $12 & $13, they are first appended e.g.: +$20= +$11 +$12 +$13 then concatenated by +$32000= +$20
The reason for the two stages it to avoid confusion, as an example of confusion, consider the short proforma below.

```
START
! +zva12=15 or whatever from some previous assignment
! +$11=sd
! +$12=zva12
! +$13==21
! +$20= + +$12
+$32000= +$11 +$20 +$13
+$32000= +$11 + +$12 +$13
FINISH
```

In the above zva12 is a subscript which currently holds 15; we are trying to build the assignment: sd15=21. The $20= + +$12 causes $20 to contain the currelt value (15) contained in the variable zva12. The first concatenation produces: sd15=21
the second concatenation produces: sd+zva12=21.

To attach a plus sign in front of an assignment e.g.
! +$32000=+ cc=zva(zp'-7)
do not provide a space between the = and + signs e.g. see sc924.pro.

When the $32000 string is used for concatenating substrings, it is sometimes necessary to leave some gaps. This may be achieved in LUCID, SPADE and CALCS by including Alt+0255 where a blank would be required. For very good reasons, SCALE treats Alt+0255 as a blank and strips it out when concatenating.

It is permissible to have more than one single string assignment on a line which is not output e.g.

```
 ! +$200=A +$210=EXAMPLE B +$220=Grid line C
```

In the above example Alt+0255 has been used in place of a space in strings $210 & $220 to convert each compound strings into a single string. Compound strings must always be on their own line e.g.

```
 ! +$230= +$210 on +$220
```

which would result in $230 containing 'EXAMPLE B on Grid line C'

It is also permissible to have numerical assignments and string assignments on the same line, providing that all the numerical assignments come before the string assignments and that no numerical assignment is used to set any following string number e.g.

```
 ! +iex=701 +$(iex)=A
```

is wrong, and would be faulted if 'iex' had not been assigned a value on a previous line, but if it had been assigned a value on a previous line then it would use that previous value in the assignment +$(iex)=A.

Although the special strings $32101-$32178 are best used for right adjustment; the following accepts the string $32001 & field width 'fw' and right adjusts $32001. It is presented here as an example of PRAXIS string operations.

```
DEFINE rtadj
! +ifb=73 +jdb=0
REPEAT
! +ifb=ifb-1
IF ASC(ifb)<>32 AND jdb=0 THEN jdb=ifb ! jdb set to string length
IF jdb>0 AND ASC(ifb)=32 THEN ASC(ifb)=0255 replace blanks by Alt+0255
UNTIL ifb=1
ENDREPEAT
! jdb points to last character position, don't adjust unless less
! than field width.  All blanks before character position 'jdb'
! have been replaced by Alt+0255
! +ifb=0 +$32000=
REPEAT
! +ifb=ifb+1
! +$32000= +$32000 +ASC(ifb)
UNTIL ifb=jdb
ENDREPEAT
! +$1878= +$32000
! Number of characters to pad front of screen +npad=fw-jdb
IF npad>0
! +ifb=0 +ASC(1)=255 +$32000=
REPEAT
! +ifb=ifb+1 +$32000= +$32000 +ASC(1)
UNTIL ifb=npad
ENDREPEAT
! +$32000= +$32000 +$1878
! +$32767= +$32000
! +$32001= +$32767
ENDIF
ENDDEFINE
```

Praxis parses each line twice, the first substituting for any expressions e.g. +3*a, or carrying out string assignments e.g. +$300=Weld detail,
the second to carry out any numerical assignments. It is recommended that string assignments are not mixed with numerical assignments; the line below will be faulted for by the above order of parsing, the string assignment +$(a)=ASDFG will be carried out before 'a' is assigned the value 423.

```
! +a=423 +$(a)=ASDFG
```

If the string assignment is not subscripted then it is permissible to have numerical assignments before the string assignment thus

```
! +a=423 +$300=ASDFG
```

would not be faulted.

10.6.22 HPGL commands for graphics display

PRAXIS supports six HPGL commands viz: IN PD PU PA LB SP. The example below draws a rectangle, puts the label RECTANGLE in the centre, and then prompts for breadth & length before returning to normal display.

```
IN
SP 4
PU
PA 2000 2000
PD
PA 4000 2000
PA 4000 6000
PA 2000 6000
PA 2000 2000
PU
PA 2500 4000
LB RECTANGLE
PU
Breadth of rectangle             +B=????
Length of rectangle              +L=????
SP 0
```

10.6.23 FILE command

Normally lines in a proforma commencing with a % are sent to the screen but not printed. The command 'FILE' in a proforma followed by a filename instructs PRAXIS to send subsequent % lines to the nominated file rather than display them on the screen. This mechanism permits output from one proforma to become data for another. Expressions may be used in subsequent lines commencing with a %, but any assignments or logic would be ignored. The FILE command may be switched off by repeating the command but omitting any filename. The 'pipe' may be redirected to another file by repeating the command with the new filename; the 'pipe' need not be switched off before redirecting to another file.

For piping from a proforma to a data file for running by NL-STRESS; within the proforma use ' |' in place of ' !'. After piping the ' !' replaces ' |'. See sc461 which pipes the data for an NL-STRESS analysis to the file sc461.dat.

10.6.24 Using PRAXIS to update files

The characters ( ) appearing as the first character in a line of a proforma permit PRAXIS to update text files with new values if the corresponding variables KLB,KRB (K Left & Right Bracket) =1. '(' causes the program to evaluate any expressions in the line and then to copy the line completely unchanged to the .CAL file. ')' causes the line to be sent unchanged to the .CAL file, then substitute for any expressions in the line and send the line again to the .CAL file with the ')' removed and ignore the next line in the procedure which is taken to be the previous 'second' line.

10.6.25 Conditional compilation lines

The characters { } appearing as the first character in a line of a proforma permit 'conditional compilation' thus:

```
{  process line unless variable KLBR=0 (KLBR= K Left BRaces)
}  process line unless variable KRBR=0 (KRBR= K Right BRaces)
```

External files (see 10.5.6) may not have conditional compilation lines but may include lines which will be ignored by starting such lines with two Alt+0255 characters. The SCALE menu file (scale.mnu) uses this device to suppress unrequired lines when the file is read in to build the SCALE Reference Manual.

10.6.26 Conditional output lines

The characters [ ] appearing as the first character in a line of a proforma permit 'conditional output' thus:

```
[  output line unless variable KLSB=0 (KLSB= K Left Square Brackets)
]  output line unless variable KRSB=0 (KRSB= K Right Square Brackets)
```

The following simple proforma in the left column and the data in the right column (held in file b20) illustrate the previous three features.

```
 START
 PAGELENGTH 4000
 ! +KLB=1 +KRB=1
 ! +KLBR=0 +KRBR=0 +KLSB=0 +KRSB=1
 bill                              ( +prev=800 +tot=950 +per=tot-prev
 ! Press Return to continue ????   ) +prev    +per     +tot
 ! +$32000= +$9 .CAL                 800      150      950
 CMD copy +$32000 b20 >nul         { Syntax check
 STOP                              } Quantities report
 DEFINE bill                       [ Starred rates report
 @b20                              ] Trade rates report
 ENDDEFINE                         etc.
 FINISH
```

Firstly the page length is set at 4000, this tells PRAXIS to omit paging and left margin in the updated file of data (b20 in the example) invoked by 'bill'. The +$32000 assignment concatenates the current heading file name with the .CAL filename extension so that the CMD copy command copies the .CAL file output by PRAXIS over the original file 'b20' thus updating it.

The file of data (shown in the right column) starts by setting the conditional compilation switches so that only those lines starting with] are included in the current report. The line starting with (is copied exactly as it is to the .CAL file, as is the line starting with) save that the line following) is discarded and replaced by a line containing the new quantities.

In addition to the above usage, [or] may be reassigned to format control characters such as '%' or '.' by setting the value of KLSB (or KRSB) = 2 for '%', 3 for '.'. The following example was taken from SC267, if the engineer responds with 1, then the '[' will be removed and the notes will be sent to both screen and the calculations file; if the engineer responds with 2, then the '[' will be replaced by a % thus causing the notes to be displayed only.

```
%Print out notes (1=Yes, 2=display only)   +KLSB=????
[
[RACKING RESISTANCE OF TIMBER FRAMED WALLS TO TRADA WOOD INFORMATION
[SHEET 45 DATED AUGUST 2001 AND BS 5268:Section 6.1
[....
```

10.6.27 Right adjustment

When PRAXIS finds an expression in a line e.g. +a it prints the value of the expression starting at the plus sign, all as fully described in section 10.6.5. It is sometimes desirable to print such a value 'right adjusted'; this is accomplished in PRAXIS by the use of a set of variables called RTA (standing for RighT Adjust) to RTZ with a one or 2 numerical digit suffix e.g. +RTA10. When PRAXIS finds such a variable it uses the numerical suffix (10 in this case) as the 'field width' and prints the value of the variable right adjusted in the field (which starts at the +). As an example, suppose the variable RTA8 held the value 12.2 then the line:

```
Level of top of shelf angle +RTA8
```

would be printed or displayed as:

```
Level of top of shelf angle     12.2
```

When an RTA-Z variable is given a three digit suffix e.g. RTB083 the first 2 digits are interpreted as the field width (8 in this example) and the third digit as the number of decimal places to be shown. Suppose the variable RTZ083 held the value 12.2 then the line:

```
Level of top of shelf angle +RTZ083
```

would be printed or displayed as:

```
Level of top of shelf angle    12.200
```

The RTA variable may be assigned to a string viz:

```
! +RTA102=12.7564
! +$20= +RTA102 mm
+$20
```

which would cause the string $20 to be printed as:

```
  12.76 mm
```

i.e. rounding to two decimal places and printing right adjusted in a field of 10 characters (leaving 2 spaces before printing starts).

For tabular reports with closely spaced columns it may be necessary to assign the RTA-Z set to character strings e.g.

```
! +$11= +RTA052
! +$12= +RTB052
! +$13= +RTC052
! +$14= +RTD052
```

which could then be used with a closer column spacing than possible when using the RTA052 - RTD052 variables directly, e.g.

```
+$11  +$12  +$13  +$14
```

RTX, RTY and RTZ behave as RTA to RTW in all respects save that any leading zero found in front of the decimal place is removed; thus .25 or -.25 would be printed in place of the normal 0.25 or -0.25. RTX, RTY and RTZ have particular use in SPADE details where space is at a premium e.g. a quarter inch weld would be shown as .25.

For right adjustment of strings, see section 10.6.21.

10.6.28 CMD and WIN commands

PRAXIS supports pseudo-invocations of command line operations and programs. Use CMD to launch a command line operations, and WIN to launch a program, e.g.:

```
CMD copy file1.txt file2.txt
WIN nls
```

Fuller examples of the use of the CMD & WIN commands are given in section 10.7.8.

10.6.29 Emboldened lines

PRAXIS supports the following pseudo-html (Hyper Text Markup Language) command: <H1> which displays the remainder of the line using a bold font (closing html tags are not required).

For html commands, the left angle bracket must be located in the first text position following any % . or other control character e.g.

```
%<H1>Single bay portal frame
```

for a heading to be displayed only but not sent to the print file.

10.6.30 DECODE for extracting information from external files

The keyword DECODE may be used to extract information from external text files e.g.

DECODE myfile.dat 24 [record-length]

would decode the text found in the 24th line of the file 'myfile.dat' saving TN numbers and TS-10 strings in D(1) -> D(TN) and $11 -> $(TS) with $10 containing the full line (record length limit is 80 characters) (the first nine strings being reserved for page headings etc.).
The filename may be a text string or text variable (e.g. +$600), the line number may be an integer, or integer variable or integer expression (e.g. +12*(n+1)). The optional file-length will cause the program to treat the file as a random-access file of that length and go directly to line 24; if the file-length is omitted the program reads from the first line counting the number of lines until it gets to the line required, in this example the 24th. Suppose the line contained:

1 THRU 3 FORCE Y UNIFORM W -3.6

then after decoding, the information extracted would be held as:

```
                              $10=1 THRU 3 FORCE Y UNIFORM W -3.6
D(1)=1                        $11=THRU
D(2)=3                        $12=FORCE
D(3)=-3.6                     $13=Y
                              $14=UNIFORM
                              $15=W
```

Before using numbers and strings extracted, use TN to see how many numbers have been assigned, and TS-10 to see how many strings have been assigned ($1-$10 excluded). If failure in reading a record, then TS=-1 and TN=-1 are returned.

In the above example a positive record number =24 is given. For long external files, where it is necessary to read from the start to find a record of interest, to avoid the decoding of substrings, and return just the record in string $10, give the record number as a negative integer, either as a number or as a variable. For this special case for importing text into $10, any assignments or expressions commencing with a plus sign followed by unassigned variables, would normally generate an error, but so that such data may be manipulated, the plus sign at the start of an assignment or expression is changed to ascii(255). The string $10 may be converted to ASCII, by assigning it to $32001.. and once in ASCII the ascii(255) may be converted back to a plus if required.

Assignment to the special string $32766 e.g.
+$32766=1 THRU 3 FORCE Y UNIFORM W -3.6
causes any substrings (either alpha or numeric) to be extracted in exactly the same manner as described above for the DECODE command.

Assignment of $11 etc. to string number $32001 would cause the ASCII values of the characters in the string to be assigned to variables ASC(1:72), see section 10.6.21. Thus after the assignment:

+$32001= +$11

```
ASC(1)=84 (84 is the ASCII value of T)
ASC(2)=72 (72 ..................... H)
ASC(3)=82 (82 ..................... R)
ASC(4)=85 (85 ..................... U).
```

The word THRU could be tested by:

```
IF ASC(1)=84 AND ASC(2)=72 AND ASC(3)=82 AND ASC(4)=85
```

or, as an alternative to the above and more directly: IF $11=THRU

10.6.31 EDIT command

Normally data is input in response to the prompt ???? in a proforma, as described in 10.6.4, with lines before the prompt line containing help (background colour normally green) to explain what is required, to show a table, or to give general advice. When the data is simple (e.g. giving sizes of a member or cross-section), or when arrays of numbers are to be input (e.g. joint coordinates), then the EDIT command provides an alternative way of inputting the data. The EDIT command takes one of two forms:

```
EDIT /W n
EDIT v(1),v(n)[,w(1),w(n)]
```

The brackets [] contain optional data, the brackets themselves should not be included in the command line.

The first form instructs the program to extract variable names for prompting from the next 'n' lines which must contain ???? prompts as described in section 10.6.4. The first 34 characters from subsequent prompt lines, are displayed on the left side of the screen; the data box is always shown on the right side. The /W switch must have a single space before it, and be an upper-case W.

For the first form only, any recognised units are displayed on the right side after the data box. Where it is required that trailing text be included, the trailing text should follow an Alt+255 which in turn should follow a single blank after the last question mark of the prompt. Option SC918 uses this feature.

The second form instructs the program to prompt for: v(1) v(2) ... v(n) and w(1) w(2) ... w(n) (n<=20) in a box on the right side of the screen with one variable per row if only v(1),v(n) are given in the command, and with two boxes on the right side of the screen if w(1),w(n) follow. The limit 'n' must be the same for both subscripted variables and must be a pre-assigned variable, or an integer value. The commas are mandatory for the 2nd form of the EDIT command. The user is reminded that v(1) and v1 are identical in PRAXIS, however the brackets must be shown in the second form even though PRAXIS omits them (to save space) when it displays the subscripted variables within the box.

In both forms, the variables may take any name; n, v and w being examples only.

The first form of the EDIT command 'looks forward' in the proforma for variables and units in the next 'n' lines and therefore is able to display any units in the data box. The second form of the EDIT command does not 'look forward' for prompt lines, it creates a box/boxes just for the names of the variables which follow the command 'EDIT' and therefore any picture on the left side of the screen should show what units are being used.

An example of the first form of the edit command:

```
EDIT /W 2
%Mark 1 calling up (8 chars max)  +$101=????
%Mark 2 calling up (16 chars max) +$102=????
```

which is taken from LU130.PRO, tells the program to scroll any picture up the screen and add a box of sufficient depth containing prompts for the string variables $100 & $101.

An example of the second form: EDIT ms(1),ms(nm),me(1),me(nm) which is taken from SC600.PRO, tells the program to leave any picture on the screen and show two columns of Praxis dialog boxes containing prompts for the nm sets of member incidences (i.e. start and end joint numbers for each member).

On leaving the box, the program runs in 'fast forward' mode. For the first form only, the fast forward mode is switched off after processing 'n' prompts. For the second form, the fast forward mode runs through to the end, only stopping and prompting for any missing data. To cancel 'fast forward' use >< as the first 2 characters in a line of the proforma; to switch back on, use >> as the first 2 characters in a line of the proforma (see 10.6.6). Examples of more general usage of the EDIT command follow.

Generally a proforma calculation is developed using the ???? prompt and when the proforma is robust, EDIT boxes are added to make the preparation of data, easier. The EDIT box itself does not get copied to the calculations.

```
The first form of the EDIT command is used when it is required to
present several consecutive associated prompts to the engineer as
one block of data e.g.

EDIT /W 4
Depth of section                  +D=???? mm
Width of section                  +B=???? mm
Thickness of web                  +t=???? mm
Thickness of flange               +T=???? mm
```

Assuming the variables D,B etc. have not been set previously, the above 9 lines of a proforma would be presented as:

```
The first form of the EDIT command is used when it is required to
present several consecutive associated prompts to the engineer as
one block of data e.g.


Depth of section                  D= 0                     mm
Width of section                  B= 0                     mm
Thickness of web                  t= 0                     mm
Thickness of flange               T= 0                     mm
```

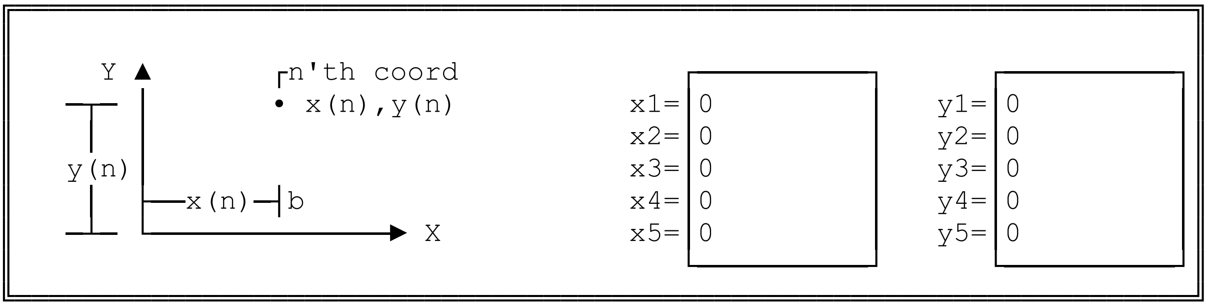
The descriptors 'Depth of section' etc. have automatically been extracted from the lines which follow the EDIT command. One improvement is to provide a figure showing D, B, t & T.

The second form of the EDIT command is best described by an example. Briefly for the computation of the section properties of a general section, the X & Y coordinates of a number of points 'np' defining the periphery of the section, must be input. The following 16 lines have been simplified from a section extracted from proforma SC650.

```
//
  Y ▲       ┌n'th coord
──┬─ │       • x(n),y(n)
  │  │
y(n) │
  │  ├──x(n)──┤
```

```
                                      X
EDIT x(1),x(np),y(1),y(np)
! +n=0
REPEAT
! +n=n+1
X-coordinate                     +x(n)=???? mm
Y-coordinate                     +y(n)=???? mm
UNTIL n=np
ENDREPEAT
><
```

The first of the 16 lines clears the screen, the second to seventh line is displayed on the screen. The eighth line tells the computer to display columns of X & Y coordinates. Assuming that np=5 has already been set but the X & Y coordinates have not been set, the engineer will be presented with:



For this second form of the EDIT command which is used for the input of one or two columns of numbers, prompts will likely be within a REPEAT -UNTIL-ENDREPEAT as in the 7 lines which follow the EDIT command in the above example. This form of the EDIT command switches the program into fast forward mode once the EDIT box has been completed, so that the engineer does not have to deal with the prompts within the loop.
If further data has to be input, it will be necessary to stop fast forward mode by the >< given in the 16'th and last line of the above example.

10.6.32 STACK and RESET commands

The STACK command causes the current set of input parameters to be sent to a file called 'STACK' so that the data may be processed elsewhere, a typical set of data follows:

```
! +ans=0 +ans1=0 +L=40.234 +can=1.524 +nk=15 +nl=14 +st=49.061E6 +ni=1
! +b1=1.8288 +b2=4.8768 +b3=7.9248 +b4=10.973 +b5=14.021 +b6=17.069
! +b7=20.117 +b8=23.165 +b9=26.213 +b10=29.261 +b11=32.309 +b12=35.357
! +b13=38.405 +a1=0 +a2=2.6426 +a3=6.1874 +a4=8.8636 +a5=12.421
! +a6=15.088 +a7=18.645 +a8=21.488 +a9=25.045 +a10=27.712 +a11=31.266
! +a12=33.924 +a13=37.49 +a14=40.234 +p1=619.76 +p2=2146.2 +p3=1607.2
! +p4=1534.5 +p5=1534.5 +p6=1474.7 +p7=1381 +p8=1476.7 +p9=1444.8
! +p10=1534.5 +p11=1534.5 +p12=1474.7 +p13=2052.6 +p14=619.76
! +k1=6.3016E-6 +k2=6.3016E-6 +k3=6.3016E-6 +k4=6.3016E-6 +k5=6.3016E-6
! +k6=6.3016E-6 +k7=6.3016E-6 +k8=6.3016E-6 +k9=6.3016E-6
! +k10=6.3016E-6 +k11=6.3016E-6 +k12=6.3016E-6 +k13=6.3016E-6
! +k14=6.3016E-6 +k15=6.3016E-6
```

10.6.33 EXIT command

The EXIT command causes the current instance of SCALE to be terminated. This command is intended to be used when SCALE is invoked to run in batch mode.

10.6.34 SKIP command

Normally lines in a proforma are sent one by one to the calculations file and to the screen buffer (see 10.6.17). The SKIP command causes the number of lines which follow the command to be skipped in the output.

```
SKIP 10             causes 10 lines to be skipped in the output
SKIP +n             causes +n lines to be skipped where  1<=n<=100
```

n may be given as an integer value; or as an integer variable name prefixed by a '+'.

10.6.35 REWIND command

The REWIND command causes the current stack file to be rewound. In combination with setting DIGITS 0, the REWIND command clears all assignments from both memory and storage in the .STK file.

10.7 Operation

This section describes - in general terms - how to respond to questions and prompts that appear when PRAXIS is in control of the computer.

10.7.1 Data files

Before PRAXIS is started the following data files should be stored on disk:

- the proforma file - its name should end with the "extension" .PRO e.g. BEAM.PRO
- the L.NAM licence file
- the menu file (see 10.7.3) which is supplied with the program.

Give the name of a file ending in .DAT where the page heading DATA is to be saved e.g. CON123.DAT. On pressing Return this response invokes a dialogue which builds a data file containing lines beginning STRUCTURE, MADEBY, DATE, REFNO collectively giving the page heading e.g.

```
STRUCTURE THEO D.LITE & LEVEL
STRUCTURE 116A High Street, Boston
STRUCTURE
STRUCTURE Job: NEW OFFICES for STILGO Inc.
MADEBY DWB
DATE Oct'15
REFNO 5089
```

Once the name of a data file has been given to the program, pressing Return in response to the prompt for the name of the data file tells the computer to use the previous name again.

10.7.2 Files created by PRAXIS

When PRAXIS is started, the screen asks for the names of the data and proforma files (CON123.DAT and BEAM.PRO in the examples above). After these names have been given PRAXIS starts copying the proforma file line by line to:

- a calculation file created automatically and given the same name as the data file but with extension .CAL (example: CON123.CAL)

and the numbers that are typed in response to the prompts to:

- a stack file created automatically and given the same name as the proforma file but with extension .STK (example: BEAM.STK).

The .CAL and .STK files are created in the working directory.

10.7.3 Menu file

When PRAXIS is started and given the name of a data file providing page headings as described in section 10.7.1, it then displays the first screen from the menu of options available and contained in the file PRAXIS.MNU (all systems which use the PRAXIS notation e.g. SCALE have their menu named by the system with .MNU as the extension e.g. SCALE.MNU).

The menu is a text file which may be amended or extended by an editor or word processor which does not include non ASCII characters and which will allow for editing while preserving a fixed record length of 80 characters (followed by Return and line feed). In operation, the menu is accessed by pressing the PageUp and PageDown keys and by clicking on the desired option with a mouse.

Remainder of section applies to Windows only:
Hovering the mouse over an option in the SCALE, LUCID or SPADE menus, causes the program to display a tooltip help item for that option, the tooltip text is contained within the library file POPUP.LIB from which an overview of the option is read and displayed on the screen. This overview text is also displayed on the Page Headings File selection screen which comes up immediately after proforma selection. Users wishing to add their own overviews to the library, will need to use the utility BUILDLIB.EXE to 'smash' the library into its constituent parts (there is a .DTL file for each option), produce a new .DTL file for the new option, add the new filename to the list of overview files in POPUP.NAM and then run BUILDLIB.EXE to rebuild the library.

USING POPUP
Get all the relevant .DTL files assembled (dwb has them in the directory \temp). Run the utility BUILDLIB thus:

BUILDLIB ↵

BUILD/SMASH LIBRARY UTILITY

ENTER: 1=BUILD LIBRARY, 2=SMASH LIBRARY

```
1
BUILD LIBRARY INVOKED

Name of file containing library list? e.g. lu440.nam
popup.nam
Name of library to be built? e.g. lu440.lib
popup.lib
Library record length (40 to 80 bytes, 72 for popup)?
72
Maximum record length =72
```

Copy the new file popup.lib into the installation directory.

10.7.4 Driving PRAXIS

For details of driving PRAXIS please see the SCALE User's Manual.

10.7.5 Restarting - the stack file

At the end of the proforma, at the editing calculations stage, the values of variables given in response to the prompts are written to a file with name ending with .STK (standing for stack); for example if the proforma was called BEAM.PRO then the values of the variables would be written to a file called BEAM.STK. When the proforma is requested next time, then PRAXIS looks for a file called BEAM.STK, and if found, extracts the old prompts and offers them to the user as default values.

If the data typed in is long and important, then it can be saved to a specific stack file, by specifying the name of the stack file at the start of the run.

The first time an option is run, accept any offer of a set of default values. Default values are provided so that the user can go through the calculation or drawing and see the information that is required without having to type in sensible values themself. If the user refuses any offer, then the program will extract values from the stack file: filename.stk if available, and offer these as 'default', values. If the user requires the program to extract values from a different stack file, they can give its name or simply press Return for normal operation.

When PRAXIS was started and BEAM.PRO again nominated, then provided that any 'standard example values' built into the proforma were refused, the old values would be offered as defaults.

10.7.6 Pre-processors for PRAXIS

A pre-processor for PRAXIS may be programmed, in any language, to write a set of assignments in a named text-file as described in section 10.6.15. For example an e.g. Python/Perl/PowerShell program could write a file of parameters and also enter the name of the data file (where page headings are to be found) followed by the required option number in the special file called FIL.NAM containing for example:

```
C702.DAT/w070
```

The pre-processor would then invoke PRAXIS which would automatically extract the headings from the data file C702.DAT, the proforma from

SC070.PRO (LU070.PRO for LUCID etc.) and 'pull-forward' the parameters from SC070.STK (LU070.STK for LUCID etc.) and offer them as defaults (see external tables and procedures in section 10.5.6 and external assignments in section 10.6.15). The w switch stands for Windows and causes the window containing the option number (70 in this example) to be invoked. If PRAXIS finds that an option number is not given in FIL.NAM (e.g. FIL.NAM contains C702.DAT/w) then PRAXIS treats this as an instruction to Stop.

In a similar way to that described above batch file operation of PRAXIS can be achieved by the /b switch e.g.

C702.DAT/bLU220.pro

would automatically extract the headings from the data file C702.DAT, the proforma from LU220.PRO (LU for LUCID) and 'pull-forward' the parameters from LU220.STK and use them as responses to each prompt automatically producing the drawing in file C702.CAL without prompting the user.

10.7.7 Post-processors for PRAXIS

A post-processor for PRAXIS may be programmed, in any language, to treat the calculations file produced by PRAXIS as a file of data.

NL-STRESS (Non-Linear STRESS) is a structural analysis program which, when invoked, reads the file FIL.NAM and if it finds a /w switch appended to the name of a data file e.g. C702.DAT/w, it automatically starts reading the data from C702.DAT and when it runs out of data opens the calculations file C702.CAL (in this example) and continues reading the data from this calculations file which has been previously created by PRAXIS. Adding switches to FIL.NAM and automatically invoking any executable program from within PRAXIS can be easily achieved by the batch facility already described (see 10.6.19). Paging of the calculations file can be suppressed by setting PAGELENGTH 2000 (see 10.6.14).

10.7.8 The SCALE interface with NL-STRESS

Proformas which may be used to "pull forward" moments, shears etc. from an NL-STRESS analysis into SCALE, commence with the command SCALE. This command causes the SCALE program to prompt for 'Member Number?'.

The information pulled-forward is not dependent on the proforma selected but is dependent on the type of structure analysed - plane frame, space frame etc. The ordered information is tabulated below for users who wish to write their own 'code check' proformas which interface with NL-STRESS.

Plane truss and plane frame:

Member number
MZ maximum moment about z axis
FY maximum shear force in local y direction
FX maximum axial force from either end (compression is positive)
Length of member
DY depth of section (if given in the analysis)
DZ breadth of section (if given in the analysis)
TY thickness of section in y direction (if given in the analysis - flange thickness for ISECTION)
TZ thickness of section in z direction (if given in the analysis - web thickness for ISECTION)

Plane grid:

Member number
MY maximum moment about local y axis
FZ maximum shear force in z direction
MX maximum moment about local x axis from either end - torsion
Length of member
DZ depth of section (if given in the analysis)
DY breadth of section (if given in the analysis)
TZ thickness of section in z direction (if given in the analysis - flange thickness for ISECTION)
TY thickness of section in y direction (if given in the analysis - web thickness for ISECTION)

Space truss and space frame:

Member number
MX maximum moment about local x axis from either end - torsion
MY maximum moment about local y axis
MZ maximum moment about local z axis
FX maximum force in local x direction from either end - axial load
FY maximum force in local y direction
FZ maximum force in local z direction
Length of member
DY depth of section (if given in the analysis)
DZ breadth of section (if given in the analysis)
TY thickness of section in y direction (if given in the analysis - flange thickness for ISECTION)
TZ thickness of section in z direction (if given in the analysis - web thickness for ISECTION)

So that each proforma can know how many values are pulled-forward, a special variable NRESP (Number of RESPonses) is passed to each proforma. NRESP is set positive when values are passed from NL-STRESS, else NRESP is zero or negative e.g.

```
NRESP=-2 if value is obtained from the .STK file
     =-1 if default values
     =0  if Changes clicked previously.
```

SCALE option 678 for guyed masts gives notes for those wishing to automate the analysis and design of special structures, and gives further information on the interface between SCALE & NL-STRESS.

As the pulled-forward "Location" is assigned to string $1, which on appearance of the first printed line is overwritten by the page headings, it is essential that the Location prompt is the first line to be printed. It is also essential that the prompts are in the above order. The assignment strings in front of the START will upset the pulling-forward process as string assignments are saved in the response stack and therefore affect NRESP; it is however permissible to have string assignments after the START, and among the prompts for the pulled-forward values, as the NRESP is set at the time of appearance of the START.

10.8 Error messages

When PRAXIS finds an error in the proforma it reports the error to the screen and to the calculations file. During development of a proforma there may be several errors; but once a proforma has been developed errors are few and usually caused by looking up values outside the range of a table. The line number given refers to an erroneous line in the proforma file. Use a text editor to correct the mistake. Error message follow together with brief notes:

ASSIGNMENT FAILURE AT LINE n
The + sign in line n has been taken as the start of an expression or assignment. The usual source of error is an expression on the right hand side of the equals sign containing a variable which does not have a value. Other possibilities are the expression containing the square root of a negative number (complex numbers are not included in the current version of PRAXIS), division by zero or unrecognised function.

DEFINE MUST FOLLOW END OF CALCS. LINE IS n
DEFINE introduces a procedure which must be placed at the end of the proforma. So that program control does not run into the procedure a STOP must precede the first occurrence of DEFINE, each procedure being terminated by ENDDEFINE before the start of the next procedure. The last line in the proforma should be FINISH.

ENDDEFINE FOUND BUT NOT STARTED AT LINE n
An ENDDEFINE has been found at line n but no previous DEFINE to start the procedure.

ERROR in Boolean expression at line n
The expression should contain no gaps and each variable should have been previously assigned a value.

ERROR IN IF-ELSE-ENDIF STRUCTURE AT LINE n
IF-ELSE-ENDIF's should be properly nested. The whole structure must be contained either outside another structure or entirely between the IF and ELSE, or entirely between the ELSE and ENDIF. IF..ELSE..ELSE would be faulted.

ERROR IN REPEAT-UNTIL-ENDREPEAT STRUCTURE AT LINE n
Error may be due to IF-ELSE-ENDIF not being properly nested within the REPEAT-UNTIL-ENDREPEAT structure (see above).

FAILURE IN EXPRESSION SUBSTITUTION IN LINE n
Either an expression involves an unassigned variable or there is insufficient space on the line to insert the numerical value/s of the expression/s within the line.

PAGE LENGTH LESS THAN MINIMUM n
An attempt has been made to reset the page length to less than n lines.

LINE NOT READ FROM MEMORY. LINE IS n
Possibly hardware failure.

LINE NOT STORED IN MEMORY. LINE IS n
Out of space in allocated memory.

LOGICAL STRUCTURES AT DIFFERENT LEVELS AT MARKER AT LINE n
In response to any ???? prompt the user may type < signifying that control has to jump back to the previous line starting with >. The > marker has been found at a different level within an IF-ELSE-ENDIF or REPEAT-UNTIL-ENDREPEAT or DEFINE-ENDDEFINE logical structure to the level of the <.

NO. OF SIGNIFICANT FIGURES OUTSIDE RANGE 1-16 AT LINE n
The DIGITS command may only be used to reset within the range 1 to 16.

ONLY ONE ASSIGNMENT PERMITTED PER LINE AT LINE n
An assignment has been found starting with a plus sign at line n. In the current version of PRAXIS only one assignment is permitted per line, unless printing has been suppressed by starting the line with an exclamation mark. This restriction is required because of the three line expansion of assignments.

OUT OF DATA AT LINE n
Has FINISH been omitted from the end of the proforma?

OUT OF TABLE RANGE AT LINE n
Either the row or column look-up value is outside the range of the table. For tables having a header such as 'greater than 6' it is useful to add an extra row or column to make extrapolation unnecessary.

PI MUST NOT BE RESET AT LINE n
An attempt has been made to reset PI which is a special variable holding the value 3.14159265358979324 and may not be reset.

PROCEDURE NOT FOUND FOLLOWING DEFINE AT LINE n
A procedure name is missing or unacceptable following the DEFINE at line n.

TABLE ASSIGNMENT EXPRESSION NOT PRINTED AT LINE n
Failure to build the table assignment statement by inserting row and column values in a line of text in the calculations file.

TABLE NOT FOUND AT LINE n
Probably a reference to a table number at line number n but the table does not appear at the start of the proforma.

TABLE PREVIOUSLY STORED WITH DIFFERENT STRUCTURE. LINE IS n
It is permissible to overwrite a table with a different set of values but the second set of values must have the same number of columns and rows as the first.

TABLE ROW OR COLUMN NUMBER IS TOO LARGE, MAX NO. IS n
There may be no more than n rows or columns in a table in the installed version.

TABLE VALUE NOT ASSIGNED AT LINE n
Probably a row or column value is out of range.

TOO MANY LINES IN THE PROFORMA FILE. MAX PERMISSIBLE IS n

TOO MANY MARKERS FOUND. MAX PERMISSIBLE IS n

TOO MANY PROCEDURES FOUND. MAX PERMISSIBLE IS n

TOO MANY TABLE ENTRIES. MAX PERMISSIBLE IS n

TOO MANY RESPONSE PROMPTS. MAX PERMISSIBLE IS n

THREE PARAMETERS NOT FOUND AFTER STORE COMMAND AT LINE n
PRAXIS expects to find a table number, number of rows and number of columns following the STORE command.
The last example shows a doubly subscripted variable bc(i,j), for which there is a need to declare its size. Doubly subscripted arrays must have the number of columns declared, i.e. one dimension needs to be set. The dimension is set by assigning it to the array name e.g. bc=3, before the first use of the array in doubly subscripted form.
Functionality is important in programming; NL-STRESS allows subscripted variables to be used in: non-subscripted form e.g. bc6

```
                        singly subscripted form    bc(a-7)
                        doubly subscripted form    bc(i,j).
```

As stated above, it is necessary to declare the dimension of an array before its first use in doubly subscripted form. NL-STRESS stores its elements left to right, top to bottom, thus if bc=3 then the array bc(,) contains:

```
┌ bc(1,1) bc(1,2) bc(1,3) ┐
│ bc(2,1) bc(2,2) bc(2,3) │
│ bc(3,1) bc(3,2) bc(3,3) │
└ bc(4,1) bc(4,2) bc(4,3) ┘
```

It follows that bc6=bc(6)=bc(2,3). One use of such functionality is that a doubly subscripted array may be assigned on a single line e.g.
ac=3 ac1=VEC(1,0,0,0,1,0,0,0,1)
which would set up a unity matrix ac(,)=

```
┌ 1 0 0 ┐
│ 0 1 0 │
└ 0 0 1 ┘
```

without the need for doubly nested loops.

As just a single dimension (for the number of columns) is set for doubly subscripted variables, the number of rows in the array may be less than, equal to, or greater than the number of columns.
The array bc() above has 3 columns, and four rows.
The width (number of columns) may be re-dimensioned as required, e.g. if bc=6 is assigned then the array bc(,) is referenced:

```
┌bc(1,1) bc(1,2) bc(1,3) bc(1,4) bc(1,5) bc(1,6)┐
└bc(2,1) bc(2,2) bc(2,3) bc(2,4) bc(2,5) bc(2,6)┘
```

i.e. two rows when previously dimensioned =3, have now been put on a single row; the order - as stated before - is always left to right, top to bottom, and has not been changed, only the referencing.

10.9 Checking the logic of proforma calculations

This section provides help and guidance to SCALE proforma 924.

10.9.1 Introduction and bugs

The purpose of this option (sc924.pro) is two fold:

- to provide a system for automatically generating hundreds of sets of engineered data for testing SCALE proformas
- to provide a system for automatically generating hundreds of sets of engineered data for testing NL-STRESS parametric models.

SCALE Proformas are written using a notation called PRAXIS. PRAXIS contains three basic programming structures viz:
IF-ELSE-ENDIF   REPEAT-UNTIL-ENDREPEAT   DEFINE-ENDDEFINE

Obviously if any of the above programming structures is not properly closed, e.g. an ENDIF is missing, then the proforma will not operate properly. Each of the above programming structures may be 'nested' to any sensible level e.g. 200, but each must be properly nested. Examples of correct nesting are given in sc997. To make a proforma check that its nesting is OK, run SCALE and when prompted for an option number, type the option number to be checked followed by /P. For example if the option number to be checked is 792, then respond to the prompt with: 792/P

When the proforma is displayed, it will show the three basic programming structures, any programming structure errors are shown by:
>>
Sometimes one programming structure error, for example a missing ENDDEFINE, will cause other errors. Such errors are called 'bugs', programming structure errors are not the only bugs, other examples of bugs are described below.

10.9.2 Default values

'Default values' mean a set of data used for running a SCALE proforma thus avoiding the engineer having to type sensible values in response to the prompts.

Most proformas start with a prompt such as:

```
Would you like a set of default values to be provided; you can
use the default values as references and type your own values
beneath to replace them.
Answer (1=Yes, 0=No)              +ans=????
```

The engineer responds 1 to be offered the first (and possibly only) set of default values, 2 for the second set and so on. Typically three sets of default values are provided.

Before each new proforma is issued, or new revision to a proforma (to accord with a new British Standard) is issued, black and yellow pencil checks are carried out for each set of default values as part of Fitzroy's QA system. Such checks ensure that the proforma will run for the various sets of default values, but not necessarily for other combinations of data such as the following simple example which contains a bug and would run for the continuous case only.

```
START
Continuous or not (1 or 0)   +con=????
IF con=1
Area of top steel            +a=???? mm2
ENDIF
Many lines of calculations...
Area of top steel provided   +a mm2
FINISH
```

If the user responds to the prompt for 'con' with zero, then the area of top steel 'a' will not be assigned; at the end of the calculation, when the summary is being printed, the 'Area of the top steel +a mm2' cannot be printed as 'a' does not have a value, in other words the proforma has a bug. The only way of detecting such bugs, is to have hundreds of sets of test data and run them with the proforma observing any error messages and editing the proforma accordingly.

Proforma sc924, which includes a procedure called 'allcom' (all combinations), automates the checking of any sensible number of different combinations of data. Typically 100 or more sets of data are generated and run and resulting calculations including errors such as '**Assignment failure...' are concatenated and included in the results (sc924.res file) for inspection.

10.9.3 The nature of data

Structural calculations need data, the items of data are of many different types, it is impossible to devise a procedure which will check the logic of a proforma calculation unless the procedure includes for the effects of the various types and range of data. Some examples of the various types of prompt for data follow. Let us assume that we wish to generate 20 sets of data to embrace the following 5 prompts - even though they are unrelated.

(1) Axial compressive load +P=???? kN
This prompt expects any 'reasonable' value, the proforma will trap excessive loads dependent on the element being designed. A maximum axial concentric compressive load of 16000 kN would be reasonable for a Universal Column but not for a rolled steel angle. To generate 20 sets of data we need only divide up the range between the lowest & highest values for P by 19, giving 20 values for 'P'.

(2) Bar diameter (12,16,20,25,32,40) +dia=???? mm
This prompt expects a reinforcing bar diameter in the range 10 to 40mm. Non recognised bar diameters would be faulted by the proforma. To generate 20 sets of data we can cycle: 12,16,20,25,32,40,12,16,20,25,32,40,12,16,20,25,32,40,12,16 thus giving 20 values for 'dia'.

(3) Continuous/not-continuous (1 or 0) +con=????
This prompt expects 1 or 0 as a response, responding 0.5 to a prompt which expects a 'true' or 'false' type answer makes no sense (unless the proforma has been written using fuzzy logic). To generate 20 sets of data we can cycle: 1,0,1,0,1,0,1,0,1,0,1,0,1,0,1,0,1,0,1,0 thus giving 20 values for 'con'.

(4) Overall depth of concrete beam +h=???? mm
(5) Effective depth of concrete beam +d=???? mm
Prompts (4) & (5) are related, as the effective depth of the beam will always be less than the overall depth (unless the reinforcement is external to the section). For a normal concrete beam of overall depth 450 mm, an effective depth of 350 to 400 mm is sensible, dependent on: fire rating, exposure condition, bar size & number of layers. To generate 20 sets of data for 'h' having a minimum value of 260 mm to a maximum of 450 mm we can cycle: 260,270,280,290,300,310,320,330,340,350,360,370,380,390, 400,410,420,430,440,450 mm. To generate 20 sets of data for 'd' having a minimum value of 210 mm to a maximum of 400 mm we can cycle: 210,220,230,240,250,260,270,280,290,300,310,320,330,340, 350,360,370,380,390,400 mm.

Obviously, the above sets of values would only test one rising set of values with another rising set of values. It is desirable to test one rising set of values with a falling set of other values. To do this we need patterns such that each set of rising values is tested against every other set of falling values. This is covered below in 'How a variable can vary".

'Dependency', of one variable on another, implies an 'order' of data e.g. the overall depth of a concrete beam should come before the effective depth so that reinforcement cover requirements, including fire, may be checked. The relationship between prompts (4) and (5) above is discussed in section 10.9.6. Although the 'order' of data is important when input is interactive (question and answer), it is not important when sets of data are provided for running in 'batch mode'. To explain the reason for this, it is necessary to define the difference between two sets of structural engineering data:

- a set of 'trivial data' consists of predominantly numeric data in which the order of the input of the data is critical e.g. when the value 5 is read it, because of its order it means that 5 sets of coordinates follow
- a set of 'parametric data' consists of a set of parameter assignments e.g. P=640 dia=32 con=1 h=800 d=740.

Obviously, if an additional item of data is added to a set of 'trivial data' e.g. a reference number is included near the start, then subsequent data will be out of synchronization for all remaining data. For a set of 'parametric data', the order of data when read as a batch is immaterial as all the items are read onto a 'stack' and numerical values and character strings are accessed by reference to their symbolic names.

It will be clear from the above, that no single 'switch' can be set to test the logic of a proforma, as the testing procedure needs to take into account the 'type', 'range' and possible 'dependency' of each variable with every other variable. Strictly speaking we should use the term 'symbolic name' rather than 'variable'. The example which follows as the symbolic name 'ans' must always hold the value zero to refuse the offer of a set of default values, thus 'ans' is a constant.

The remainder of this help describes how to prepare a table containing all parameters which can be used by a standard program (sc924) to test the logic of a proforma, or an NL-STRESS model, for robustness.

10.9.4 Example

Let us imagine the following five lines constitute a complete proforma.

```
Default values (1=Yes, 0=No)        +ans=????
Axial load                          +P=???? kN
Continuous/not-continuous (1 or 0)  +con=????
Bar diameter (16,20,25)             +dia=???? mm
Exposure condition (1 to 5)         +xpo=????
```

and we wish to test all combinations of data ("All of the colours in all of the sizes"). Obviously we could run the proforma and type in small and large bar diameters, and for each: continuous or not-continuous, and for each all the exposure conditions. Such an exercise would take a considerable amount of dedication when the data is not just 5 numbers, as above, but extends to say 50 numbers. It is desirable to be able to automate the process.

The process of providing 100 sets of data for a proforma containing 50 prompts requires 5000 items of data to be assigned. It is considered that such an exercise would be too onerous for most authors (including the writer), so a shorthand method has been devised. Usually engineers prefer 'examples' to 'syntax', so there now follows test data for the simple example given at the start.

First we need to provide the symbolic names of all the prompts:

```
ans  P  con  dia  xpo
```

Next to provide a start and end numerical value for each prompt.

```
Start values:  0  100  1  16  1
End values:    0  400  0  25  5
```

For ease of maintenance it is convenient to provide such data in tabular form thus:

```
PARAMETER  Start   End     Type    Dependency conditions
No. name   zst()   zen()   zty()
  1 ans    0       0
  2 P      100     400
  3 con    1       0
  4 dia    16      25
  5 xpo    1       5
```

The keyword PARAMETER starts the table of verification data, a blank line terminates the data. So far we have tabulated that the name of the first variable is 'ans' which starts and ends with zero, the name of the second variable is 'P' which starts at 100 and ends at 400; the name of the third variable is 'con' which starts at 1 and ends at 0; the name of the fourth variable is 'dia' which starts at 16 and ends at 25; the name of the fifth variable is 'xpo' which starts at 1 and ends at 5. Although this is not a complete definition of the data, the exercise has been made simple.

Beneath the columns headed Start & End in the PARAMETER table are zst() & zen() respectively. When the table is read the start values for the five variables are stored in zst(1) to zst(5), similarly the end values are stored in zen(1) to zen(5). For simplicity we will assume that each variable takes 3 values. The variable 'zni' holds the number of increments e.g. if zni=3 then P takes values 100, 250 & 400.

The names of variables: zni, zst(), zen() etc. start with a 'z' to distinguish them from other variables such as: ans P dia con xpo. The names of the variables are stored in string numbers starting at $27000 & above to distinguish them from other string numbers, thus the names of variables: ans P con dia xpo, are stored in $27001 to $27005 respectively.

Once the above exercise has been completed, we have broken-the-back of the problem. There remains the need to define how each variable can vary. The first symbolic name 'ans' needs to be a constant of value zero so that sets of default values, at the start of any proforma, are ignored.

For the second variable 'P', by linear interpolation, three increments of: 100 kN, 250 kN, 400 kN would be sensible, we can say that its type=0 (remember 0=Ordinary).

For the third variable 'con' then - by interpolation - three increments of: 1, 0.5, 0 would not be sensible, but if we say that its type =2 (remember 2 or more means 'cycle' - bicycle for Queen fans) then cycling the values 1 & 0 for 3 increments would be sensible. For zni=1 to 3, then 'con' would take values: 1 0 1, for zni=1 to 4, then 'con' would take values: 1 0 1 0, and so on.

For the fourth variable 'dia', by linear interpolation, three increments of: 16 mm, 20.5 mm, 25 mm would not be sensible; but if we say that its type =1 meaning that after linear interpolation we round down to the nearest integer value (remember 1=Integer), then bar diameters of: 16 mm, 20 mm, 25 mm would be computed and would be sensible for certain conditions e.g. main steel for a suspended floor slab, providing that the number of increments is always 3 i.e. zni=3. If zni=4 then bar diameters of: 16 19 22 & 25 mm, would be computed and would not be sensible, so we could say that its type =3 meaning always cycle for three values, which would cycle bar diameters of: 16 mm, 20.5 mm, 25 mm. To say that the three values have also to be integer, we will say that its type =-3.

For the fifth variable 'xpo' then - by interpolation - three increments of: 1, 3 & 5 would be OK but not thorough, if we say that its type=5 then cycling the values 1,2,3,4,5 would be sensible. Adding the types for the five parameters gives the following.

PARAMETER No.	name	Start zst()	End zen()	Type zty()	Dependency conditions
1	ans	0	0	0	
2	P	100	400	0	
3	con	1	0	2	
4	dia	16	25	-3	
5	xpo	1	5	5	

Summarising Type: 'zty()' defines how the parameter is to be varied between the start value zst() and the end value zen(). The () denote subscripted with the parameter number, e.g. zty(4)=-3. During verification, the engineer responds to the prompt for zni where zni is the number of increments between the start and end values inclusively. Briefly zty() is classified as follows.

- 0 means no modification i.e. zst() - zen() in zni equal increments inclusively.
- 1 means as for 0, but make all values integer (remember by 1=I)
- 2 means split into 2 increments and cycle with the 2 increments, similarly for 3,4,5... When it is required to cycle AND make all values integer, use -2,-3,-4...
- 91 to 99 means split into 1 to 9 increments and cycle with the 1-9 increments always progressing from zst to zen.
- -91 to -99 means split into 1 to 9 increments and cycle with the 1-9 increments always progressing from zen to zst.
- 200,300...20000 means make the computed value for each increment exactly divisible by 2,3...200 respectively.
- 100 means interpret the zst() & zen() as pointers to the start and end elements in za(), if za(1)=VEC(6,8,10,12,16,20,25,32,40,50), zty()=100, zst()=3, zen()=8 then the 6 values 10,12,16,20,25,32 will be extracted and cycled, dependent on 6 being <=> zni.
- 101 to 125 similar to 100 but for zb() to zz().
- 126-151 are used for importing small sets of numbers, see chapter 13. The small sets of data are stored in za'(1), za'(2)... for zty() =126 up to zz'(1), zz'(2)... for zty() =151.
- 152 means treat the parameter as a 'component number', e.g. see vm452.dat.

There are some similarities between zty=101-125 & zty=126-151. Both are for specifying small sets of numeric data; for zty=101-125 the set of data must be specified in proforma sc924.pro; for zty=126-151 the set of data is given in the parameter table.

10.9.5 Patterns

The foregoing treatment will produce test data for running every parameter over a range of values specified by the engineer; but does not run a rising range of one parameter against a falling range of another parameter. To run a rising parameter against a falling parameter, or vice-versa, for each and every parameter the procedure needs to reverse the start and end values of every single variable with respect to every other variable. This can be achieved by having a pattern for direction of the range, cf. a pattern for considering live load on a continuous beam.

For five variables, a pattern of (1,1,1,1,1) means that all ranges go in the initial direction, where we define the initial direction as that specified by the engineer when setting zst() & zen(). In the previous PARAMETER table 'con' decreases (starts at 1 for continuous, and ends at 0 for non-continuous) whereas dia increases; as the engineer has specified them, we say that 1 represents the initial value for con and 0 represents the final value. A pattern of (1,0,1,0,1) means that even numbered parameters go in the reverse direction.

To run each parameter with its reverse, a mathematician would use a unity matrix, such as that shown below:

```
   1 0 0 0 0    where the first row says run the Initial value of the
   0 1 0 0 0    first parameter with the reverse of parameters 2 to 5;
   0 0 1 0 0    the second row says run the initial value of the second
   0 0 0 1 0    parameter with the reverse of the first and parameters
   0 0 0 0 1    3 to 5; and so on.  Five parameters would require five
                patterns additional to the basic  1 1 1 1 1 ...
```

It follows that fifty parameters would require fifty additional patterns. Engineers engineer problems by opting for simplicity, practicality at the expense of rigour e.g. using one depth for beams, rather than 16 different depths were the beams to be designed for 16 different sets of bending moments, shear forces and deflections. It is desirable to:

- limit the number of runs to avoid producing over a million pages of calculations
- run the initial range of each parameter with the reverse of every other parameter
- run the initial range of each parameter with the reverse of every pair of parameters
- run the initial range of a pair of parameters with the reverse of every other parameter
- and so on.

As an example, a rectangular beam of width b range 200-800mm, and depth d range 200-800mm, with the number of increments zni=7, will run a width of 200mm with a depth of 200mm, width of 300mm with a depth of 300mm, and so on. To run a width of 200mm with a depth of 800mm, or vice-versa, the procedure needs to reverse the start and end values of every single parameter with respect to every other parameter. This can be achieved by having a pattern for direction of the range, cf. a pattern for considering live load on a continuous beam.

For five variables, a pattern of 1 1 1 1 1 means that all ranges go in the initial direction, where we define the initial direction as that specified by the engineer when setting zst(n) & zen(n), respectively the start and end values of the range for the n'th parameter.

```
   PARAMETER  Start   End     Type    Dependency conditions
   No. name   zst()   zen()   zty()
     1 ans    0       0        0
```

```
  2 P      100     400      0
  3 con    1       0        2
  4 dia    16      25      -3
  5 xpo    1       5        5
```

In the PARAMETER table above 'dia' increases, whereas 'con' decreases (starts at 1 for continuous, and ends at 0 for non- continuous), but as the engineer has specified them, we say that 1 represents the start value for 'con' and 0 represents the end value. A pattern of 1 0 1 0 1 means that even numbered parameters go in the reverse direction.

Considerations need to be given to patterns of variation which do not loose sight of the engineering. For simplicity let us assume that we require a minimum number of patterns which will guarantee that every parameter which is increasing, need only be considered with every parameter which is decreasing, and that a minimum number of patterns is required.

```
Parameter No. 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 ...
Pattern 1     0 1 0 1 0 1 0 1 0  1  0  1  0  1  0  1  0  1  0  1 ...
Pattern 2     1 1 0 1 1 0 1 1 0  1  1  0  1  1  0  1  1  0  1  1 ...
Pattern 3     0 1 1 0 1 1 0 1 1  0  1  1  0  1  1  0  1  1  0  1 ...
```

Reading pattern 1 in association with the parameter numbers above, where 1 indicates a parameter is going in the original direction, 0 indicates the parameter is reversing, adjacent parameters go in opposite directions, parameters which step by odd numbers e.g. 1 4 7 10 13... 2 5 8 11 14... go in opposite directions as do 1 6 11 16... 2 7 12 17... 1 8 15... 2 9 16... and so on. An easy way to check the veracity of this is to use the first & little finger of the left hand as pointers to the figures in pattern 1 and check the parameter numbers above. By mathematical induction, pattern 1 shows that all parameters which have an odd step between, go in opposite directions. For patterns which have an even step between, repeat the exercise but reading patterns 2 & 3 in conjunction. Again by mathematical induction, patterns 2 & 3, read in conjunction for parameter numbers having an even step between, can provide opposite directions. When pattern 2 does not provide opposite directions, then pattern 3 will. To provide opposite directions for a parameter step of 2 i.e. parameter Nos. 1,3,5,7,9,11... pattern Nos. are: 2 2 3 2 2 3 2 2 3 2... For step equals 4 i.e. parameter Nos. 1,5,9,13,17... pattern Nos. are: 3 2 2 3 2 2...

Patterns 2 to 3 above, do not quite cover for the case of adjacent spans going in the original direction e.g. parameter numbers 3 & 4 always go in opposite directions. Obviously patterns 2 and 3 are the same but out of alignment by 1 parameter number, it seems sensible to add another pattern, shifted by one parameter to complete the set of 3 patterns of type: 1 1 0 1 1 0 ... which will provide for all adjacent pairs of parameters going in the original direction, prefaced & followed by parameters going in the reverse direction. Pattern 1 only considers even parameter numbers going in the original direction with odd parameter numbers going in the reverse direction. To provide for the case when adjacent parameters are of different types, the opposite to pattern 1 will be added, i.e. odd parameters going in the original direction. When building the definition table for parameters, the engineer has control over the primary direction of each parameter. All the patterns discussed above, are for varying the engineer's original choice of direction. For good functionality, a pattern which keeps to the original directions specified by the engineer will be added. Collecting all six patterns together and rearranging them into the familiar order for dealing with dead & live (imposed) load patterns on a continuous beam, we have:

```
1 1 1 1 1 1 1 1 1 1 ...  The 6 patterns shown to the left will cover
1 0 1 0 1 0 1 0 1 0 ...  for all parameters ranging over the original
```

0 1 0 1 0 1 0 1 0 1 ... direction specified, and for each parameter
1 1 0 1 1 0 1 1 0 1 ... considered with the reverse of every other
0 1 1 0 1 1 0 1 1 0 ... parameter, and for all sets of: 1 1 0 1 1 0
1 0 1 1 0 1 1 0 1 1 ... patterns where 1=initial direction, 0=reverse.

Number theory and coverage are important when devising patterns for engineering systems, but engineering is of paramount importance when dealing with sets of data for engineering models which are beset with discontinuities and interdependency between the parameters.

10.9.6 Dependency

Some parameters are dependent on other parameters e.g. the depth of a structural member is dependent on its length. Such dependencies are shown to the right of the PARAMETER table and commence with > = or <, usually followed by an expression. When the expression is too long to be contained on the same line as the parameter to which it refers, the > = or < are provided as flags to say: the expression is given on a line by itself following the table, in the order of occurrence (l to r, t to b) of any isolated > = or < flags given in the table. The > and < are used in the algebraic sense e.g. a<b<c or c>b>a meaning b is bounded by the limiting values of a & c.

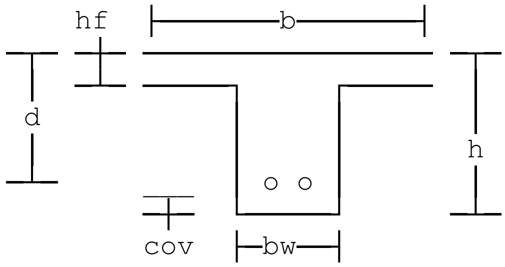
When a dependency condition is long and is given on a line following the parameter table as described above, internally, values of zva() are shortened to Z() to maximise the number of components in the expression. The routine 'showpar' which is normally suppressed, is invoked when the number of increments is given as a negative number, provides a summary of the parameters for debugging purposes. The summary shows Z() values for dependency conditions which follow the parameter table, rather than zva() values, the two symbolic names are synonymous and numerically equal. The absolute length limit for an expression which follows the parameter table is 70 characters, but an error will be reported if, substituting Z() values in place of the parameter names, causes the expression to exceed 70 characters in length. If an error is reported, it is usually straightforward to split the expression into two or more shorter expressions and use one or more dummy parameters to model the full expression. Inspection of the expression in the summary will reveal if the expression has been truncated.

As an example of dependency, let the effective depth 'd' be given as: d=INT(h-cov-dial-dia/2) where: h=overall depth (200-3000mm), cov=concrete cover to outside of links (20-80mm), dia=diameter of main bars (16-25mm), dial=diameter of links (8-12mm). For these five parameters, assuming 'h' is say the eleventh parameter, then the partial PARAMETER table below, describes sensible dependency conditions.

PARAMETER No. name	Start zst()	End zen()	Type zty()	Dependency conditions	
...					
11 h	200	3000	1		
12 cov	20	80	1	>h/10	<h/40
13 dia	16	25	-3		
14 dial	8	12	3	>dia/2	<0.75*dia
15 d	170	2950	1	=INT(h-cov-dial-dia/2)	
...					

When parameters are subscripted e.g. ax(1) to ax(10), ax is shown as the parameter name, preceded by its first and last parameter numbers. Assuming ax is the second parameter and the first is not subscripted, then the parameter numbers for ax(1) to ax(10) will be 2 to 11.

So far we have developed a compact (10-15 lines) treatment to define the start and end values for a range, for each and every input parameter for the design of a component; we have defined how each parameter may vary within its range, classifying it as a type 0, 1, 2 etc.; we have provided a system for dealing with the approximately one percent of parameters which have irregular ranges; we have identified patterns to be applied to vary each parameter with every other parameter for each pairing going from the start of its range to the end of its range, and the other going from the end of its range to the start. From reviewing a few hundred calculations, approximately 15% of the parameters are dependent on the current values of one or more other parameters. As another example of dependency, consider the following reinforced concrete Tee beam.



Dependency conditions
bw>=b/5 bw<=b/2
hf>=h/5 hf<=d/2
d=h-cov-20

PARAMETER No. name	Start zst()	End zen()	Type zty()	Dependency conditions	
...					
1 cov	20	75	500		
2 h	200	3000	500		
3 d	150	2900	1	=h-cov-20	
4 b	300	3000	500		
5 bw	150	1500	500	>b/5	<b/2
6 hf	75	1450	500	>h/5	<d/2

In the table above, 'cov' varies from zst(1)=20 to zen(1)=75mm, zty(1)=500 rounds to the current increment to 5mm; 'h' varies from zst(2)=200 to zen(2)=3000mm; 'd' is computed from 'h-cov-20' then checks that it is in the range 150 to 2900 adjusting for compliance; b is computed in the range 300 to 3000 according to the increment number, then rounding to the nearest 5mm; bw is computed in the range 150 to 1500 according to the increment number, then rounding to the nearest 5mm then checks that it is in the range b/5 to b/2 adjusting for compliance; hf is computed in the range 75 to 1450 according to the increment number, then rounding to the nearest 5mm, then checks that it is in the range h/5 to d/2 adjusting for compliance. When the foregoing data is run by a standard procedure, the following sets of test values are produced, rearranged below for clarity. The table below includes 6 patterns and 4 increments, all values for the parameters are given as numbers. The first table shows the sets of test data when all dependencies are suppressed, the second table includes the dependencies. Errors caused by ignoring the dependencies are shown starred. Were 3 increments chosen then the test data for Increment 2 would show that all six patters are identical, this follows from the fact that with just three increments, the middle increment values will always be computed as average of the start and end values whatever the direction of parameter increase. In a proper test for verifying the correctness of the logic of a calculation, the engineer would normally set a higher number of increments.

TABLE OF SETS OF TEST DATA IGNORING DEPENDENCIES

	INCR 1	INCR 2	INCR 3	INCR 4
	SET 1	SET 2	SET 3	SET 4
	cov=20	cov=40	cov=55	cov=75
	h=200	h=1135	h=2065	h=3000
PATTERN	d=150	d=1066	d=1983	d=2900
1	b=300	b=1200	b=2100	b=3000
	bw=150	bw=600	bw=1050	bw=1500
	hf=75	hf=535	hf=990	hf=1450

	INCR 1	INCR 2	INCR 3	INCR 4
	SET 5	SET 6	SET 7	SET 8
	cov=20	cov=40	cov=55	cov=75
	h=3000	h=2065	h=1135	h=200
PATTERN	d=150*	d=1066	d=1983*	d=2900*
2	b=3000	b=2100	b=1200	b=300
	bw=150*	bw=600	bw=1050*	bw=1500*
	hf=1450	hf=990	hf=535	hf=75

	INCR 1	INCR 2	INCR 3	INCR 4
	SET 9	SET 10	SET 11	SET 12
	cov=75	cov=55	cov=40	cov=20
	h=200	h=1135	h=2065	h=3000
PATTERN	d=2900*	d=1983*	d=1066*	d=150*
3	b=300	b=1200	b=2100	b=3000
	bw=1500*	bw=1050	bw=600	bw=150*
	hf=75	hf=535	hf=990	hf=1450

	INCR 1	INCR 2	INCR 3	INCR 4
	SET 13	SET 14	SET 15	SET 16
	cov=20	cov=40	cov=55	cov=75
	h=200	h=1135	h=2065	h=3000
PATTERN	d=2900*	d=1983*	d=1066*	d=150*
4	b=300	b=1200	b=2100	b=3000
	bw=150	bw=600	bw=1050	bw=1500
	hf=1450*	hf=990*	hf=535	hf=75*

	INCR 1	INCR 2	INCR 3	INCR 4
	SET 17	SET 18	SET 19	SET 20
	cov=75	cov=55	cov=40	cov=20
	h=200	h=1135	h=2065	h=3000
PATTERN	d=150	d=1066	d=1983	d=2900
5	b=3000	b=2100	b=1200	b=300
	bw=150*	bw=600	bw=1050*	bw=1500*
	hf=75	hf=535	hf=990	hf=1450

	INCR 1	INCR 2	INCR 3	INCR 4
	SET 21	SET 22	SET 23	SET 24
	cov=20	cov=40	cov=55	cov=75
	h=3000	h=2065	h=1135	h=200
PATTERN	d=150*	d=1066*	d=1983*	d=2900*
6	b=300	b=1200	b=2100	b=3000
	bw=1500*	bw=1050*	bw=600	bw=150*
	hf=75*	hf=535	hf=990*	hf=1450*

TABLE OF SETS OF TEST DATA CONSIDERING DEPENDENCIES

	INCR 1	INCR 2	INCR 3	INCR 4
	SET 1	SET 2	SET 3	SET 4
	cov=20	cov=40	cov=55	cov=75
	h=200	h=1135	h=2065	h=3000
PATTERN	d=160	d=1075	d=1990	d=2900
1	b=300	b=1200	b=2100	b=3000
	bw=150	bw=600	bw=1050	bw=1500
	hf=75	hf=535	hf=990	hf=1450

	INCR 1	INCR 2	INCR 3	INCR 4
	SET 5	SET 6	SET 7	SET 8
	cov=20	cov=40	cov=55	cov=75
	h=3000	h=2065	h=1135	h=200
PATTERN	d=2900	d=2005	d=1060	d=150
2	b=3000	b=2100	b=1200	b=300
	bw=600	bw=600	bw=600	bw=150
	hf=1450	hf=990	hf=530	hf=75

	INCR 1	INCR 2	INCR 3	INCR 4
	SET 9	SET 10	SET 11	SET 12
	cov=75	cov=55	cov=40	cov=20
	h=200	h=1135	h=2065	h=3000
PATTERN	d=150	d=1060	d=2005	d=2900
3	b=300	b=1200	b=2100	b=3000
	bw=150	bw=600	bw=600	bw=600
	hf=75	hf=530	hf=990	hf=1450

	INCR 1	INCR 2	INCR 3	INCR 4
	SET 13	SET 14	SET 15	SET 16
	cov=20	cov=40	cov=55	cov=75
	h=200	h=1135	h=2065	h=3000
PATTERN	d=160	d=1075	d=1990	d=2900
4	b=300	b=1200	b=2100	b=3000
	bw=150	bw=600	bw=1050	bw=1500
	hf=80	hf=540	hf=535	hf=600

	INCR 1	INCR 2	INCR 3	INCR 4
	SET 17	SET 18	SET 19	SET 20
	cov=75	cov=55	cov=40	cov=20
	h=200	h=1135	h=2065	h=3000
PATTERN	d=150	d=1060	d=2005	d=2900
5	b=3000	b=2100	b=1200	b=300
	bw=600	bw=600	bw=600	bw=150
	hf=75	hf=530	hf=990	hf=1450

	INCR 1	INCR 2	INCR 3	INCR 4
	SET 21	SET 22	SET 23	SET 24
	cov=20	cov=40	cov=55	cov=75
	h=3000	h=2065	h=1135	h=200
PATTERN	d=2900	d=2005	d=1060	d=150
6	b=300	b=1200	b=2100	b=3000
	bw=150	bw=600	bw=600	bw=600
	hf=600	hf=535	hf=530	hf=75

To specify that a start and/or end value is to be computed from an expression, give its start or value as 1E20, and enter the expression/s as dependencies commencing with a '>' prefixing the expressions for the start value and a '<' prefixing the end value. When the program reads the value 1E20 for zst(n) then it evaluates the expression prefixed by '>' and uses that instead of 1E20; similarly when the value 1E20 is read for zen(n) then it evaluates the expression prefixed by '<' and uses that instead of 1E20. Care must be taken when using this device as the limits will change dependent on the parameters contained in the expressions.

Note that this is a different procedure to that used when expressions follow '>' and/or '<' under 'Dependency conditions' but zst(n) and/or zen(n) do not contain 1E20; for this case the value of the expression contained in the strings is used to act as a limit to the zva(n) computed from the numbers held in zst(n) & zen(n). Generally the computed value zva(n) lies within the range zst() and zen(), the exception being when either contains 1E20.

10.9.7 Running the sets of data twice

As values are provided for ALL prompts, then data may have been provided for cases when the prompt is within a programming structure e.g. DEFINE-ENDDEFINE and should not have been provided for the set (e.g. depth to compression steel would not be prompted-for unless the calculation showed that compression steel was required). The solution to this problem is to run the complete set of data (which may contain superfluous data), then rerun the problem using the .STK file alone, as the stack file produced from the first run contains only those items of data which have been 'prompted-for' during the first run, the second run will only have access to the set of data that is needed. The following gives an example, the double asterisk marks the bug which would pass the first run but not the second:

```
Rectangular (1) or Tee-beam (2)  +btype=????
Depth of beam                    +dbeam=????
IF btype=2
Depth of top flange              +dtop=????
ENDIF
IF dtop>dbeam/2 **
STOP Depth of top flange exceeds half of beam depth.
ENDIF
```

Obviously if values were provided for all parameters e.g. btype=1, dbeam=400, dtop=100, then the above would work; but if btype=1 then 'dtop' would not be 'prompted-for' and in consequence the Boolean expression 'dtop>dbeam/2' would fail. The second 'run' would pick up the bug, the first 'run' would not.

Running the defaults twice introduces a complication; if the calculation has been terminated before the end because say a stress combination has exceeded unity, then prompted-for values which follow would not have been input, and thus data for the second run would be missing from the .STK file written at the end of the first run. Thus using the .STK from an incomplete calculation, and modifying the data slightly to correct for an incomplete calculation, will not guarantee a successful second run free for errors which start with **; for this reason it is recommended that time be spent on choosing ranges of values for the parameters so that at least 50% of every set of calculations runs through to completion.

10.9.8 Robustness

Even quite trivial programs with less than 10 programming structures, can have thousands of different paths through their logic. The suggested strategy given above will not guarantee that every single path through a 'proforma' is tested, but it does provide a reasoned approach to testing and it is straightforward to add the PARAMETER table near to the start of a proforma.

Praxis (the notation in which SCALE, LUCID... proformas are written), tells the engineer when it finds an error in an assignment or Boolean expression and reports the line number at which the error occurred (errors occur because one of the parameters has not been set).

To avoid errors in looping, Praxis keeps a count of the number of times each and every line in the current proforma is accessed, and if the number exceeds 128000, then an error in looping is reported. In procedure 'allcom' there are four nested loops, from outside to inside:

- the number of patterns (always 6)
- the number of increments (specified at run time)
- the number of parameters (typically 10 to 50)

Routine 'allcom' currently assumes a limit of 996 files will be produced (scrtch.001 to scrtch.996) containing the sets of parameters. For 6 patterns then the number of increments must be <= 996/6 =166, 'allcom' faults when number of increments exceeds 166. Thus the maximum number of times that any line is read will be in the order of: 6 x 166 x 50 = 49800; but will exceed the 128000 limit if the number of parameters increases to 150.

Incorporation of the PARAMETER table into a proforma - as described above - allows for the automation of the testing of many different combinations of the parameters, saving time, increasing the 'coverage' tested and increasing the robustness of the logic.

10.9.9 Typical component design

The classification of types of structural data in Chapter 4 for the automatic generation of sets of data to test the logic of proforma calculations is equally applicable to the automatic generation of sets of data to test the logic of a model for the structural analysis of a framework i.e. the PARAMETER table is appropriate to both. Nevertheless, the data required for structural design is often more complicated than that required for the structural analysis of a framework. The nature of structural analysis is that for a given model, variation of any parameter is accompanied by continuous behaviour of the model. Structural design models are often associated with discontinuous behaviour, for example the table which follows, taken from sc385, is typical of the discontinuous behaviour of the structural design processes.

<table>
<tr><th colspan="3">Table 22 Nominal effective length for a compression member</th></tr>
<tr><th colspan="2">Conditions of restraint at ends</th><th>Effective Length</th></tr>
<tr><td rowspan="2">Effectively held in position at both ends</td><td>Restrained in direction at both ends</td><td>K = 0.7</td></tr>
<tr><td>Partially restrained in direction at both ends</td><td>K = 0.85</td></tr>
</table>

	Restrained in direction at one end	K = 0.85
	NOT restrained in direction at either end	K = 1.0

Restraint factor z-z axis +Kz=????
Restraint factor y-y axis +Ky=????

Most structural engineers are familiar with the set of bar diameters: 6,8,10,12,16,20,25,40 & 50. This set gives rise to other parameters having discontinuous values e.g. reinforcement cover, lap lengths etc. which are defined in codes of practice in terms of bar diameter. Fire resistance requirements of 0.5, 1, 2 & 4 hours is another parameter having discontinuous values. Strategies are required for dealing with such parameters including both short and long tables e.g. steel section properties.

Before the advent of computers, the analysis of structural frameworks was considered difficult and the province of engineers, but the structural design of members was considered to be straightforward and accordingly was delegated to technicians. This division of work between engineers and technicians has now changed in some firms, with technicians preparing the data for global models, and engineers being rightly concerned with materials and the local design of structural components such as beams and columns. Today the longhand design of say a steel beam is protracted, and even when carried out using computer software, the concepts in the design are far from the concepts of BS449.

All the types of engineering data classified in Chapter 4 are equally applicable to the structural design of components and to the analysis of structural frameworks.

For manufactured components such as Universal Beams etc. further types of data are required to cater for say a 203x133x25 UB. Tools for handling such items of data are developed below.

10.9.10 Engineers' arithmetic

One important requirement for software is that the logic be robust. To test the logic against a thousand sets of data for a structural program requires that the data be appropriate and non- trivial. An example of trivial data is a square hollow section beam 400x400x15, spanning 3 m and carrying a minimum distributed load of 1 kN/m and a maximum distributed load of 10 kN/m. To avoid such trivial data, cognizance of structural engineering is required i.e. the ability to produce back-of-envelope calculations, also known as engineers' arithmetic.

The tools of engineers' arithmetic include:

- limiting maximum span:deflection ratio to say 384
- limiting span:depth ratio to be in the range 8 to 24
- limiting the maximum bending moment such that the design strength times area of steel in tension times depth of section is not exceeded
- assuming area of steel in tension = cross-sectional area/3
- assuming aspect ratio (depth:breadth) of beam varies from 1 to 3 with a typical value =2.

The order in which these tools are applied depends on the order of known values e.g. if the maximum bending moment and the span are known, then the equivalent distributed load may be found using M=w*L^2/8 rearranged as w=8*M/L^2.

Rearranging delta=5*w*L*L^3/(384*E*I), span:deflection = L/delta =384=384*E*I/(w*L^3), knowing E for the material, I may be found from I=w*L^3/E.

Knowing D=L/14 where D is the section depth, & I=At*D^2, where At is the area of steel in tension, then At=I/(L/14)^2 =196*I/L^2.

Knowing A=3*At, then the cross sectional area A is known.

Knowing D/B=2 then breadth of section B=0.5*D, and flange thickness T=At/B=196*I/(B*L^2).

These expressions provide dependency conditions which may be used in the PARAMETER table to avoid trivial data or data which exceeds the model limits i.e. those under the column headings zst() and zen() in the PARAMETER table.

A large proportion of routine structural calculations are for the sizing of beams, slabs and columns. A large proportion of beams are simply supported. The maximum bending moment in a simply supported beam carrying a uniform total load of W kN is W*L/8 kNm; for the load W kN concentrated at the centre of the beam, the maximum bending moment is WL/4 kNm i.e. twice the bending moment for the load if it were uniformly distributed. For the approximate sizing of a beam, engineers work out W*L/8 and then allow a percentage increase depending on how the loading is distributed. Once the bending moment is known, it is divided by the lever arm (assumed as 75% of the beam depth for a reinforced concrete beam or 90% for a structural steel section) to give the tension or compression force, which for known permissible stress, gives the area of reinforcement, or steel beam flange size. Such engineers' arithmetic is carried out by engineers for both initial (scheme) design stage and for checking. Engineers' arithmetic relies on concepts such as: lever arm, modular ratio, load factor (typically 1.5) etc. Engineers' arithmetic is not limited to scheme design and checking, but has considerable use in setting sensible dependency conditions for the PARAMETER table, accordingly engineers' arithmetic is used in the design of the sets of test data for increasing the robustness of structural design calculations.

10.9.11 A Typical proforma for steel component design

SC385 for the design of stainless steel square and rectangular hollows sections is used to describe the process of building the PARAMETER table. For ease of reference, it is recommended that flowcharts be printed for both SC385 and for SC3800 which is invoked by SC385. To do this run SCALE and in response to the prompt for Option number, type 385/P and press Enter and click the Print button; repeat but type SC3800.PRO/P and press Enter and click the Print button.

Inspection of the printout shows that there are 47 prompts for parameters: $92 Mz Fv F L D' B' T' t' My stype Lz Ly Kz Ky Ae moment restz Mz betaMz mz2 mz3 mz4 M24 resty My betaMy my2 my3 my4 My24 LT refno udl betaM m2 m3 m4 grade E $27569 sd(ssd1) sb(ssd1) st(ssd1) sd(srd1) sb(srd1) st(srd1). Each of these parameters is input interactively e.g.

```
Factored bending moment axis zz   +Mz=???? kNm
```

which causes the engineer to be offered the previous response, which may be accepted by pressing Enter, or replaced by the engineer typing a new value. As an alternative to interactively providing data, the proforma may be run in batch mode for which all values are extracted from a stack. Thus the requirement is to automate the production of hundreds of stack files containing the parameters, each stack file preferably being unique, and run all the stack files with the model, providing a self check at the end of each model with automatic reporting on results of the check for each run.

It will be apparent from inspection of prompts, that the task is more difficult than that for the analysis of structural frameworks.

The Proceedings of the 17th International Conference, Edinburgh, July 2005 on Computer Aided Verification contains 54 papers. All papers are written by experts in informatics, 'tools' deal with running a program and checking that the results are as expected, none deal with the verification of engineering software.

Inspection of the simplified flowchart for sc385/sc3800 shows that 'stype' takes the value 1 or 2, dependent on a square or rectangular hollow section being required. The model also allows a rectangular hollow section to be used in portrait or landscape orientation, this complication is mentioned as an example for demonstrating that an engineer is required for verifying engineering models. Although there are firms specialising in verification and self-checking software, verifying software for the structural analysis of frameworks and the structural design of components, requires engineering expertise.

To write a program that generates values for 47 parameters would be trivial if all parameters were continuous and each was independent from all others. To write a program that generates sensible values for 47 parameters

- with a large proportion being dependent on one or more other parameters
- with some parameters obtaining their values from tables and other indirect methods
- with the full set of parameters passing all error checks in the model, would be difficult but straightforward if each model had its own program for generating data.

To write one program for generating sets of values for interdependent parameters for 800 models, as described above, has been achieved in SC924.

10.9.12 Dummy and subscripted parameters

Dummy parameters are parameters which are not input, but which are required in the parameter table so that they may be used in the self checking logic and/or the dependency conditions. An example of a table which requires dummy parameters is given below. If the permissible design strength of the structural material is known, then the design strength is a parameter and may be used for both the input data and for the material strength used in the self- check. Occasionally, the authors of codes refer to types of materials rather than strengths of materials e.g. stainless steel to BS EN 10088-2 described by:

Grade	Number	Type
Basic chromium-nickel austenitic	1.4301	1
Molybdenum-chromium-nickel austenitic	1.4401	2
Molybdenum-chromium-nickel austenitic	1.4404	3
Duplex Steel	1.4362	4
Duplex Steel	1.4462	5

```
Steel type (1,2,3,4 or 5)              +grade=????
```

In the above table, the design strength will be dependent on the Type chosen. For Types = 1 to 5, the design strengths in BS EN 10088-2 are: 210,220,220,400,460 N/mm2 respectively. If a model were to input the design strength directly e.g. 220 N/mm2, then the molybdenum-chromium-nickel austenitic grade would be ambiguous, as its Number could be either 1.4401 or 1.4404. Accordingly the steel is best referred to by its type, with the model consulting a short table to give the design strength and Number.

It will be evident that for structural calculation models, parameters have to be set which may or may not be required e.g. for sc385, properties for both square hollow sections and rectangular hollow sections are required dependent on 'stype' taking the value 1 or 2. To carry out a thorough check on the logic of a model to detect the presence of unassigned variables etc., two runs are necessary; the first including all parameters which may or may not be required for the run, the second including only data actually used in the first run. Thus for the first run with 'stype=1' which specifies a square hollow section, section dimensions for depth, breadth & thickness will be provided for both square & rectangular hollow sections; for 'stype=1' section dimensions for rectangular hollow sections are one type of dummy parameter. For the second run with 'stype=1' which specifies a square hollow section, section dimensions for depth, breadth & thickness will only be provided for square hollow sections.

Another type of dummy parameter is needed when a parameter, such as design strength, is required for the self-check and that parameter has not been input directly but looked-up or computed during the first run. For such a situation, the parameter will not normally be passed to the second run. To force a pass from the first run to the second run, a dummy prompt is needed, preceded by a fast forward (>>) and followed by a stop fast forward (><) e.g. for the design strength:

```
>>
!Design strength                 +py=????
><
```

The three lines above are not seen in the calculations, but cause 'py' to be written to the stack of values, just as though its value had been input by the engineer.

Occasionally it is advantageous to use subscripted parameters e.g. for a continuous beam having 10 spans, say s(1) to s(10) then assuming s(1) was the 4th parameter, then the follow section from a parameter table would suffice.

```
PARAMETER  Start   End     Type    Dependency conditions
No. name   zst()   zen()   zty()   and notes.
...
4 13 s     1.5     6       0       Spans s(1) to s(10)
14 ...
```

Optionally the 13, corresponding to s(10) may be omitted from the table as it may be assumed to be one less than 14 i.e. the next parameter number in the table. When a subscripted parameter is the last in the table, both parameter numbers must be shown.

10.9.13 Storage of short tables

To look-up a table and extract the design strength would normally require a procedure; when the table is short, as for the design strengths in BS EN 10088-2 (210,220,220,400,460 N/mm2) corresponding to the five types of steel given in the previous table, then the table may be incorporated into the PARAMETER table e.g.

```
PARAMETER  Start   End     Type    Dependency conditions
No. name   zst()   zen()   zty()   and notes.
...
10  grade  1       5       5
11  py1    210     210     2
12  py2    220     220     2
13  py3    220     220     2
14  py4    400     400     2
15  py5    460     460     2
16  py     210     460     0       =zva(10+grade)
...
```

In the program for generating the sets of data, zva(n) is the current value of parameter 'n', thus if grade=1 then py is set to the current value of zva(11) i.e. 210; if grade=2 then py is set to the current value of zva(12) i.e. 220 and so on. A further and more compact method of storing values for 'py' follows.

```
PARAMETER  Start   End     Type    Dependency conditions
No. name   zst()   zen()   zty()   and notes.
...
13  grade  1       5       5       For grades (1:5) py as below.
14  py     1       5       126     210 220 220 400 460
...
```

The above extract is taken from the parameter table for proforma sc385. Parameters 13 & 14 are fixed so that they are both in their ascendancy or descendancy. For confirmation, run option 385 with a negative sign before the number of increments; this produces tables of sets of data from which it can be seen that when: grade=1 py=210, grade=2 py=220, grade=3 py=220, grade=4 py=400, grade=5 py=460.

The very short parameter table below is an example of how to test a feature, in this case 'mapping' a set of six structural hollow section types to formula numbers used by 'sprops' which is the section properties procedure contained in sc924.pro. Firstly copy the heading and the lines to be tested to the start of the parameter table leaving a gap before the main table. The gap following the mini-table ensures that only the mini table is tested.

PARAMETER No. name	Start zst()	End zen()	Type zty()	Dependency conditions and notes.
1 stype	1	6	6	HFSHS,HFRHS,HFCHS,CFSHS,CFRHS,CFCHS.
2 formula	1	6	126	2 2 4 2 2 4

Secondly run proforma sc924.pro, selecting opn=2 for testing a SCALE proforma and giving the proforma/parameter table number to be tested as 392, and the number of patterns=1 and the number of increments=-1. The minus tells proforma/program 924 to tabulate the generated data only and not to test proforma 392. When the results are inspected the data for the six runs will be seen to contain the following in which the correct formula will be seen to be selected for 'stype' taking values 1 to 6.

```
run=1          run=2          run=3          run=4
stype=1        stype=2        stype=3        stype=4
formula=2      formula=2      formula=4      formula=2

run=5          run=6
stype=5        stype=6
formula=2      formula=4
```

Having found the correct formula for procedure 'sprops', we need to tell 'sprops' that the symbolic name 'formula' contains the required formula number. The start & end values defining the range of the parameter, by the parameter table rules, must be numeric. Arguments passed to a procedure are given under 'Dependency conditions'. The 'Type' of the pseudo parameter for calling a procedure, such as 'sprops' is + or -1E40. When it is given as 1E40 then the numerical constants given under zst() & zen() in the parameter table are passed to the procedure. Occasionally it is necessary to pass a variable value to a procedure; this may be done by giving the 'Type' as -1E40, which is interpreted as "Don't pass the numerical constants under zst() & zen(), instead treat the constants as pointers to parameter numbers higher up in the table and pass the current values of those parameters to the procedure". This is achieved in sc924.pro by:

```
IF zst>0 AND zst<zp' AND zst=INT(zst) THEN zst=zva(zst) ENDIF
IF zen>0 AND zen<zp' AND zen=INT(zen) THEN zen=zva(zen) ENDIF
```

in other words if zst is greater than zero and on a line higher up in the table (zp' being the current parameter number) and is an integer value, set zst to the current value of the 'higher up in the table parameter'. The parameter table sc392.prm uses the above.

10.9.14 Storage of long tables

Although the provision of structural properties or structural geometry is sufficient to define the cross-sectional area, moment/s of inertia etc. for the members of a framework to be analysed, the data required for the design of structural components such as: I-sections, structural hollow sections etc. presents another type of data. As a example, consider the following table from sc3800.

Size	Available thickness t mm									
40 x 40	2	3								Stainless
50 x 50	2	3	4							Square
60 x 60	2	3	4	5						Hollow
80 x 80	2	3	4	5						Sections
100 x 100		3	4	5	6					
125 x 125		3	4	5	6	8				
150 x 150		3	4	5	6	8				
175 x 175			4	5	6	8	10			
200 x 200			4	5	6	8	10			
250 x 250				5	6	8	10	12		
300 x 300				5	6	8	10	12		
350 x 350					6	8	10	12	15	
400 x 400					6	8	10	12	15	

```
Serial depth                     +sd(ssd1)=???? mm
Serial breadth                   +sb(ssd1)=???? mm
Thickness                        +st(ssd1)=???? mm
```

Although it is a trivial exercise to generate sensible integer values for the sizes, responding to the three prompts with 250,250,7 would not be acceptable data, as such a section does not exist. The checking in the model will detect that a section does not exist and advise the engineer accordingly, but generating a thousand triads for sd(),sb(), st() would not be helpful as the great majority will fail unless:

(a) logic is written to describe the table
(b) the table is provided
(c) a procedure is provided to give a best fix for a generated triad.

(a) For the simple table above, many lines of logic would be needed. Although logic could be devised; as from time to time manufacturers have to change section sizes in response to changes in codes of practice and/or changes in market conditions, maintaining the logic would be a long term commitment.

(b) Look up tables are already present in the model itself, it would be better to avoid duplication.

(c) A best fix for the sd(),sb(),st() triad, would be easier to maintain than storing the table. If the sd(),sb(),st() triad is combined into a single number: sd()*10^6+sb()*10^3+st(), then reading left to right, top to bottom in the table, we have:

40040002, 40040003, 50050002, 50050003, 50050004,
60060002, 60060003, 60060004, 60060005, 80080002,
80080003, 80080004, 80080005, 100100003, 100100004,
100100005, 100100006, 100100008, 125125003, 125125004,
125125005, 125125006, 125125008, 150150003, 150150004,
150150005, 150150006, 150150008, 175175004, 175175005,
175175006, 175175008, 175175010, 200200004, 200200005,

200200006, 200200008, 200200010, 250250005, 250250006, 250250008, 250250010, 250250012, 300300005, 300300006, 300300008, 300300010, 300300012, 350350006, 350350008, 350350010, 350350012, 350350015, 400400006, 400400008, 400400010, 400400012, 400400015.

These values can be stored, using the VEC() function thus:

```
+tri(1)=VEC(40040002,40040003,50050002,50050003,50050004)
+tri(6)=VEC(60060002,60060003,60060004,60060005,80080002,
+tri(11)=VEC(80080003,80080004,80080005,100100003,100100004)
+tri(16)=VEC(100100005,100100006,100100008,125125003,125125004)
+tri(21)=VEC(125125005,125125006,125125008,150150003,150150004)
+tri(26)=VEC(150150005,150150006,150150008,175175004,175175005)
+tri(31)=VEC(175175006,175175008,175175010,200200004,200200005)
+tri(36)=VEC(200200006,200200008,200200010,250250005,250250006)
+tri(41)=VEC(250250008,250250010,250250012,300300005,300300006)
+tri(46)=VEC(300300008,300300010,300300012,350350006,350350008)
+tri(51)=VEC(350350010,350350012,350350015,400400006,400400008)
+tri(56)=VEC(400400010,400400012,400400015)
```

In a similar manner, the rectangular hollow sections in Table 10.7b can be stored.

```
+tri(1)=VEC(50025001.5,50025002,60030002,60030003,80040002)
+tri(6)=VEC(80040003,80040004,100050002,100050003,100050004,
+tri(11)=VEC(100050005,100050006,150075003,150075004,150075005)
+tri(16)=VEC(150074006,150075008,150100003,150100004,150100005)
+tri(21)=VEC(150100006,150100008,200100004,200100005,200100006)
+tri(26)=VEC(200100008,200100010,200125004,200125005,200125006)
+tri(31)=VEC(200125008,200125010)
+tri(33)=VEC(250125006,250125008,250125010,250125012,250125015)
+tri(38)=VEC(250150006,250150008,250150010,250150012,250150015)
+tri(43)=VEC(300150006,300150008,300150010,300150012,300150015)
+tri(48)=VEC(300200006,300200008,300200010,300200012,300200015)
+tri(53)=VEC(350175006,350175008,350175010,350175012,350175015)
+tri(58)=VEC(350200006,350200008,350200010,350200012,350200015)
+tri(63)=VEC(400200006,400200008,400200010,400200012,400200015)
+tri(68)=VEC(400250006,400250008,400250010,400250012,400250015)
```

Although the above will need to be maintained, the maintenance will be much simpler than maintaining the many lines of logic needed to describe the table.

10.9.15 The PARAMETER table for SC385

The PARAMETER table for SC385 follows, and in turn is followed by a detailed description of each parameter, using the parameter number as a reference.

PARAMETER No.	name	Start zst()	End zen()	Type zty()	Dependency conditions and notes.
1	L	0.3	6	0	Length of member in m.
2	stype	1	2	2	stype=1 is SHS, stype=2 is RHS
3	ssd1	11	11	0	Constant for SHS table number
4	sd11	40	400	1	>L*1000/24 <L*1000/8
5	sb11	40	400	1	=sd11 for SHS
6	st11	2	15	1	>sd11/66.667+1 <sd11/12+1
7	tri	3	58	1E40	! Calls procedure tri.
8	srd1	12	12	0	Constant for RHS table number
9	sd12	40	400	1	>L*1000/24 <L*1000/8
10	sb12	20	200	1	=sd12/2 say for RHS
11	st12	2	15	1	>sd12/66.667+1 <sd12/12+1
12	tri	3	72	1E40	! Calls procedure tri.
13	grade	1	5	5	For grades 1:5, py as below.
14	py	1	5	126	210 220 220 400 460
15	sd	40	400	0	=sd11*(2-stype)+sd12*(stype-1)
16	sb	20	400	0	=sb11*(2-stype)+sb12*(stype-1)
17	st	1.5	15	0	=st11*(2-stype)+st12*(stype-1)
18	zrn	4	0.667	1E40	! Creates zrn(1:4) Σ=0.667
19	Mz	0.1	1000	0	=sb*st*py*sd/1E6*zrn1 kNm
20	My	0.05	1000	0	=sd*st*py*sb/1E6*zrn2 kNm
21	Fv	1	1000	0	=4*Mz/L*zrn3 kN
22	F	1	1000	0	=(sd+sb)*2*st*py/1E3*zrn4 kN
23	Lz	300	6000	1000	>L*1000/6 <L*1000 mm
24	Ly	300	6000	1000	>L*1000/6 <L*1000 mm
25	Kz	0.7	1	3	
26	Ky	0.7	1	3	
27	Ae	5	221	0	<(sd+sb)*2*st/100 cm2
28	moment	1	0	2	
29	restz	1	0	2	
30	betaMz	0.1	1000	0	<Mz kNm
31	mz2	0.1	1000	0	<Mz*zrn1 Must write zrn1 etc.
32	mz3	0.1	1000	0	<Mz*zrn2 here and not write
33	mz4	0.1	1000	0	<Mz*zrn3 zrn(1) etc.
34	M24	0.1	1000	0	<Mz*zrn4
35	resty	1	0	2	
36	betaMy	0.05	1000	0	<My
37	my2	0.05	1000	0	<My*zrn1
38	my3	0.05	1000	0	<My*zrn2
39	my4	0.05	1000	0	<My*zrn3
40	My24	0.05	1000	0	<My*zrn4
41	LT	1	6	0	<L
42	refno	1	10	1	
43	udl	1	2	2	
44	betaM	0.01	1000	0	<Mz*zrn1
45	m2	0.01	1000	0	<Mz*zrn2
46	m3	0.01	1000	0	<Mz*zrn3
47	m4	0.01	1000	0	<Mz*zrn4
48	E	200	200	0	Young's modulus N/mm2
49	NRESP	0	0	0	Avoids importing from NL-STRESS
50	ans	0	0	0	ans=0 refuses default values

Parameter 1. The length L is allowed to vary from 1 to 6 m; as zty(1)=0 an integer or real value will be generated. Dependency conditions may not be given for the first parameter; dependency conditions may only be specified in terms of parameters listed previously in the table.

Parameter 2. As zty(2)=2, the section type 'stype' takes only 2 values i.e. stype =1 or =2. If the number of increments 'zni=5' the values generated for 'stype' will be: 1,2,1,2,1.

Parameter 3. Proforma sc385 prompts for sd(ssd1)=???? for which ssd1 is a constant =11, which must be set before the serial sizes are input.

Parameter 4. The serial depth 'sd11' is specified to be an integer value by zty(4)=1, lying in the range 40 mm to 400 mm, with dependency condition >L*1000/24 which for L=1 m gives a minimum size of 42 mm and dependency condition <L*1000/8 which for L=6 m gives a maximum size of 750 mm which would be reduced to zen(4)=400 mm.

Parameter 5. The serial breadth 'sb11' for a square hollow section lies in the range zst(5)=40 mm to zen(5)=400 mm but constrained by the dependency condition '=sd11' to be the same size as the serial depth, as necessary of a square hollow section.

Parameter 6. The serial thickness 'st11' is specified to be an integer value by zty(6)=1, lying in the range 2 mm to 15 mm, with dependency condition >sd11/66.667+1 which for sd11=40 mm gives a minimum thickness 1 mm and dependency condition <sd11/12+1 which for sd11=400 mm gives a maximum thickness of 34 mm which would be reduced to zen(6)=15 mm.

Parameter 7. The setting of zty(7)=1E40 tells the program to invoke a procedure named 'tri', passing the values zst(7)=3 and zen(7)=58 to the procedure as 'arguments'. The procedure 'tri' is given in section 10.9.16 with an explanation; briefly the serial depth, breadth & thickness i.e. sd11, sb11 & st11 are fixed to be those of an available square hollow section for stainless steel, see section 10.9.14.

Parameters 8-12 are similar to Parameters 3-7 except that they apply to a rectangular hollow section rather than a square hollow section.

Parameters 13-19 are discussed in sections 10.9.12 & 10.9.13.

Parameters 20-22, respectively set the current serial: depth, breadth and thickness, regardless of whether the section is square or rectangular. None of these parameters appear in sc385, but all are needed to be able to do engineers' arithmetic to compute sensible forces e.g. bending moments etc. The dependency conditions do the necessary assignments viz.

Expression for evaluation	stype=1	stype=2
=sd11*(2-stype)+sd12*(stype-1)	=sd11	=sd12
=sb11*(2-stype)+sb12*(stype-1)	=sb11	=sb12
=st11*(2-stype)+st12*(stype-1)	=st11	=st12

Parameter 23. For a serial depth 'sd', breadth 'sb' & thickness 'st', the ultimate force in the flange for bending about the z axis using engineers' arithmetic =sb*st*py where py is the yield strength, thus the ultimate bending moment =sb*st*py*sd Nmm =sb*st*py*sd/1E6 kNm. Obviously this ultimate bending moment ignores stability.

23	Mz	0.5	1000	0	<sb*st*py*sd/1E6/4 kNm
24	My	0.5	1000	0	<sd*st*py*sb/1E6/4 kNm
25	Fv	0.5	1000	0	<sd*2*st*py/SQR(3)/1E3/4 kN
26	F	0.5	1000	0	<(sd+sb)*2*st*py/1E3/4 kN

For a factored bending moment Mz, applied to a square or rectangular hollow section of length L, for a span:depth ratio of 8 to 24, the section depth varies between sd=L/8 to L/24 as used in Table 10.9. From section 10.7, for hollow sections the depth:thickness ratio varies between 66.66 (for 400:6) and 12 (for 60:5), as used in Table 10.9.

```
   20  sd     40      400     0     =sd11*(2-stype)+sd12*(stype-1)
   21  sb     20      400     0     =sb11*(2-stype)+sb12*(stype-1)
   22  st     1.5     15      0     =st11*(2-stype)+st12*(stype-1)
```

In the above extract from a parameter table, for square hollow sections the serial breadth is made equal to the serial depth. For rectangular hollow sections sb() is constrained by the aspect ratio sd()/sb() with a maximum =2 and minimum =1.5, but because rectangular hollow sections may be rotated, sb() varies between sd()/2 and 2*sd(). For both square and rectangular hollow sections, the serial thickness is constrained by the depth:thickness ratio. From inspection of the table for square hollow sections, the depth:thickness ratio has a maximum value of 66.66 (for 400:6) and a minimum value of 12 (for 60:5). These values are used for the dependency conditions in the PARAMETER table.

The problem of nominating a section has been reduced to one of taking the sd,sb,st triad defined in the parameter table, combining all three component values into a single number and picking the nearest value in shs(1:58) or rhs(1:70) and then separating the combined number back to the sd,sb,st components for use by the model. To invoke a procedure for doing this, we use a pseudo parameter called tri i.e. the name of the vector containing the table of values. The value 3 in the new parameter in the table, under the column headed zst(), tells the system that the previous 3 parameters must be fixed. The values 58 or 70 in the table under zen() tells the system that there are 58 or 70 values in the tables shs() or rhs() used for the fixing, the values 1E40 in the table under zty() tells the system that the line in the table is that for a pseudo parameter and therefore the line must be treated differently to other parameters.

10.9.16 The procedure 'tri'

Procedures such as 'tri' below provide an escape path from the parameter table and back again. A procedure in a parameter table is distinguished by its type being 1E40. Numeric data may be passed to the procedure via zst & zen values; arguments follow the 1E40. If it is required that a description should be provided in place of arguments, then the description MUST follow an exclamation mark.

When it is required to suppress the procedure name from the stack of values generated, set ztm=1E40 to let sc924.pro know that the procedure name e.g. tri, is to be omitted.

```
DEFINE tri ! Procedure for fixing triad preceding parameter zp'.
! +triad=zva(zp'-3)*10^6+zva(zp'-2)*10^3+zva(zp'-1)
IF zen=58  ! ──────────────────────────────────────────────────┐
! +tri(0)=VEC(0,40040002,40040003,50050002,50050003,50050004)
! +tri(6)=VEC(60060002,60060003,60060004,60060005,80080002)
! +tri(11)=VEC(80080003,80080004,80080005,100100003,100100004)
! +tri(16)=VEC(100100005,100100006,100100008,125125003,125125004)
! +tri(21)=VEC(125125005,125125006,125125008,150150003,150150004)
! +tri(26)=VEC(150150005,150150006,150150008,175175004,175175005)
! +tri(31)=VEC(175175006,175175008,175175010,200200004,200200005)
! +tri(36)=VEC(200200006,200200008,200200010,250250005,250250006)
! +tri(41)=VEC(250250008,250250010,250250012,300300005,300300006)
! +tri(46)=VEC(300300008,300300010,300300012,350350006,350350008)
! +tri(51)=VEC(350350010,350350012,350350015,400400006,400400008)
! +tri(56)=VEC(400400010,400400012,400400015)
ENDIF ! ───────────────────────────────────────────────────────┘
IF zen=72  ! ──────────────────────────────────────────────────┐
! +tri(0)=VEC(0,50025001.5,50025002,60030002,60030003,80040002)
```

```
! +tri(6)=VEC(80040003,80040004,100050002,100050003,100050004)
! +tri(11)=VEC(100050005,100050006,150075003,150075004,150075005)
! +tri(16)=VEC(150075006,150075008,150100003,150100004,150100005)
! +tri(21)=VEC(150100006,150100008,200100004,200100005,200100006)
! +tri(26)=VEC(200100008,200100010,200125004,200125005,200125006)
! +tri(31)=VEC(200125008,200125010)
! +tri(33)=VEC(250125006,250125008,250125010,250125012,250125015)
! +tri(38)=VEC(250150006,250150008,250150010,250150012,250150015)
! +tri(43)=VEC(300150006,300150008,300150010,300150012,300150015)
! +tri(48)=VEC(300200006,300200008,300200010,300200012,300200015)
! +tri(53)=VEC(350175006,350175008,350175010,350175012,350175015)
! +tri(58)=VEC(350200006,350200008,350200010,350200012,350200015)
! +tri(63)=VEC(400200006,400200008,400200010,400200012,400200015)
! +tri(68)=VEC(400250006,400250008,400250010,400250012,400250015)
ENDIF !
! +i=0
REPEAT
! +trista=tri(i) +i=i+1 +triend=tri(i)
UNTIL triad>trista AND triad<=triend OR i>=zen
ENDREPEAT
! +trinew=triend  Save revised values.
IF triad-trista<triend-triad THEN trinew=trista ENDIF
! +zva(zp'-3)=INT(trinew/10^6+.5) +trinew=trinew-zva(zp'-3)*10^6
! +zva(zp'-2)=INT(trinew/10^3+.5)
! +zva(zp'-1)=trinew-zva(zp'-2)*10^3 +ztm=1E40 to suppress 'tri'.
ENDDEFINE
DEFINE zrn ! Generates zst(zp') random numbers, Σ=zen(zp'), in
! zrn(1:zen(zp')). +znn=zst(zp') +zsig=zen(zp') +zir=0 +ztot=0
REPEAT
! +zir=zir+1 +zrn(zir)=RAN(27) +ztot=ztot+zrn(zir)
UNTIL zir>=znn
ENDREPEAT
! +zir=0
REPEAT
! +zir=zir+1 +zrn(zir)=zsig*zrn(zir)/ztot
UNTIL zir>=znn
ENDREPEAT
ENDDEFINE
```

10.9.17 Replacing logic with a numerical expression

Occasionally there is a requirement that a parameter 'p' has to be the least of two previously assigned parameters e.g. p<=a and p<=b or greatest of two previously assigned parameters e.g. p>=a and p>=b. In a computer program, such as NL-STRESS, this appears to be straightforward, we could simply write IF p<=a AND p<=b THEN ...
or IF p>=a AND p>=b THEN ...
Either line would be incorrect if 'p' had not been assigned previously. Assuming that 'a' and 'b' and 'p' had been assigned previously, and that the value held in 'p' was not required subsequently, we could

```
write:    p=a                    Test with a=12 & b=27
          IF b<=a THEN p=b       p=12, as 'b' is not< 'a' then p=12.
                                 Test with a=42 & b=27
                                 p=42, as 'b' is < 'a' then p=27.
```

or to assign the greatest of a & b to p, we could

```
write:    p=a                    Test with a=12 & b=27
          IF b>=a THEN p=b       p=12, as 'b' is > 'a' then p=27.
                                 Test with a=42 & b=27
                                 p=42, as 'b' is not< 'a' then p=42.
```

When building the PARAMETER table, it is desirable to omit Boolean programming structures such as those shown above as the table could become indecipherable. It is desirable to have one expression that results in 'p' returning the minimum and one returning the maximum. A little bit of thought and the expedient usage of SGN (Signum) will suffice. It will be remembered that:

```
ABS Absolute value.  ABS(2.5) and ABS(-2.5) both return 2.5, ABS(0)
returns 0
SGN Signum.  Returns 1 if the argument is positive, -1 if
negative, 0 if zero.  SGN(0.01) returns 1, SGN(-270) returns -1.

The astute will soon figure out that:
the minimum of a,b   =b-SGN(SGN(b-a)+1)*(b-a)
    maximum of a,b   =b-SGN(SGN(a-b)+1)*(b-a)
the switch SGN(SGN(b-a)+1) evaluates to 1 when b>=a
                                   or 0 when b<a.
```

The following short SCALE proforma may be used to test the correctness of the above.

```
START
+a=????
+b=????
min(a,b) +pmin=b-SGN(SGN(b-a)+1)*(b-a)
max(a,b) +pmax=b-SGN(SGN(a-b)+1)*(b-a)
FINISH
```

Typical calculations from the above proforma follow:

```
a=300.45
b=-47.6
min(a,b) pmin=b-SGN(SGN(b-a)+1)*(b-a)
             =-47.6-SGN(SGN(-47.6-300.45)+1)*(-47.6-300.45)
             =-47.6
max(a,b) pmax=b-SGN(SGN(a-b)+1)*(b-a)
             =-47.6-SGN(SGN(300.45--47.6)+1)*(-47.6-300.45)
             =300.45
```

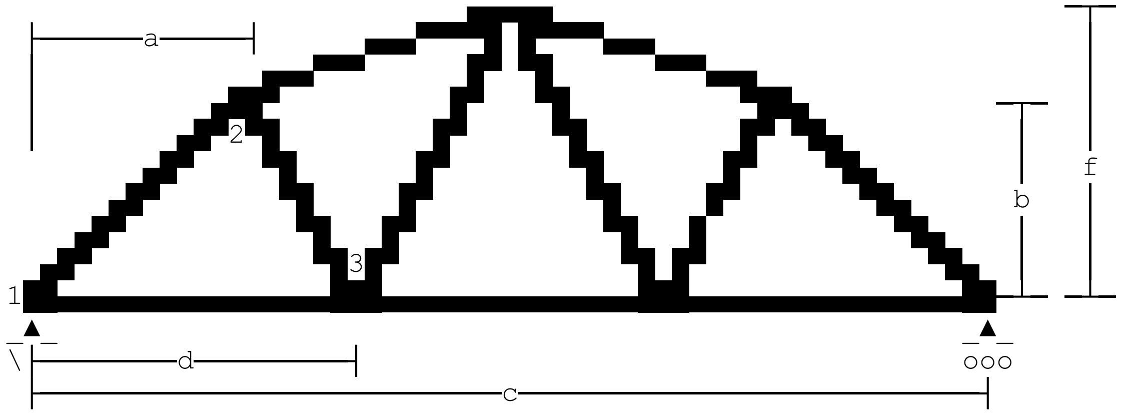
```
The forgoing demonstrates that instead of including: p=a
                                                     IF b<=a THEN p=b
```

in the parameter table, we can achieve the same effect by including b-SGN(SGN(b-a)+1)*(b-a) in the parameter table. The following first few lines of a parameter table were taken from verified model vm181.ndf. Comparison of the Mansard (or hog back) roof truss figure (verified model vm181.ndf) with the Parameter table shows that dimension 'a' should be >0.1*d and <b-SGN(SGN(b-d)+1)*(b-d). If 'a' is less than 0.1 m then the truss is not a Mansard.

The expression b-SGN(SGN(b-d)+1)*(b-d) selects the minimum dimension from 'b' and 'd' to avoid the triangle bounded by joints 1,2 & 3 having an obtuse angle.

PARAMETER

No.	Name	Start	End	Type	Dependency conditions	
1	c	3	12	0		
2	f	1	6	0	>c/5	<c
3	d	0.1	5	0	>c/20	<0.4*c
4	b	0.5	3	0	>0.7*f	<0.9*f
5	a	0.1	6	0	>0.1*d	<b-SGN(SGN(b-d)+1)*(b-d)



The so called 'mod' function is generally used in programming for finding the remainder when a value such as a=8 is divided by a base say b=5, obviously the remainder =3. The function takes two forms:

```
(i) mod=a-INT(a/b)*b      For b=5 & a=0 then remainder=0-INT(0/5)*5=0
                          For b=5 & a=1 then remainder=1-INT(1/5)*5=1
                          For b=5 & a=2 then remainder=2-INT(2/5)*5=2
                          For b=5 & a=3 then remainder=3-INT(3/5)*5=3
                          For b=5 & a=4 then remainder=4-INT(4/5)*5=4
                          For b=5 & a=5 then remainder=5-INT(5/5)*5=0
                          For b=5 & a=6 then remainder=6-INT(6/5)*5=1
                          For b=5 & a=7 then remainder=7-INT(7/5)*5=2
                          For b=5 & a=8 then remainder=8-INT(8/5)*5=3
                              and so on.
(ii) mod=a-INT((a-1)/b)*b For b=5 & a=0 remainder=0-INT((0-1)/5)*5=0
                          For b=5 & a=1 remainder=1-INT((1-1)/5)*5=1
                          For b=5 & a=2 remainder=2-INT((2-1)/5)*5=2
                          For b=5 & a=3 remainder=3-INT((3-1)/5)*5=3
                          For b=5 & a=4 remainder=4-INT((4-1)/5)*5=4
                          For b=5 & a=5 remainder=5-INT((5-1)/5)*5=5
                          For b=5 & a=6 remainder=6-INT((6-1)/5)*5=1
                          For b=5 & a=7 remainder=7-INT((7-1)/5)*5=2
                          For b=5 & a=8 remainder=8-INT((8-1)/5)*5=3
                              and so on.
```

Whatever the value of 'a' the expression 1-ABS(SGN(a-b)) evaluates to zero except when a=b the expression evaluates to 1. Let b=8, then 1- ABS(SGN(a-8)) evaluates to zero except when a=8 which causes the expression to evaluate to 1. The following shows that whatever the value of 'a', the expression evaluates to zero unless a=b, for which the expression evaluates to 1.

```
If a=0 then   SGN(0-8)=-1   ABS(-1)=1 and 1-1=0,
if a=1 then   SGN(1-8)=-1   ABS(-1)=1 and 1-1=0,
if a=7 then   SGN(7-8)=-1   ABS(-1)=1 and 1-1=0,
if a=8 then   SGN(8-8)=0    ABS(0)=0  and 1-0=1,
if a=9 then   SGN(9-8)=1    ABS(1)=1  and 1-1=0,
if a=16 then  SGN(16-8)=1   ABS(1)=1  and 1-1=0,
if a=999 then SGN(999-8)=1  ABS(1)=1  and 1-1=0, and so on.
```

It has been demonstrated that the above provides a means of actioning something just once. In an ordinary computer program, logic for the above could be described by say:

```
a=467
b=8
IF a=b THEN ...
```

The following SCALE proforma (from START to FINISH) may be extracted and run by SCALE to test that all Booleans return 'true=1' or 'false=0' as expected.

```
START
CONVERSION OF BOOLEAN TO ALGEBRAIC-LOGIC BY ROTE
First value for testing   +a=????
Second value for testing  +b=????
Expressions 'ex1' to 'ex6' should give true=1 or false=0 as expected,
 dependent on values a & b.

The Boolean a>=b, returns 1 if true, zero if false. To convert to
algebraic-logic, write Boolean as a-b>=0 and enclose in signum thus
SGN(a-b), add 1 and enclose in another signum to give the switch
+ex1=SGN(SGN(a-b)+1)

The Boolean a>b, returns 1 if true, zero if false.  To convert to
algebraic-logic, write Boolean as a-b>0 and enclose in signum thus
SGN(a-b), add 1 and enclose in another signum to give the switch
+ex2=SGN(SGN(a-b-1E-40)+1)

The Boolean a=b, returns 1 if true, zero if false.  To convert to
algebraic-logic, write Boolean as a-b=0 and enclose in ABS and
enclose in signum and extract from 1
+ex3=1-SGN(ABS(a-b))

The Boolean a<=b, returns 1 if true, zero if false.  To convert to
algebraic-logic, write Boolean as b-a<=0 and enclose in signum thus
SGN(b-a), add 1 and enclose in another signum to give the switch
+ex4=SGN(SGN(b-a)+1)

The Boolean a<b, returns 1 if true, zero if false.  To convert to
algebraic-logic, write Boolean as b-a>0 and enclose in signum thus
SGN(b-a), add 1 and enclose in another signum to give the switch
+ex5=SGN(SGN(b-a-1E-40)+1)

The Boolean a<>b, returns 1 if true, zero if false.  To convert to
algebraic-logic, write Boolean as a-b=0 and enclose in ABS and
enclose in signum
+ex6=SGN(ABS(a-b))

Example: IF tf>=tl AND stg<460 THEN fy=(stg-20)*1E3
In the example there are two Booleans separated by an AND, each
Boolean must return true=1, thus the addition must equal 2.  By
comparison with ex1 to ex6, it can be seen that examples ex1 and ex5
are required.  Assigning:
Thickness of flange         +tf=???? m
Limiting thickness          +tl=???? m
Steel grade                 +stg=????
Steel grade limit           +sgl=????
Two Booleans combined +lhs=SGN(SGN(tf-tl)+1)+SGN(SGN(sgl-stg-1E-40)+1)
FINISH
```

Suppose it is required that the following three lines of logic are required when running model vm643.dat.

```
IF tf<=0.04 THEN fy=stg*1E3
IF tf>0.04 AND stg<460 THEN fy=(stg-20)*1E3
IF tf>0.04 AND stg=460 THEN fy=(stg-30)*1E3
```

As shown above, we could convert the Boolean expressions into algebraic-logic and thereby incorporate the three lines into the Parameter table. It is simpler if the logic is included in model vm643.dat and only the steel grade is included in the Parameter table.

The following parameter table solves the problem.

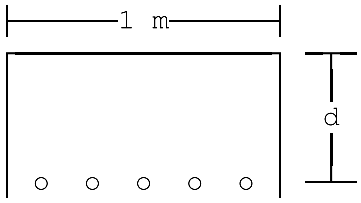
PARAMETER

No.	Name	Start	End	Type	Dependency conditions
1	tfmin	0.004	0.004	0	
2	tfmax	0.04	0.04	0	
2	stg	275	275	1	
3	fy				=stg*1E3
4	h	0.1	1	0	=tf/0.04 Ratio.
5	b	235	460	0	=460/stg Ratio.
6	p	0.1	460	0	<b-SGN(SGN(b-d)+1)*(b-d)

10.9.18 Some parametric dependency devices

SELECTING A SUITABLE SLAB DEPTH
The following extract is taken from the parameter table in sc080.pro. Given a design bending moment M, the effective depth 'd' has to ensure that the strength of the concrete is not exceeded and that the amount of reinforcement is within a practical percentage e.g. within 2% to 4%. The extract achieves both requirements, an explanation follows the extract.

No.	Name	Start	End	Type	Dependency conditions
...					
4	M	5	400	0	Ultimate moment of resistance.
5	fcu	30	60	4	Characteristic concrete strength.
6	fy	250	460	3	Characteristic steel strength.
...					
10	dconc	75	375	1	=SQR(M*1E6/(0.1*1000*fcu))
11	d	1E20	1E20	1	=dconc > <
...					



M kNm known, lever arm =0.75*d.
Let steel fraction of area =p,
then Ast (mm2) = d*1000*p and

$$M = \frac{Ast*fy*0.75*d}{10^6} = \frac{d*p*fy*0.75*d}{10^3}$$

$$\text{Rearranging} \quad d = SQR\left[\frac{M*10^3}{p*fy*0.75}\right]$$

Obviously an increase in steel fractional area 'p' causes a reduction in 'd' and vice versa. Taking a maximum fractional area of steel reinforcement of 0.02 i.e. 2%, & a minimum fractional area of 0.004 i.e. 0.4%, to give a minimum & maximum slab thickness respectively, then:
d >= SQR(M*1E3/(0.75*fy*0.02)) and d <= SQR(M*1E3/(0.75*fy*0.004))
as used in the PARAMETER table for sc080.pro.

For minimum slab depth based on the concrete alone, supply dummy parameter, the 10th, dconc=SQR(M*1E6/(0.1*1000*fcu)). Then 'd', the slab depth and 11th parameter, is set to 'dconc', the 1E20 for the start & end values means dynamically set them by the > and < which follows, i.e. (SQR(M*1E3/(0.75*fy*0.02)))
(SQR(M*1E3/(0.75*fy*0.004)))
these dependency conditions follow the table because of shortage of space on the 11th line.

SELECTING AN EXPOSURE CONDITION

Selecting an exposure condition is straightforward as the maximum exposure condition is dependent on just the concrete cube strength. An extract from the parameter table for sc080.pro follows, which in turn is followed by a description.

```
PARAMETER  Start   End     Type    Dependency conditions
No. name   zst()   zen()   zty()
...
 5  fcu    30      60      4        Char. concrete strength.
...
23  expos  1       5       5       >1 <INT(fcu/10)-1 Exp. condn.
```

Inspection of the parameter 'expos', the 23'rd parameter in the parameter table for sc080.pro, shows that when fcu=30, 'expos' is an integer number varying between 1 and INT(30/10)-1=2; when fcu=60, 'expos' is an integer number varying between 1 and INT(60/10)-1=5.

SELECTING THE NEAREST BAR DIAMETER

Selecting a sensible bar diameter has to meet two requirements:

- the bar must be selected from the set: 6 8 10 12 16 20 25 40 50
- the bar must be a sensible size for the section depth e.g. if section depth =100, then a bar diameter of 50 mm would be wrong
  =400, then a bar diameter of 6 mm would be wrong.

zty()=100 to zty()=125 accesses data stored in vectors za() to zz() respectively. Currently reinforcing bar diameters are stored in za() thus to access the bar diameters, zty()=100 i.e. Type=100.

```
             ┌3'rd          ┌8'th
za(1)=VEC(6,8,10,12,16,20,25,40,50)
             └start value   └end value
```

The line from the parameter table which follows, tells the program to pick a bar diameter from the range 10 mm to 40 mm.

```
12  dia    3       8       100    =INT(d/12) Tens bar diameter.
```

If we divide section depth by 12 say and use as a trial diameter:

```
   section depth =100, then bar diameter of 100/12 =8.33 mm
                 =400, then bar diameter of 400/12 =33.33 mm.
```

Obviously we need to use the trial diameter to select an available size from the set za(1)=VEC(6,8,10,12,16,20,25,40,50). This is done automatically when a dependency condition is provided e.g.

```
automatically when a dependency condition is provided e.g.─┐
zva(3)=10mm┐        ┌zva(8)=40mm       ┌─────────────────────┘
12  dia    3       8       100    =INT(d/12) Tens bar diameter.
```

which is interpreted as: evaluate d/12 and find the nearest (in this case bar diameter) within the range za(3) to za(8).

SELECTING A SUITABLE SPAN

Selecting a sensible span has to meet two requirements:

- the span must give a sensible span:depth ratio for the continuity specified e.g. fixed, pinned ends or cantilever
- the span:depth ratio must be found from the location reference given in response to the prompt for the parameter 'ans0' with values 1 to 5 corresponding to five location descriptions for the section.

An extract from the parameter table for sc080.pro follows, which in turn is followed by a description.

```
PARAMETER  Start   End     Type    Dependency conditions
No. name   zst()   zen()   zty()
...
 2  ans0   1       5       5       Location of section.
 3  sdr    1       5       126     20 7 20 23 26 Span:depth ratios.
...
21  span   2       20      0      =0.7*d*sdr/1000 Effective span.
```

From inspection of proforma sc080.pro it can be seen that parameter 2 offers the user options 1 to 5 corresponding to beam span:depth ratios respectively: 20 7 20 23 & 26. Parameter 3 stores the five span:depth ratios so that when ans0=1 sdr=20, when ans0=2 sdr=7, when ans0=3 sdr=20, when ans0=4 sdr=23, when ans0=5 sdr=26. Types=126-151, the pattern i.e. increasing or decreasing from zst to zen is made the same as the previous parameter 2. Parameter 3 is a dummy parameter i.e. it is not used in the model; but it is used in the expression given in parameter 21 to compute a sensible span for the current set of data which is being built.

RESTRICTING DISTRIBUTED LOADING LENGTHS TO BE WITHIN VARYING SPANS
An extract from a parameter table follows, which in turn is followed by a description.

```
PARAMETER  Start   End     Type    Dependency conditions
No. name   zst()   zen()   zty()   and notes.
...
2  ns      7       7       0       Number of spans.
3  sp      1       6       0      =RAN(29)*5+1  Beam spans l. to r.
10 uj      1       7       1      =INT(RAN(47)*ns+0.5) Span No.
17 23 splu 0       20      0      =+sn=zva(zp'-7),ztm=sp(sn)
```

Parameter 2 sets the number of spans 'ns' to 7. Parameter 3 sets seven spans i.e. sp(1) to sp(7) to random spans in the range 1 to 6 m. It is permissible to include the end parameter number on the line as in:

```
3  9 sp    1       6       0      =RAN(29)*5+1  Beam spans l. to r.
```

where parameter 3 refers to sp(1) and parameter 9 refers to sp(7); or to omit the '9' as in the extract. When the '9' is omitted it is deduced by the program. Parameters 10 to 16 select a random set of seven spans, where parameter 10 refers to uj(1) and parameter 16 refers to uj(7). The problem is to find the span lengths corresponding to the random span numbers uj(1) to uj(7). This requires two stages:

- selecting the span numbers stored in uj(1) to uj(7) by sn=zva(zp'-7), where zp' is always the current parameter number
- setting the current value of the parameter, which is always 'ztm' to sp(sn).

The two (or more) assignments are concatenated, with comma/s as separator/s. A plus sign follows the first equals sign to let the program know that multiple assignments follow.

ASSIGNING VALUES TO PARAMETERS FOR INCLUSION IN DEPENDENCY CONDITIONS
Occasionally it is necessary to assign constants to variables, these may be specified in a similar manner to that described above, starting the first assignments with =+ and separating each with a comma. When several constants need to be set, replace the line following the PARAMETER line with the assignments e.g.

```
PARAMETER  Start   End     Type    Dependency conditions
=+a1=0.065,a2=0.068,a3=0.070,a4=0.075,a5=0.077,a6=0.079
```

The values a1 to a6 may be used in the dependency conditions, and reset to other values when required without the need to edit the values in the table.

An extract from the parameter table for sc632.pro follows, which in turn is followed by a description.

```
   PARAMETER  Start   End     Type    Dependency conditions
   No. name   zst()   zen()   zty()
   ...
   6  typ    1       6       1      =zp'-5 typ1=1 typ2=2 etc.
   12  typ7   0       0       0       typ7=0 to stop load looping.
   13  w     -20      20      0       Uniformly distributed load.
   ...
```

This proforma has 6 types of loading numbered typ(1) to typ(6). As zp' is always the current parameter number, then zp'-5 refers to typ(1)=1, typ(2)=2 and so on. To stop the load types being cycled for ever, typ(7) is set to zero.

An extract from the parameter table for sc650.pro follows, which in turn is followed by a description.

```
   PARAMETER  Start   End     Type    Dependency conditions
   No. name   zst()   zen()   zty()
   ...
    8  x0     1       10      126     0 3.5 3.5 0 0 0 0 0 0 0
    9  x     -100     100     0      =+ztm=za'(zp'-8)
   19  y0     1       10      127     0 0 4 4 0 0 0 0 0 0
   20  y     -100     100     0      =+ztm=zb'(zp'-19)
   29 ...
```

The eighth parameter 'x0' is used to set 10 numbers starting with 0 and ending with 0, storing them in za'(1) to za'(10). zp' is always the current parameter number: 9 for x1, 10 for x2 and so on e.g. when zp'=9, then za'(zp'-8) equals za'(1). The variable 'ztm' is the current value for the current parameter. The nineteenth parameter 'y0' is used to set 10 numbers starting with 0 and ending with 0, storing them in zb'(1) to zb'(10). zp' is always the current parameter number: 20 for y1, 21 for y2 and so on e.g. when zp'=20, then zb'(zp'-19) equals zb'(1). The variable 'ztm' is the current value for the current parameter.

Occasionally a proforma calculation has several similar procedures but with overlapping parameters e.g. sc660.pro for yield line analysis, has nine options corresponding to nine different types of slab each with its own set of parameters. For such a situation, it is simpler to have nine parameter tables corresponding to the nine options, the first of which follows. When a table number is found following the keyword PARAMETER, the user is prompted for the required table number. The first table which follows covers options 1 to 3, the second table covers option 4 alone.

Six tables were needed to cover all nine options included in proforma calculation sc660.pro.

PARAMETER 1	Start	End	Type	Dependency conditions
No. name	zst()	zen()	zty()	and notes, for options 1 to 3.
1 ans	0	0	0	Default values (1=yes,0=no).
2 option	1	3	1	Rectangular slab s.s. edges.
3 L	2	6	0	Length of slab.
4 A	1	6	0	>L/2 <L Width of slab.
5 w	4	40	0	Ultimate UDL kN/m2.

PARAMETER 4	Start	End	Type	Dependency conditions
No. name	zst()	zen()	zty()	and notes, for option 4.
1 ans	0	0	0	Default values (1=yes,0=no).
2 option	4	4	1	Rectangular slab s.s. edges.
3 b'	2	6	0	Length of slab.
4 a	1	6	0	>b'/2 <b' Width of slab.
5 p	4	40	0	Ultimate UDL kN/m2.
6 mu	0.7	1	0	
7 10 i	1	2	0	

SELECTING THE LEAST OR GREATEST OF SEVERAL EXPRESSIONS
To find the least of several values or expressions, in a procedural language containing symbolic names h, d, e and b we may write

```
b=1500
IF h<b THEN b=h
IF d<b THEN b=d
IF e<b THEN b=e
```

which would cause b to contain the least of the values h, d & e. Incorporation of the above into a parameter table, would be awkward. A condensed form of the above code would be desirable.

An extract from a parameter table follows, in which it is required to set b as the least value of h, d & e. The parameters should be enclosed in brackets, separated by commas, and preceded by < which is taken to denote "select the least value from the parameters or expressions enclosed in brackets". Although it is permissible to omit the brackets when single dependency conditions follow a > or <, when two or more dependency conditions follow a > or < it is essential to enclose each expression in brackets with a comma between as below for the fourth parameter 'b'.

PARAMETER	Start	End	Type	Dependency conditions
No. name	zst()	zen()	zty()	and notes.
1 h	250	1500	1	
2 d	1500	250	1	>(h/2) <(2*h)
3 e	1500	250	1	
4 b	250	1500	1	<(h),(d),(e)

The second parameter 'd' is restricted to be >h/2 & <2*h, see section 6 entitled 'Dependency', which means d must not be less than h/2 and not greater than 2*h. The fourth parameter 'b' has a dependency condition which is interpreted similarly viz: 'b' must not exceed (h), (d) or (e), i.e. the maximum value of 'b' is the least of (h), (d) & (e). Firstly, the value of 'b' is computed dependent on the number of increments and the Type. If the current value of 'b' exceeds either (h), (d) or (e) then it is set to the 'least value'. This device is not supported for parameter types >=100, or where either zst() and/or zen() is set to 1E20 i.e. pointers to dynamic range limits.

The data generated from the parameter table is designed for engineers; consider the table:

```
PARAMETER  Start   End     Type    Dependency conditions
No. name   zst()   zen()   zty()   and notes.
 1  d      30      200     0
 2  b      0       200     0       >(0.5*d)   <(0.8*d)
```

```
It will be clear from the table that if the number of increments is 4
then values for 'd' will be            30     86.667     143.33     200
and those for 'b' will be initially     0     66.667     133.33     200
applying >0.5*d gives                  15     43.333     71.667     100
applying <0.8*d gives                  24     69.333     114.66     160
resulting in                           15     66.667*    114.66**   160
```

The application of the conditions gives redundant values e.g. for d=86.667, then b could be 66.667 i.e. the initial value or 43.33 to satisfy the condition >0.5*d or 69.333 to satisfy the condition <0.8*d. The rule chosen is 'keep the initial value unless it needs to be modified to satisfy a condition'. Thus the second value i.e. 66.667 with a star is left unmodified, the third value i.e. 133.33 does not need to be modified to satisfy >0.5*d but it does need to be modified to satisfy <0.8*d hence the third value, shown with ** is adjusted as required. Let us suppose that there are two greater than conditions and two less than conditions to be satisfied. To keep the explanation simple to follow we will make the second condition for each trivial. The greater than conditions given below i.e. >(0.5*d),(0.3*d) should result in the same set of values for 'b' as above, because >(0.5*d),(0.3*d) means >(0.5*d). Similarly the less than conditions given below i.e. <(d),(0.8*d) should result in the same set of values for 'b' as above, because <(d),(0.8*d) means <(0.8*d). Testing confirms the results to be as expected.

```
PARAMETER  Start   End     Type    Dependency conditions
No. name   zst()   zen()   zty()   and notes.
 1  d      30      200     0
 2  b      0       200     0       >(0.5*d),(0.3*d)   <(d),(0.8*d)
```

SWITCHING BETWEEN SMALL SETS OF DATA TO BE LOOKED UP

Consider the table below for which ansk9=1/0 for softwood/hardwood. First a dummy variable needs to be defined for the number of pieces from 1-4, then conditional upon ansk9=1/0 the appropriate column in the table below needs to be looked up and a value of K9 extracted. Of course such a procedure can be made into a subroutine, but it is more convenient to have the logic included in the parameter table.

Modification factor K9 to minimum modulus of elasticity for trimmer joists and lintels		
Number of pieces	Softwoods	Hardwoods
1	1.00	1.00
2	1.14	1.06
3	1.21	1.08
4 or more	1.24	1.10

The following stores the 8 values as described previously on the first line following the PARAMETER table. The variable name must not exceed 3 characters in length, and have a lower case letter before the start of the numerical subscript. The brackets around the subscript have been omitted for reason of space.

```
PARAMETER  Start   End     Type    Dependency conditions
=+a1=1,a2=1.14,a3=1.21,a4=1.24,a5=1,a6=1.06,a7=1.08,a8=1.10
```

We need to choose a value for K9 dependent on 'ansk9' & the number of pieces. To do this we first need to pick a subscript say 'sst' for a(), then assign K9 with the value i.e. K9=a(sst). Inspection of the expression: sst=4*(1-ansk9)+INT(4*RAN(23))+1 will reveal that it will do the job, where the (23) is a seed for the random number generator.

The PARAMETER table is a powerful tool for testing the logic of programs but needs care in its construction. Patterns are important and are used to ensure that all the nooks and crannies of the logic are tested. A description of 'patterns' and the reason for their use, is given in Section 5 of sc924.hlp. Briefly, pz'=1 refers to the initial direction i.e. zst to zen, and pz'=0 refers to the reverse direction i.e. zen to zst.

```
10  stype  1      2      2     Sect type, single angle=1, double=2.
11  numd   2      3      2    =stype+1 No. of dims is two or three.
12  nump   340    350    2    =340+10*(stype-1)  Proc is sec340/350.
```

Consider the above three lines; the first line is straightforward and associates stype=1 with a single angle for which there are two geometric properties in its section designation i.e. serial depth and thickness; stype=2 refers to a double angle for which there are three geometric properties in its section designation i.e. serial depth, breadth and thickness. For stainless steel the procedure number required for a single angle is sec340, and for a double angle is sec350, see sc924.pro for these two procedures. The numerical part nump=340 or 350 is dependent on the section being a single or double angle respectively. Obviously there is a potential to cause chaos:

- associating three dimension with a single angle or associating two dimensions with a double angle
- associating the procedure sec350 with a single angle or associating procedure sec340 with a double angle.

To avoid chaos we need to nullify the effect of patterns, this is done by the two expressions: =stype+1 and =340+10*(stype-1). If these two expressions were omitted then either or both values of 'numd' & 'nump' would be wrong when 'stype' was 'ascending' and both 'numd' & 'nump' were 'descending'.

An inspection of proformas sc388.pro & sc3800.pro (which is invoked by sc388.pro) reveals that the serial depth, breadth & thickness are referred to by sd(ead1),st(ead1) for single angles, but referred to by sd(dad1),sb(dad1),st(dad1) for double angles. It is desirable to avoid having the complexity of both section designations in the parameter table, for most proformas, including sc387, the form is:

```
tbn     11      12      2      =stype+10
...
sd(tbn) 100     600     0      >L/24  <L/8 Serial depth.
sb(tbn) 50      600     0      =sd(tbn)*stype      Serial breadth.
st(tbn) 3       15      1      >sd(tbn)/66.667+1  <sd(tbn)/12+1
```

which generates assignments such as: tbn=11, sd(tbn)=200, sb(tbn)=200 & st(tbn)=12. As only one subscript i.e. tbn is being used there is no problem. When the subscript changes from 'ead1' to 'dad1' a different strategy is needed thus:

```
13  ead1   11      11     2     Constants ead1 & dad1 are used in
14  dad1   12      12     2     sc3800.pro for single/double angles.
15  tbn    11      12     2    =stype+10
16 sd(zva15) 100   600    0    >L/24  <L/8 Serial depth.
17 sb(zva15) 50    600    0    =sd(tbn)*stype      Serial breadth.
18 st(zva15) 3     15     1    >sd(tbn)/66.667+1  <sd(tbn)/12+1
```

The (zva15) is interpreted as an instruction to look-up the current value of the 15'th parameter and replace the zva15 with, in this case, either 11 or 12. The data generated being e.g. sd(11)=200, sb(11)=200 & st(11)=8 or sd(12)=400, sb(12)=200 & st(12)=8.

SELECTING ONE OF TWO SETS OF VALUES
Occasionally it is required that a range of lengths needs to be modified in accordance with a condition. See the PARAMETER table for the design of a portal frame held in the file vm453.dat. For example 'height to eaves' varying from 3 to 8 metres in 1 metre intervals, is sensible for portal spans of say 30 m, but not sensible for a portal frame having 203*133x25 UB rafters. For 203x133x25 UB rafters, it is unlikely to have columns which are 8 m high. A switch is required to change the set of heights to the eaves from 3 to 8 m in 1 m intervals to say 2.5 to 5 m in 0.5 m intervals. In the partial parameter table which follows, the first parameter 'hc' varies within the range 0.127 m to 1.0361 m.

The two sets of column heights to eaves, where hte refers to the 'height to eaves', are given in the second & third line in the parameter table. The Type of parameter for the second line is 128 which denotes that the set of values 3 4 5 6 7 8 is to be stored in zc'(). The Type of parameter for the third line is 129 which denotes that the set of values 2.5 3 3.5 4 4.5 5 is to be stored in zd'().

```
PARAMETER
No. Name   Start   End     Type    Dependency conditions
1  hc      0.127   1.0361  0       Depth of beam, switch at O.42 m.
2  hte     1       6       128     3 4 5 6 7 8
3  hte'    1       6       129     2.5 3 3.5 4 4.5 5
4  pntr    2       3       2      =INT(hc*7.1429)
5  ht      2.5     8       0      =zva(zp'-pntr)
```

The fourth line is a pointer which may take only the integer numbers 2 & 3. The =INT(hc*7.1429) equates to 3 for section depths of 0.42 m and above. The =INT(hc*7.1429)=2 for section depths less than 0.419 m.

The fifth line limits the height to eaves to be in the range 2.5 m to 8 m. The zp' always refers to the current parameter number, in this case the fifth parameter. When pntr=2 then =zva(zp'-pntr) i.e. =zva(5-2)=zva(3) refers to the set 2.5 3 3.5 4 4.5 5. When pntr=3 then =zva(zp'-pntr) i.e. =zva(5-3)=zva(2) refers to the set 3 4 5 6 7 8.

To test the validity of the above parameter table, remove the exclamation mark preceding the keyword PARAMETER; run SCALE proforma 924 and give responses to the prompts thus: opn=1 nsref=253 npat=1 mode=2 zni=-48. The -ve value for the number of increments tells SCALE to generate the sets of data but not to run the NL-STRESS data file vm453.dat.

SUMMARISING
Over a three year period, it has been found that with a little bit of thought, it is possible to engineer dependency conditions for most situations which arise, only having to resort to writing procedures for tables of steel section properties.

10.9.19 Numerical devices and other matters

SCALE uses the parameter table for generating data for different portal spans and different sectional properties for UB's.
Numerical devices in parameter tables include ABS(), INT() & SGN().
ABS Absolute value. ABS(2.5) and ABS(-2.5) both return 2.5, ABS(0) returns 0
INT Integral part by truncation of the absolute value. INT(2.9) returns 2, INT(-2.9) returns -2, INT(0) returns 0
SGN Signum. Returns 1 if the argument is positive, -1 if negative, 0 if zero. SGN(0.01) returns 1, SGN(-270) returns -1. For switches (programming devices) using Signum, see sc924.hlp.

As 'rnn' takes integer values 1 to 1000, the expression
INT((i-INT((rnn-1)/8)*8)-1)/2+1 takes the values 1 1.5 2 2.5 3 3.5 4 4.5 1 1.5 2 2.5 3 3.5 4 4.5 1 1.5 2 2.5 3 3.5 4 4.5 and so on.

As 'rnn' takes integer values 1 to 1000, the expression
INT((rnn-INT((rnn-1)/4)*4)-1)/4+1 takes the values
1 1.25 1.5 1.75 1 1.25 1.5 1.75 1 1.25 1.5 1.75 and so on.

As 'rnn' takes integer values 1 to 1000, the expression
INT((rnn-INT((rnn-1)/8)*8)-1)/4+1 takes the values
1 1.25 1.5 1.75 2 2.25 2.5 2.75 1 1.25 1.5 1.75 2 2.25 2.5 2.75
1 1.25 1.5 1.75 2 2.25 2.5 2.75 and so on.

As 'rnn' takes integer values 1 to 1000, the expression
1-ABS(SGN(rnn-5))
takes the values 0 0 0 0 1 0 0 0 0 0 0 0 0 0 0 all zeroes following.

As 'rnn' takes integer values 1 to 1000, the expression SGN(SGN(b-a)+1) evaluates to 1 when b>=a or =0 when b<a.

As 'rnn' takes integer values 1 to 1000, the expression
INT((rnn-1)/7)+1
takes the values 1 1 1 1 1 1 1 2 2 2 2 2 2 2 3 3 3 3 3 3 3 and so on.

When it is required that a parameter be repeated a fixed number of times. e.g. 18, starting with 1 and generating the sequence:
1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4
5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 and so on, this may be achieved by:
INT((rnn-1)/18)+1 where rnn takes integer values 1,2,3 and so on.

When it is required that a parameter be repeated a fixed number of times. e.g. 18, starting with 1 and generating the sequence:
1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2
3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 3 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4
5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 and so on up to 18 and then restarting from 1, this may be achieved by:
INT(INT((rnn-INT((rnn-1)/108)*108)-1)/18+1)
where rnn takes integer values 1,2,3 and so on.

When it is required that a parameter be repeated a fixed number of times, e.g. 18, starting with 18 and generating the sequence:

```
18 18 18 18 18 18 18 18 18 18 18 18 18 18 18 18 18 18
17 17 17 17 17 17 17 17 17 17 17 17 17 17 17 17 17 17
16 16 16 16 16 16 16 16 16 16 16 16 16 16 16 16 16 16
15 15 15 15 15 15 15 15 15 15 15 15 15 15 15 15 15 15
14 14 14 14 14 14 14 14 14 14 14 14 14 14 14 14 14 14
13 13 13 13 13 13 13 13 13 13 13 13 13 13 13 13 13 13
18 18 18 18 18 18 18 18 18 18 18 18 18 18 18 18 18 18
17 17 17 17 17 17 17 17 17 17 17 17 17 17 17 17 17 17 and so on
```

down to 13 and then restarting from 18, this may be achieved by:

```
19-INT(INT((rnn-INT((rnn-1)/108)*108)-1)/18+1)
```

or as above starting from l'

```
l'-INT(INT((rnn-INT((rnn-1)/108)*108)-1)/18+1)
```

where rnn takes integer values 1,2,3 and so on.

When it is required that a parameter be repeated a fixed number of times, e.g. 18, starting with 64 and generating the sequence:

```
64 64 64 64 64 64 64 64 64 64 64 64 64 64 64 64 64 64
59 59 59 59 59 59 59 59 59 59 59 59 59 59 59 59 59 59
54 54 54 54 54 54 54 54 54 54 54 54 54 54 54 54 54 54
49 49 49 49 49 49 49 49 49 49 49 49 49 49 49 49 49 49
44 44 44 44 44 44 44 44 44 44 44 44 44 44 44 44 44 44
39 39 39 39 39 39 39 39 39 39 39 39 39 39 39 39 39 39 and so on
```

down to 39 and then restarting from 64, this may be achieved by:

```
span+inc-inc*INT(INT((rnn-INT((rnn-1)/108)*108)-1)/18+1)
```

where 'span' is the starting span and 'inc' is the span increment where rnn takes integer values 1,2,3 and so on.

OTHER MATTERS

The names of NL-STRESS variables normally commence with a lower case letter; when a variable commences with an upper case letter the variable must be prefaced with a '.' when it is on the left side of an assignment in the NL-STRESS data file.

For NL-STRESS parameter tables, the parameter table should not start with '.' when the first alphabetic letter starts with A-Z. SC924.PRO automatically inserts '.' in front of A-Z when it copies the parameter assignments to cc924.stk.

Rounding is not generally a problem unless data is written to a file and then read from it, as happens in sc924, which writes sets of parameters, subsequently reading in the sets. As a numerical example consider the dimension 'Na1' in sc111, which limits the dimension to a maximum of l/6 in the parameter table, where l is the span. If l=1 then the maximum value for Na1 is 0.166666666666667. When the data gets written, then assuming the default setting of DIGITS 5, then the value written will be 0.16667.

Checking input data for errors is by procedure 'chkrng' which is called with three arguments viz: chkrng l/6 0.1 Na1
where l/6 is the maximum and 0.1 is the minimum for Na1. Within chkrng, the arguments become chk1=l/6=0.166666666666667, chk2=0.1, and chk3=Na1=0.16667 as read from disk. The first check for Na1 being in the required range is:

```
IF chk3>chk1   (i.e. IF 0.16667>0.166666666666667)
Data +chk3 entered, exceeds maximum expected value +chk1
! +OK=0 i.e. failure due to the data chk3 being outside its range.
ENDIF
```

To cater for such occurrences, fix the fifth significant digit by:

```
IF chk3>chk1*1.0001   (i.e. IF 0.16667>0.166683333333334)
```

which will not cause Na1 to be reported out of range.

Praxis (the notation in which SCALE, LUCID... proformas are written), in section 10.6.16 describes a numerical device for testing whether a parameter is on the response stack. This device which involves assigning a parameter with the value -1E39, has use in verifying the logic of proforma calculations using sc924.pro. When sc924.pro generates sets of data, the sets are copied one at a time to the stack file e.g. if proforma sc083.pro is being tested then the sets of data held in scrtch.001 et seq. are copied to sc083.stk and SCALE is invoked to run the data. Sometimes the author of the proforma, sets the value of a parameter before the prompt e.g. in proforma sc083 the number of link legs is usually set by '+nlegs=2' such that the prompt '+nlegs=????' accepts the value =2. Thus the data generated by sc924 which is copied to sc083.stk, will be overwritten by any reassignment e.g. '+nlegs=2' prior to the prompt '+nlegs=????'. There is a way to avoid this overwriting by use of the -1E39 numerical device which uses the fact that the response stack has priority over the numerical stack. This 'priority' is necessary as inexperienced proforma writers prompt with '+b=???? m' and subsequently write '+b=b*1000 mm'. When the calculation is rerun and the prompt '+b=???? m' appears again, the value of 'b' in the response stack which holds 'b' is offered rather than the value of 'b*1000' in the numerical stack thus avoiding confusion by the user.

If the sets of data generated by sc924 for testing proforma sc083 are inspected it will be seen that .stk file contains e.g.

```
...
+nlegs=5
   +nlegs=-1E39
+dial'=10
...
```

causing the value nlegs=5 to be put on the numerical stack, then the indented '+nlegs=-1E39' is interpreted as "copy the current value of nlegs =5 from the numerical stack to the response stack". The result is that if the proforma sets the value '+nlegs=2', then when the prompt '+nlegs=????' appears, the value 5 from the response stack will be that offered in response to the prompt. Two runs of SCALE are normally used by sc924, but for the special case of proformas which preset prompts e.g. '+nlegs=2' before the prompt '+nlegs=????' then a single run of SCALE is carried out. The reason for the single run is that the .stk file produced at the end of the first run, only contains one assignment for 'nlegs'.

For further information on refining a design procedure given in a parameter table, see the notes at the end of sc083.prm.

There are two difficulties with generating sets of data for testing steelwork proformas which are not a problem with concrete proformas.
(a) Steelwork proformas such as for the component design for a beam typically prompt for the type of beam: UB, UC, SHS etc. whereas proformas for the design of a concrete component are specific to the type of section e.g. rectangular.
(b) Steelwork proformas must ensure that the section chosen be available, whereas for a concrete proforma the engineer specifies the size of section required.

For steelwork proformas, the parameter table must allow for the generation of several different types of section. Rather than launch into the production of a parameter table for every type of steelwork section available, as an introduction, consider proforma sc386 for the design of stainless steel sections: CHS=1 SHS=2 RHS=3 [=4][=5.

An extract from the parameter table sc386.prm follows.

6	stype	1	5	5	CHS=1 SHS=2 RHS=3 [=4][=5.
7	bintri	1	5	127	2 2 3 3 3
8	ref	1	5	128	300 310 320 330 330
9	widf	1	5	129	1 1 0.5 0.3 0.6
10	minf	1	5	130	66.7 66.7 66.7 24 24
11	maxf	1	5	131	12 12 12 8 8
12	hsd	11	15	5	=stype+10
13	sd(hsd)	40	400	0	>L*1E3/24 <L*1E3/8 Serial depth.
14	sb(hsd)	40	400	0	=sd(hsd)*widf Serial breadth.
15	st(hsd)	2	15	0	>sd(hsd)/minf <sd(hsd)/maxf Thick.
16	chosec	7	8	-1E40	! Calls procedure 'chosec'.
17	D	0	400	0	=sd(hsd)*(1-ABS(SGN(stype-1)))
18	t	0	15	0	=st(hsd)*(1-ABS(SGN(stype-1)))

From inspection of the part table above, three parameters: sd(hsd), sb(hsd) & st(hsd) set an approximate serial zize for the section dependent on the length of the section. The procedure 'chosec' to choose a section is invoked by either setting the type to be +1E39 or -1E39. When the type is 1E39 then the values zst & zen are passed to the procedure; when the type is -1E39 then the values zst=7 & zen=8 are treated as parameter numbers and looked up. In this example zst=7 causes the current value of bintri (either 2 or 3) to be passed to chosec; in this example zst=8 causes the current value of ref (300, 310, 320, 330 & 330) to be passed to chosec.

The engineer author of sc386 chose to treat stainless CHS as a special case by giving parameter names D & t. Obviously this produces redundant data in the parameter table; but because the proforma is run twice, the redundant data gets filtered out for the second run. To avoid confusion, when the chosen section is not a CHS, then the values of D & t are set to zero. This is achieved by the switch (1-ABS(SGN(stype-1))) which takes the value 1 when stype=1, else zero.

Some tables are published going from the largest section size to the smallest e.g. UB's. Some tables are published going from the smallest section size to the largest e.g. RHS's. When tables need to be 'reversed' include the following example:

```
! +m=lst+1
REPEAT
! +m=m-1
+zAX(m)
UNTIL m=1
ENDREPEAT
```

Occasionally there is a requirement to cycle a value thus:
10 10 10 10 10 10 20 20 20 20 20 20 30 30 30 30 30 30 then repeating
10 10 10 10 10 10 20 20 20 20 20 20 30 30 30 30 30 30 and so on.

No.	Name	Start	End	Type	Dependency conditions
1	rnn	1	36	36	Run No. give zni=End=540=15*36
2	tmp	1	3	1	=INT((rnn-INT((rnn-1)/18)*18)-1)/6+1
3	ang	10	30	1	=tmp*10

In the parameter table extract shown above, the generation of the set of values 10 10 10 10 10 10 20 20 20 20 20 20 30 30 30 30 30 30 is carried out in two stages. The second parameter 'tmp' is evaluated from INT((rnn-INT((rnn-1)/18)*18)-1)/6+1 and computes the set of values 1 1 1 1 1 1 2 2 2 2 2 2 3 3 3 3 3 3. The parameter 'ang' multiplies the current value of 'tmp' by 10. The reason for the two stage computation is that PRAXIS has a limit of 25 components in any single expression.

10.9.20 Parameter table for test purposes

A double slash sign following an option number e.g. 72// given in response to the option number prompt, causes a 1st draft PARAMETER table to be produced for the automatic generation of sets of data. Go through the proforma accepting the first set of default values, when the calculation is complete, click 'Changes' and 'Edit stack' to leave SCALE; the parameter table will be found in e.g. sc072.stk.

```
@cc924.stk ! Imports run number for multiple test runs.
PARAMETER  Start   End     Type    Dependency conditions
=+a1=1,a2=1.14,a3=1.21,a4=1.24,a5=1,a6=1.06,a7=1.08,a8=1.10
 1  ansk9  1       0       2        Softwood=1, hardwood=0.
 2  sst    1       8       1       =4*(1-ansk9)+INT(4*RAN(23))+1
 3  K9     1       1.24    0       =a(sst)
 4  set    0       0       0       =+d1=20,d2=25,d3=32,d4=40
 5  dia    20      40      1       =+ra=INT(4*RAN(17)+1),ztm=d(ra)
 6  sp     1       6       0       =RAN(29)*5+1  Beam spans l. to r.
16  pj     1       10      1       =INT(RAN(47)*10)+1 Span No.=1-10.
26  sc     0       6       0       =+cc=zva(zp'-10),ztm=zva(cc+5)
36  xc     0       6       0       =RAN(3)*zva(zp'-10) Dist. to load.
46  x0     1       11      126      0 4 4 0 0 3 3 1 1 0 0
47  x     -100     100     0       =+ztm=za'(zp'-46) Set x(1) etc.
58  y0     1       11      127      0 0 4 4 3 3 1 1 3 3 0
59  y     -100     100     0       =+ztm=zb'(zp'-58) Set y(1) etc.
70  M      50      5000    0        Design moment after any redis.
71  fy     250     460     2        Char. strength of longit. steel.
72  d      200     800     1  > <  Effective depth.
73  dia    1       10      100     >INT(d/20) <INT(d/17) Bar diams.
74  b      200     1100    1  > <  Breadth of section.
75  nbart  2       20     19    >  Number of tension bars.
76  expos  1       5       5       <INT(fcu/12) Exposure condition.
(0.6*(M*1E8/fy)^(1/3))
((M*1E8/fy)^(1/3))
(M*1E6/(0.0225*fy*d^2))
(M*1E6/(0.0075*fy*d^2))
(INT(M*1E6*5.33/(fy*PI*dia^2*d))+1),(INT(b/125)+1)
```

Modification factor K9 to minimum modulus of elasticity for trimmer joists and lintels		
Number of pieces	Softwoods	Hardwoods
1	1.00	1.00
2	1.14	1.06
3	1.21	1.08
4 or more	1.24	1.10

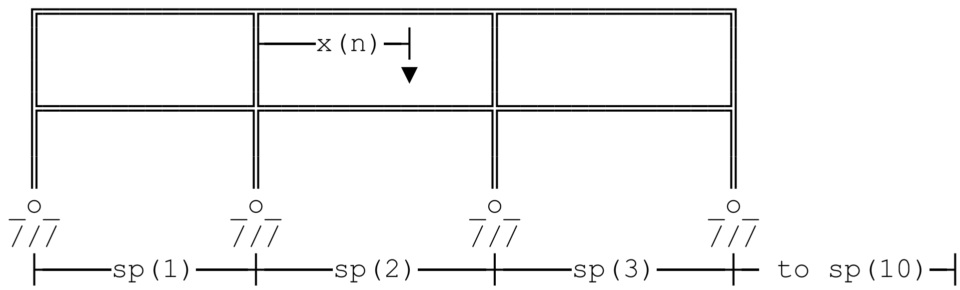
Choosing an appropriate value for modification factor K9 in BS5268, dependent on the timber and the number of pieces, is carried out in two stages. In the parameter table below the line following PARAMETER saves four values for softwoods followed by the four values for hardwoods in a(1) to a(8) where the brackets have been omitted for reason of space. The =+ tells the program that assignments follow. The second stage, to choose an appropriate value from a(1) to a(8), is carried out by lines numbered 1 to 3. Line 1 sets ansk9 to 1 or zero. Line 2 limits sst to be an integer in the range 1 to 8. The assignment sst=4*(1-ansk9)+INT(4*RAN(23))+1 has tow components, the first part 4*(1-ansk9) takes the value 0 for softwood and 4 for hardwood, the second part INT(4*RAN(23))+1 picks a random number of pieces in the range 1 to 4, where 23 is a seed for the random number generator which generates a number in the range 0 to 0.99999.. Combining the two components results in sst taking an integer value in the range 1 to 8. Finally K9 is assigned a value in the range 1 to 1.24 by the =a(sst). It will be clear that it is insufficient to choose a value for K9 picked from a(1) to a(8), as the probability is that 50% of the values would be wrong unless the type of wood was taken into account.

```
@cc924.stk ! Imports run number for multiple test runs.
PARAMETER  Start   End     Type    Dependency conditions
=+a1=1,a2=1.14,a3=1.21,a4=1.24,a5=1,a6=1.06,a7=1.08,a8=1.10
 1  ansk9  1       0       2        Softwood=1, hardwood=0.
 2  sst    1       8       1       =4*(1-ansk9)+INT(4*RAN(23))+1
 3  K9     1       1.24    0       =a(sst)
```

In the section of table above, assignment of values a(1) to a(8) was on the line following PARAMETER, it is also permissible to assign values under the dependency conditions heading; as before the set of assignments commence with =+. The fourth line of the section of the table below sets bar diameters 20,25,32,40. The fifth line assigns the variable ra with a random number in the range 1 to 4, ztm=d(ra) assigns dia with a random bar diameter selected from the set 20,25,32 & 40. It is not permissible to replace ztm=d(ra) with dia=d(ra) as dia is subsequently assigned to ztm.

```
 4  set    0       0       0       =+d1=20,d2=25,d3=32,d4=40
 5  dia    20      40      1       =+ra=INT(4*RAN(17)+1),ztm=d(ra)
```

Another use for random numbers is for choosing span dimensions for multi storey frames and for setting distances to concentrated loads from the left end of each span. Partial distributed loads follow the same pattern.




```
 6  sp     1       6       0       =RAN(29)*5+1  Beam spans l. to r.
16  pj     1       10      1       =INT(RAN(47)*10)+1 Span No.=1-10.
26  sc     0       6       0       =+cc=zva(zp'-10),ztm=zva(cc+5)
36  xc     0       6       0       =RAN(3)*zva(zp'-10) Dist. to load.
46  ...    0       0       0
```

Random span lengths sp(1) to sp(10) of between 2 & 6 m are generated by =RAN(29)*5+1 where the 29 is a seed for the random number generator which generates numbers in the range zero to 0.999999.. thus the spans generated vary from 1 to 5.999999.. Random span numbers pj(1) to pj(10) of between 1 and 10 are generated by =INT(RAN(47)*10)+1 where the 47 is a seed for the random number generator which generates a number in the range zero to 0.999999.. thus the spans numbers generated vary from 1 to 10.

Span lengths sc(1) to sc(10) contain the span lengths corresponding to the span numbers held in pj(1) to pj(10). The two assignments =+cc=zva(zp'-10),ztm=zva(cc+5) start with =+ to say that one or more assignments follow, assignments being separated by commas. For non-subscripted parameter names, assignments may contain any parameters which have been set previously i.e. parameters higher up in the table. Subscripted parameters names such as cj may not be used in an assignment as cj is undefined, existing only as cj(1),cj(2)... The parameter number, whether subscripted or not is held in zp', the corresponding value of the parameter is zva(zp'). The current value of any parameter is ztm, thus ztm=zva(cc+5) assigns the current parameter with the value held in zva(cc+5). The assignment cc=zva(zp'-10) assigns cc with the value of the parameter 10 locations before the current, thus cc contains the span number. Finally the distance to the load =RAN(3)*zva(zp'-10) sets the value for xc, such that xc is within the randomly chosen span length.

```
46  x0     1      11     126     0 4 4 0 0 3 3 1 1 0 0
47  x     -100    100    0      =+ztm=za'(zp'-46) Set x(1) etc.
58  y0     1      11     127     0 0 4 4 3 3 1 1 3 3 0
59  y     -100    100    0      =+ztm=zb'(zp'-58) Set y(1) etc.
70  ...
```

Occasionally it is more convenient to assign a set of numbers to a subscripted parameter rather than assign them, one at a time, following an =+. As an example, parameter 46 is a dummy parameter, setting type=126 tells the program that any numbers listed at the end of the line are to be stored in za'(1), za'(2) to za'(11). Parameters 47 to 57 assign the values of za'(1), za'(2) to za'(11) to x(1), x(2) to x(11). The ztm=za'(zp'-46) says give the current parameter the value of za'(zp'-46), for example if the current parameter is 47 i.e. x(1) then the assignment is ztm=x(1)=za'(47-46), if the current parameter is 57 i.e. x(11) then the assignment is ztm=x(11)=za'(57-46). The assignment of y(1) to y(11) follow in a similar manner to that for x(1) to x(11).

Structural design models generally ask the engineer for design bending moments, shear forces etc. and the sizes of the structural component being designed e.g. depth & breadth. For the production and running of a thousand sets of test data, it is important that the sets of data are sensible, this means that engineers' arithmetic is essential. The section of table below computes sensible values for the depth and breadth of reinforced concrete beams to resist a design moment.

```
70  M      50     5000    0       Design moment after any redis.
71  fy     250    460     2       Char. strength of longit. steel.
72  d      200    800     1  > <  Effective depth.
73  dia    1      10      100    >INT(d/25) <INT(d/17) Bar diams.
74  b      200    1100    1  > <  Breadth of section.
75  nbart  2      20     19    >  Number of tension bars.
76  expos  1      5       5      <INT(fcu/12) Exposure condition.
(0.6*(M*1E8/fy)^(1/3))
((M*1E8/fy)^(1/3))
(M*1E6/(0.0225*fy*d^2))
(M*1E6/(0.0075*fy*d^2))
(INT(M*1E6*5.33/(fy*PI*dia^2*d))+1),(INT(b/125)+1)
```

For 2% reinforcement of characteristic strength fy N/mm2, for a beam of depth 'd' & breadth '2/3*d' & lever arm=d*3/4 we may write:

```
              ┌───Ast────┐
     depth─┐  ┌─b─┐  2%┐     ┌───┬lever arm
     M*1E6=d*d*2/3*0.02*fy*d*3/4, rearranging we get
d=(M*1E6*2/0.02/fy)^(1/3) =(M*1E8/fy)^(1/3)
```

which gives a sensible effective depth for the section. Limits for 'd' are typically from half of this value to twice this value dependent on other parameters. See the first two lines which follow the parameter table which give the minimum and maximum limits respectively for the effective depth. When the expression is too long to be contained on the same line as the parameter to which it refers, the > = or < are provided as flags to say: the expression is given on a line by itself following the table, in the order of occurrence (l to r, t to b) of any isolated > = or < flags given in the table. The > and < are used in the algebraic sense e.g. a<b<c or c>b>a meaning b is bounded by the limiting values of a & c.

Having found a sensible section depth, again from engineers' arithmetic: lever arm≈0.75*d mm, for p% reinforcement, area of steel Ast≈b*d*p/100 mm2 For p% reinforcement, tensile force in steel tfs≈fy*b*d*p/100 N.

```
Bending moment M*1E6=tfs*0.75*d ≈(fy*b*d*p/100)*0.75*d Nmm
                                ≈0.0075*fy*b*d^2*p Nmm
Rearranging                   b ≈M*1E6/(0.0075*fy*d^2*p) mm
For say 1% tensile steel      b ≈M*1E6/(0.0075*fy*d^2) mm
....... 3% .............      b ≈M*1E6/(0.0225*fy*d^2) mm
```

as used in the third and fourth lines which follow the parameter table. Tensile force in steel tfs=M*1E6/(0.75*d)

```
                                  =fy*nbart*PI*dia^2/4 N
```

Thus No. of tension bars nbart=INT(M*1E6*5.33/(fy*PI*dia^2*d))+1 To control crack widths at tension steel level, a sensible limit for the maximum bar centres is 125 mm thus the number of tension bars must be greater than INT(b/125)+1, thus there are two condition which the number of tension bars must exceed i.e. (INT(M*1E6*5.33/(fy*PI*dia^2*d))+1),(INT(b/125)+1) as given on the fifth line following the parameter table. These expressions must be enclosed within brackets and separated by a comma.

It would be inappropriate to use 40 mm dia. bars in a beam of depth 200 mm, or use 6 mm bars in a beam of depth 800 mm, accordingly parameter 73 has both > and < limitations on the bar diameter. For a number of increments zni=4 then the beam depths would be 200, 400, 600 & 800 mm. For the two conditions below only one bar diameter satisfies both conditions.

Effective depth of beam	200	400	600	800
Greater than condition INT(d/20)	10	20	30	40
Less than condition INT(d/17)	11	23	35	47
Diameter satisfying both conditions	10	20	32	40

For the conditions below, two bars satisfy both conditions:
for d=200, 8 or 10 mm bars, for d=400, 16 or 20 mm bars,
for d=600, 25 or 32 mm bars, for d=800, 32 or 40 mm bars.
The two conditions are applied sequentially, thus the less than condition predominates.

Effective depth of beam	200	400	600	800
Greater than condition INT(d/25)	8	16	24	32
Less than condition INT(d/17)	11	23	35	47
Diameter satisfying both conditions	10	20	32	40

Certain parameters are restricted by material strengths e.g. codes require an increase in concrete strength for concrete in severe exposure conditions or an increase in cover to reinforcement for an increase in fire resistance. Code requirements are usually in the form of tables, generally simple expression/s can be devised, e.g. for the five exposure conditions: mild, moderate, severe, very severe, extreme, then <INT(fcu/12) for the 75'th parameter will ensure that sets of data generated from the parameter table will not usually be faulted. 'Usually' is apposite, it is OK if one or two sets of data are failed, as it shows that the error checking in the model is working. It is unhelpful if every run is terminated with a message saying that the concrete strength is inadequate for the exposure condition.

For an example of a beam subjected to two types of loading see sc253:
Case 1 - biaxial bending due to loading Wxd, Wxl, Wyd & Wyl applied in the directions of the principal axes
Case 2 - loading Wvd & Wvl applied in the vertical direction to a beam which is rotated through an angle theta.
To determine a typical loading, it is necessary to separate out the two cases thus: ((2-lcase)*(Wxd+Wxl+Wyd+Wyl)+(lcase-1)*(Wvd+Wvl))
as when lcase=1, applied loading =Wxd+Wxl+Wyd+Wyl
when lcase=2, applied loading =Wvd+Wvl.
The combined expression is long and therefore follows the PARAMETER table, referenced by an isolated equals sign in the table.