ACES
BRIDGE ANALYSIS
SYSTEM

USER MANUAL

February 2007

This page intentionally left blank

ACES HELP SYSTEM

CONTENTS

PART 1 GETTING STARTED

1.1 Setup
1.2 Menus
1.3 User Interface
1.4 Units
1.5 Coordinate System
1.6 Sign Convention
1.7 Saving Model Data Files
1.8 Retrieving Model Files
1.9 System Limitations & Restrictions

PART 2 CREATING THE MODEL GEOMETRY

2.1 How to Create a Model
2.2 Parametric Modelling Using Templates
2.3 Changing the Model Geometry
2.4 Defining Lanes
2.5 Member Properties
2.6 Adding New Sections to the Data Base
2.7 Section Properties Calculator
2.8 Composite Sections Calculator
2.9 Element Properties

PART 3 STATIC & MOVING LOADS

3.1 Creating & Deleting Load Cases
3.2 Moving Vehicle Loads
3.3 Patch & Lane Loads
3.4 Nodal Loads
3.5 Member Loads
3.6 Element Loads
3.7 Hydrostatic Loads
3.8 Temperature Expansion/Contraction Loads
3.9 Other Static Loads & Dead Load

PART 4 ANALYSIS

4.1 Analysis Options
4.2 Frame & Beam Analysis
4.3 Grillage Analysis
4.4 Finite Element Analysis
4.5 Dynamic Analysis
4.6 Second Order Analysis

ACES HELP SYSTEM

CONTENTS

PART 5 RESULT & REPORTS

5.1 Graphical Results
5.2 Tabular Reports
5.3 Envelopes
5.4 Combinations
5.6 Finite Element Results

PART 6 INCREMENTAL LAUNCHING MODULE - Main Index

PART 7 CONTINUOUS BEAM MODULE - Main Index

PART 8 MODEL TEMPLATES - Main Index

PART 1
GETTING STARTED

PART 1.1 Setting up and Uninstalling ACES

1.0 Installation

Insert the installation CD into the PC then bring up ***Explorer*** (from the *Start/Programs* menu). List the contents of the CD drive and locate the *Setup* program. Double click on *Setup.exe* to run the installation program then follow any instructions that may be given.

ACES will be installed by default into the ***C:/Program Files/Aces6*** folder. If you wish to install it into another folder (e.g. *C:/Aces*) refer to the installation instructions that came with the CD or contact ACES Analysis Systems for assistance.

2.0 Creating an ACES Icon on the Desktop

Use Windows *Explorer* to display the contents of the folder into which ACES was installed (e.g. *C:/Program Files/Aces6*). Locate the file called ***aces6.exe*** (it will be prefixed with a small bridge icon), click onto it with the *right* mouse button then drag it out to a free spot on the desk-top and release the button.

A short menu will pop up on the screen. Scroll down the list of items with the *left* button and select ***Create Shortcut(s) Here***. Windows will create an ACES icon at that spot on your desktop. You can now move it to another position on your desktop and rename it if you wish.

3.0 Removing (Un-Installing) ACES

Click on *Start / Settings / Control Panel / Add-Remove Programs* , highlight *ACES Bridge Analysis System*, click *Add/Remove..* then respond accordingly. Note that the full contents of the ACES folder may not always be deleted **-** you may need to use *Windows Explorer* to complete the task.

PART 1.2 Menus & User Interface

1.0 GENERAL OVERVIEW

The ACES graphical interface is based on the standard Windows environment with its now familiar menu structure, tool bar layout and message prompts.

Pull-down menus are generally self-explanatory. Most have a menu option labelled ***MenuHELP..*** that, when selected, displays a dialogue box containing a brief description of all other options in that menu. Some of the key menus also have an option called ***HowTo..*** that provides a brief description of the more frequently asked questions.

All help dialogue boxes can be printed using the commands ***File/Print/Last help file***.

2.0 MENU BAR OPTIONS

File

This menu option enables a range of file-related operations to be performed. Refer to ***File / Menu HELP*** for a short description of all file options in this menu.

Structure

This menu option enables changes to be made to the base model. Refer to the main contents page for a complete list of all features or to ***Structure / Menu HELP*** for a short description of all options in this menu.

Loads

This menu option enables loadings to be applied to the model. Refer to the main contents page for a complete list of all features or to ***Loads / Menu HELP*** for a short description of all options in this menu.

Analyse

This menu option contains options for performing a range of different types of analyses (frame, grillage, finite element, second order and dynamic). Refer to *PART 4.1* for details.

Results

This menu option enables results of the analysis to be viewed graphically. Refer to *PART 5.1* for detailed instructions in doing this.

Reports

This menu option produces tabular reports (i.e., results are presented as numeric text values in a table rather than graphically). Refer to *PART 5.2* for detailed instructions in creating tabular reports.

View

This menu option enables the structural model and graphical results to be viewed from many angles. It also allows you to view any lanes that may have been created and to toggle on the background grid if required.

Zoom-Pan

This menu option allows you to zoom and pan around the model.

Activate

This menu option enables parts of the structure to be activated and de-activated i.e., groups and blocks of nodes, members and elements can be selectively "suppressed" (not shown on the screen or in reports). Note that they are not *deleted* from the model but merely de-activated.

This can be very useful in complex 3D frames and FE models where you wish to view and interrogate only one visible part of the model in order to better see its attributes or results. Once you have created these special "views" they can be saved to a database and recalled later.

Inquire

This menu option enables "spot" inquiries to be made on a large number of model attributes including:

- Exact coordinates of selected nodes
- Distances between any two nodes
- The number of a selected node, member or element
- The largest node, member or element number in the model and its location
- The location of a node, member or element of a specified number

- ▪ Type and length of a specified member
- ▪ Element type and area

Shortcuts

This option saves the last ten menu commands used. In effect it mimics a macro that continuously retains the last series of menu accesses made by you during the current session. It is particularly useful if you are repeating a series of actions that are deeply nested within several layers of menus and submenus (such as adjusting the path of a vehicle or selecting a new member range).

LastMenu

This displays the last menu or submenu that was accessed. It is particularly useful if you are repeating a series of actions that are deeply nested within several layers of menus and submenus.

MyMenu

This option allows you to display a menu of up to 15 of your favorite command strings and actions. It can be customised to suit your own requirements by selecting the menu options ***Settings/Customise MyMenu***. Refer to *Customise MyMenu* in *PART 1.3* of this help system for further details.

Settings

The *Settings* option in the menu bar allows a large number of diagram and model attributes and defaults to be customised then saved for future sessions. These include:

- ▪ Screen windows, diagram and text colours and line styles
- ▪ Background grid
- ▪ Display of symbols
- ▪ Node, member and element numbers and local axes
- ▪ Member and element property type colours
- ▪ Customising *MyMenu*
- ▪ Decimal places
- ▪ Axes

Refer to *PART 1.3* for more information

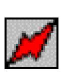
Help

This menu option provides information about the current ACES system and contains the help system.

3.0 TOOL BARS

3.1 Main Tool Bar

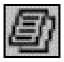
This extends across the top of the screen, immediately below the menu options. To quickly determine the function of a tool bar icon, let the cursor hover over it for a moment. A yellow description bar will pop up. Most buttons are self-explanatory and will not be described in any further detail here. However, the following require some elaboration.



Begin the analysis. The problem will be solved using the current analysis settings.

Displays the attributes dialog box for the last-drawn, or currently selected, results diagram. It allows attributes such as vector type, colour, scale and so on to be selected. The contents of the dialog box will depend on the nature of the current, or previously drawn, results diagram i.e., whether it was a moment diagram, shear, reaction, displacement diagram etc.

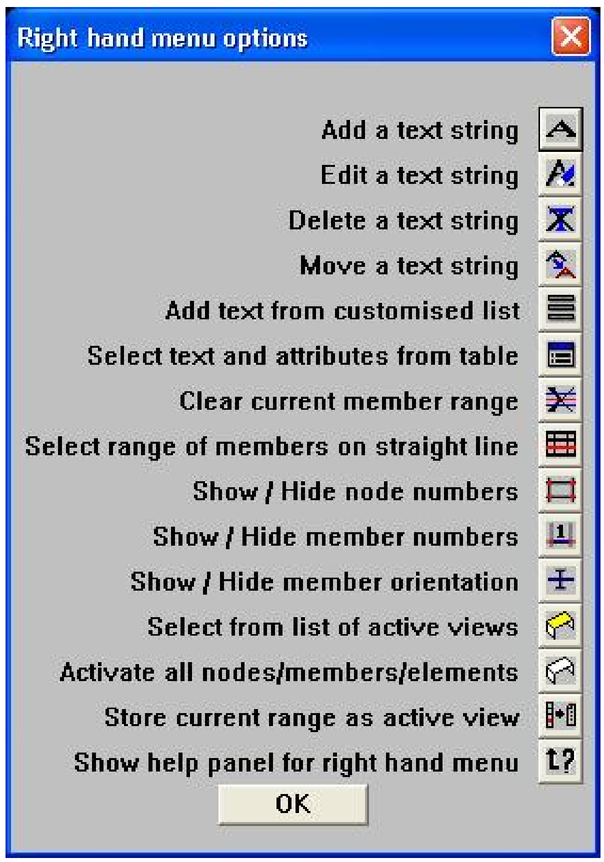
Redraws the graphical results diagram that was last displayed (i.e., before it was cleared). The diagram will be redisplayed using the currently active settings and attributes. It is particularly useful after a zoom action is performed.

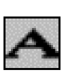


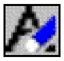
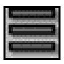
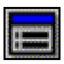
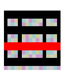
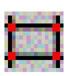
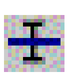
Redisplays the tabular report that was last generated.

PART 1.2 Text Editing Options & Side Menu

The side menu contains icons to assist in adding, editing, moving and deleting text and other icons to allow member ranges, active views and member and element node numbers to be easily displayed. Click the ***Up-arrow*** button at the bottom of the right-hand (side) menu to bring up the following icon help panel:



Detailed Description of Icons	
	***Add a new text string***. Clicking this icon will display a text entry dialog box. Type in the text you would like to place on the current diagram, select a font size and colour, then click OK to insert the text. ACES will prompt you to click on the spot in your screen drawing where you would like the text placed. Note that the text string will also be inserted into a text library for re-use on other drawings (refer to the last icon for details).

	***Edit a text string***. ACES will ask you to click on the first character of the text string you wish to edit. The string will then be echoed into a text edit panel that will allow you to perform edit operations.
	***Delete a text string***. After clicking this icon ACES will prompt you to click on the first character of the string that you would like to delete. Continue deleting other strings or select another operation.
	***Move a text string***. After clicking this icon ACES will prompt you to click on the first character of the string that you would like to move. Click and drag the string to its new location. Continue moving other strings or select another operation.
	***Add text from a list***. Displays a menu selection of up to 10 frequently used text strings. New strings can be added, but they must already exist in the text library (refer to the next icon, shown below). Once a button is clicked, the text is displayed in an editing box and can be modified as required or applied as is. ACES will then prompt you to click on the spot on the drawing at which you want the text to be placed.
	***Select text from table:*** Displays a dialog box of all available text strings. Check or uncheck strings that you would like to appear on the current diagram. Click ***Apply*** to display the selected text strings on the diagram. Note that all strings can be manually edited within this dialog box and their location and attributes changed if required. New strings can be added or unwanted strings deleted. Whenever a new string is created using the "***Add a new string***" icon, it is also added to the next available line in this list (together with all its attributes). A maximum of only 20 strings can be stored in the table.
	***Clear current member range***. Click this button to clear the currently selected member range.
	***Select a range of straight member***s. Click this button to select one or more members to add to the current range. Note that this will select all members lying on a continuous straight line. If you wish to select only a part of a full line of members then use the menu options: *Results / Select a range of members / Members on part of a straight line*. Note that to add another member to the current member range all members in the model must be active. To do this, click the **Activate all members/nodes/elements** button described below.
	***Show/Hide node numbers***. This button acts as an On/Off toggle. When clicked the first time it will show all nodes and their respective numbers. When clicked again the node numbers will disappear.
	***Show/Hide member numbers***. This button acts as an On/Off toggle. When clicked the first time it will show all member numbers. When clicked again the member numbers will disappear.
	***Show/Hide member orientation***. This button acts as an On/Off toggle. When clicked the first time it will show the orientation of all member in the model. When clicked again the member orientation symbols will disappear. Note that for 2D structures ACES will display the model in as a 3D perspective in order that the orientation symbols can be properly viewed.
	***Select from a list of active views***. Up to 20 active views can be stored at various stages of building and interrogating the model then retrieved using this option. For example, the currently selected range of members can be saved as an active view using the commands: *Results / Select a range of members / Add current range to active views list*.
	***Activate all members/nodes/elements***. This button will make all nodes, members and elements in the model active. (Note that parts of the model will be de-activated if an active view is selected using the ***Select from a list of active views*** icon described above).

PART 1.3 The ACES User Interface

1.0 SETTINGS

The *Settings* option in the menu bar allows a large number of diagram and model attributes to be customised then saved for future sessions. Not all will be described here-under since they are relatively self-evident. The following, however, warrant comment:

1.1 Customise 'MyMenu'

This option allows *MyMenu* to be customised. In order to use this feature, however, the menu actions that you wish to include in *MyMenu* must first exist in the *Shortcuts* menu. In other words, you will have to manually perform the required actions on your model at least once in order that they can be 'recorded' and placed into the *Shortcuts* menu.

Once there, *Customise MyMenu* can be used to copy them across into *MyMenu*. To allocate a command string to one of the menu buttons, click on the target button then select the required action from the *Shortcuts* menu list that will appear. Click *Save* to save the entries in *MyMenu* for future sessions.

1.2 Background Grid

This option allows the colour, mesh spacing and reference point of the background grid to be changed. By default ACES will draw the grid using the origin (*X=0, Y=0*) as the reference point. This can be changed to any defined node (or point) on the model.

1.3 Line Styles

ACES creates all model drawings and results diagrams using a palette of 33 line "styles". Line styles are not only used in drawing the model but also for displaying member types, axes, labels, added text, graphical results and so on. This feature allows the default attributes for these 33 line styles to be changed. Lines can be changed in terms of their colour, thickness and style (solid, dashed etc).

1.4 Member, Element, Results Colours

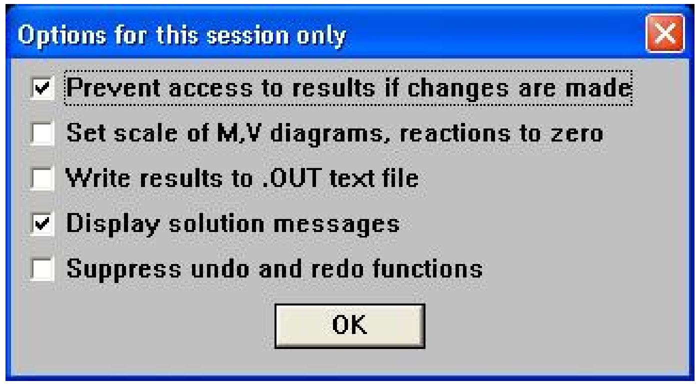
This option allows default colours to be assigned to member and element property types and to graphical results (moments, shears, deflections etc).

1.5 Save/Retrieve Settings

This option allows you to save your customised settings to one of two attribute sets - a *normal* set and an *alternate* set. The normal set is the one ACES always uses when it starts up. If you wish to use the alternate set it must be activated either just after a new model has been generated or once an existing model has been loaded from disk. *Retrieve Settings* allows either of the two attribute sets to be reloaded.

1.6 Session Options

1.7 This feature enables a number of global options to be set for the current session. Note, however, that they will <u>*not*</u> be saved as system defaults settings for future use. The following dialog box will be displayed:



- • *Prevent access to results if changes are made*: If any of the member/element properties or load case menus are accessed, or the respective values viewed or edited and this option is checked *on*, ACES will require that you reanalyse the problem before access is allowed to results or reports.

- • *Set scale of M,V diagrams & reactions to zero:* If this option is checked *on* ACES will always reset the scale of moment, shear and torsion diagrams to zero prior to displaying the results i.e., ACES will automatically select a scale for the diagrams.

- • *Write results to OUT output file*: ACES saves all model and results data to two different files - a binary graphics file (*PLT* extension) and a text file (*OUT* extension). Only the graphics file is used by the program to display model and results diagrams - the text file is ignored (it may be thought of as a “backup” option to be used as and when needed).

  For large models with thousands of members and nodes and hundreds of vehicle loadings the *OUT* output file may take a long time to write. This will not only slow the analysis considerably but will also create a very large (and unnecessary) additional file in the *...\output* directory.

- • *Display solution messages:* If this option is checked *on* ACES will display the progress of the analysis via messages on the bottom status line.

- • *Suppress Do and Redo functions:* If this option is checked *on* ACES will deactivate the *Do* and *Redo* functions (which may consume significant disk space and, for large models being processed on older PCs, slow the modelling).

1.7 Structure Colours

This option allows the colour in which nodes, members, elements, supports and their associated numbers are displayed on the drawing.

1.8 Text & Member Axes Sizes

This option allows the size, angle and line style for node, member, element and results values to be set . It also allows the length of member local axes to be specified. All sizes are in *mm* units. Element axes are automatically scaled as a proportion of the element size.

2.0 Creating and Manipulating Text

Text can be added to the diagram and manipulated by using the series of icons on the right-hand vertical tool bar. Refer to *Part 1.2* for an explanation of what the text icons do and how they can be used.

PART 1.4 Units

1.0 Geometry, Loadings & Results

Although ACES is essentially non-dimensional it recognises and uses the standard range of metric and imperial units. Irrespective of which units are used, however, they *must* be consistent. For example, if metres (*m*) and kilonewtons (*kN*) are chosen as the desired units, output will be given as follows:

kN	for forces
kN.m	for moments in linear members
kN.m/m	for bending moments at centroids of elements
Mpa	for stresses in finite elements
Radians	for rotations
Degrees	for principal angles

2.0 Finite Element Moments

Moments at centroids of finite elements are given as *moments per unit width*. Rotations are always in radians and the principal angles are given in degrees:

3.0 Lane Loads

Lane loading is given as a value per linear metre spread over the full lane width. (E.g. 12.5 kN/m over a 3 metre lane width. This is equivalent to 4.17 kN/m2)

PART 1.5 Coordinate System

1.0 Global Coordinate System

Structural models in ACES are based on the *RIGHT-HANDED ORTHOGONAL CARTESIAN* coordinate system as shown in Figure 1. However, for interpreting output results, distinction must be made between the global (or nodal) co-ordinate system and the local (member or element) co-ordinate system. In the terminology to be used in this section of the User Guide the parameters *U1, U2, U3* will correspond to forces and displacements while *U4, U5, U6* correspond to moments and rotations.

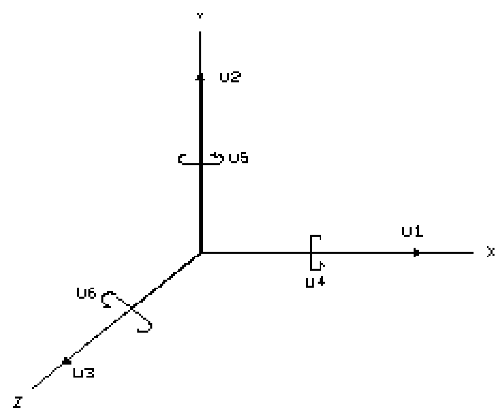


Figure 1 Global Coordinate System

The global coordinate system (X, Y, Z) is an arbitrarily chosen system for the entire structure. The origin of the system may be located at any arbitrary point and the direction of the axes are chosen to suit the structure. All input data is described with respect to the global co-ordinate system and all nodal distortions, forces, and reactions are similarly given. Two-dimensional structures are always assumed to be contained in the X-Y plane.

2.0 Local Co-Ordinate System for Members

A local co-ordinate system (***x, y, z***) is associated with each member, and all member input data and member output forces are specified in terms of this system. The local **x** axis coincides with the long-itudinal axis of the member running from the *Start* to the *End* of the member, while the direction of the other principal axes are based on the right-hand-rule. To determine the direction of member axes select *Settings/Show Symbols* and place a tick in the *Member local axes* item.

Unless the member is axially symmetric (as is the case in all parametrically generated models), the orientation of one of the principal axes of the member with respect to the global axes needs to be specified. This quantity is called the ***BETA*** angle (*B* in Figure 2). For plane structures ***BETA*** should be omitted or specified as zero.

Referring to Figure 2 on the next page, let *A* be a plane containing the member **x** axis and a line parallel to the global *Y* axis. Let **y'** be a vector in this plane and perpendicular to the **x** axis. The direction of **y'** is taken so that the projection of **y'** on the *Y* axis is in the positive *Y* direction. Then *B* is the angle from **y'** to **y**, positive by the right-hand rule around **x**.

This definition is not sufficient if the **x** axis is parallel to the *Y* axis, in which case the plane *A* is indeterminate. Then *B* is the angle from the **-***X* axis to the **y** axis if the **x** axis is in the same direction as the *Y* axis, and from the *+X* axis if not. The default value if *B* is not specified is zero.

For plane structures, the local axis and the direction of **y** is determined according to the right-hand rule. Because of this convention, the direction of the local **y** axis will not necessarily always be in the positive direction of the global *Y* axis.

WARNING: Note that the directions of the local axes may be changed if a plane structure is redefined and analysed as a space structure.

Viewing & changing member axes sizes: To view member axes select *Settings/Show Symbols* and place a tick against *Member local axes*. To change the size in which member axes are displayed on the diagram select *Settings/Text & local axes size/Member local axes* and enter a size in *mm* units.

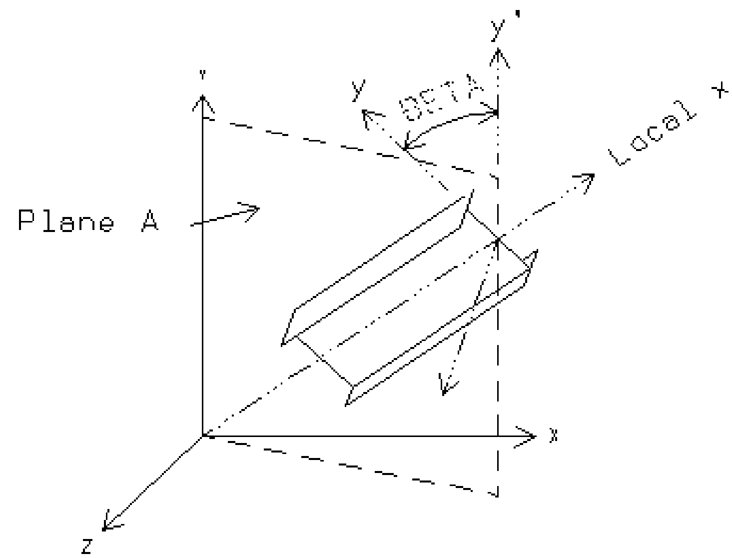


Figure 2: Local Coordinate System for Members

3.0 Local Coordinate System for Finite Elements

A similar local coordinate system (**x' y' z'**) is associated with each element. For plane stress, plane strain, plate bending and tridimensional cases, this local co-ordinate system will coincide with the global coordinate system. To determine the direction of element axes select *Settings/Show Symbols* and place a tick in the *Element local axes* item.

For shells, the **x'** axis (local) will be parallel to the line joining the first numbered node to the second numbered node. The **y'** axis is perpendicular to the **x'** axis, lies in the plane of the element, and is directed towards the remaining node(s). The **z'** axis is normal to the plane of the element (refer to Figure 3).

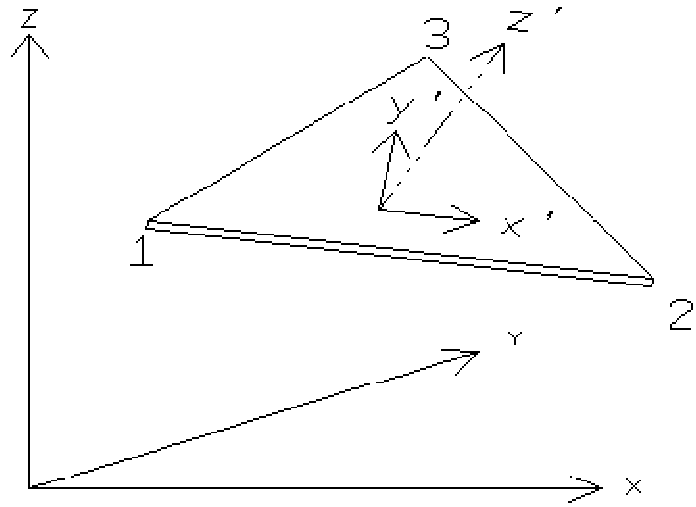


Figure 3: Local Coordinate System for Finite Elements

PART 1.6 Sign Convention

1. Nodal Deflections, Rotations, Forces & Moments

These are given with respect to the *Global* coordinate system. Their positive directions are in accordance with the right-handed coordinate system. To view nodal results in 2D grillage and slab models you may need to change the viewing angle. Click on the *Isometric view* icon in the tool bar then on the *Redraw last-diagram* icon to do this.

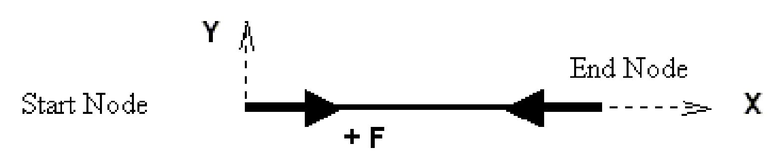
2. Reactions

These are given with respect to the *Global* coordinate system. Their positive direction is in accordance with the right-handed coordinate system. To view reactions in 2D grillage and slab models you may need to change the viewing angle. Click on the *Isometric view* icon in the tool bar then on the *Redraw last-diagram* icon to do this.

3. Member End Forces

Member end forces (moments, shears, torsions, axial forces), are given with respect to the *Local* coordinate system. Positive directions are in strict accordance with the right hand rule when applied to the *local* coordinate system. A *positive* axial force at the start node of a member will therefore correspond to *compression* in that member.

In a 2D frame, for example, where the *Global Y* axis is upwards and the *Z* axis is out of the page, a *positive* bending moment *Mz* at the start node corresponds to bending *tension* stresses on the positive *Y* face of the member. In numeric, or printed, output reports, therefore, a member under <u>*constant*</u> member force will experience a change in sign of the force between its start and end nodes. (Results displayed *graphically* already makes allowance for this).



Force Sign Convention in a 2D Frame Member

4. Element Bending Moments

These are given with respect to the *local* element axis. Positive element moments will induce tension on the **+z** (local system) face. Note with care that ***Moment X*** calculated for an element is *parallel* to the local **x** axis (i.e. it is about the local y-axis). For two dimensional analysis, the *local* axis system always coincides with the *global* system. Results are given per unit length (e.g. kN.m/m).

NOTE: When creating contour or solid shadediagrams it is very important that local axes of all elements are all orientedin the same direction, otherwise the diagrams will becomemeaningless.

To check their orientation click **Settings/Show symbols** from the top menu and tick the **Element local axes** box.If any element axis does not align itself with the corresponding global axes then such elements will need to be considered separately. Although ACES has routines for reversing and rotating local element axes and aligning the axes of groups of elements, (refer to *Structure/Finite Elements*), it may not always be possible to align them exactly, particularly if irregularly shaped triangles are used. Alternatively, use only the principal values rather than the basic ***x*** and ***y*** values.

▲

5. Element Stresses & Strains

Element stresses and strains are given with respect to the *local* axis system. Stresses (and strains) are considered positive if, on the '*positive faces*', they are in the positive direction of the axes. Thus, tensile stresses are always considered as positive and compressive stresses as negative. This convention remains true for two dimensional analysis, in which only *Stress-X*, *Stress-Y* and *Stress-XY* (same as *Stress-YX*) will be present. For two dimensional analysis, the *local* axis system always coincides with the *global* system.

Stresses and strains are normally calculated at element centroids, unless the analysis option for displaying values at element nodes is invoked. Note that in bending cases, stresses and strains are given in accordance with plate bending theory (i.e. *'Moment X'* means the moment parallel to the local **x**-axis).

NOTE: When creating contour or solid shadediagrams it is very important that local axes of all elements are all orientedin the same direction, otherwise the diagrams will becomemeaningless.

To check their orientation click **Settings/Show symbols** from the top menu and tick the **Element local axes** box.If any element axis does not align itself with the corresponding global axes then such elements will need to be considered separately. Although ACES has routines for reversing and rotating local element axes and aligning the axes of groups of elements, (refer to *Structure/Finite Elements*), it may not always be possible to align them exactly, particularly if irregularly shaped triangles are used. Alternatively, use only the principal values rather than the basic ***x*** and ***y*** values.

Element Shears: Refer to *Part 5.5* for a detailed discussion regarding the interpretation of shear forces and shear stresses.

6. Curvatures/Rotations

Curvatures are the reciprocals of radii of curvature. A positive curvature will correspond to a positive strain on the **+z** face.

Rotations are measured around an apprpriate axis. For example, if the main girders are parallel to the *Global X* axis then the main rotation would be a "*Rotation-z*" for a 2D frame model (*PLANE FRAME* type) , or "*Rotation-y*" for a 2D grillage model (*PLANE GRID* type).

7. Horizontal Patch Loads

Face the direction in which the patch extends on the model (ie from its start coordinates to its end coordinates). The positive y-direction will then be to your left and the negative y-direction to your right (i.e. the convention follows the right hand rule). You can check the direction that ACES has used by selecting <*Results*><*Applied Loadings*> but remember that the patch loads are applied as nodal loads in global directions.

In situations where a patch, (or sub-patch in the case of a curved patch load), is not parallel to either the global-X or global-Y axis, then two nodal forces are generated which, when added vectorially, equal the intended load.

PART 1.7 Saving & Retrieving Model Data Files

1. Saving Job Data

To save all model data and any results files that may have been generated click in turn on *File / Save all model data* or *Save model data as..* and provide file information as requested. Note that the graphical results files (designated with extensions *PLT, PLM, PLF*) can be quite substantial if the structural model is very large **-** you may not be able to save them to a 1.4Mb floppy disk. Periods, (the "." symbol), are permitted in data file names.

To check the size of files generated by ACES use *Windows Explorer* to view the contents of the folder in which output results are stored (e.g. *C:\Program Files\Aces\Outpdat,* OR *C:\aces5\outpdata* etc). Use the option, *File / Set project directory* to specify the default directory for your ACES model and results files (the initial default path will be set to ..*\USERDATA*).

If changed, the new path will be written to a file call ***Proj_dir.dat*** in the ..*\TEMPDATA* subfolder. It will be read and used in subsequent runs. Note that if you open a previous job and change the project directory, the previous job's default file path will be retained when you try to save it again.

2. Retrieving Saved Job Data

At start-up ACES will give you the option to load the last file worked on. If you wish to retrieve a model file that was generated and saved earlier use one of the two *File* options: *Open an existing file* or *Open a file last used*. If a results file is available ACES will ask if you want it loaded into the system. (Some results files can be very large and may not be required in the immediate session).

PART 1.9 System Limitations & Restrictions

1.0 MODEL LIMITS

1.1 Geometric & Loading Limits

1.1.1 General Modelling Limitations

The following geometric limits generally apply to all models (refer also to *Section 1.3* for limitations associated with Dynamic Analysis):

Nodes	**6000**
Members	**6000**
Elements	**6000**
Member Property Types	**500**

Although ACES has the ability to handle mixed 2D/3D frame and FE models (such as slab grillages and pier/pile cap/pile combinations), there are a number of significant structure types that it is not capable of modelling viz:.

- Variable modulus elasticity across the section (i.e., non linear element/member stiffness)
- Members subject to tension or compression only (such as cables)
- One-way supports (such as non-uplifting bearings)
- Non-linear behaviour
- Forced-function dynamic response

1.1.2 Loading Limitations

ACES allows up to 99 individual ***load cases*** to be created with up to five (5) identical or different vehicles in each load case. Vehicles in any one load case can either move in exactly the same path or along different paths. The static location of all vehicles at one particular point along their paths within a *load case* constitutes an individual ***vehicle loading***. Refer also to *Section 3.1* for further details regarding vehicle loads. The table below summarises the total number of individual vehicle *loadings* that can be generated in any one ACES run:

2D Grillages	2000
3D space frames and FE shell models	1000
Trusses	3000

1.1.3 Envelope Limitations

The maximum number of envelopes that can be created in any one run is 50.

1.2 Moving Load Analysis Problems

The limitations given in Section 1.1 above will vary depending on the total number of individual loadings generated by all load cases containing moving vehicle loads. (Actual limits are dependent on the total number of discrete nodal loads). Unfortunately, there is no simple way of determining these limits. If the program aborts during analysis it is probably due to the fact that there are too many vehicle loadings. In this event you may need to take one or more of the following actions:

- Reduce the number of moving load cases
- Remove vehicles from multi-vehicle load cases
- Shorten vehicle travel paths
- Reduce vehicle movement increments
- Remove large patch loads from vehicle load cases
- Don’t include Dead Load in a moving vehicle load case

1.3 Dynamic Analysis Limitations

Due to the way in which ACES has traditionally implemented dynamic analysis there are severe limitations placed on the total number of nodes that the model can have. The following node limits apply to different model types when performing Dynamic Analysis:

Space trusses	**455**
Space frames	**227**
Shell elements	**227**
All other types	**500**

If your model is much larger than this you may consider creating a scaled-down version for performing the dynamic analysis prior to generating the more complex version.

2.0 PARAMETRIC MODELS

When changes are made to the base model parameters ACES always regenerates the model “from scratch” using its own default material property types and assignment rules. Therefore, if you change any of these basic parameters (such as the number of spans or main girders, the skew angle, and so on), you may affect the member (and element) property assignments throughout the regenerated model. So, always check your model carefully after parameters are changed or modified.

Note also that any sub or super-structure manually added to the original base model geometry (or, indeed, any additions or deletions of nodes, members and elements that you may have made), will also be lost.

3.0 MOVING VEHICLE LOADS

3.1 Swapping Movement Direction

To swap the direction of movement of a vehicle *don’t* set the *Start X* coordinate of the vehicle path to be larger than the *End X* coordinate – the vehicle will be flipped over and the reference axis will be aligned with the top wheels of each axle. Instead, use the menu commands: ***Loads / Edit vehicle loads / Vehicle # / Swap movement direction*** where **#** represents the vehicle number whose direction of movement is to be swapped.

This page intentionally left blank

PART 2

CREATING THE MODEL GEOMETRY

HOW TO CREATE A MODEL

1.0 GETTING STARTED

This section of the Manual will step you through the process of creating and analysing a basic 2-dimensional grillage, slab, frame or continuous beam structure using the parametric modelling technique (often referred to as the “template” method). It is based on the concept of selecting a “template” that most closely resembles the structure you are attempting to model and entering parameters and dimensions expressed in real engineering terms. The program then generates the model geometry.

For a quick overview of the modelling process refer to the document: *Bridge Modelling Made Easy* (it's in Microsoft *WORD* format).

Otherwise, follow the more detailed tutorial outlined here-in. Run ACES, either from the *Start* Menu or from the icon on your desktop (if you have one). A dialogue box will appear with a number of options:

- *Start a completely new job*
- *Reload the last job that was run on the PC you are now using*
- *Open an existing job file that was previously created and saved to disk*
- *Select the Section Properties, Continuous Beam or Incremental Launching module*
- *Select a job file from a list of up to five previously run jobs*
- *Exit from ACES altogether*

Click the option that suits your requirements.

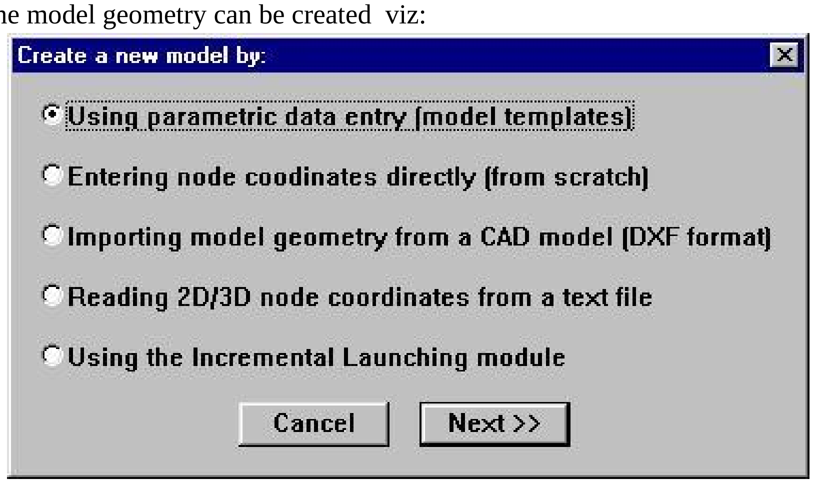
1.1 Job Identification

A job identification window will pop up. This allows basic job identification information to be entered. A heading/title of some description *must* be provided - all other Job ID data is optional.

When finished, click *Next>>*. Note that if you press *ENTER* after any data is keyed in this will have the same effect as pressing the *Next>>* button. All data currently entered will be accepted and control will transfer to the next dialogue box.

1.2 Selecting a Modelling Option

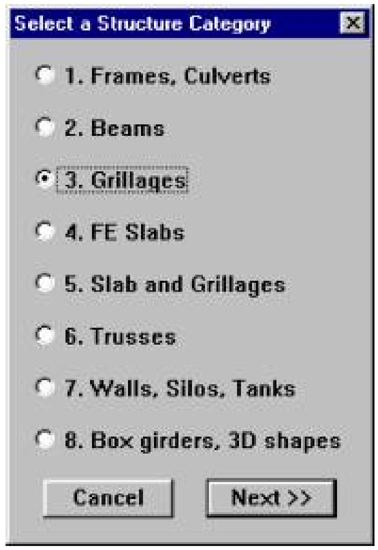
A menu of modelling options will be displayed. They provide a number of different ways in which the model geometry can be created viz:



By default the first option, *Using parametric data entry*, is already highlighted. Click *Next>>* to accept it and to continue with the modelling process. If you wish to create the model geometry by reading in a list of 2D/3D node coordinates refer to *Reading Node Coordinates & Member Numbers*.

1.3 Selecting a Structure Category

A panel displaying a range of generic structure types will appear, together with a list of options labelled *Select a structure category* (as shown on right). Option 3, *Grillages*, should be highlighted. If you wish to create a grillage model click *Next>>* to accept the selection.



If you are creating a different model type (such as a slab, frame or continuous beam), highlight that option with the left mouse button then click *Next>>* to continue.

Note carefully that if you select the wrong structure category at this point there is no way of tracking back through the previous options. You will have to continue through to the last panel in the model generation wizard then start from the beginning with *File / New Job...* (The last dialogue box is the *Tips* panel).

Refer to PART 8 for all available generic structure types together with a complete list of parameters relevant to the templates representing each model type.

1.4 Selecting a Structure Type

After selecting the required structure category (e.g. the *Grillage Category*), a panel with nine different grillage types will be displayed together with a dialogue box describing each type. Similar panels will be displayed for continuous beams, frames and slabs. Click on the required grillage type (e.g Type 2 grillage with parallel girders and uniformly skewed supports) and press *Next>>*. (*Part 8* of the User Manual lists all model templates available in ACES and the parameters required to define them).

Note that the template icons displayed under the various structure categories represent the overall layout of the structural model and not its intricate details. They are, in effect, large pictorial "icons" that show only the main girders and principal support lines. Exact support conditions, transverse grillage members, bearing points, diaphragms and so on are not shown (*Part 8* of the User Manual detail all parameters required for a given model template). Click here to display the next step in the modelling process.

Section 2 below describes other ways in which a model can be created.

2.0 OTHER WAYS OF CREATING A MODEL

2.1 Entering Node Coordinates Directly

Option 2 in the modelling options dialogue box (refer to Section 1.2) enables the basic model geometry to be created by manually entering *X*, *Y* (an Z) node coordinates directly into the system. The geometry manipulation tools in the *Structure* menu are then used to add members, elements, supports and so on. Member and element property types will also have to be created and manually assigned to the different parts of the model. Refer to the relevant parts of this Manual for instruction in doing this.

If vehicle loads are to be applied to the model, you will also be required to specify a running surface for these vehicles. This must be done by activating the relevant nodes, members and elements just prior to performing the analysis.

2.2 Importing Node Geometry From a CAD Package

Option 3 in the modelling options dialogue box (refer to *Section 1.2* above) enables basic model geometry to be imported from a CAD package in *DXF* format. However, the only DXF commands recognised by this module are as follows:

POINT
LINE
POLYLINE
SEQUEND
AcDb2dVertex

Therefore, if the model imported into ACES does not look right, you may need to modify the CAD drawing so that it contains only points and lines. For example, structural elements represented by circles, arcs and curves would need to be converted to short line segments. Once read into ACES you will also have to manually create and assign member and element property types to the various model elements. (Alternatively, you could dump point coordinates to a text file then use the ACES functions *File/Merge/2D-3D Node Lists* and *File/Merge/Members* to generate the model - refer to Section 2.3 below for details)

If vehicle loads are to be applied to the model, you will also be required to specify a running surface (Section 3.2) for these vehicles. This must be done by activating the relevant nodes, members and elements just prior to performing the analysis.

2.3 Reading Node Coordinates & Member Attributes from Text Files

Option 4 in the modelling options dialogue box (refer to Section 1.2) enables the basic model geometry to be created by reading in a list of 2D or 3D node coordinates. Node coordinates can be determined using a spreadsheet, calculator or another CAD package and must be saved to a text file in the format given below.

After the coordinates have been created another text file containing members with their associated node numbers and property type attributes can be read in to complete the model using the *File/Merge/Members* menu options (refer to *Section 2.3.2* below).

Note that this feature also allows "topographical" contours and shaded contours of the values represented by the nodal "Z" values to be produced. It could, for example, be used to monitor the shape of an embankment, highway slope or distorted bridge deck. Refer to *Section 5.1* (subsection 1.5) for a detailed description of how this technique can be used.

2.3.1 Node Coordinates File

Either spaces or commas can be used as delimiters between node numbers and coordinate values. Any number of nodes may be included in the list. There is no requirement to specify the exact number in the file nor is there a limit on the total number of nodes that can be read in.

Note that line 2 is only a label - the actual number of coordinates is entered in line 3. If a 3-dimensional list of nodes is to be read in, line 3 should read ‘3’ and each node must have an *X,Y* and *Z* coordinate.

Line 1: Header information (e.g. Coordinates of substructure)
Line 2: The following label **- “*Number of dimensions*”**
Line 3: The number of dimensions expressed as an integer (e.g. 2)
Line 4: The node number and X,Y coordinates of the first node in the list
Line 5: The node number and X,Y coordinates of the second node in the list
“ “ “ “ “
“ “ “ “ “

2.3.2 Member Attributes File

After the coordinates have been created another text file containing members with their associated node numbers and property type attributes can be read in to complete the model using the *File/Merge/Members* menu options. Either spaces or commas can be used as delimiters between member and node numbers and the property type attribute. Any number of members may be included in the list. There is no requirement to specify the exact number of members in the file nor is there a limit on the total number that can be read in.

Note that line 2 is only a label - the actual number of coordinates is entered in line 3. If a 3-dimensional list of nodes is to be read in, line 3 should read ‘3’ and each node must have an *X,Y* and *Z* coordinate.

Line 1: Header information (e.g. Coordinates of substructure)
Line 2: Member number, start node number, end node number, member property type
Line 3: Member number, start node number, end node number, member property type
“ “ “ “ “
“ “ “ “ “

PART 2.2 Parametric Modelling Using Templates

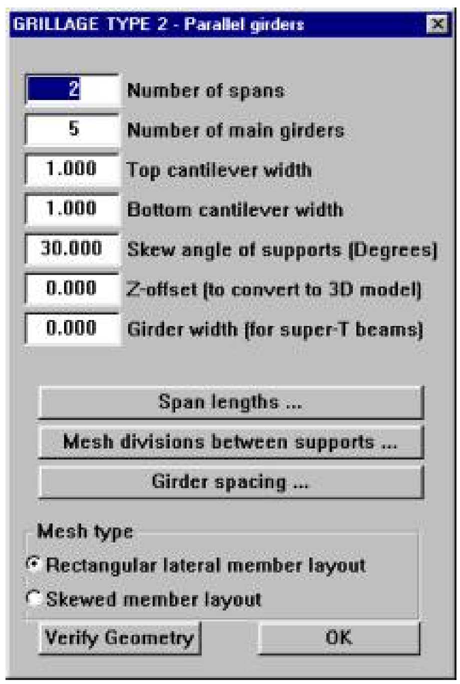
2.0 ENTER GEOMETRY PARAMETERS

2.1 Grillages

2.1.1 Grillage Types 1-6 and8

The default template displayed by ACES is a schematic representation of a bridge deck with uniformly skewed supports together with a data entry window labelled *GRILLAGE TYPE 2 - Parallel Girders* (refer to an example of the dialogue box below). Note that *Part 8* of the User Manual lists all model templates available in ACES and the parameters required to define them. For a quick overview of the whole modelling process refer to the document: *Bridge Modelling Made Easy* (it's in Microsoft *WORD 2000* format).

Default values have been preset by ACES for most parameters in the table. They may all be changed to suit your own unique requirements. If any part of the schematic diagram is obscured by the data entry window, simply click anywhere within the diagram to bring it to the foreground. To continue entering data into the parameter fields click on any visible part of the dialogue box.



At this stage you may wish to experiment with the parametric modeller by selectively changing some of the parameters and observing the resultant effect on the generated geometry. Enter a value into the ***Number of Spans*** field (e.g *2*). Note that nothing happens. A scaled representation of the current model geometry can only be viewed by clicking the *Verify Geometry* button.

Referring to the schematic representation of a uniformly skewed grillage structure, enter other parameters as appropriate. From time to time click the *Verify Geometry* button to see the effect your data values have on the geometric layout. Note in particular the way in which the model is meshed when skew angles are varied and the ***Mesh Type*** option is toggled between *Rectangular lateral member layout* and *Skewed member layout*.

Select the lateral member layout that best suits your own model - ACES does not presume to suggest the best representation of a grillage mesh. That is left for you, as the designer, to do. The system will issue a warning if you attempt to create an unacceptable structural model.

Z-Offset - This parameter is used to convert the basic 2D model to 3D. It does *not* represent the distance between the centroids of the deck slab and longitudinal girders. If a non-zero value (either positive or negative) is entered it will be appended as a *Z* coordinate to all generated 2D nodes, effectively converting the model into a 3D structure. (Refer to *Section 2.4* for a more detailed explanation).

Girder Width - This parameter should only be used if the main longitudinal girders are Super-T sections and you wish to more accurately model the transverse effects caused by the much stiffer top soffit and flanges of that type of section. Check the effect this parameter has on the grillage mesh by entering a value and clicking the *Verify Geometry* button. (Refer also to PART 6, *Grillage Types 1,2,3*)

To enter span lengths click the ***Span lengths..*** button. A list of all spans will be displayed with default lengths preset to 20 metres (or feet, depending on the length unit you are working with). Change the lengths by clicking in the required data fields and entering the appropriate dimensions.

Alternatively, enter a value into the field labelled *Click Apply button to set all Span Lengths to this value* and click *Apply*. Once all span lengths have been defined, click *OK* to return to the menu.

TIP: *If most spans have the same length, set the global span length first then change individual span values to suit.*

The ***Mesh divisions between supports..*** and ***Girder spacing..*** parameters are specified in exactly the same way as span lengths. Mesh divisions determine the density of transverse members and hence the accuracy of the solution. They are set to 10 by default, but this value can be changed to suit your modelling requirements (a value of 10 means that nine transverse beam elements will be generated between support points in that particular span). Parameters that are not required in the structure (such as cantilever overhangs in grillage models) should be set to zero.

2.1.2 Grillage Types 7 and9

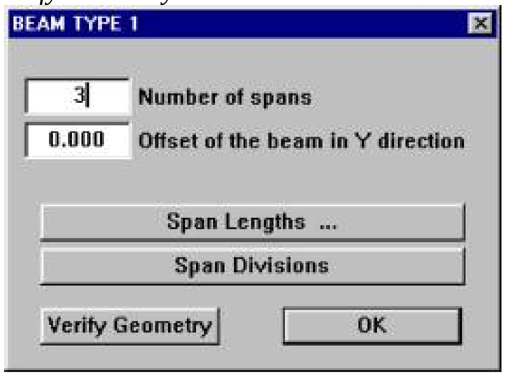
Part 8 of the User Manual lists all available grillage model templates and the parameters required to define them. The main difference between these two grillage types and the others lies in the way in which girder and cantilever beam spacings are applied to the model.

In grillage types 7 and 9 the girder and cantilever spacings are assumed to be measured along a straight line orthogonal to the X axis. These spacings are first summed up then converted into a series of *proportions*. The position of each cantilever and girder along every support line is then calculated as a proportion of the overall width of that line. While support lines are defined by the ***X,Y*** coordinates of their end points, the actual location of girders and cantilevers along the support lines is calculated as a fixed proportion of the total distances between those points. Note that these proportions remain the same for all support lines.

EXAMPLE: If the top and bottom cantilevers have a spacing of 2m each and 4 main girders are at 3 m centres measured along a notional vertical line, their proportional spacing ratios are *0.125 (2/16), 0.25 (4/16), 0.25, 0.25* and *0.125* while the spacing along any abutment or pier line is calculated as: *0.125L, 0.25L* etc., where *L* is the length between their end points.

2.2 Continuous Beams & Frames

The general comments made for *Grillages* also apply to continuous beams and frames. For example, entering a value into the ***Number of Spans*** field (e.g *3*) will have no effect on the currently displayed diagram. A scaled representation of the actual model geometry will only be displayed if the *Verify Geometry* button is clicked.



When modelling frames, click the *Verify Geometry* button from time to time to see the effect your data values have on the geometric layout. The system will issue a warning if you attempt to create an unstable or unacceptable structural model.

The parameter ***Offset of the beam in Y direction*** is used to convert the continuous beam model into a 2D frame structure. It does *not* represent the distance between the centroids of the deck slab and longitudinal girders. If a non-zero value (either positive or negative) is entered, it will be appended as a *Y* coordinate to all generated beam nodes, effectively converting the model into a 2D structure. For a more detailed discussion of this parameter refer to *Section 2.4* in this Manual.

To enter span lengths click the ***Span lengths..*** button. A list of all spans will be displayed with default lengths preset to 20 metres (or feet, depending on the length unit you are working with). Change the lengths by clicking in the required data fields and entering the appropriate dimensions. Alternatively, enter a value into the field labelled *Click Apply button to set all Span Lengths to this value* and click *Apply*. Once all span lengths have been defined, click *OK* to return to the menu.

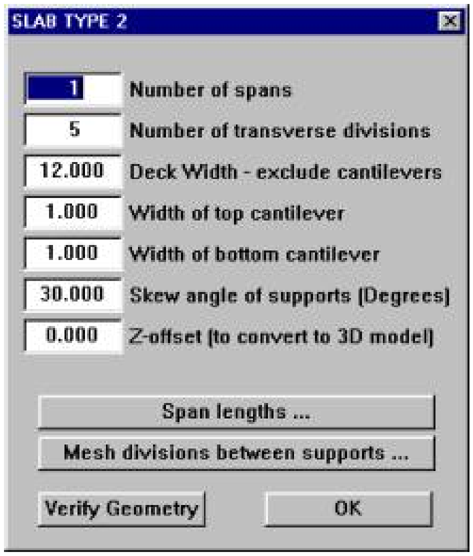
TIP: *If most spans have the same length, set the global span length first then change individual span values to suit.*

Parameters in the ***Span Divisions..*** panel are specified in exactly the same way. ACES performs member and vehicle load distribution to nodes and calculates results at nodal points using a default value of 10 span divisions. If you require greater accuracy or smoother bending moment, shear and deflection diagrams then change this value to suit.

2.3 FE Slabs

The general comments made for *Grillages* apply also to FE slabs. For example, entering a value into the ***Number of Spans*** field (e.g *3*) will have no effect on the displayed model image. A scaled representation of the current model geometry will only be displayed if the *Verify Geometry* button is clicked.

A schematic representation of a bridge deck with uniformly skewed supports will be displayed together with a data entry window labelled *SLAB TYPE 2*. Default values have been preset by ACES for most parameters in the table. They may all be changed to suit your own unique requirements.



If any part of the schematic diagram is obscured by the data entry window, simply click anywhere within the diagram to bring it to the foreground. To continue entering data, click on any visible part of the dialogue box.

At this stage you may wish to experiment with the parametric modeller by selectively changing some of the parameters and observing the resultant effect on the generated geometry. Enter a value into the ***Number of Spans*** field (e.g *2*). Note that nothing happens. A scaled representation of the current model geometry can only be viewed by clicking the *Verify Geometry* button. Try it now.

Referring to the schematic representation of a uniformly skewed slab structure shown in the drawing window, enter other parameters as appropriate. Note that plate bending elements are used to model a finite element slab structure.

From time to time click the *Verify Geometry* button to see the effect your data values have on the geometric layout. Note in particular the way in which the model is meshed when skew angles and mesh divisions are varied. The system will issue a warning if you attempt to create an unacceptable structural model. Click on *Mesh divisions between supports..* and enter the mesh density for each span that suits your own model - ACES does not presume to suggest the best representation of a finite element mesh. That is left for you, as the designer, to do.

The ***Z-Offset*** parameter is used to convert the basic 2D model to 3D. If a non-zero value (either positive or negative) is entered it will be appended as a Z coordinate to all generated 2D nodes, effectively converting the model into a 3D structure. Refer to Section 2.4 for a more detailed description of this parameter and how it is used.

The ***Deck Width*** dimension represents the structure width *excluding* cantilevers (if any). Check the effect this parameter has on the FE mesh by entering a value and clicking the *Verify Geometry* button.

To enter span lengths click the ***Span lengths..*** button. A list of all spans will be displayed with default lengths preset to 20 metres (or feet, depending on the length unit you are working with). Change the lengths by clicking in the required data fields and entering the appropriate dimensions. Alternatively, enter a value into the field labelled *Click Apply button to set all Span Lengths to this value* and click *Apply*. Once all span lengths have been defined, click *OK* to return to the menu.

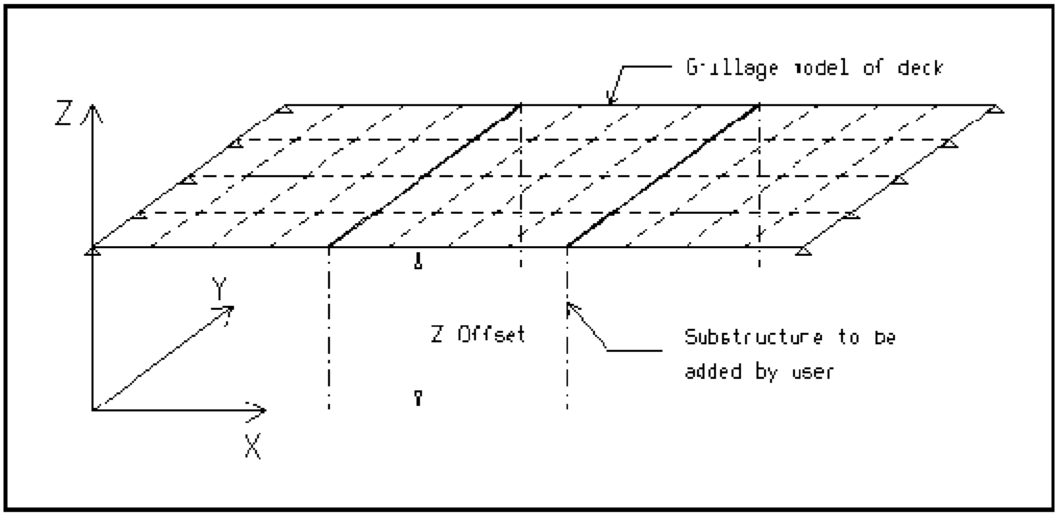
TIP: *If most spans have the same length, set the global span length first then change individual span values to suit.*

The ***Mesh divisions between supports..*** parameters are specified in exactly the same way.

2.4 Z-Offset (Y-Offset)

A number of structure templates employ the ***Z Offset*** parameter (or, for beam models, the ***Y Offset*** parameter). This is an optional dimension that allows the basic 2-Dimensional structure to be quickly and easily converted into a 3-Dimensional model. (For beam types the conversion is from 1-D to 2-D).

The diagram below indicates the concept. Mesh generation of grillages and slabs is normally performed in the *X-Y* plane where the *Z* coordinate is effectively zero. If a *Z-Offset* value is specified it will be added to all generated nodes, in effect creating a 3-D model. (For continuous beams the offset value is a Y-coordinate which is added to every node to create a 2-D model).



Note that any substructure (or superstructure) that forms part of the model must be manually added by the user using the geometry tools provided in the *Structure* menu (refer to PART 2.3 for instruction in adding nodes, members and elements to the model). Vehicles applied to the structure will still move over the deck in the normal way but the analysis will be performed on the complete 3D model.

3.0 Completing the Model

After all parameters have been defined click *OK*. The screen will be cleared and a *Tips* window will be displayed to give you some guidance in assigning member properties and loads to the model. Click *OK* on the *Tips* panel. ACES will now generate and display a scaled representation of the full model geometry.

Note that all members belonging to a logical structural element type will be displayed in their own unique colour c.f. main longitudinal girders will be shown in one colour, cantilever edge beams in another and so on. Each colour represents a pre-defined section and material property type, with default properties set to either zero or 1. To define and assign member properties refer to Part 2.5 (Member Properties).

PART 2.3 Changing the Model Geometry

1.0 INTRODUCTION

After the “base” geometry has been generated (either by using templates or by other means) it can be modified to suit your own requirements. Unwanted supports can be deleted (or new supports added) and new nodes, members and finite elements can be created or existing elements removed from the model. End releases and rigid links can be specified and other options in the *Structure* menu are available to create “layers” of nodes and members and to mirror the active parts of the model about any plane.

2.0 NODES

Nodes can be added, deleted and changed using the options found in the *Structure/Nodes..* menu. Note, however, that if you add nodes that lie out of the original 2D plane generated by the ACES parametric modelling templates you will have to change the model type to 3D (unless the ***Z-Offset*** parameter was specified, in which case ACES will do this automatically). To change the model type, use *Structure / Change coord system* and set the coordinate and analysis attributes to the appropriate type.

To display nodes on the model either click the node icon in the right hand menu or select *Settings/Show symbols* from the main menu and tick the ***Node symbols*** option.

2.1 Merging Nodes into the Model

To merge node coordinates that have been calculated separately and saved to a text file (e.g. in a spreadsheet) use the options: *Structure / Merge / 2D-3D node list* or *File / Merge / 2D-3D node list*. Data in the merged file must conform to the following format:

Line 1: Header information (e.g. Coordinates of substructure)
*Line 2: The following label - “**Number of dimensions**”*
Line 3: The number of dimensions expressed as an integer (e.g. 2)
Line 4: The node number and X,Y coordinates of the first node in the list
Line 5: The node number and X,Y coordinates of the second node in the list
“ “ “ “ “
“ “ “ “ “

Note that line 2 is a label and must be typed in as shown. If a 3-dimensional list of nodes is to be merged, Line 3 should read ‘3’ and each node must have an *X*,*Y* and *Z* coordinate. Either spaces or commas can be used as delimiters between node numbers and coordinates. Any number of nodes may be included in the list - there is no need to specify how many.

3.0 MEMBERS

Members can be added, deleted and changed using the features found in the *Structure/Members..* menu. Note, however, that if you add new members that lie out of the original 2D plane generated by the ACES parametric modelling templates you will have to change the model type to 3D (unless the ***Z-Offset*** parameter was specified, in which case ACES will do this automatically). To change the model type, use *Structure / Change coord system* and set the coordinate and analysis attributes to the appropriate type.

To view the currently assigned member property types click *Structure / Display members*. All members having the same property type will be displayed in the same colour.

Note that there are two ways of selecting a rectangular member range. If you drag out out a rectangle from left to right ACES will include only those members that are wholly within the selected area. When dragging from right to left <u>*all*</u> members, or any part of a member, within the boxed area will be selected.

3.1 Adding Members

Members can only be added between predefined points, or nodes, on the model. To add a member between nodes that already exist select *Structure / Members /Add members / Between existing nodes* then click on the first node with the left mouse button and drag an elastic line out to the second node.

If a node does not exist at the required location along a member you will first need to create it. Use the menu options *Structure / Members / Divide /Add node at specified distance from start of member* to do this. The default distance is the mid-point along the member. To determine the *start node* for the member use the *Settings / Show symbols / Member local axes* options. (You may need to zoom in to see the member local axes clearly).

To add members using a background grid of points, first turn the grid on using *View/Grid on*, change the grid density or reference node (if required) via menu options *Settings/Background grid/Change parameters*, then add members with the commands *Structure/Members/Add members/Snap to a grid.*

EXAMPLES

3.1.1 Add a member at right angles to another member

To add a member at ***right angles*** to another member (i.e. at right angles to an existing member and passing through a target node that also exists elsewhere on the model):

- • Select *Structure/Members/Add members/Normal to an existing member & end node*
- • Click on the required member with the left mouse button then, holding the button down, drag an elastic line out to the target node. Release the button. *(TIP: The dragged line need only be approximately at right angles to the member).*
- • If the dragged line crosses any other members ACES will ask you if you wish those members to be connected to the new member (generally you will). Reply accordingly.
- • The program will then realign the dragged line so that it passes through the selected target point and is at right angles to the source member.
- • If you had replied affirmatively to the prompt for connecting all intersecting members, ACES will create nodes at the intersection points of the new member with all other members it crosses.
- • All intersecting members will be automatically sub-divided into shorter members lying between these points.
- • To view the newly created nodes and members select: *Settings/Show symbols* and toggle on the appropriate drawing attributes (such as node and member numbers, node symbols, member axes etc).

3.1.2 Add a member offset by a relative distance from an existing node

To add a member ***offset by a relative distance from an existing node*** (e.g to create a vertical pier at a support node and at right angles to the deck):

- • Select *Structure/Members/Add members/With end points offset from an existing node*

- • Click on the required node with the left mouse button then enter the offsets from that node. To create other members with the same offsets at other nodes, simply click on those nodes.

3.2 Deleting Members

To delete one or more members from the model click in turn *Structure / Members / Delete members* then select one of the options from the submenu. For example, to delete the edge beams generated by ACES for a model with cantilever footpaths, select *… / Delete a line of members* then click in turn on any member lying along each of the two edge lines. Both lines will disappear.

Note that there are two ways of selecting a rectangular member range. If you drag out out a rectangle from left to right ACES will include only those members that are wholly within the selected area. When dragging from right to left *all* members, or any part of a member, within the boxed area will be selected.

3.3 Renumbering Members

To renumber a group of members use the options: *Structure / Members / Renumber/..* then select the range that you wish to renumber viz:

- • Individual members
- • Whole or partial lines of members
- • Blocks of members
- • Members selected by passing a fence line through them
- • By member property type

ACES will first display the current number of the first member in the nominated range then prompt you to enter a new number. Renumbering is performed from left to right and top to bottom.

If you retain the default analysis setting of ***No renumbering*** prior to the solution being performed, ACES will not check for duplicate member numbers - refer to PART 4.1, *Analysis Options*, (*Renumbering the Model)*, for further information.

3.4 Merging Members into the Model

To merge a list of members (and their associated nodal links) that have been separately compiled and saved to a text file (e.g. in a spreadsheet) use the options *File / Merge / Members*. Data in the file must conform to the following format:

Line 1: Header information (e.g. Substructure for Job)
Line 2: Member number, start node number, end node number, member property type
Line 3: Member number, start node number, end node number, member property type
“ “ “ “ “
“ “ “ “ “

Either spaces or commas can be used as delimiters between the data values in each line. Any number of members may be included in the list - there is no need to specify the exact number.

4.0 ELEMENTS

4.1 Modifying Elements

Elements can be added, deleted and changed using the options found in the *Structure/Elements..* menu. Note, however, that if you add elements that lie out of the original 2D plane generated by the ACES parametric modelling method you will have to change the model type to 3D (unless the ***Z-Offset*** parameter was specified, in which case ACES will do this automatically).

To change the model type, use *Structure / Change coord system* and set the coordinate and analysis attributes to the appropriate type.

To view the currently assigned element property types click *Structure / Display elements*. All elements having the same property type will be displayed in the same colour.

To release plate element forces refer to [Section 6.1](#) for details.

4.2 Adding Elements

To add elements to the model select *Structure / Finite Elements /Add Elements / ..* then click on the option you want.

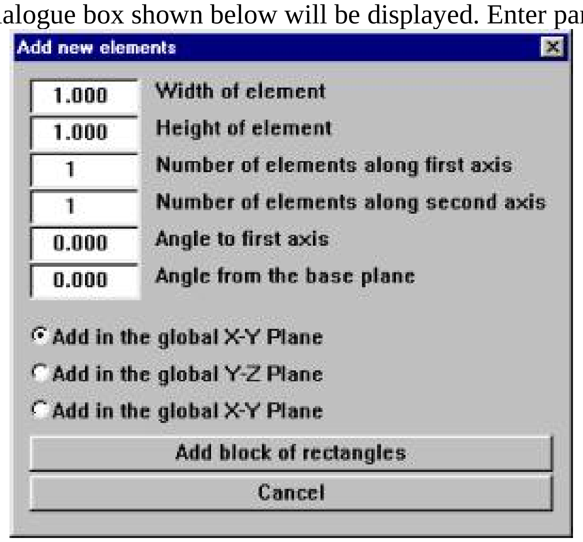
4.2.1 Adding Elements Using a Background Grid

To add elements using a background grid of points, first turn the grid on using *View/Grid on*, change the grid density or reference node (if required) via menu options *Settings/Background grid/Change parameters*, then add elements with the commands *Structure/Finite Elements/Add elements/Single triangular element by snapping to grid point*. Ditto for rectangular elements.

4.2.2 Adding a Block of Elements

To add a block of elements to, say, the top edge of the model:

- • Select *Structure/Finite Elements/Add Elements/Block of rectangular elements in any plane*
- • Select a starting node for the block (click on it with the crosshairs)
- • The dialogue box shown below will be displayed. Enter parameters as required.



- • Note that if you want the new elements to line up exactly with those already existing along the horizontal or vertical line you must enter the exact value of element width or height.
- • The direction and side of the line in which element generation will occur will depend on the sign of the element *width* and *height*. If both are positive, elements will be generated in the (+)ve *X,Y* directions. If negative, they will run in the -*X* and -*Y* directions.
- • Click the button labelled: *Add block of rectangles*

4.3 Deleting Elements

To delete one or more elements from the model click in turn *Structure / Finite Elements / ..* then select one of the deletion options from the submenu. Prompt messages on the bottom line will give you instruction on how to do this.

4.4 Aligning Element Local Axes

When new elements are added to the model their local *x-y* axes may not be aligned in the same direction as existing elements. This may cause problems later when displaying and interpreting results. It is important, therefore, that the direction of element axes are aligned prior to performing the analysis.

To check the alignment select: *Settings/Show symbols* and toggle on the display of element axes. Now use one of the alignment options : *Structure / Finite Elements /Rotate..* or *Reverse..* or *Align..* to align element axes in the direction you require.

4.5 Renumbering Elements

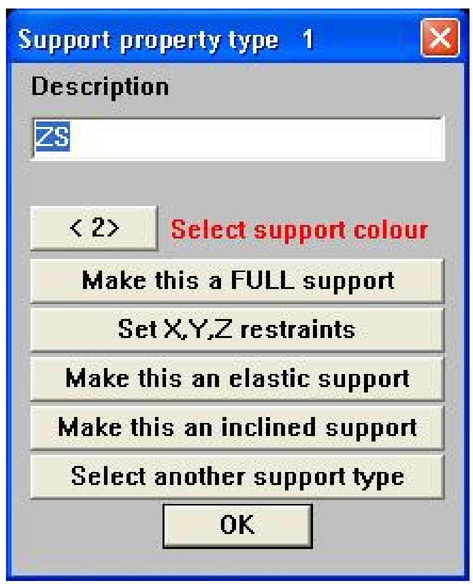
To renumber all elements in the model use the options: *Structure / Finite Elements / Renumber* then enter the element number from which renumbering is to begin. Renumbering is performed from left to right and top to bottom.

If you retain the default analysis setting of *No renumbering* prior to the solution being performed, ACES will not check for duplicate element numbers - refer to *PART 4.1*, *Analysis Options*, *Renumbering the Model*, for further information.

5.0 SUPPORTS

5.1 Creating & Editing Support Types

To create a new support type select *Structure / Supports / Create support Type.* The following dialog box will appear:



To change the ***colour*** of the support click the button labelled < *2* >. The current support colour can be identified by the number on the first button as well as the colour of the text label associated with that button. To make this support type fully restrained simply click the button labelled *Make this a FULL support*. To change

To make this support type ***fully restrained*** simply click the button labelled *Make this a FULL support*.

To change or individually set ***support conditions*** click the button labelled *Set X,Y,Z restraints*. For a 2D grillage generated using one of the standard templates and subject only to loadings normal to the *X-Y* plane, ACES will automatically create a support type that is only restrained in the vertical Z direction. If you apply non-orthogonal loads (such as in-plane braking forces) you will need to change the support conditions appropriately (and also the structure type - from a 2D grillage to a 3D space frame).

To create an ***elastic support*** click the button labelled *Make this an elastic support*. Refer to *Section 5.3* for details.

To create an ***inclined support*** click the button labelled *Make this an inclined support* and enter the parameters in accordance with your requirements.

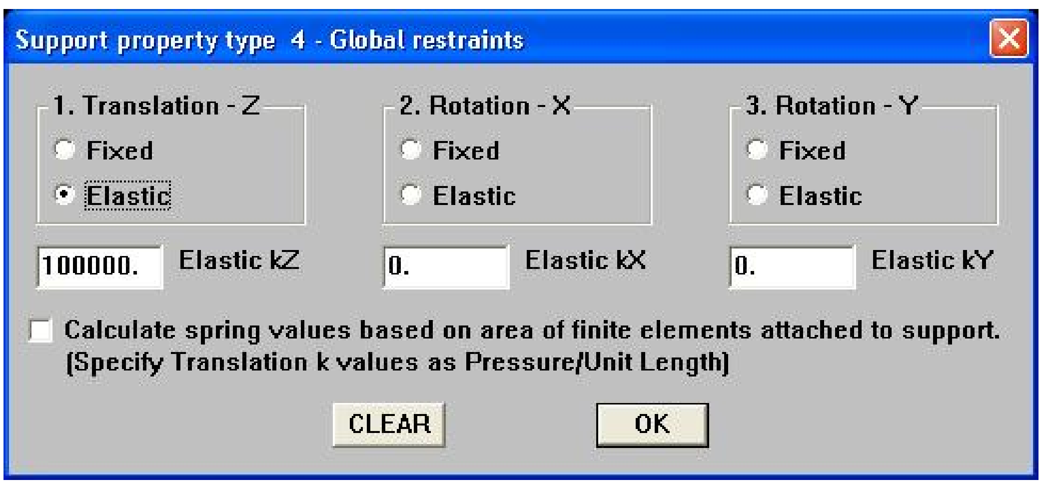
5.2 Adding and Deleting Supports

To add one or more supports to the model (or delete one or more supports) select *Structure / Supports / Add supports* (or *Delete supports*) then indicate the range of supports that are to be so affected. For example, to delete all supports generated under cantilever edge beams select *… / Delete supports / From a line of nodes* then successively click on any two nodes lying along the nominated edge beam. All supports along that line will be deleted. *(NOTE: This option only works for a straight line of nodes)*.

To add a support at any point along a member you will first need to create a node at the required location on that member. To do this, use the menu options *Structure / Nodes / Add node at a specified distance from start of member* to first create a node along the member then apply a support at that node. The default distance is the mid-point along the member.

5.3 Elastic (Spring) Supports

To create an elastic support click *Structure / Supports / Create support type / Make this an elastic support*. To edit an exisiting elastic support click *Structure / Supports / Edit support propsand* select the support type you would like to change. In either case, the following dialog box will be displayed:



Change any values and default elastic support conditions to suit your requirements. The tick box labelled "*Calculate spring values...*" is a flag that indicates if you would like ACES to calculate the equivalent nodal eleastic forces for you. If you tick the box, these values will be calculated using the subgrade reaction that you enter into the ***kZ*** field and the element areas connected to each support node that has this support type assigned to it.

An example of when this feature may be useful is an FE model of a raft resting on an elastic foundation. In this case it will probably be convenient to get ACES to determine the elastic spring constant based on the subgrade reaction of the soil and the area supported by the node.

Note, however, that this flag can only be used in plate-bending and shell type FE models where all elements joined at the support nodes contribute to the spring support.

If, for example, the nodes along the junction between a slab and a wall are connected by vertical elements (in the wall) and horizontal elements (in the slab) then this flag cannot be used for supports at these nodes. In such cases the spring constant must be calculated manually and entered into a new elastic support type.

Note on Bearings

If spring constants have been specified they can be derived from the physical properties of the bearing. Note that even pot bearings have a certain amount of elasticity and it is often beneficial to account for this in your model.

For example, the current Australian Bridge Design Code (see, for example, *Section 4, Commentary*), gives details of the physical properties of a range of elastomeric bearing sizes. For bearing *040902R* (page *C4.10*), the "*Calc. Comp Stiffness at zero shear*" is 657E3 kN/m. If you want to model this bearing in a 2D grillage model (*PLANE GRID*) and your units are kilonewtons and metres, then you would need to specify an elastic ***Kz*** value of *657E+03*. For pot-type bearings you can usually calculate an approximate stiffness using the serviceability load rating, the rotational capacity and the dimensions of the bearing (the rotation and dimensions are used to calculate an approximate deflection).

6.0 END RELEASES

To specify end releases use the options: *Structure / End releases / Specify release conditions* to define the type and location of the end release(s) followed by *Structure / End releases / Apply a member release /..* to apply those conditions to selected members. (You will be prompted to click on the members to which the end release conditions are to apply).

6.1 Releases for Plate Elements

ACES does not have plate release facilities but there are several ways that hinges in finite element models can be modelled.

First, an extra line of nodes will need to be created with either a horizontal or vertical offset (say 50mm) from the hinge nodes. Using a vetical offset will allow you to model the hinge more accurately, but the drawback is that it will introduce extra plate bending if in-plane forces are significant. (Note that a vertical offset would not be allowed in 2D problems).

Once the extra lines of nodes is created, the existing elements to the right (or left) of the hinge will need to be deleted and replaced with ones which connect to the new offset nodes. The offset nodes then need to be connected to the original hinge nodes. There are three ways of doing this:

(1) Using rigid links with the approriate direction/rotation links

This is probably the most mathematically correct way of modelling the hinge but I have found that they use a lot of dynamic array space and reduce the number of loadings that can be solved. Also, result values across the hinge nodes will not be available.

(2) Using a line of thin finite elements

Being thin the elements will transmit only a small moment, but the modulus of elasticity will need to be "tweaked" to avoid shear displacements (if the offset is horizontal) or axial displacements (if the offset is vertical). This is probably the quickest but least accurate of the hinge models. However, it is possible to derive results values close to the hinge and this may be important.

(3) Using a line of appropriately sized members with a line of member releases at the hinge

Member properties can be readily calculated and result values will be available at the hinge, but this is the most time-consuming way to model the joint. If this option is chosen, the analysis type for the frame analysis option must be set to the appropriate one.

The final choice of which option to use depends mainly on your own preference.

7.0 MODIFYING THE PARAMETRIC MODEL

7.1 Generating a Non-Standard Mesh

The parametric templates only allow “standard” structure types to be generated i.e., those confined to the limitations imposed by the parameters themselves. However, certain “tricks” can be employed to create a parametric model that can then be massaged to produce the desired final structural model.

For example, if a finer grillage or Finite Element mesh is required in the vicinity of supports, “dummy” parametric spans can be initially specified that reflect the different mesh densities within the “real”spans. The unwanted supports can then be removed or reassigned once the model geometry has been generated leaving, in effect, a final mesh configuration that displays variable mesh divisions within each span.

Similarly, additional lines of “dummy” members can be added to a slab and grillage structure in the parametric model in order to allow for slab haunches between girders. These dummy members can subsequently be removed using the member deletion tools.

7.2 Changing Parameters of an Existing CAD Model

During the model geometry generation phase, ACES will preset a number of model attributes. Support, member and element property types will be assigned to the model in accordance with predefined defaults that have been determined by the developers using "the most likely case" scenario.

For example, in the case of grillages, vertical supports are placed under each longitudinal girder at the support lines, member property Type 1 is assigned to all longitudinal girders, Type 2 to all transverse girders, Type 3 to cantilever edge beams and Type 4 to support diaphragms. Key member properties (areas, inertias etc) are also preset to 1.

After the final model has been completed, or "detailed", (by defining and assigning member properties, applying static and moving vehicle loads etc), it can be modified again by changing the original parameters. For example, the skew angle of one of the supports may need to be changed or an additional girder added to the structure (or removed).

Although changes to any of the parameters can be made, the resultant model must be carefully checked, since the parametric modelling process will regenerate all nodes, members, elements and supports using the new set of parameters. Element and member property types, vehicle load groups, load cases and envelopes which were originally created and defined will remain unchanged.

However, any nodes, members, elements, releases, rigid links or supports that were added to, or deleted from, the original model will be lost. They will have to be recreated. Similarly, the assignment of additional property types (over and above those automatically created by ACES), will be lost. They too will have to be re-assigned (but not recreated - all property type groups are retained by the system).

If you envisage having to change any of the parameters it is advisable, therefore, not to make significant changes to the final model geometry.

PART 2.4 Defining Vehicle Traffic Lanes

1.0 General

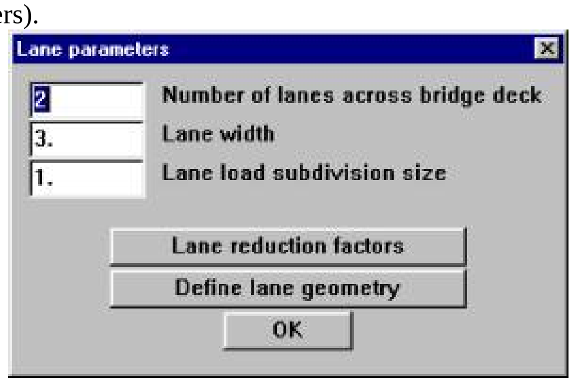
If you wish to use any of the automated Austroads vehicle load cases, or if you intend applying patch and vehicle loads to lanes manually, you must first define the location of lanes you intend using.

Note that, once lanes have been defined, it is not strictly necessary to use them in load cases. ACES will simply ignore them if they are not referred to.

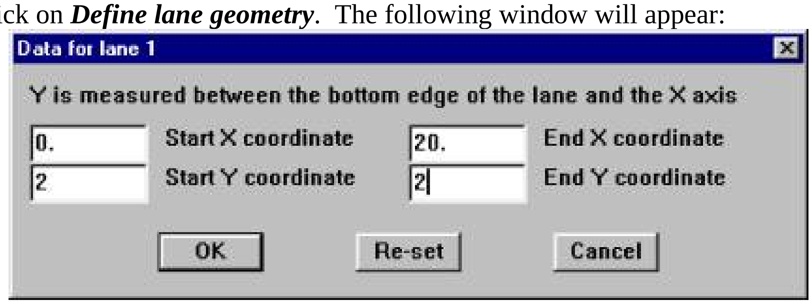
WARNING: Lanes cannot be defined for frame or continuous beam structures

2.0 Defining Lanes

To define one or more traffic lanes, select ***Loads / Define lanes*** then click on the appropriate buttons to enter the lane parameters. (Note that these same options can be used to edit or modify lane parameters).



Click on ***Define lane geometry***. The following window will appear:



Lanes are defined by start and end *X,Y* coordinates, where Y is measured between the bottom edge of the lane and the X axis. The *Re-set* button in the lane data panel simply sets the start and end X coordinates of the lane to the values calculated at the left and right abutments.

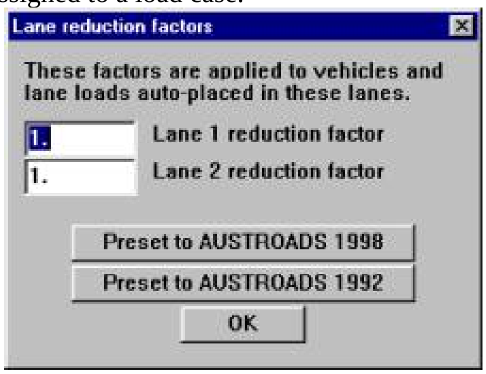
Lanes can begin and end off the structure. This situation may arise in cases where you require auto-generated Austroads vehicle loads to traverse a path that requires them to start well before, and end well clear of, the deck.

The parameter ***Lane load subdivision size*** is specified in the same units as the X,Y coordinates and represents the patch size into which any applied lane load will be subdivided. Lane load is represented by ACES as a uniformly distributed load which is first subdivided into small patches when being distributed to adjacent nodes in the model. This parameter should therefore be less than the horizontal transverse grillage member spacing and also less than the smallest girder spacing in the model.

Default lane colours can be preset using ***Settings / Lane colours*** and current lane locations can be viewed by selecting ***View / View lanes***.

3.0 Lane Load Reduction Factors

To specify lane load reduction factors click onto ***Lane reduction factors*** and select the option you require (or enter the factors manually to suit your requirements. Note that AUSTROADS lane reduction factors will be applied to Austroads vehicles and Austroads lane loads that have been "automatically" assigned to a load case.



They will also be applied to vehicles that are manually assigned to a lane within a load case (see PART 3, *Moving Vehicle Loads*, for details).

PART 2.5 Member & Material Properties

1.0 INTRODUCTION

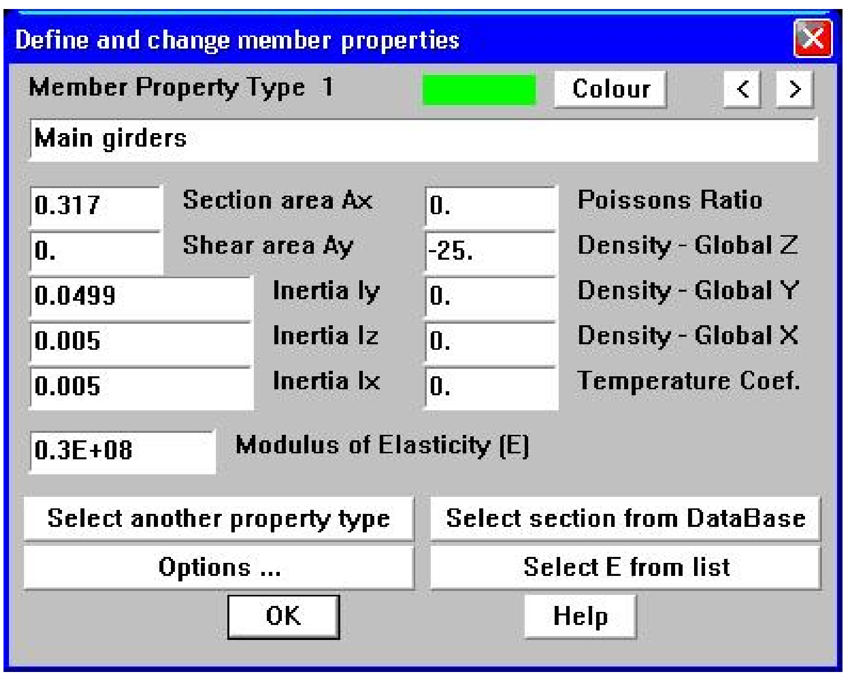
This section describes the manner in which section and material properties are defined and assigned to members. When generating a grillage model ACES automatically assigns unique colours (representing member property types) to each structural element of the model (main girders, transverse beams, cantilever edge beams and support diaphragms - refer to Section 2.2, Parametric Modelling Using Templates for further details).

If the result is exactly what you require, then all you need do is enter the appropriate properties for each of the member property types. Otherwise, you will need to create new property types and assign them to the relevant members in the model.

2.0 DEFINING MEMBER PROPERTIES

Either click the member property icon on the top menu bar (shown as an **I**-beam) or select the following menu options: *Structure / Member properties / Create-Edit.* A maximum of 500 member property types can be defined.

A dialogue box listing all currently defined member property types will appear. Highlight the required member property type (e.g. *Main girders*) then click *Edit*. A member properties panel will be displayed as shown below, with default values of *1.0* assigned to all major property values (with the exception of *E*, density and shear area, *Ay*, all of which are set to zero). Change these to suit your own requirements. To navigate between property types Click the left and right arrow buttons. The colour and line style assigned to the current member property type is shown in a colour bar adjacent to the *Colour* button (click it to change these attributes).



Uniform Properties

Enter values for *Ax*, *Ay* (shear area), *Iy*, *Iz*, *Ix* (torsion) and Poisons' ratio as appropriate. For properties of tapered sections refer to Section 3.4 (***Properties of Tapered Sections***).

Shear Area (Ay)

ACES uses the value in the ***Ay*** field to define the member shear area in either the ***y*** or ***z*** directions, depending on the structure type being modelled. For example, in a 2D frame structure, ***Ay*** will represent the shear area in the vertical (***y***) direction. In a 2D grillage, where both global and member ***Z*** axes are oriented normal to the grillage plane, ***Ay*** will represent the area in the vertical ***Z*** direction. Consequently, if you require shear deflections in a 2D grillage, you will need to enter a non-zero value into the ***Ay*** field.

Inertias (Ix, Iy, Iz)

Ix represents the torsional stiffness and ***Iy, Iz*** the stiffness about the corresponding axis. For example, in a 2D frame model where the global *X,Y* axes lie in the plane of the structure, the local member z axis will be oriented normal to this plane. ***Iz*** will therefore need to be defined since it represents the moment of inertia about the member ***z-z*** axis. In a 2D grillage, where both global and member ***Z*** axes are oriented normal to the grillage plane, ***Iy*** will need to be defined since the principal local member axis is in the ***y-y*** direction. For further discussion of the axis system used in ACES please refer to *Section 1.5*

Densities

Enter density as *Force/Unit^3* (e.g. *kN/m^3*) and note that for models where the deck lies in the global X-Y plane they must be given in the correct sense. For grillage models generated parametrically, density must correspond to ***Density - Global Z*** and must be negative, since the global *Z* axis is *out* of the plane of the screen e.g. *Density - Global Z = -77 (kN/m3)*. For frames and continuous beams enter a value for *Density - Global Y*.

Temperature Coefficient

This parameter represents expansion /contraction effects and *not* vertical temperature gradients.

Select section from the DataBase

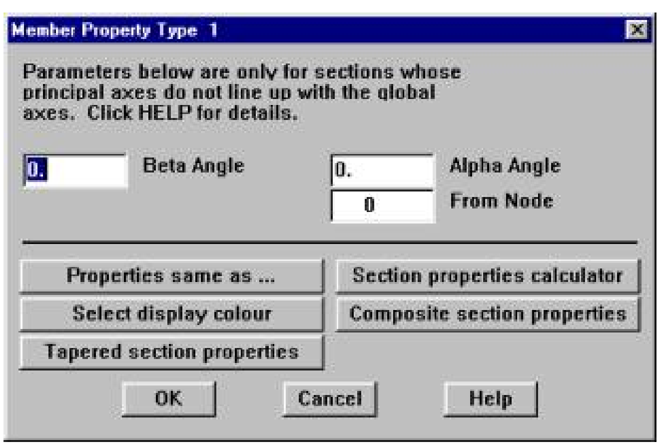
This option allows a section to be selected from a data base of standard BHP and Austroads sections. You may add other sections to the data base by editing the text file *SECPROPS.DAT* in the *Program Files\Aces* folder using any text editor (e.g *NOTEPAD* or similar). Refer to *Section 2.6, Adding New Sections to the Data Base*.

Select E from list

This button allows the Modulus of Elasticity (*E*) to be selected from a list of standard values for concrete, steel & timber.

3.0 MEMBER PROPERTIES OPTIONS

The ***Options..*** button provides access to other section properties features as shown in the panel below:



3.1 Section Properties Calculator

The *Section Properties Calculator* allows properties to be calculated for a range of rectangular, circular, octagonal and general shapes. Rectangular and circular sections can be hollow (achieved by entering a wall thickness) while octagonal sections are solid only (this is useful for determining section properties of solid octagonal piles). The general section properties calculator also allows properties of arbitrary shapes to be determined.

Arbitrary shapes are created by specifying the number of internal and external nodes in the section, a symmetry option (if any) and the system to be used for defining internal and external node coordinates (absolute values or offsets from previous nodes). Internal nodes need only be specified if the section is hollow. Coordinates are then entered separately for each of the external and internal nodes.

An option has been provided to calculate the torsional inertia in one of two ways: either using the bubble analogy (membrane) theory or a simple formulation based on rectangular plate theory. Once calculated, the geometry and properties for the section can be saved to a file for future use. Refer to *Section 2.7, Section Properties Calculator*, for details.

3.2 Composite Section Calculator

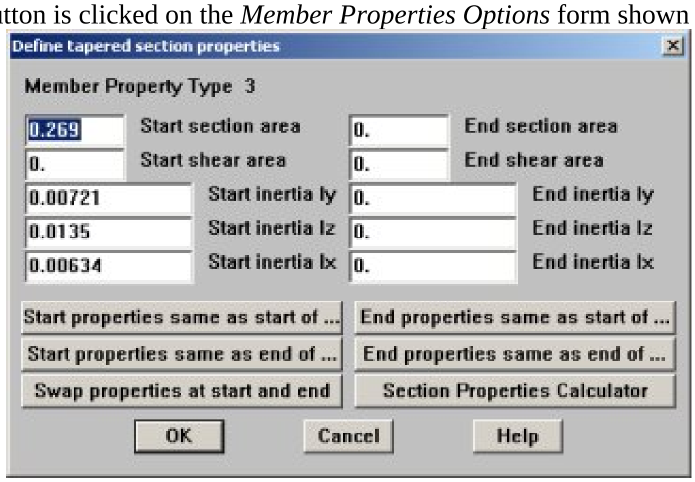
The *Composite Section Calculator* allows slab properties to be specified for composite girders. ACES will calculate the composite properties and display the resultant values. Note that this feature is entirely optional - you may choose to either calculate the composite properties yourself, (entering them directly as *Ax*, *Ix*, *Iy*, *Iz* values), or you can allow ACES to do this for you. In this case the values entered into the main properties panel will be assumed to represent the girder properties only, while those supplied in the *Composite Section* panel represent the slab properties only. Refer to *Section 2.8, Composite Section Calculator*, for details.

3.3 Properties of 3D Sections

For 3D frames and grillage models having members whose principal axes do not line up with the global axes (or 2D grillages with sections lying out of the Y-Z plane) section properties will have to be specified using *Alpha* and *Beta* angles. Refer to *PART 1.5, Coordinate System*, of the User Guide for a definition of these angles.

3.4 Properties of Tapered Sections

This option allows section properties for the current property type to be specified at the beginning and end of the member. If start and end properties are different, ACES will perform a proper tapered section analysis. The following dialog box will be displayed when the *Tapered Section Properties* button is clicked on the *Member Properties Options* form shown in *Section 3.0* above:



When the dialog box is displayed, the start properties are imported from the main *Member Properties* dialog box (refer to *Section 2* above). Note that start and end properties can be made the same as those of any other existing member propery type by clicking the appropriate buttons. Alternatively, the end property can be calculated using the *Section Properties Calculator* button.

HINT: If you really require the start properties to be, in fact, the end properties on entry into this dialog box, first calculate the end properties then click the button labelled ***Swap properties at start and end*** to change them!

3.5 Properties Same as Another Section

This option allows section properties for the current property type to be set to those of another member property type.

3.6 Displaying Member Property Attributes

3.6.1 MemberNumbers:

Either click the *Member Number* icon on the right menu bar or, from the top menubar, click onto *Settings / Show symbols.* In the dialog box check the box labelled *Member number*s. Note that the member number icon is a toggle that switches the numbers on and off with repeated presses.

3.6.2 MemberColours :

Click onto *Structure / Display Members* to view the model drawn in its constituent member colours. Alternatively, click the member properties icon on the top menu bar and scroll through all member property types. The corresponding colour is show in a colour bar at the top of the dialog box.

3.6.3 Member LocalAxes :

To view the member local axes click onto *Settings / Show symbols* and, in the dialog box, check the box labelled *Member local axes*. ACES displays the axes with a small axis symbol that has arrow heads on the ends of the stems that represent the local ***x*** and ***y*** directions. When the model is viewed in 3D the arrow head is left off the axis in the local ***z*** direction. Using the right hand rule it then becomes possible to quickly determine the orientation of the axes (the middle finger of the right hand represents the local ***z*** axis).

3.6.4 MemberOrientation :

To determine the orientation of member localaxes (particularly when working on 3D space frame models), eitherclick the *Member Orientation* icon on the right menu bar or, from the top menubar, click onto *Settings / Show symbols.* In the dialog box check the box labelled *Member orientation*. ACES represents all members using an "**I**" girder shape where that the largest moment of inertia is about the local axis that runs parallel to the top and bottom "flanges" of the section. Note that the member orientation icon is a toggle that switches the numbers on and off with repeated presses..

3.6.5 Member PropertyTypes :

Click onto *Structure / Display Members* to view the model drawn in its constituent member property type colours. To display the property type *numbers* on the model diagram, click onto *Settings / Show symbols from the main menu bar* and, in the dialog box, check the box labelled *Member type number*. (Note that a maximum of 500 member property types can be defined).

3.6.6 Members of a select property typeonly :

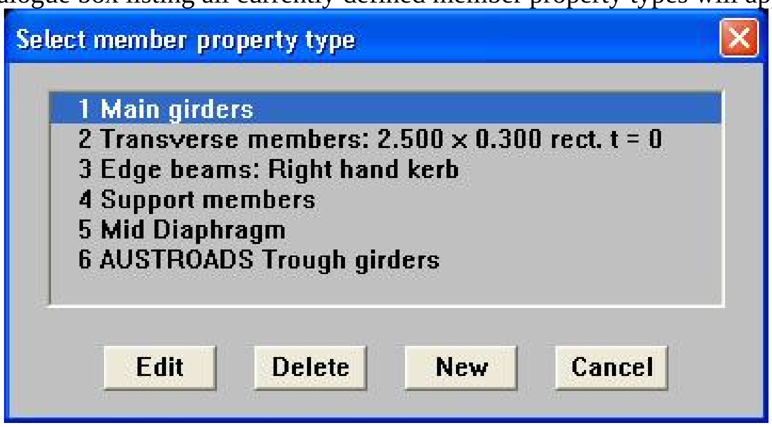
Although each member is displayed in a unique property type colour, for very dense meshes you may wish to quickly check whether member property types have, in fact, been assigned correctly. To do this click onto *Structure / Member properties / Display* then, for each of the *Member*

Types shown in the dialogue box, click on each in turn. Only those members will be separately highlighted.

3.7 Creating & Deleting Member Property Types

To create a new member property type either click the member property icon (designated by the **I**-beam shape) or select the following menu options: *Structure / Member properties / Create-Edit.* Note that a maximum of 500 member property types can be defined.

A dialogue box listing all currently defined member property types will appear as shown below:



Click the *New* button. The main member properties panel will be displayed (go to *Section 2.0* for a display of this dialog box and for guidance as to how section properties can be entered into it). Clicking the *Delete* button will remove the highlighted member property type from the list.

4.0 ASSIGNING MEMBER PROPERTY TYPES

Properties are assigned to members in one of two ways:

- • By selecting a property type first then assigning it to a specified range of members
- • By selecting a range of members first then assigning a specified property type to all members in that range

4.1 Select a Property Type & Assign it to a Range

Select a property type by clicking on the section icon and choosing one of the already defined property types. It will be echoed on the status line just below the drawing area. Scroll in turn through the menu options: *Structure / Member properties / Assign current type to..* and select the member range you require from the submenu of range options.

Now follow any instructions that may appear on the lower prompt line (they will help you to identify the range). Once the range has been delineated, all members in the range will be assigned the currently active property type. They will be drawn in the current member type colour.

Note that there are two ways of selecting a rectangular member range. If you drag out out a rectangle from left to right ACES will include only those members that are wholly within the selected area. When dragging from right to left *all* members, or any part of a member, within the boxed area will be selected.

4.2 Select a Range of Members & Assign a Property Type to Them

Select a range of members in the model by cascading through the menu options: *Structure / Member properties / Assign selected type to..* and select the member range you require from the submenu of range options. Follow any instructions that may appear on the lower prompt line (they will help you to identify the range). Once the range has been delineated a list of current member property types will be displayed. Select the property type you wish to assign to the members in the range then click *OK*.

Note that there are two ways of selecting a rectangular member range. If you drag out out a rectangle from left to right ACES will include only those members that are wholly within the selected area. When dragging from right to left <u>all</u> members, or any part of a member, within the boxed area will be selected.

4.3 EXAMPLES

- • To assign a property type to a ***straight line*** of members the line is selected by clicking on any member lying along the line.
- • To assign a property type to only one ***part of a straight line*** of members click on the first member in the required range and drag out an elastic band to the last member in the range.
- • To assign a property type to only one ***single member*** select the range as *..individual, select members*.
- • To assign a property type to a ***rectangular block*** of members place the crosshairs at some point on the screen, click the *LEFT* button once, drag out a box of a size that encompasses the required members then release the button. All members whose centroids fall within the delineated area will be changed to the current (or selected) member property type.
- • To assign a property type to all members whose centroids lie within an arbitrary ***trapezoidal area***, the area must be defined using two unconnected but opposing straight lines. Each line must be drawn in an anti-clockwise sense, but the lines themselves need not be parallel - they can be oriented in any general direction. However, the area delineated by their notional "boundaries" must incorporate that part of the model within which the target members lie. All members whose centroids fall within the delineated "area" will be changed to the current (or selected) member property type.

  Lines are created by clicking at a point on the screen that is to represent one corner of the notional area then, holding down the left mouse button, dragging out a fence line in an anti-clockwise direction to an end point representing the adjacent corner.

4.4 Merging Material Properties

To merge material properties that have been defined and saved in a previous model, use the options: *Structure / Merge / Member properties* or *File / Merge / Member properties* (ditto for element properties). This feature can sometimes be useful in situations where an iterative process is being used to refine a structural model (in preliminary design, for example). Once generated in the first iteration, material property types can be saved then reloaded into the next iteration of the model.

File / Merge / Member Properties will allow you to read a member properties type file formatted in one of two ways: either with the properties of each member type listed on the same line separated by commas; or with a single property per line. The option will appear *<u>after</u>* you have selected the data file containing the property types to be merged into the model. Note that gaps can be left in the member type numbering and that current properties for existing types are not zeroed.

If you <u>do</u> leave gaps in the numbering and one of the new types is greater than the number of currently defined types, then all of the types between the current maximum and the new type number will be zeroed.

When reading/merging members from a text file into the model a warning will be given if a member property type assigned to an imported member does not exist (i.e. if it is undefined).

When enquiring on a member whose type has not been defined, ACES will warn you that a property type has not been assigned.

PART 2.6 Adding New Sections to the Data Base

1.1 How to Add a New Section to the Data Base

Section properties are stored in a text file called *Secprops.dat* located in the *\Program Files\Aces* folder. The contents of this folder may be viewed using *Windows Explorer*. Locate *Secprops.dat* then double click on it to load it into *Notepad* or, alternatively, use any other text editor to perform the editing.

Scroll through the file to get an appreciation of its structure and layout. You will notice a number of instructions at the top of the file. Please read these carefully and follow all directions exactly as instructed. Note that the file contains a number of unique section CATEGORIES, each category consisting of a specified number of individual sections. You can add one or more new categories at any point in the file providing you adhere to the correct layout and formatting viz:

- • Each category must have a label or name consisting of ***no more than*** 30 alpha-numeric characters separated from the previous data by at least one blank line (e.g. *WELDED BEAMS*).
- • The total number of sections in the category must appear on the title line & separated from the title by a comma (e.g. *WELDED BEAMS*, *23*). Section properties are entered beneath the title, the first record only separated from the title by at least one blank line. Do *not* use blanks between each individual section.
- • Properties must be entered in the ***exact*** format and positions shown in the data below and, for each section, in the following order:

Area	(current units are *mm2*)
Ix	(current units are *mm2*)
Iy	(current units are *mm2*)
J	(current units are *mm2*)

1.2 Example

To insert a new category into the file immediately following *WELDED COLUMNS* leave a blank line after the last record in that category then insert your own data as shown below.

```
          ………
          ………
SUPER-T BEAMS, 3

  ST-1600-115   14600.0000  1150.0000   41.7000   0.8880
  ST-1600-130   16600.0000  1400.0000   52.1000   1.5100
  ST-1800-150   19100.0000  1710.0000   65.2000   2.6900

  UNIVERSAL BEAMS, 28
          ………
          ………
```

SECTION 2.7 - Section Properties Calculator

2.7.1 Introduction

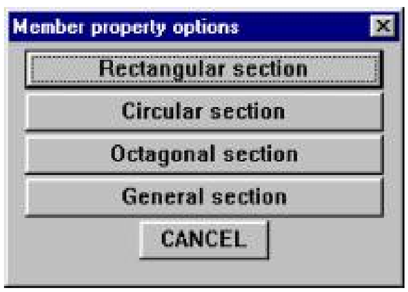
The *Section Properties Calculator* allows properties to be calculated for a range of rectangular, circular, octagonal and general shapes. Rectangular and circular sections can be hollow (achieved by entering a wall thickness) while octagonal sections are solid only (this is useful for determining section properties of solid octagonal piles).

The general section properties calculator allows properties of arbitrary shapes to be determined. It can be accessed either from the opening dailog box when the program first starts or from its own icon on the tool bar. When accessed in this way the module will act in stand-alone mode. If accessed from the member properties dialog box (*Structure/Member Properties/Other Options*) the calculated properties will be automatically transfered into the relevant properties fields.

Please read the comments in Section 2.7.6 if you elect to change any of the torsional parameters.

2.7.2 Standard Shapes

Selecting the *Section Properties Calculator* from the *Member Properties* dialog box will bring up the following panel.

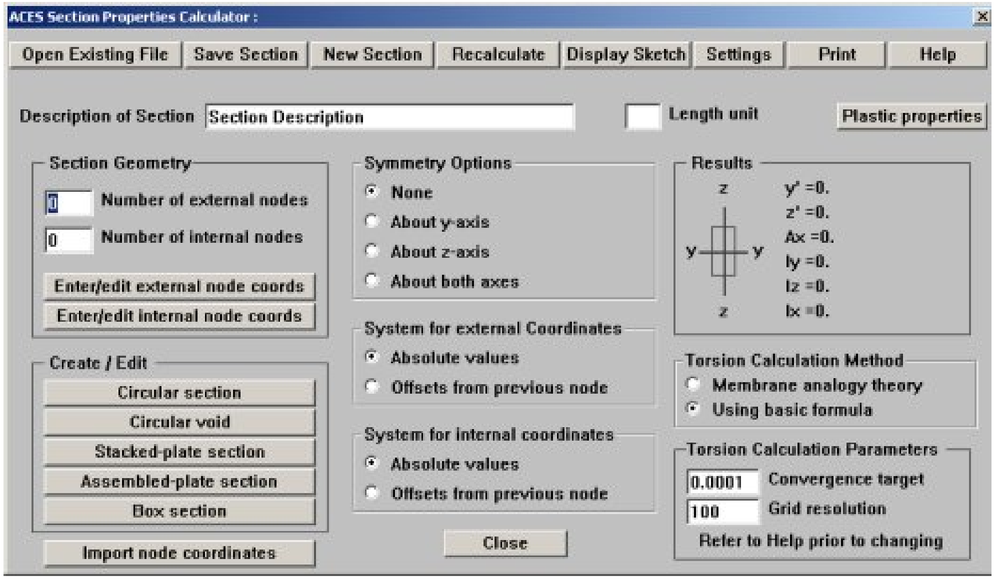


The first three options allow simple solid and thin-walled shapes to be quickly generated. (Voided sections can be generated for rectangular and circular sections by entering a membrane thickness). Only solid sections are available for the *Octagonal* shape.

2.7.3 General Shapes

The general section properties calculator allows properties of arbitrary shapes to be determined. It can be accessed either from the opening dailog box when the program first starts or from its own icon on the tool bar. When accessed in this way the module will act in stand-alone mode.

If accessed from the member properties dialog box (*Structure/Member Properties/Other Options/General Section*) the calculated properties will be automatically transfered into the relevant properties fields in the member properties dialog box. Which-ever method is selected, the following dialog box will appear:



Description of Section

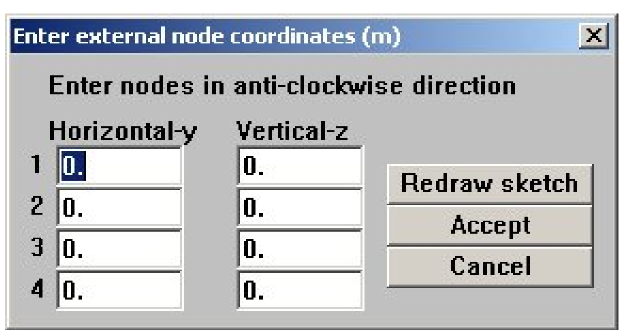
Enter a description as required.

Length Unit

Enter a length unit as required. When accessed through the member properties dialog box the relevant unit will automatically appear in this field. Note that in stand-alone mode this field is a label only - calculations will be performed in accordance with standard shape theory using the values strictly as entered by the user (i.e., the system does not actually "know" that the label "m" designates "metres").

Section Geometry

To generate an arbitrary section you must first enter the number of external nodes defining the shape (*Number of external nodes*), specify a symmetry option (*Symmetry Options*) and indicate the system you wish to use for specifying the node coordinates (*System for external Coordinates*). Finally, click the *Enter/edit external node coords* button and enter the coordinates into the dialog box that will be displayed (shown below). Node coordinates should be defined entered in the anti-clockwise direction.



Note that the shape will only be drawn once you click the *Redraw sketch* button - it is not dynamically displayed as coordinates are being entered. Click *Accept* to lock-in the coordinates and return to the main dialog box.

Hint: There is no need to specify nodes on the axes if they do not contribute to the shape of the section. The exception to this is if there is symmetry about any or both axes, in which case a minimum of 3 nodes is required.

Voided Sections

If a void is inserted into the model (either circular or non-circular), both internal and external node coordinates must be defined in the anti-clockwise direction. In some programs a void is modelled by defining its node coordinates in the opposite sense to that of the external nodes. However, ACES has adopted a different convention - both external *and* internal nodes must be defined in the *same* direction, preferably anticlockwise. Internally, ACES will automatically reverse the direction of the void coordinates (this is transparent to the user). Consequently, the properties of the voided shape will be subtracted from those of the external section.

If the external node coordinates are entered in one direction but the internal nodes are entered in the other direction, ACES will internally reverse the order of the internal coordinates. As a consequence, both internal and external coodinates will be aligned in the same sense. In this situation the properties of the internal (voided) shape will be *added* to those of the external section.

Both external and internal nodes must be defined in the same direction, preferably anti-clockwise!

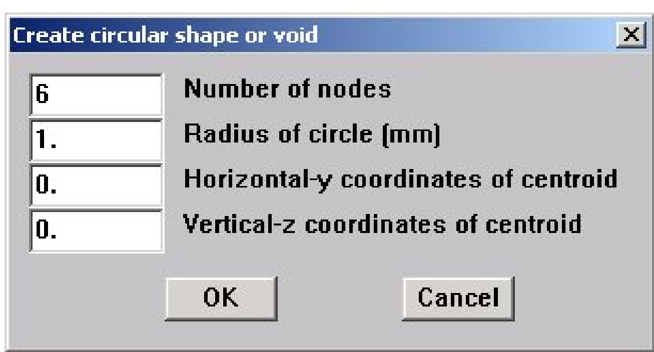
Note that any symmetry options defined for the external shape will also apply to the internal void. Hence if the external shape is symmetrical but the internal void is not (or vice-versa), then do not use any of the symmetry options.

Calculation of Torsional Constant

Refer also to *Section 2.7.6* for further details, particularly if you elect to calculate ***Ix*** using membrane theory and wish change the default parameters.

2.7.4 Circular Shapes & Voids

To create a circular shape or a circular internal void just click the appropriate button. Note that a circle in ACES is approximated by straight line segments connecting node points, the number of segments being determine by the number of nodes you specify. Therefore the more nodes you use, the smoother will be the curve.



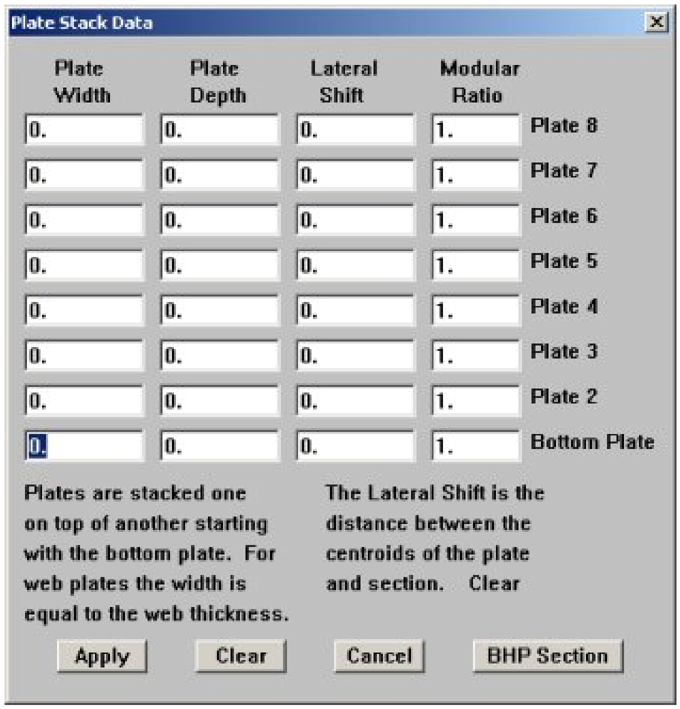
Note: Only a single circular void is allowed in the section.

Nodes are generated in the anti-clockwise direction. Therefore, if using a circular void with an arbitrarily defined external shape the external nodes *must* also be defined in the anti-clockwise direction.

Note that once the section has been created you can manually edit any of the generated nodes by clicking either of the two buttons *Enter/edit external (or internal) node coords* and entering the required coordinates into the dialog box that will be displayed (similar to the one shown above). This allows more complex sections to be created. (Refer to *Section 2.7.3* above for further details).

2.7.5 Stacked Plate Sections

To create or edit a plated section click the button labelled *Create/edit - Stacked Plate Section*. The dialog box shown below will be displayed:



Up to eight plates can be stacked on top of each other to create a section. Web plates can be modelled by assuming the plate width is equal to the web thickness. The ***Lateral Shift*** represents the distance between the centroids of the plate and the section reference axes. It allows non-symmetrical shapes to be created.

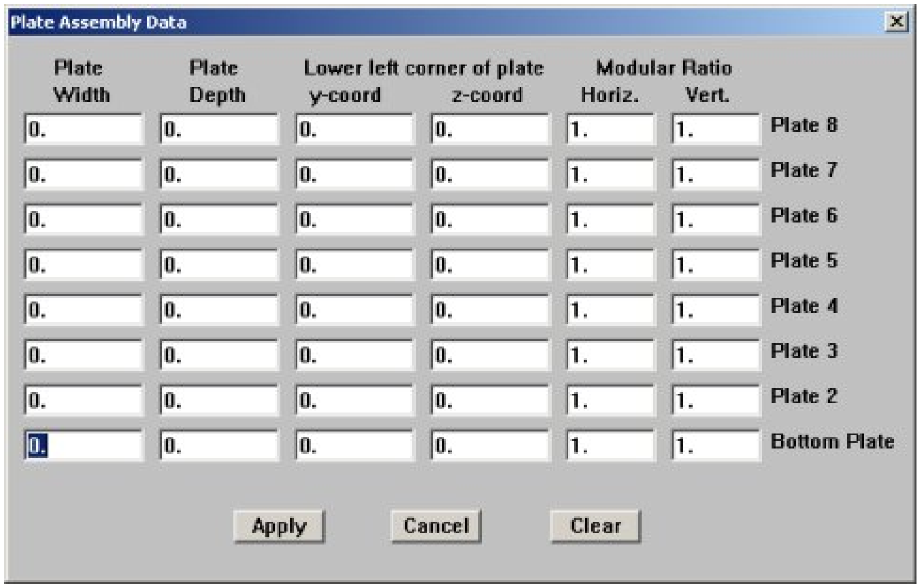
Note that if a standard BHP section is selected the bottom flange of the section is referenced as *Plate 2*. *Plate 1* is intentionally left zero in order to allow you to spacify a bottom plate cover (if required). ACES will ignore this entry if the dimensions are left zero.

Composite sections can be created by entering a modular ratio for the relevant plate(s). In calculating section properties ACES will reduce the width of these plates in proportion to the specified modular ratio(s).

Click ***Clear*** if you wish to delete all values from the table and reset them to zero.

2.7.6 Assembled Plate Sections

To create or edit a section made up of a series of horizontal and vertical plates click the button labelled *Create/Edit - Assembled Plate Section*. The dialog box shown below will be displayed:



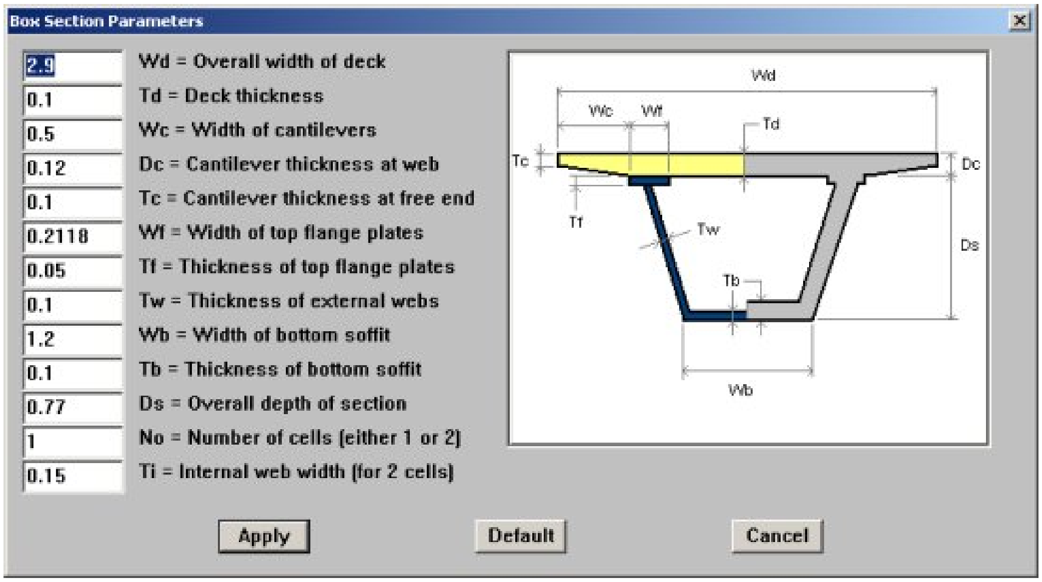
A mixture of up to eight horizontal and/or vertical plates can be arranged in any order to create a section (e.g. a steel box with flanges and a concrete deck on top). Vertical web plates can be modelled by setting the actual plate thickness to the *Plate Width*. Plates are located and oriented in space by specifying the position of their lower left corner.

Warning! *If modular ratios are specified for a plate, its transformed shape may not be properly located on the section when the diagram is displayed. This is due to a shortcoming in the drawing algorithm and does not necessarily mean that all section properties are incorrect.*

For example, in a typical composite section that has a top slab symmetrically located about the centroidal Y-axis, the transformed deck section will be shown in the diagram at the position specified by the bottom left corner of the slab i.e., it will appear offset from the centroid of the section. Although calculated properties will be correct about the horizontal Y-axis they will *not* be correct about the vertical Z-axis. This may not, however, be important to you.

2.7.7 Box Sections

To create or edit a plated section click the button labelled *Create/Edit - Box Section*. The dialog box shown below will be displayed:



Enter box parameters as required. Note that for a composite steel-concrete section no provision has been made for a concrete modular ratio, so the calculation of true composite properties is not possible.

Limitations & Restrictions

A maximum of only two cells can be specified.

A top flange width and depth must be specified.

2.7.8 Torsional Constant & Plastic Properties

(a) Torsional Constant

The torsional constant can be calculated in one of two ways:- either using an exact formula for simple shapes; or standard membrane theory for more complex sections. However, it should be noted that except for a few basic shapes (triangles, rectangles and circles), there is no direct way of calculating the torsional constant.

Simple Shapes

For simple shapes ACES uses the standard relationship: $Ix = Ax^4 / (40 * Ip)$ where *Ax* is the section area and *Ip* is the polar moment of inertia.

Membrane Theory

Membrane theory is based on the property that a membrane subjected to a uniform pressure, (such as a soap bubble), will take on the same shape as the distribution of shear stress of a section under torsion. However, there is no exact formula that can be used to calculate this distribution - an iterative (finite difference) technique must therefore be used.

A fundamental requirement of this method is that the complete section is divided into a large number of small squares. These squares are either included or excluded from the calculation, depending on whether their centroid lies inside or outside of the section boundary.

The default torsion calculation parameters are generally fine for geometrically small sections (I beams etc). On larger, more complex, sections (particularly box girder shapes) a much finer grid and a smaller convergence tolerance should be used. There is generally no exact solution for ***Ix*** for an arbitrarily shaped voided section - the iterative finite difference solution only approximates the true value and its accuracy depends on a number of factors (such as the shape factor and void ratio).

Open & Closed Sections

When modelling voided sections (such as box girders) care must be taken not to create an "open" section (rather than the proper closed section) by inadvertently using re-entrant nodes i.e., where the nodes defining the void are part of the total number defining the external shape and form part of a continuous, unbroken, "line" that wraps back onto itself. (Other CAD packages allow this to be done in order to be able to calculate properties of complex voided shapes).

The problem comes when you attempt to calculate ***Ix***. ACES will assume there is a "slit" through the section making it, in effect, "open" (even if the gap is infinitesimally small). The torsional stiffness of an "open" section is significantly lower than that of a closed section, resulting in the wrong torsional stiffness.

Care must therefore be taken when using the symmetry option with voided shapes. The correct way of modelling a closed, voided, section using the symmetry/mirror options is to create the outside face using external nodes and the inside face using internal nodes. Note that the inside face must be defined in the same direction as the outside face.
If an open section is intended then the gap should be of a reasonable size.

Convergence Target

The finite difference technique is based on the calculation of the required property using successive iterations. In theory this can continue indefinitely, but for practical purposes some target value is required. This is set using the convergence factor, a value that represents the difference between successive iterations. The default value (0.0001) is considered adequate, but should be changed if the section becomes more complex.

Note, however, that if the *Grid Resolution* is increased by 100 then the *Convergence Target* should also be decreased by a factor of around 10. For example, a *Grid Resolution* of 200 will generally require a *Convergence Target* of .00001 (1.0E-5). A resolution of 300 will require a convergence factor of .000001 (1.0E-6) and so on. (Linear interpolation between these values is permitted). The maximum allowable grid resolution is 600, but the solution time and diagram drawing time for grid resolutions above 200 on complex shapes start becoming very long, even on a fast PC.

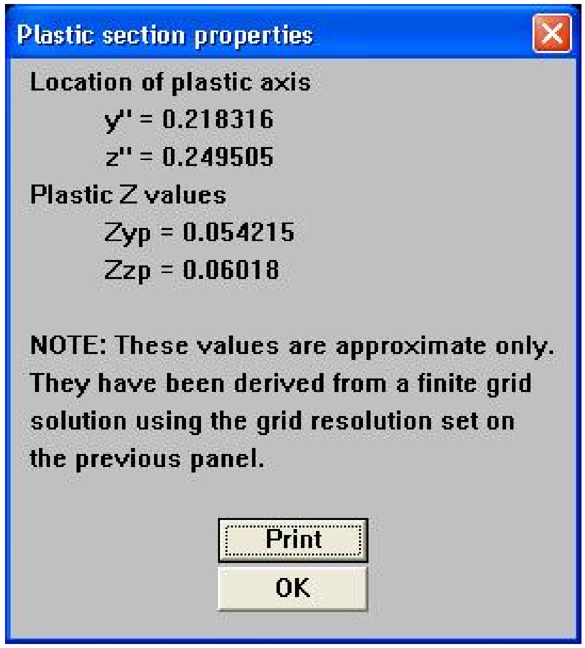
Grid Resolution

This parameter controls the number of horizontal and vertical squares that will be mapped onto the section during the finite difference calculation process. The larger this number, the more "accurate" will be the calculation of the torsional constant. On some shapes, small changes to the resolution can have a significant effect on the result. You may find, under some circumstances, a difference in the calculation of Ix when a complex section (such as a box girder) is created with and without using one of the symmetry options. There are a number of reasons why this may occur:

- The internal geometric "construction" of the model (influenced by the use of symmetry options);
- The precision with which lines and nodes are stored and manipulated (the section properties module only uses single precision arithmetic); and
- The magnitude of the torsional membrane parameter values (for example, the default parameters of 100 for grid resolution and 0.0001 for convergence are generally considered to be "coarse" for complex shapes). If these values are changed to 500 and 0.000001 respectively, the calculated Ix results are identical to about the 3rd or 4th decimal place.

(b) <u>Plastic Properties</u>

Plastic properties are calculated by clicking the *Plastic properties* button on the main panel. Note, however, that the only plastic properties calculated are the location of the origin of the plastic axis and the plastic ***Z*** values. More-over, results are approximate only, since they are derived by using a finite grid analysis based on the grid resolution specified in the main panel (Section 2.7.3).



2.7.9 Importing Node Coordinates

This option allows a section to be defined outside of the module then imported into it for further refinement and calculation of member properties. The section geometry can be imported in one of two ways:

Text File

Create a text file using *Notepad*, *Excel* or any other text editor. The file must be formatted in accordance with the instructions given below:

Line 1: Header information (e.g. Kerb section)
*Line 2: The following label - "**Number of dimensions**"*
*Line 3: The number of dimensions expressed as an integer (e.g. **2**)*
Line 4: The node number and X,Y coordinates of the first node in the list
Line 5: The node number and X,Y coordinates of the second node in the list
" " " " "
" " " " "

From the ACES Interface

Run ACES in the normal way, select *Start a new job* from the opening dialog box then create a new model by *Entering node coordinates directly*. Create a shape using the CAD tools available under the *Structure* menu option, then select the *Section Properties* icon on the top icon bar. Click the *Import node coordinates* button then select *From the ACES interface*. The current shape shown in the main ACES drawing window will be imported into the Section Properties module. You can now modify it as you wish.

2.7.10 Printing Results & Saving & Retrieving Section Data

Click the *Print* button to obtain a report that contains all of the following information:

- A list of all section properties
- A scaled diagram of the section
- Offsets of all nodes from the centroidal *X* and *Y* axes

NOTE: If you do <u>not</u> have a colour printer and some or all of the output is not visible then refer to <u>Section 2.7.9</u> (Setting Display & Print Colours). Ditto if your colour printer is only printing in black.

To save or retrieve a section click the relevant buttons.

NOTE: Sections created in versions earlier than 5.3 may not have all attributes present. Please check section details carefully and recalculate all properties.

2.7.11 Setting Display & Printing Colours

To change the default colours in which diagrams are displayed on screen and results printed in hard copy click the *Settings* button. Two separate windows will be shown together:

A Dialog Box of Current Default Colours: This panel lists the colours of all drawing attributes that you can customise to suit your system viz:

- *Section Outline and Section Fill colours*
- *Main Headings and Axes colours*
- *Table Headings, Outlines and Values colours*

The number on the button adjacent to each attribute corresponds to a line style shown in the second window (*Current Line Styles*). To change the colour, or style, of one of the attributes click the button adjacent to its decription. A line style selector window will pop up. Click the radio button of the number corresponding to the style you require followed by *OK*.

Note: If an attribute colour is selected that is identical to the background colour of the dialog box the attribute description will not be visible on the dialog box! (Since it's the same colour).

A Palette of Available Line Colours & Styles: This panel shows all currently available line colours and styles. It serves no other purpose than to help you to identify and match the default line style numbers with their corresponding colours.

Set all printed values to black: If you do <u>*not*</u> have a colour printer you will need to tick this check box, otherwise very light colours on a black screen background may not be visible when results are printed on white paper.

Print node numbers: By default ACES will not print node numbers on the diagram of the section. Tick this check box if you want these numbers printed.

Select colour of drawing window: Click this button if you wish to change the background colour of the drawing window.

Save colour settings: Click this button if you wish to save the settings for future use.

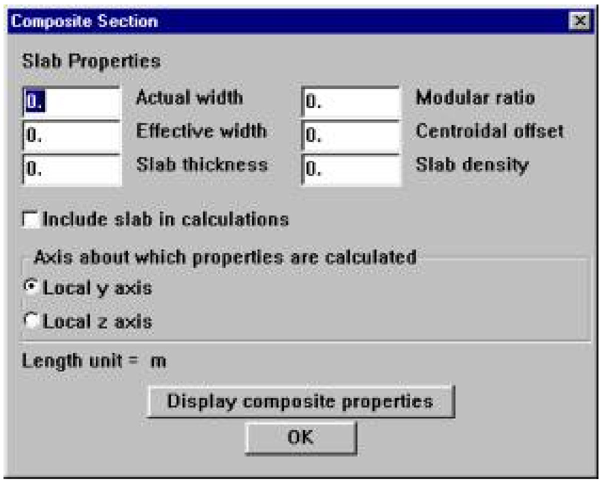
PART 2.8 Composite Section Calculator

1.0 INTRODUCTION

The *Composite Section* option allows composite section parameters to be defined and included as part of the overall girder cross-section. ACES assumes the slab section is made up of a single rectangular block offset from the girder centroid by a user-specified distance. The parameters entered in this panel are used to convert the concrete slab into an equivalent steel slab whose properties are then added directly to those of the girder just prior to the analysis.

2.0 SECTION PARAMETERS

The following panel is displayed when the *Composite Section* buttoen is pressed:



Actual width

The actual width of the deck slab is used to calculate the self weight (dead load) of the slab. Note that if you are modelling a *grillage* structure and you elect ACES to calculate DL effects for you, then you should set the density of transverse grillage members to zero - otherwise their self-weight will be included twice.

Effective width

The effective width of the slab is used to calculate the longitudinal stiffness of the girder. Under certain conditions design codes will not allow the full slab width to be used in the analysis owing to factors such as shear lag, cracking and so on. (Refer to your Design Code for guidance).

Modular ratio

The Modular Ratio (defined as *E girder/E slab*) is used to convert the concrete slab into an equivalent steel section - its properties are then added directly to those of the girder.

Slab density

The slab density is used to calculate the dead load (self-weight) of the slab. If the structure is being modelled as a combined *Slab+grillage* type the slab weight will *not* be included in the girder weight, since it is taken into consideration during the finite element modelling process. Nor will its in-plane contribution to the overall girder stiffness be considered (for the same reason). Refer to the discussion regarding the theoretical basis on the following page.

Centroidal offset

The centroidal offset is the distance between the centroids of the girder and slab. Typically the slab will be cast directly onto the top flange of the girder or, as often happens, a haunch is included (which increases the effective slab offset further). In this event, the offset should be calculated as the girder-slab centroidal distance and include the haunch depth (if any).

Axis about which properties are calculated

This indicates the direction of the local axis when calculating composite properties. For 2D/3D grillage models the appropriate local axis is likely to be the ***y*** axis, while for 2D frame structures it will be the ***z*** axis. ACES will issue a warning if it considers the selected axis direction is not appropriate. You may, however, ignore the warning.

Include slab in calculations

Not-with-standing the fact that parameters may be present in this panel the section will not be considered composite unless the parameter *Include slab in calculations* is checked on (i.e. ticked). Note too that the girder and slab properties will always be displayed separately in the *Member properties* dialogue box **-** they will only be combined into a single composite section just prior to the analysis.

Display composite properties

To view the composite properties click the button labelled *Display composite properties*.

(Note that the *ACES.INP* input file created by the system just prior to the analysis will reflect the composite properties and not the separate girder and slab properties **-** a message will be included in the file to remind you of this fact).

3.0 THEORETICAL BASIS & ASSUMPTIONS

Knowing the effective width, thickness and modular ratio the system transforms the slab area into an equivalent steel area (*As*). Using the slab-girder *Centroidal Offset* the program then determines the location of the centroid of the composite section and calculates the distances *Yg* and *Ys*. Finally it uses the relationship given below to calculate the composite moment of inertia of the transformed section:

$$Ic = Ig + Ag.Yg.Yg + Is + As.Ys.Ys$$

Ic, ***Ig*** and ***Is*** represent the moments of inertia of the composite section, the girder and the slab respectively. For *Slab+Grillage* models ***Is*** will be excluded from ***Ic*** since the stiffness of the slab is taken into consideration during the finite element modelling process. Similarly, the slab area is excluded from the total area. The torsional constant for the composite section is approximated by considering the section made up of narrow rectangles and summing the total.

PART 2.9 Element Properties

1.0 INTRODUCTION

This section will describe the manner in which section and material properties are defined and assigned to elements. As already pointed out in Section 2.2, when generating an FE slab model ACES automatically assigns unique colours (representing element property types) to each structural element of the model (main deck, cantilevers etc).

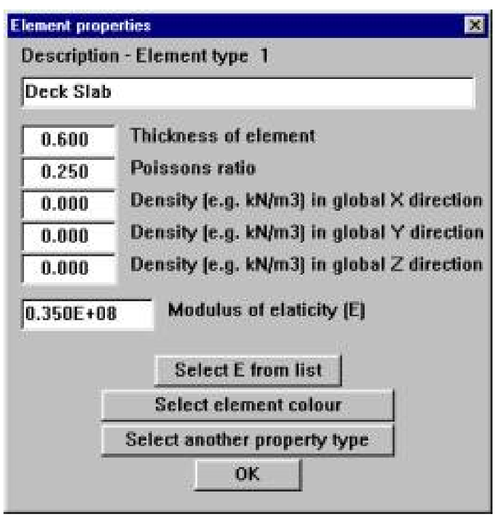
If the result is exactly what you require, then all you need do now is enter the appropriate properties for each of the element property types. Otherwise, you will need to create new property types and assign them to the relevant elements in the model.

2.0 DEFINING ELEMENT PROPERTIES

Either click the element property icon (shown as a small rectangular block) or select the following menu options: *Structure / Element properties / Create and change.*

A dialogue box listing all currently defined element property types will appear.

Highlight the required element property type (e.g. *Deck slab*) then click *Edit*. An element properties panel will be displayed as shown, with default values of *1.0* assigned to all major property values (with the exception of *E* and density, which are both set to zero). Change these to suit your own model requirements.



Densities

Enter as *Force/Unit^3* (e.g. *kN/m^3*) and note that for models where the deck lies in the global *X-Y* plane they must be given in the correct sense. For FE slab models generated parametrically (using one of the model templates), density must correspond to ***Density - Global Z*** and must be negative, since the global *Z* axis is *out* of the plane of the screen e.g. *Density* - *Global Z = -24 (kN/m3)*

Select E From List

This button allows the Modulus of Elasticity (*E*) to be selected from a list of standard values for concrete, steel & timber.

3.0 DISPLAYING ELEMENT PROPERTY TYPES

Once all properties have been entered for element property *Type 1* click onto the *Select another property type* button to choose the next element property type to edit (or to create a new property type). Repeat the above steps for as many property types as required. Once all element property types have been defined click *OK* to exit from the section properties panel and to redisplay the model geometry.

Although each element is displayed in a unique property type colour, for very dense meshes you may wish to quickly check whether element property types have, in fact, been assigned correctly. To do this click onto *Structure / Element properties / Display same type* then, for each of the *Element Types* shown in the dialogue box, click on each in turn. Only those elements will be separately highlighted.

4.0 CREATING NEW ELEMENT PROPERTY TYPES

To create a new element property type take one of the following two actions: either click the element property icon (designated by the brick shape) or select the following menu options: *Structure / Element properties / Create and change.*

A dialogue box as shown in Section 2 above listing all currently defined member property types will appear. Click the *New* button. An element properties panel will be displayed. Now return to Section 2 for guidance in specifying actual section properties.

4.1 Assigning Property Types to Selected Elements

Select a range of elements in the model by cascading through the menu options: *Structure / Element properties / Assign to..* and select the element range you require from the submenu of range options. Follow any instructions that may appear on the lower prompt line (they will help you to identify the range). Once the range has been delineated a list of current element property types will be displayed. Select the property type you wish to assign to the element in the range then click *OK*.

- To assign a property type to only one single element select the range as ***..single*** then click the cross-hairs on the element(s) you wish to change.

- To assign a property type to a ***rectangular block*** of elements place the crosshairs at some point on the screen, click the *LEFT* button once, drag out a box of a size that encompasses the required elements then release the button. All elements whose centroids fall within the delineated area will be changed to the current (or selected) element property type.

- To assign a property type to all elements whose centroids lie within an arbitrary ***trapezoidal area***, the area must be defined using two unconnected but opposing straight lines. Each line must be drawn in an anti-clockwise sense, but the lines themselves need not be parallel - they can be oriented in any general direction. However, the area delineated by their notional “boundaries” must incorporate that part of the model within which the target elements lie. All elements whose centroids fall within the delineated “area” will be changed to the current (or selected) elements property type.

  Lines are created by clicking at a point on the screen that is to represent one corner of the notional area then, holding down the left mouse button, dragging out a fence line in an anti-clockwise direction to an end point representing the adjacent corner.

PART 3

STATIC & MOVING LOADS

PART 3.1 Creating Load Cases

1.0 LOAD CASES & LOADINGS

Loadings are applied to the structure via the ***Loadings*** option in the menu bar. It is important that the distinction between *load cases* and applied (or generated) *loadings* is clearly understood, particularly as it applies to moving vehicle loads.

Load cases can be a mixture of self-weight, nodal loads, member loads, patch loads and vehicle loads. In practice, of course, it is best to separate them into separate load cases then combine or envelope them once the analysis has been performed. A maximum of 99 load cases can be created and applied to the model in any one run.

However, any one *load case* can consist of up to 5 individual moving vehicle loads that in themselves might generate many hundreds of separate *loadings* as they move along and across the structure. In this situation each generated *vehicle loading* represents a snapshot of the structure showing the position of all vehicles in that load case at that instant as they travel along their designated paths.

1.1 Loading Restrictions

Although ACES allows models of up to 6000 nodes, members and elements to be created the practical limits may be smaller if large numbers of generated loadings are to be applied. Up to 99 individual ***load cases*** can be created with the following total number of generated vehicle loadings permitted in any one run (the static location of all vehicles at one particular point in their paths within a load case constitutes a ***loading***):

2D Grillages	2000
3D space frames and FE shell models	1000
Trusses	3000

In reality, the limitation imposed on the system is the *total number of nodal loads* generated by all load cases in the model. It is important, therefore, to avoid including uniformly distributed loads (such as lane loads) within a vehicle load case, since they have the potential for generating large numbers of nodal loads. If it is essential to combine lane loads with moving vehicle loads, create a separate “patch” load case then envelope it with the relevant vehicle load cases when interrogating the results.

2.0 CREATING, EDITING & DELETING LOAD CASES

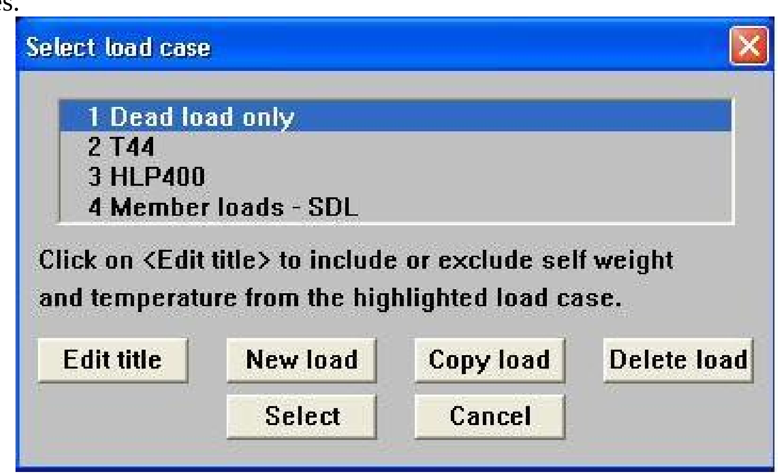
2.1 Creating a Load Case

To apply loads to the model you must first define at least one load case. This can be done by either selecting *Loads/Add-Edit load case* or by clicking on the ***Select a load case*** icon (the button displaying a large down-arrow and the letter ’***F***’).

If this is the *first* load case the identification panel will be displayed:

You need not enter a load case title, or description, but it may make identification of that load case difficult later. You may also indicate in the check box provided if the structure self-weight (dead load) is to be included as part of the load case. Generally, dead load should be treated as a separate load case and should *not* be included as part of a moving vehicle loading (refer to *Section 1.1* above for details).

If this is *not* the first load case a dialogue box will appear showing all of the currently defined load cases.



You may either:

- • Edit the title of an existing load case using ***Edit title***
- • Create (add) a new load case using ***New load***
- • Copy all loadings from the currently highlighted load case into a new load case using ***Copy load***
- • Delete one of the existing load cases using ***Delete load*** (but refer to Section 2.2 for details)
- • Select a load case which you would like to view, edit or to which other loadings are to be applied. Just highlight it in the list then click ***Select***.

Once a load case has been selected, (or activated), any loadings associated with that load case (if any) will be displayed on the model. You may now apply new, or additional, loads to that load case or delete any of the existing loadings. To **delete all loads** in the load case select *Loads/Delete loadings/All loads*.

2.2 Editing a Load Case

To edit an existing load case, click on the loading icon (the button showing the down-arrow and the red letter '***F***') and highlight the load case you wish to edit. Press ***Select***. All loadings associated with the selected load case will be graphically displayed on the model. You can now add other loadings to the model or delete any that are no longer required.

HINT: To change the value of a static nodal or member loading you must first delete that loading from the appropriate nodes and/or members then reapply new value(s).

2.3 Deleting a Load Case

To delete a complete load case (as against individual *loadings* within it), click on the loading icon (the button showing the down-arrow and the red letter '***F***') and highlight the load case you wish to remove. Press ***Delete load***. ACES will warn you of the consequences of doing this and allow you to abort the operation. The main thing to be aware of is that if you delete a load case that forms part of existing envelopes or combinations, those envelopes or combinations will no longer be valid.

Note that to *delete all loadings* in a particular load case you must use the menu commands: *Loads/Delete loadings/All loads*.

PART 3.2 Moving Vehicle Loads

1.0 MODELLING MOVING VEHICLE LOADS

Vehicles can be applied to a load case in one of two ways:

- • By choosing one or more vehicles from the data base and manually defining their paths
- • By selecting one of the standard Austroads loading templates (automatic generation of paths)

In both cases ACES will expand each vehicle load case at solution-time and create a series of individual loadings that each represent a snapshot of the position of all vehicles on the structure in that load case at any point of travel along their designated paths. Results values (moments, shears, torsions, reactions and deformations) are calculated and stored for every one of these generated vehicle loadings, enabling any one of them to be viewed and interrogated at any stage. The distribution of the moving wheel and patch loads is done only in the XY plane and the Z coordinates are ignored.

Note that vehicle wheel loads in 3D models are always displayed as if they are located in the X-Y plane (i.e. at Z = 0). If the running surface has a non-zero Z-coordinate value and an isometric view is selected, the defined vehicle(s) may sometimes appear to be applied at the wrong location. This is only a display problem and not a load modelling issue - vehicle loads will always be distributed to nodes with non-zero Z values providing those nodes are valid participants in the distribution process. To confirm that the vehicle location is correct, you should view the model in 2D (by clicking the "*Plan*" icon on the toolbar).

For a comprehensive over-view of the way in which vehicles can be applied and moved over the structure refer to Section 4 (changing the vehicle path attributes) and Section 5 (changing the vehicle movement attributes). Note also the limitations and restrictions imposed on moving vehicle loads as discussed in Section 6

1.1 Distribution of Wheel Loads

Wheel loads are distributed to the nearest three or four nodes using a mathematical algorithm based on statics. The formulation ensures that the centroid of the loads distributed to adjacent nodes lies at the same point as the position of the wheel itself. In the vicinity of skewed supports there will generally be three nearest nodes while for continuous beams and 2D frames only two nodes.

Modified or Non Parametric Models:

If a model is *not* generated using one of the standard parametric templates, or if other nodes are added to it, the associated internal moving load distribution keys will also not be auto-generated. Therefore, when the model is analysed the system has to generate them using a temporary finite-element "overlay" mesh based on all available active nodes. This results in a pseudo set of triangular elements which is somewhat random (and is displayed during anlaysis) but, more importantly, results in wheel loads being distributed only to the closest three nodes rather than the normal four. This can produce a "skewed" distribution of loaded nodes, where three may be loaded but the fourth is not.

A better way to distribute moving loads to non-generated models (or to highly modified models) is to explicitly define a set of finite elements linking the nodes that you want to be included in the moving load distribution (i.e. the "running surface"). This is because ACES uses the elements as the moving load keys, hence giving you better control of how the loads will be distributed. Note, however, that prior to the analysis you must first set the analysis type to "*No finite element* *analysis*". This device forces the system into using the finite elements when distributing the moving vehicle loads but ignores them in the actual analysis of the structure.

1.2 Horizontal & Transverse Trains of Moving Loads

Both horizontal and transverse moving loads, as well as the usual vertical wheel loads, can be applied to the model, either individually or simultaneously. Horizontal and transverse trains of wheel loads can represent longitudinal braking and centrifugal force effects respectively. They must, however, be created and saved as individual, user-defined vehicles. Refer to Section 7, *Create & Save a New User Vehicle*, for further information.

1.3 Defining the Vehicle Running Surface

When model templates are used to create the model geometry ACES automatically assigns a running surface based on the type of parametric model selected. If you create the model geometry in another way (e.g., by importing node coordinates from a CAD package) the system has no inherent ability of determining the surface on which any applied vehicles are to run. To do so it is necessary to define an active plane in which the vehicles will run. This can be done by using the *Activate / Nodes /…* or *Activate / Mixed…* menu options and suppressing or enabling those parts of the model that will either form, or be excluded from, the vehicle running surface.

Normally this should be done just prior to the analysis, after which the entire structure can be made active again (for the purpose of interrogating results). It should also be noted that the process of de-activating parts of the model does **not** exclude those parts from the analysis process. It only ensures that wheel loads are distributed to active nodes during load case generation.

Once an active view of the running surface has been created it can be stored for future use via the menu commands: *Activate / Store* (ACES will ask that you provide a descriptive name for this view). To recall the view at a later time simply click the penultimate icon on the right hand menu bar and select it from the list of all available active views.

Note on viewing vehicle loads: Note that vehicle wheel loads in 3D models are always displayed as if they are located in the X-Y plane (i.e. at Z = 0). If the running surface has a non-zero Z-coordinate value and an isometric view is selected, the defined vehicle(s) may sometimes appear to be applied at the wrong location. This is only a display problem and not a load modelling issue - vehicle loads will always be distributed to nodes with non-zero Z values providing those nodes are valid participants in the distribution process. To confirm that the vehicle location is correct, you should view the model in 2D (by clicking the "*Plan*" icon on the toolbar).

1.4 Circular Curved Paths

For circular vehicle paths ACES now assumes that the full vehicle reference line, from one end of the vehicle to the other, is also circular i.e., all axles will now point radially towards the centre of the circle. This ensures that all axles follow the line delineated by the circular vehicle path. (Consequently the rear portions of long vehicles will no longer "swing" outside the path).

It is important to bear in mind, however, that the mathematics used to calculate the wheel positions of vehicles on circular paths differs from that of a straight vehicle. For a radius of 10,000m and a structure length of 20 metres, for example, a vehicle can track 5mm off a straight line. This may be enough to make a 1 or 2 kNm difference in the maximum girder moments. Shear and reaction values may also differ noticeably, since they are very dependant on the exact location of vehicle wheels.

2.0 MANUALLY APPLYING VEHICLES

To add one or more user-nominated moving vehicle loads to the model:

- • Create a load case as described in *PART 3.1* of the user guide
- • Select ***Loads/Apply vehicle loads/User selected vehicles/Vehicle #***
- • Select the vehicle number (#) that is to be added e.g. *Vehicle 1*
- • Choose ***Select vehicle from file***, highlight the required vehicle in the list box and click ***Open***
- • Specify the vehicle path details in the dialogue box (refer to Section 4, *Change Vehicle Path*)
- • Click ***OK*** to generate the vehicle and display its path on the model
- • Repeat the above steps to add other vehicles to the same load case
- • To change vehicle movement characteristics refer to Section 5, *Change Vehicle Movement*
- • To display the vehicle(s) moving across the structure select *Loads / Display vehicle movement*
- • Add other nodal or member loads to the load case if required (but see Section 6 first!)
- • To locate a vehicle in a static position on the model refer to Section 5.3

Vehicles in any load case can all be identical or all mixed. A single T44 truck, for example, can constitute one load case while a T44 truck in combination with an HLP400 moving independently across the bridge structure can be considered as another. Yet a third load case could consist of two T44 trucks moving simultaneously with an HLP.

To delete a vehicle from the load case select *Loads/Delete loadings/Vehicle loads/Vehicle #*, where **#** refers to the vehicle number you wish to remove. Refer also to Section 7, *Create & Save a New User Vehicle*, if you wish to create your own vehicles and save them to the data base.

3.0 APPLYING AUSTROADS VEHICLES

To add an AUSTROADS vehicle to the model you must first define at least one lane on the model (refer to *PART 2.4* for instruction in doing this) then:

- • From the menu bar select ***Loads/Apply vehicle loads***
- • Select the required AUSTROADS loading code followed by the vehicle type
- • Specify a vehicle movement increment (be careful not to make the increment too small!)
- • Indicate the lanes to which a vehicle is to be added (not applicable to continuous beams)
- • Indicate if the lane reduction factors are to apply to vehicles in this load case
- • Read carefully the message in the dialog box
- • Click ***OK*** to continue.

ACES will generate the vehicle loads and display their travel paths (represented by a series of dashed lines). The chosen AUSTROADS vehicle will be added to all nominated lanes. Each vehicle will have the same start and end X coordinates and the same longitudinal movement increment. If you wish to modify the path or movement characteristics for any of the auto-generated vehicles refer to Section 4, *Changing Vehicle Path* and Section 5, *Changing Vehicle Movement Attrbutes*.

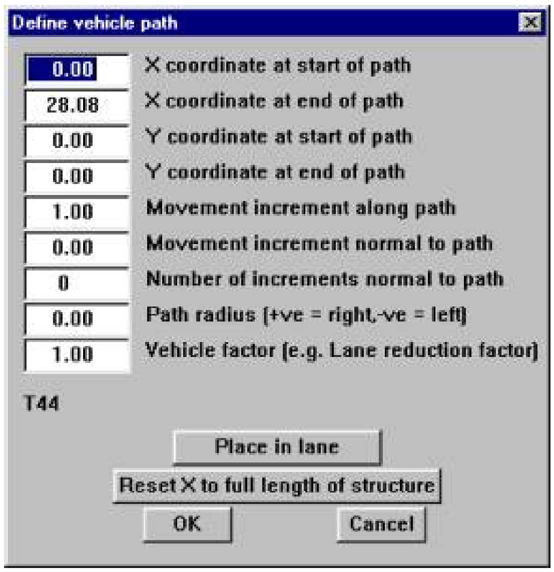
If the lane reduction flag has been ticked on, vehicles in all lanes will be multiplied by the appropriate lane factor (select *Structure/Define lanes/Lane Reduction Factors* to check these factors).

WARNING! If this is not a new load case then any existing vehicle loadings in this load case will first be deleted before the new vehicle loads are applied.

4.0 CHANGING THE VEHICLE PATH

If the *Define vehicle path* dialogue box is already displayed, skip these bullet points. Otherwise, to change the vehicle path:

- Select ***Loads/Edit vehicle loads/Vehicles # /Specify path of vehicle*** (where # represents the vehicle number whose path is to be changed).
- Alternatively, if you have previously edited the path of any *Vehicle #,* click on ***Shortcuts*** in the menu bar and select that command string from the pop-up menu.
- Change the path parameters to suit your requirements (refer to the parameter definitions below).
- Click ***OK*** to apply the changes to the vehicle path and exit from the dialogue window.
- To locate the vehicle in a static position on the model refer to Section 5.3



X,Y coordinates at start & end of path

These coordinates define the vehicle travel path. Coordinates are measured with respect to the position of the reference line (the dashed line) where it touches the leading (front) axle of the vehicle. By default ACES positions the leading axle at the origin (select ***Settings/Place axes at origin*** to see where this is) and moves the vehicle from left to right along the structure. The dashes represent, to scale, the vehicle movement increments. (The Y-coordinate will be suppressed for continuous beams and frames).

Note that vehicle paths may begin and end off the model - ACES keeps track of where they are during the load distribution process. If you want the vehicle to move in the opposite direction, refer to Section 5, *Changing Vehicle Movement Attributes*.

Movement increment along path

This parameter governs the number of vehicle movement increments in the longitudinal direction i.e. along the structure. Movement increments are shown to scale as dashed lengths of line representing the vehicle path. Take care not to use too small an increment, since the number of generated vehicle loadings may exceed the capacity of the system (refer to Section 6 for a discussion of this issue).

If the path is not parallel to the X-axis this parameter represents the incremental (diagonal) distance *along* the path, the positive direction of movement being in the same general sense as the X axis.

Movement increment normal to the path *and* ***Number of increments normal to path***

These parameters specify the number of parallel paths that will be swept by the vehicle in the transverse direction across the structure. They do not apply to continuous beams and frames. For example, if the *Number of increments normal to path* is set to *1* and a value is entered for *Movement increment normal to path*, ACES will:

- • Move the vehicle along the structure using the start and end *X,Y* co-ordinates then
- • Return the vehicle to its start *X,Y* position;
- • Increment the start and end *Y* co-ordinates by the offset distance normal to the path and
- • Move the vehicle along the new path to its end coordinates, then stop.

Path radius

If the vehicle is to move on a circular curve enter the path radius. Note that if the vehicle is very long it will not articulate properly as it moves along the path. It may therefore be necessary to sub-divide the vehicle into several shorter components which can then be all added to the load case as individual vehicles (up to 5 vehicles are permitted in any one load case).

In this event it will be necessary to offset the path of each individual vehicle “component” by an appropriate longitudinal distance such that, when taken in total, they model the entire vehicle.

Vehicle Factor

This factor can be used to specify a lane load reduction value, a dynamic load allowance or a vehicle multiplier (e.g. if only 0.5 of a T44 is to be applied to the structure). It should not be confused with the load factors used in creating final envelopes **-** they normally represent limit state factors prescribed by design codes. The default value for the *Vehicle factor* is 1. Note that any lane load reduction factor associated with the lane in which the vehicle is placed will be reflected in this field. If two vehicle factors are to be applied simultaneously (e.g. lane reduction and dynamic impact) they will have to be manually multiplied then entered into the *Vehicle factor* field.

Place in lane

Click this button if the vehicle path is to be centrally located within a previously defined lane. ACES will prompt you to choose the required lane number if more than one lane has been defined. To check on lane numbers select *View/View lanes*. Note that any lane load reduction factor associated with that lane will be applied to the vehicle loading and will be echoed in the *Vehicle factor* field (see above). This option is not relevant to continuous beams and 2D frames.

Reset X to full length of structure

Click this button if you want to reset (or set) the end X coordinate to the full length of the structure.

To edit another vehicle path, repeat the above steps or click *Shortcuts* or *LastMenu* to retrieve the last series of commands relating to vehicle path editing.

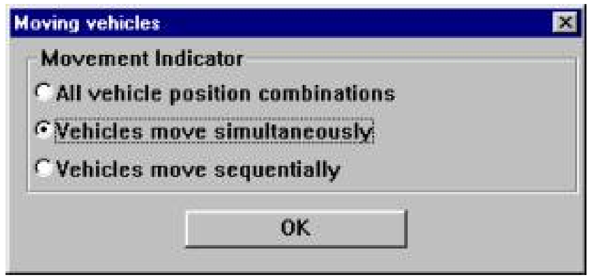
5.0 CHANGING VEHICLE MOVEMENT ATTRIBUTES

5.1 Reverse Vehicle Movement Direction

Select *Loads/Edit vehicle loads/ Vehicle # /Swap movement direction*, where "#" represents the vehicle number whose direction you wish to reverse. The direction of movement will be reversed. Repeat as often as you wish, either for the same vehicle or for other vehicles.

5.2 Specify Movement of Multiple Vehicles

To choose the way in which all vehicles in a multiple-vehicle load case are to move select ***Loads/Edit vehicle loads/Set type of movement*** and select the required movement mode from the table:



Sequential movement will move each vehicle in the load case individually along its own path i.e. the first vehicle will move across and off the structure before the next begins moving.

Simultaneous movement means just that - all vehicles in the load case will move across the structure together and by the same increment, albeit within their own paths. Vehicle movement increments and distance travelled are governed by the path attributes of the *first* vehicle in the group. This option is useful for moving articulated locomotives, particularly if the combined effects of vertical load and braking effects need to be modelled.

All vehicle position combinations will generate loadings of all possible vehicle positions. In this mode one vehicle is fixed while the others are incrementally moved along their entire paths. The position of the “fixed” vehicle is then incremented and the others are again moved over their full paths. While this mode is ideal for ensuring that a blanket envelope is created of all possible vehicle combinations, it has the potential for generating an enormous number of loadings that may exceed the capacity of the system (refer to Section 6 for details).

5.3 Display Vehicle Movement

To check that the vehicle(s) will move correctly over the specified path select *Loads / Display vehicle movement* and enter appropriate parameters into the dialogue box. Since there is no way of stopping the animated display once it has begun the number of animation increments should be selected with care. Note also that this feature is only meant to provide a visual check on the correct orientation of the vehicle path(s) - load generation is *not* performed at this stage.

Note on viewing vehicle loads: Note that vehicle wheel loads in 3D models are always displayed as if they are located in the X-Y plane (i.e. at Z = 0). If the running surface has a non-zero Z-coordinate value and an isometric view is selected, the defined vehicle(s) may sometimes appear to be applied at the wrong location. This is only a display problem and not a load modelling issue - vehicle loads will always be distributed to nodes with non-zero Z values providing those nodes are valid participants in the distribution process. To confirm that the vehicle location is correct, you should view the model in 2D (by clicking the "*Plan*" icon on the toolbar).

5.4 Static Positioning of the Vehicle

If the vehicle is to be analysed in a static location on the structure set the X,Y starting coordinates to the required static position of the lead axle then set the end coordinates to values that will orient the vehicle in the left-to-right sense on the bridge deck (note that the end ***X*** coordinate *must* be larger than the start ***X*** coordinate). Finally, set the movement increment along the path to zero.

If you want the vehicle oriented in the opposite sense (facing from right to left across the structure), first set it to face in the positive direction then reverse the direction of travel as described in Section 1.4.1.

6.0 LIMITATIONS & RESTRICTIONS

Although ACES allows models of up to 6000 nodes, members and loadings to be created the practical limits may be smaller if large numbers of generated loadings are to be applied. While it is difficult to prescribe exact limits, a model with 3000 nodes and members, for example, may be restricted to approximately 1000 generated vehicle loadings.

In reality, the limitation imposed on the system is the *total number of nodal loads* generated by all load cases in the model. It is important, therefore, to avoid including uniformly distributed loads (such as lane loads) within a vehicle load case, since they have the potential for generating large numbers of nodal loads. If it is essential to combine lane loads with moving vehicle loads, create a separate "patch" load case then envelope it with the relevant vehicle load cases when interrogating the results.

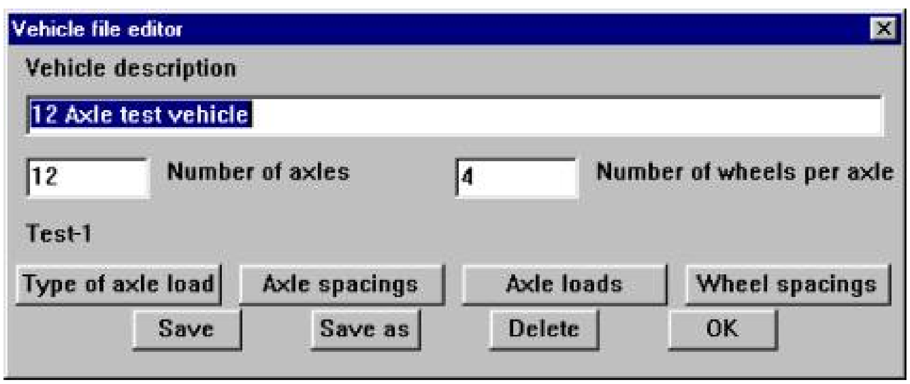
Modifying Vehicles

Be aware that when a vehicle is edited then resaved to the vehicle database ACES will *not* automatically change any of those vehicles that may have already been applied to the model. Whatever vehicle configuration has already been applied to an existing load case (or cases) is irrevocably tied to that load case and will be stored in the model as such. If you edit a vehicle and want those changes to be reflected in load cases that contain that vehicle you must first delete the said vehicle from those load cases then reapply them. Alternatively, make changes to your vehicle first before creating the load cases.

7.0 CREATING & SAVING USER DEFINED VEHICLES

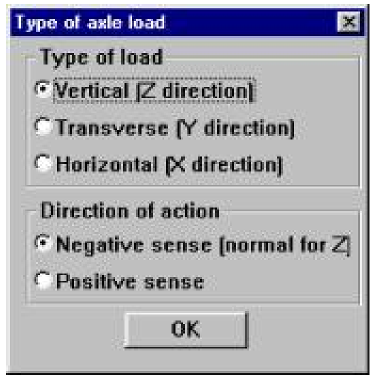
To create a new vehicle and save it to the data base for future use select in turn *Loads/Create-Edit vehicles/Create*. Enter a vehicle file name and click *OK*. The vehicle editor dialogue box shown below will be displayed. Input parameters as required. Note that the vehicle will be redrawn to scale as each parameter (or group of parameters) is entered.

Be aware that when a vehicle is edited then resaved to the vehicle database ACES will *not* automatically change any of those vehicles that may have already been applied to the model. Whatever vehicle configuration has already been applied to an existing load case (or cases) is irrevocably tied to that load case and will be stored in the model as such. If you edit a vehicle and want those changes to be reflected in load cases that contain that vehicle you must first delete the said vehicle from those load cases then reapply them. Alternatively, make changes to your vehicle first before creating the load cases.



Type of Axle Load

This parameter allows horizontal and transverse trains of moving loads to be defined as well as the usual vertical loads (the default). The following panel will be displayed:



The negative, (downwards), sense is considered the normal direction when creating models of vehicles that represent vertical wheel loads. ACES makes allowance for load signs with this loading type.

Axle Spacing

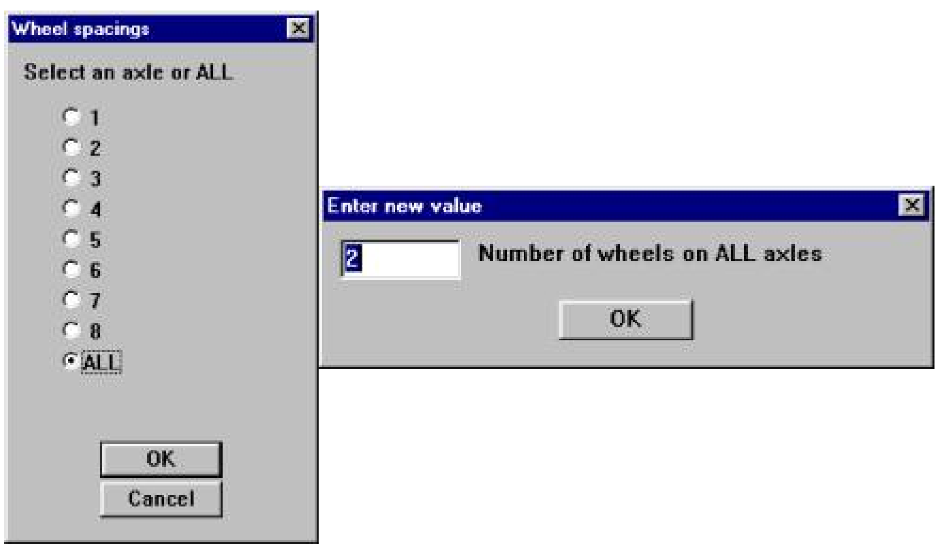
Axle spacing is measured relative to the previous axle. If most axles are uniformly spaced enter a value in the top left field then click the *Apply* button to set all axle distances to that value. Now change the spacing of axle 1 to zero and set any others to their true values. Note the positive sense of axle spacings (shown in the bottom right corner of the drawing area).

Axle Loads

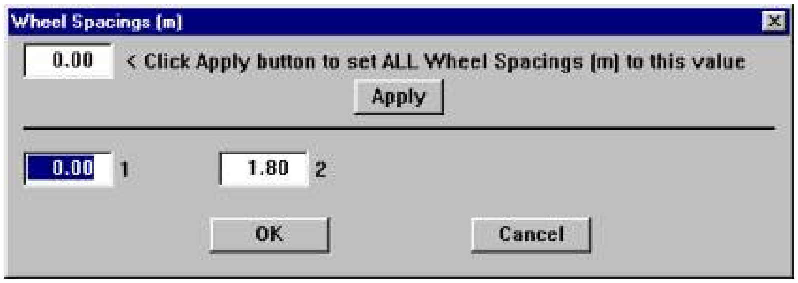
If most axles have the same load enter a value in the top left field then click the *Apply* button to set all axle loads to that value. Now change the load on any axle to its true value.

Wheel Spacing

Wheel spacings for all axles can either be all uniform or every axle can have a different spacing configuration. The panel on the left will be displayed when this option is selected:



The ***ALL*** option assumes that all axles will have the same wheel spacing. Clicking OK will display the panel on the right. This allows the number of wheels per axle to be entered, after which the window below is shown:



Wheel offsets are given relative to the immediately preceeding wheel, with a negative offset value causing wheels to be positioned *downward* from the preceeding wheel and positive values positioning them *upwards* from the preceeding wheel. (The positive sense of wheel spacings is shown in the bottom right corner of the drawing area). Note that the first wheel can also have a *negative* offset, in which case it will be located *below* the horizontal vehicle reference line.

When applying vehicle loads to the model, ACES positions the vehicle on the structure using this datum line. Locating the reference line at the centre of the axle will therefore allow vehicles to be placed on the model with respect to the centre of the leading axle and not the bottom wheel.

Save / Save As

To save the vehicle to the vehicle data base with the name originally specified when entering this procedure click *Save*. To save it under a new file name click *Save as*

PART 3. 3 Patch & Lane Loads

1.0 INTRODUCTION

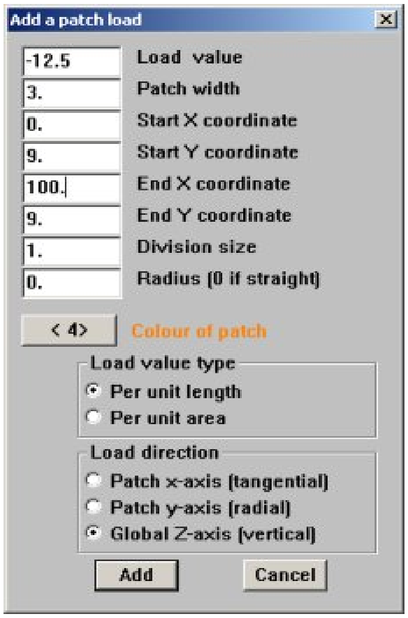
Although patch loads are strictly static loads they are most often used in ACES to model lane loads and are, therefore, considered as a form of "vehicle loading". (Note that patch loads cannot be applied to 2D frames and continuous beams). Patch loads can be added to the model in two different ways:

- • By defining and applying them manually (refer to *Section 2.1* below)
- • By selecting one of the standard Austroads lane loading templates (refer to *Section 2.5*)

2.0 ADDING PATCH & LANE LOADINGS

2.1 Adding Patch Loads Manually

Create a load case as described in *Section 1.1*, then select the following menu options: ***Loads/Apply static loads/Patch loads/Add***. Change the patch parameters in the dialogue box to suit your requirements then click the ***Add*** button.



Load Value

This is the uniformly distributed load per unit length that is to be applied over the specified width of the patch. For example, the default value of -12.5 kN per metre length shown in the dialog box above would be applied by ACES across the full patch width specified in the *Patch Width* field of 3m (see below). In reality this is equivalent to a UDL of $-12.5/3 = -4.167\ kN/m^2$. The negative sign indicates that the load is acting *downwards* (in the negative global Z direction).

Patch Width

This is the width across which the patch is to be distributed. (Refer to the example given under *Load Value* above).

Start & End X,Y Coordinates

These coordinates define the beginning and end points of the patch load. The patch can begin and end off the structure - the system will ensure that only those portions on the structure will form part of the actual applied load (but see the note on *Division Size*). Patch loads can only be rectangular - skewed patches are not available in this version of ACES.

Division Size

During the load distribution process (just prior to analysis) the patch is subdivided into a series of sub-patches that are assigned a width and length specified by the *Division size* parameter. If the centroids of these sub-patches are located on the structure their associated load will be applied to the model. This could, in the case of skewed structures, lead to slight over or under estimation of applied load in the nodes lying along the support lines.

The load in each sub-patch is aggregated and notionally applied to the structure at the sub-patch centroid. It is then distributed to the nearest nodes using the method of least squares.

Radius

This parameter allows curved patch loads to be applied to the structure. The patch radius is measured with respect to the patch definition points (generally the bottom edge of the patch). If left zero the system assumes a straight, parallel sided patch.

Creating the Patch Load

After clicking the button labelled ***Click here to apply changes*** the loaded patch will be displayed to scale on the model with the unit value of the loading shown in the middle of the patch. Multiple patch loads can be added to the same load case by repeating the command options described above.

Load Value Type

This parameter allows you to apply patch loads that act either across the whole width of the patch or per unit area.

Load Direction

The "direction" of the patch load must be specified when adding or changing patch loads as follows:

(a) *Patch x-axis (Tangential)*: This will apply the load parallel to the axis of the patch load if the patch is straight, or tangential to the path radius of each sub-patch if the patch is curved. Typical uses for this would be the modelling of braking or acceleration forces. See also the note below on the sign convention adopted for patch loads applied in the x-axis (horizontal) direction.

(b) *Patch y-axis (Radial)*: This will apply the load at right angles to the axis of the patch load if the patch is straight, or radial to each sub-patch if the patch is curved. Typical uses for this would be the modelling of centrifugal force.

(c) *Global Z-axis (Vertical)*: This is the default loading direction. If you read in model files that contain patches created in previous versions of ACES the orientation of all patche loads should default to this direction.

To display the direction set for each patch, click on the *Display load vector* item in the *Load display options* dialog box.

***2.1.1 Sign convention for horizontal patch loads*:**

Face the direction in which the patch extends on the model (ie from its start coordinates to its end coordinates). The positive y-direction will then be to your left and the negative y-direction to your right (i.e. the convention follows the right hand rule). You can check the direction that ACES has used by selecting <*Results*><*Applied Loadings*> but remember that the patch loads are applied as nodal loads in global directions.

In situations where a patch, (or sub-patch in the case of a curved patch load), is not parallel to either the global-X or global-Y axis, then two nodal forces are generated which, when added vectorially, equal the intended load.

***WARNING! If you create models with patch types (a) or (b) above, they will not be read correctly by previous versions of ACES*.**

2.2 Deleting a Patch Load

To delete a patch load, select ***Loads/Delete loadings/Patch loads*** and click onto the patch you would like to delete. Note that for circular curves you must place the cursor in the vicinity of the origin of the patch (generally at the bottom tip of the left side of the curve) in order to delete it **-** *not* at the apex of the curve!

2.3 Changing a Patch Load

To change a patch load click onto *Loads/Apply static loads/Patch loads/Change* and click onto the required patch with the cursor crosshairs. Edit the patch parameters then click the button labelled ***Click here to apply changes***.

2.4 Adding Multiple Patch Loads

Multiple patch loads can be added to the model by repeating the steps described in Section 2.1 or by selecting the appropriate command string from the ***Shortcuts*** menu option. Patch loads can be superimposed over one another if necessary.

2.5 Adding AUSTROADS Lane Loads

This is done in the same way as applying a vehicle load viz. **Select *Loads / Apply vehicle loads / AUSTROADS ..*** then choose the type of lane load you require. A dialogue panel will appear with a field for entering the lane load value (expressed as a force/unit length applied over the full width of the lane). A note is also displayed indicating that the patch loads will be applied along the entire length of every defined lane and that they will be modified by the lane load reduction factors (refer to *PART 2.4* for lane reduction factor details).

When ACES generates Austroads lane loads, the loading is not applied as one long, continuous patch whose length is equal to the lane length. Rather, each patch is subdivided into segments equal to the span length that the lane traverses. Consequently, it becomes possible to quickly generate various patterns of alternating loaded lanes by deleting unwanted segments of patch loads.

3.0 ADDING MOVING VEHICLES TO PATCH LOADS

Although moving vehicles can be applied concurrently with patch loads to simulate complex lane load conditions, it is advisable in very large models not to do so (refer to *PART 3.1* for a discussion on this issue). However, any other static loads in any numbers can be safely applied.

To apply moving vehicle loads to a patch load refer to *PART 3.2*

PART 3.4 Nodal Loads

1.0 APPLYING NODAL LOADS

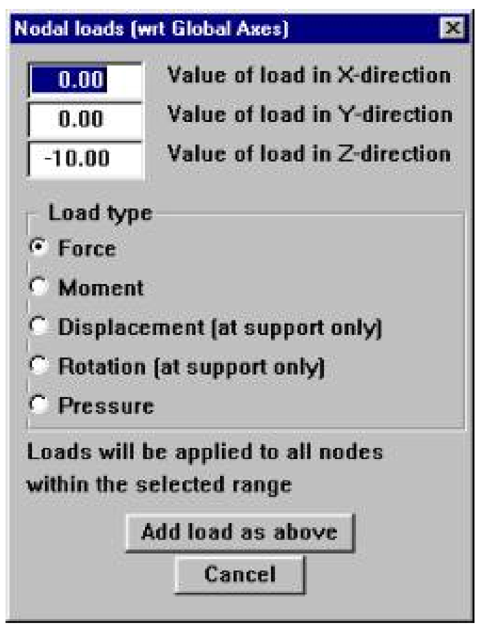
Nodal loading options cascade down to separate menus that contain comprehensive facilities for applying either single instances of such loads or multiple blocks of loads. Multiple static nodal loads can be included in the same load case as moving vehicles and patch loads.

Note that for bridge structures modelled as 2D grillages these loads will generally be acting downwards (or into the plane of the screen) i.e., in the *negative* ***Z*** direction. Therefore negative values should be specified for nodal loads unless you specifically require the load to be acting upwards.

To add nodal loads to a load case select in turn ***Loads/Apply static loads/Add nodal loads (constant)/ To a ..*** and choose a range of nodes from the pop-up menu.

For example, if you wish to add a concentrated load to only a few, widely separated points on the model, select ***..single node***. If the target nodes all lie on the same line in a continuous sequence select either ***.. a full line of nodes*** or ***.. a partial line of nodes***.

In all cases you will be prompted to identify the target node or range of nodes, after which a dialogue box as shown below will be displayed. It will allow you to select the load type, direction of action and load magnitude. Note that all nodal loads must be applied with respect to the *Global* axis system.



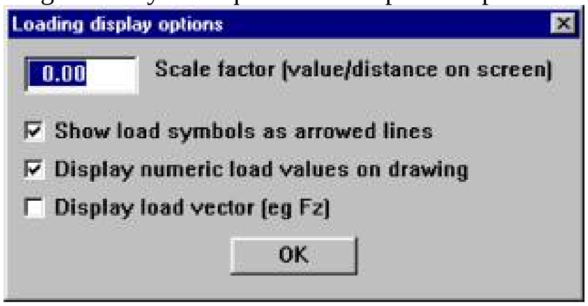
If, for example, you wish to add a concentrated load normal to the plane of a 2D grillage model, set the load type to *Force* and change the value of the load in the Z direction to, say, *-150* (kN). Now click the button ***Add load as above.*** The load as specified in the dialogue box will be applied to the previously selected range of nodes.

You will then be prompted to choose another node or range of nodes. This procedure can be repeated as often as required or terminated by clicking the right mouse button (or selecting another function from the menu or tool bars).

2.0 NODAL LOAD OPTIONS

2.1 Load Display Options

To better view a graphical representation of nodal loads, select ***Loads/Load display options*** and change the settings to suit your requirements as per the options in the panel viz:



If all check boxes in the above panel are ticked each nodal load would be represented by an arrowed line and its corresponding value and vector identifier (e.g. *[Fz] 150.*)

To improve the viewing angle of the loading diagrams choose a different orientation or perspective for the model e.g. either click the ***Isometric view*** button on the tool bar or select ***View/..*** from the menu bar and choose another viewing angle.

2.2 Deleting Nodal Loads

To delete a nodal load select ***Loads/Delete loadings/Nodal loads/From..*** and choose the range of nodes from which loads are to be deleted. Note that *all* loads will be deleted from the selected range of nodes.

2.3 Changing Nodal Loads

It is not possible to change or modify the value of a nodal load by simply clicking on a load vector and changing its value. The process must be done in two steps; viz:

You must first delete that loading from the node(s) in question then reapply it with a new value(s)

PART 3.5 Member Loads

1.0 INTRODUCTION

Member loading options cascade down to separate menus that contain comprehensive facilities for applying either single instances of such loads or multiple blocks of loads. Multiple static member loads can be included in the same load case as moving vehicles and patch loads.

Note that for bridge structures modelled as 2D grillages these loads will generally be acting downwards (or into the plane of the screen) i.e., in the *negative* ***Z*** direction. Therefore negative values should be specified for member loads unless you specifically require the load to be acting upwards.

2.0 APPLYING MEMBER LOADS

Select in turn ***Loads/Apply static loads/Add member loads/To ..*** and choose a range of members from the pop-up menu. Member loads can also be added to a specified list of member numbers. Each individual number, or number group, must be separated by a comma. A dash symbol (*not* the underscore) designates a contiguous group of members. e.g. 4, 5, 6, 25-35, 49

2.1 2.1 Selecting a Range of Members

Single Member: To add a *uniformly distributed load* to only a few, widely separated members on the model, select **..a *single member***.

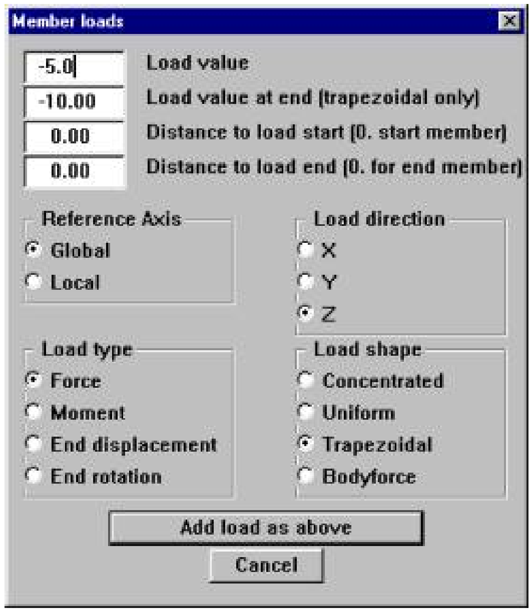
Line of Members: If the target members all lie on the same line in a continuous sequence select either ***.. a full line of members*** or ***.. a partial line of members*** or ***to all members of same property type***. The latter is particularly useful when adding dead and superimposed dead loads to main longitudinal girders and edge beams.

List of Members: Member loads can be added to a specified list of member numbers by using the option ***To a range of members***. A dialog box will appear with a single data entry field. Enter a the required list of member numbers into this field. Each individual member number, or number group, must be separated by a comma. A dash symbol (*not* the underscore) designates a contiguous group of members (e.g. 4, 5, 6, 25-35, 49).

In all cases you will be prompted to identify the target member or a range of members, after which a dialogue panel, (similar to that shown below), will be displayed. It will allow you to select the loading type, direction of action and the magnitude of the load.

Range selection for a *partial line* of members is accomplished by dragging out an elastic line between two nodes on the target line. Note that the two nodes must define the precise range required. For a *full line* of members simply click on any one member lying in the required line.

Rectangular and arbitrary areas must encompass member centroids if the specified load is to be applied to those members. Refer also to PART 2.5, *Member Properties*, for a description of the way in which the area of these ranges is defined.



Once all parameters have been entered click the button *Add load as above.* The loading specified in the dialogue box will be applied to the selected range of members and you will then be prompted to choose another range.

If adding member loads to a *full line* of members you will have the option of either: redisplaying the member loads dialogue box by left clicking on a member in another line; or, by *right* clicking on it, applying the current loads to that selected line of members immediately (without first displaying the member loads dialogue box).

To add multiple member loads to the model in the same load case this procedure can be repeated as often as required or terminated by clicking the right mouse button. Note, however, that the right mouse button cannot be used to terminate load addition to a full line of members - this can only be done by selecting another function from one of the menus or from the tool bar).

2.2 2.2 Member Load Parameters

Referring to the previous screen panel, the member load parameters are described below:

Reference Axes

Member loads can be applied with respect to the *Global* or *Local* axis system.

Load Values

For *concentrated* or *uniform* member loads enter a load value in the first field only (values in the other fields will be disregarded). *Trapezoidal* loads will require both values (refer also to Section 2.3, Trapezoidal Loads, below). The *Distance to load start* must also be provided for *concentrated* loads, otherwise ACES will assume that the load is applied at the beginning of the member.

Distance to load Start & End

For a *single* member subject to uniform or trapezoidal loading, *Distance to load start* represents the distance from the *Start* node of the member to the beginning of the applied force, while *Distance to load end* represents the distance from the *Start* node to the end of the force.

If the load is to be applied over a *line* of members, (full line or part of a line), *Distance to load start* only refers to the *first* member in the range and *Distance to load end* to the *last* member. If the load is to be applied from the beginning of the first member to the end of the last, set both of these values to zero.

To determine the *Start* of a member, select ***Settings/Show symbols/Member local axes*** and observe the direction of the local *x-axis*. It always points towards the member *End* node.

Add load as above

Once all parameters have been entered, clicking this button will cause all loadings specified in the dialogue box to be applied to the selected range of members, after-which you will be prompted to choose another range of members.

2.3 **2.3 Trapezoidal Loads**

This feature is useful for situations where the loading varies trapezoidally along a line of members (when modelling, for example, soil pressure on the sides of a culvert or wind loading on the external columns of a building). First read the comments made in the previous paragraph regarding the parameters ***Distance to load start*** and ***Distance to load end***. Note that the trapezoidal loading will be applied with the first load value placed on the lowest member number and the ***Load value at end*** placed on the larger member number.

2.4 **2.4 Prestress Loads**

Member prestress loading refers to those prestress moments that produce parasitic (secondary) effects in the structure. ACES does not determine stress profiles across the section due to applied prestress.

2.5 **2.5 Load Display Options**

To better see the graphical representation of member loads, select ***Loads/Load display options*** and change the settings to suit. To improve the viewing angle of the loading diagrams choose a different orientation or perspective for the model e.g. either click the *isometric view* button on the tool bar or select ***View/..*** from the menu bar and choose another viewing angle.

3.0 DELETING MEMBER LOADS

To delete member loads select ***Loads/Delete loadings/Member loads/From..*** and choose the range of members from which loads are to be deleted. Note that *all* member loads will be deleted from the selected range of members.

4.0 EDITING MEMBER LOADS

To change or edit an existing load case that contains member loadings, click on the loading icon (the button showing the down-arrow and the red letter ***F***) and highlight the load case you wish to edit. Press *OK*. All loadings associated with the selected load case will be graphically displayed on the model and can now be changed, added to or deleted.

TIP: To change a static member load you must first delete the existing loadings from the relevant members then reapply them with the new, required, values

PART 3.6 Element Loads

1.0 INTRODUCTION

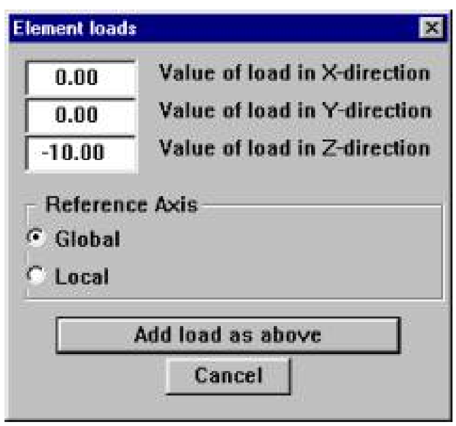
Element loading options cascade down to separate menus that contain comprehensive facilities for applying either single instances of such loads or multiple blocks of loads. Multiple static element loads can be included in the same load case as moving vehicles and patch loads.

Note that for bridge structures modelled as 2D finite elements these loads will generally be acting downwards (or into the plane of the screen) i.e., in the *negative* ***Z*** direction. Therefore negative values should be specified for element loads unless you specifically require the load to be acting upwards.

2.0 APPLYING ELEMENT LOADS

Select in turn ***Loads/Apply static loads/Add uniform element loads/To ..*** and choose a target range of elements from the pop-up menu. For example, if you wish to add a uniformly distributed load to only a few, widely separated elements on the model, select ***..a single element***. Rectangular and fenced areas must encompass element centroids if the specified load is to be applied to those elements. (Refer to PART 2.9, *Element Properties*, for a description of the way in which the range of an arbitrary area is defined).

Once a range has been identified, a dialogue panel similar to that shown below will be displayed. It will allow you to select the direction of action and the magnitude of the applied element load(s).



Load Values

Element load values must be applied in the appropriate direction. They are applied as *uniformly distributed* loads normal to the plane of the element.

Reference Axes

Element loads can be applied with respect to the *Global* or *Local* axis system.

Add load as above

Once all parameters have been entered click the button ***Add load as above.*** The loading specified in the dialogue box will be applied to the selected range of elements and you will then be prompted to choose another range of elements. To add multiple element loads to the model in the same load case this procedure can be repeated as often as required or terminated by clicking the right mouse button (or by selecting another function from the menu or tool bars).

2.1 Load Display Options

To better see the graphical representation of element loads, select *Loads/Load display options* and change the settings to suit. To improve the viewing angle of the loading diagrams choose a different orientation or perspective for the model e.g. either click the *isometric view* button on the tool bar or select *View/..* from the menu bar and choose another viewing angle.

3.0 DELETING ELEMENT LOADS

To delete element loads select ***Loads/Delete loadings/Element loads/Delete from..*** and choose the range of elements from which loads are to be deleted. Note that *all* element loads will be deleted from the selected range of elements.

4.0 EDITING ELEMENT LOADS

To change or edit an existing load case that contains element loadings, click on the loading icon (the button showing the down-arrow and the red letter ***F***) and highlight the load case you wish to edit. Press *OK*. All loadings associated with the selected load case will be graphically displayed on the model and can now be changed, added to or deleted.

TIP: To change the value of an element load you must first delete the existing loadings from the relevant elements then reapply them with the new, required, values

PART 3.7 Hydrostatic Loadings

1.0 CREATING A HYDROSTATIC LOAD

To add hydrostatic loads to tanks and walls it is first necessary to identify a node that lies on the free surface of the structure. Do this using the menu commands:

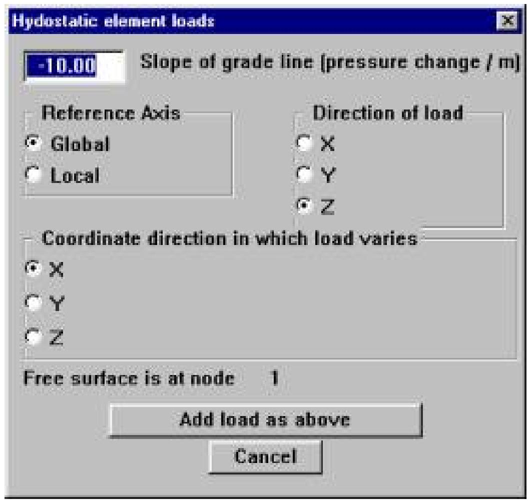
Loads / Apply static loads / Add hydrostatic element load / Select node at zero surface

Generally any node along the top edge of the tank or wall can be nominated as the free-surface node. Once this has been done, select the element range to which the hydrostatic load will be applied. For example, use the command string: ***Loads / Apply static loads / Add hydrostatic element load / To a rectangular block of elements***

You will then be asked to identify the rectangular block of elements. If hydrostatic water pressure is to be applied to the internal walls of a tank, for example, drag out a rectangular area that encompasses the entire structure. Once this has been done, a dialogue box similar to that shown below will pop up.

1.1 Hydrostatic Pressure Parameters

The dialog box shown below has a number of parameters:



Slope of grade line

This parameter defines the pressure change per unit length. It is multiplied by the distance of each element centroid from the free surface and applied as a pressure at the element centroid.

The pressure will have the same sign as the slope if the element centroid lies *below* the free surface of the structure (i.e., if it lies in the global *negative* direction from the free surface). The converse will also apply.

Reference Axis

The hydrostatic pressure may be applied relative to either axis system. In the case of parametrically generated tanks, the wall elements have all been organised with their local axes already pointing in the "outside" direction. Therefore, selecting *Local* axes with a positive pressure slope will model the internal water pressure automatically.

Direction of load

The actual direction of the applied vector depends on the reference axis selected. If the *Local* axis system is used then the *Z* direction of the load will apply a pressure normal to the element surface (i.e., in the element local ***z***-direction).

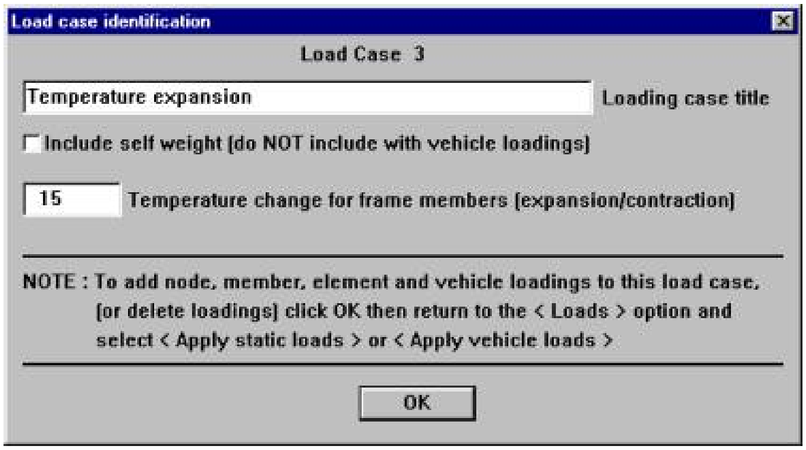
Coordinate direction in which load varies

This parameter essential identifies the orientation of the free surface plane. For the tank example used above, the free surface would be parallel to the global *X*-*Y* plane, in which case the variation of the hydrostatic load will occur in the global *Z* direction. For 2-dimensional structures the free surface is likely to be parallel to the global *X* axis and the variation will therefore be in the global *Y*-direction (e.g. when applying earth pressure to 2D culvert models).

PART 3.8 Temperature Expansion-Contraction

1.0 CREATING THE LOAD CASE

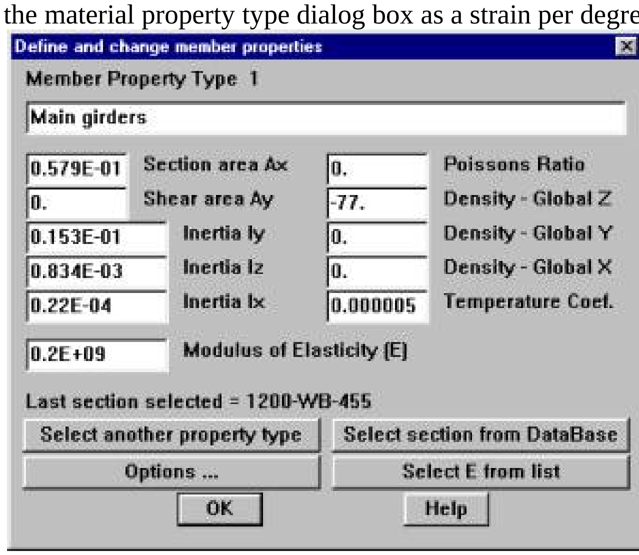
To add a temperature expansion or contraction loading, create a load case and on the load case title dialog box (shown below) enter the temperature change in degrees: ***(+)*** for expansion, ***(-)*** for contraction.



Note that this is applicable for frame and grillage members only (not recommended for FE slabs).

1.1 Temperature Coefficient

The coefficient of temperature expansion/contraction is a material property and as such must be entered in the material property type dialog box as a strain per degree rise/fall viz:



PART 3.9 Dead Load

1.0 DEAD LOAD (SELF-WEIGHT)

Select ***Loads/Add-Edit Load Case..*** or click the loading icon. If this is the *first* load case you are creating (generally recommended), the dialog box shown below will appear. Enter a load case title then tick the box labelled ***Include self weight ...***

Dead load is normally specified as a separate load case and should *not* be included as part of a moving vehicle load. Note that ACES makes no allowance for the "doubling" effect of self-weight at member intersection points. This may, or may not, be an important factor in deciding whether to use this option.

Alternatively, you may wish to model dead load effects using strings of member loads (*Part 3.5*). Refer also to *PART 3.1, Creating Load Cases*, for further discussion of this issue, particularly as it relates to concurrent application with moving loads.

This page intentionally left blank

PART 4

ANALYSIS

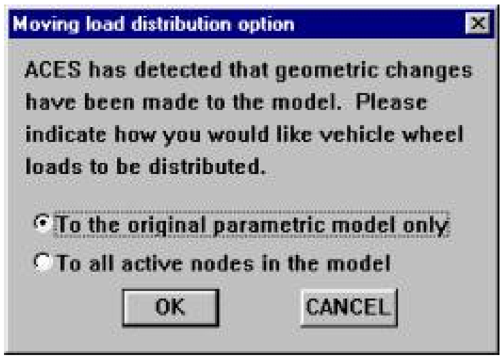
PART 4.1 Analysis Options

1.0 INTRODUCTION

If the base 2D parametric model has not been modified to 3D and there are no options to be set, simply select ***Analyse / Begin normal analysis*** to solve the problem. This holds true for all structure types (see sections 1.2 – 1.4). Alternatively, click the button with the red lightning symbol to initiate analysis.

1.1 Distributing Wheel Loads to a Modified Parametric Model

If geometric changes have been made to the structure (for example, by deleting some of the parametrically generated members and supports and adding new members and support nodes), *and* the model contains one or more moving vehicle load cases, the system will issue a warning regarding the manner in which vehicle wheel loads are to be distributed to nodes viz:



You may elect to have the wheel loads distributed to the original, parametrically-generated, model or to all *active* nodes in the modified structure. The method you choose will depend entirely on whether it is important that vehicle loads be distributed to the modified parts of the structure.

If, for example, a sub-structure, (such as piers), has been added to a grillage model representing a bridge deck, then providing no new nodes were created in the deck the vehicle wheel loads can be distributed to the original parametric model. However, if the piers were connected to transverse diaphragms at nodes that were newly created for this purpose, then the distribution should be to all active nodes in the deck. In some cases it may be necessary to de-activate parts of the structure in order that load distribution does not occur to them.

1.2 Frame & Beam Analysis

If no changes have been made to the original parametric model then simply click onto the ***Analysis*** icon (unless you wish to renumber parts of the model – refer to *Section 2.2* here-in).

If the original 2D frame model is modified with the addition of Z coordinates it will need to be set to a 3D space-frame model prior to performing the analysis. Refer to Section 2.1, *Analysis Type*.

If moving vehicle load cases have been defined, ACES may also display a warning message regarding the way in which wheel loads will be distributed to nodes (refer to *Section 1.1* above). Providing no new nodes were created in the deck, the vehicle wheel loads can be distributed to the original parametric model. However, if the piers are connected to transverse diaphragms at nodes that were newly created for this purpose, then the distribution should be to all active nodes in the deck. In some cases it may be necessary to de-activate parts of the structure in order that load distribution does not occur to them.

1.3 Grillage Analysis

If no changes have been made to the original parametric model then simply click onto the ***Analysis*** icon (unless you wish to renumber parts of the model – refer to *Section 2.2* here-in).

If a sub or super-structure, (e.g. piers), has been added to a 2D grillage model representing a bridge deck, then it will need to be set to a 3D space-frame model prior to performing the analysis. Refer to Section 2.1, *Analysis Type*.

If moving vehicle load cases have been defined, ACES may also display a warning message regarding the way in which wheel loads will be distributed to nodes (refer to *Section 1.1* above). Providing no new nodes were created in the deck, the vehicle wheel loads can be distributed to the original parametric model. However, if the piers are connected to transverse diaphragms at nodes that were newly created for this purpose, then the distribution should be to all active nodes in the deck. In some cases it may be necessary to de-activate parts of the structure in order that load distribution does not occur to them.

1.4 Finite Element Analysis

If no changes have been made to the original parametric model then simply click onto the ***Analysis*** icon (unless you wish to renumber parts of the model – refer to *Section 2.2* here-in).

Element Shears

Prior to performing the analysis ACES will ask if element out-of-plane shear forces will be required. If you reply affirmatively, they will be calculated. However, the analysis will take longer and the resultant output files will be considerably larger (in particular *ACES.OUT*).

If a sub or super-structure, (e.g. piers), has been added to a 2D FE slab model representing a bridge deck, then it will need to be set to a 3D shell type model prior to performing the analysis. Refer to Section 2.1, *Analysis Type*.

If moving vehicle load cases have been defined, ACES may also display a warning message regarding the way in which wheel loads will be distributed to nodes (refer to *Section 1.1* above). Providing no new nodes were created in the deck, the vehicle wheel loads can be distributed to the original parametric model. However, if the piers are connected to transverse diaphragms at nodes that were newly created for this purpose, then the distribution should be to all active nodes in the deck. In some cases it may be necessary to de-activate parts of the structure in order that load distribution does not occur to them.

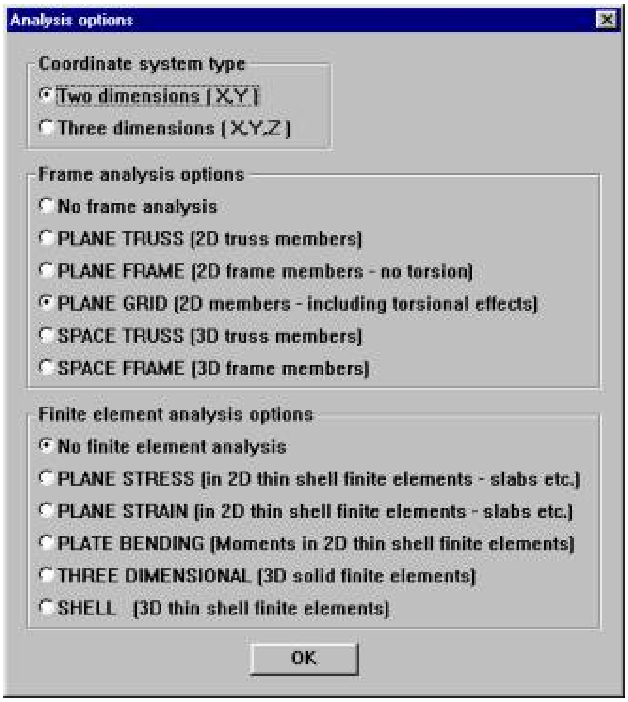
2.0 ANALYSIS OPTIONS

2.1 Analysis Type

The facilities in this menu option need only be accessed if the basic geometric model is changed from its original parametrically generated form. Examples of this include:

- Out-of plane supports or loads are applied to a 2D slab or grillage model
- A sub- or super-structure is added to a 2D parametric model
- Finite elements are introduced into a 2D/3D frame model

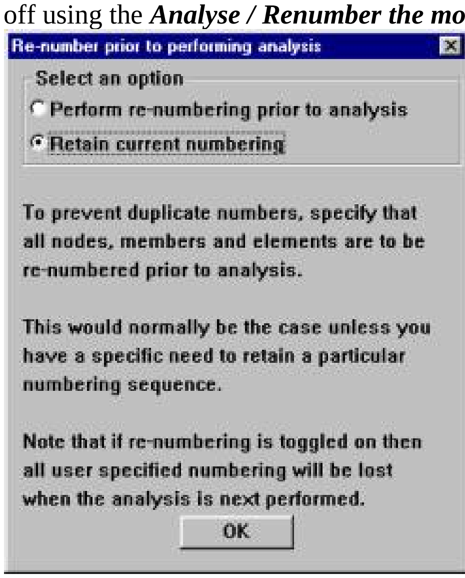
In such cases you will have to set the appropriate coordinate system and structural type using this option.



If a sub or super-structure is added to a 2D parametric model it will be transformed into a 3D space frame etc.

2.2 Renumbering the Model

By default ACES will retain the current node, element and member numbering system unless this action is switched off using the ***Analyse / Renumber the model*** option.



WARNING: *If the numbering system is retained, no check will be performed on duplicate node and member numbers, resulting in possible errors or run-time termination of the analysis*

If you do select the renumbering option then any new node or member numbers that may have been created by you will be renumbered and your sequences will be lost.

3.0 ERROR REPORTING

By default all input data and errors encountered during the analysis are written to the text file *ACES.OUT* in the *../Outpdata* folder. If the solution is error-free, model data and results for all loadings, including *every* position of every vehicle in all moving load cases, will be written to a binary output file labelled *ACES.PLT*.

If an error is encountered during data validation or analysis, ACES will beep and display a diagnostic message. In the event that the on-screen error message is not helpful in isolating the problem you may need to consult the *ACES.OUT* file for more detailed information. This can be done by selecting ***Results / View current output file*** and scrolling through it until the error message is reached.

PART 4.5 Dynamic Analysis

1.0 INTRODUCTION

The Dynamic (or Vibration) Analysis Module is used to determine the natural modes and frequencies of vibration of structures. It will calculate and display the deflected shape vectors for the number of modes requested, either as a static maximum-displacement diagram or in animated form. These vectors are pre-multiplied so that the largest value is 1.

The Dynamic Analysis Module determines the natural modes and frequencies of vibration of the structure assuming that all structural mass is located at the nodes. The more finely the structure is divided, the more closely the result will approach the theoretical value.

Extra lumped masses which do not contribute to the structural stiffness may be simulated using nodal loads (refer to *PART 3.4* for instruction in applying nodal loads).

The calculation of natural frequencies is based on the method detailed in WANG, C.K., "*Computer Methods in Advanced Structural Analysis*". *Section 4* summarises the theory.

1.1 Limitations

Due to the way in which the theoretical methodology has been implemented within ACES, the dynamics module has an inherent limitation in the model size that can be analysed. The total number of nodes in the model must not exceed:

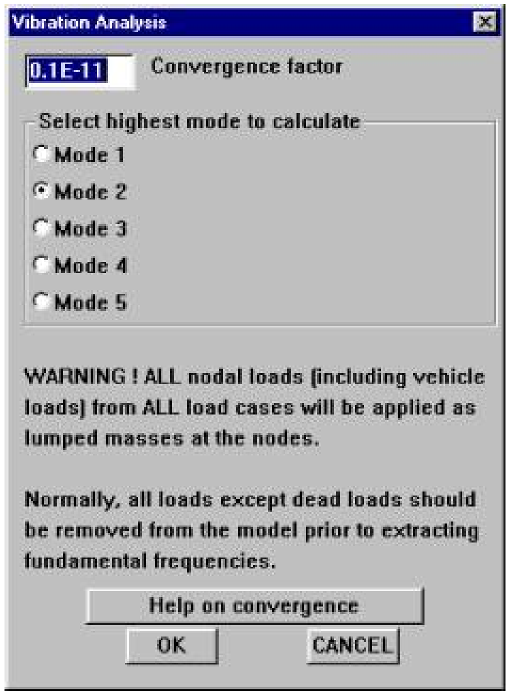
455 *For space trusses*
225 *Space frames and finite element shell structures*
500 *All other structure types*

If the model you wish to analyse exceeds these limits you will need to simplify it for the purpose of obtaining the natural frequencies. In most cases this will not produce significant error, particularly if small structural detailing is excluded.

WARNING: All vehicle load cases should be deleted prior to performing the analysis, otherwise the nodal loads from all vehicle loadings will be applied as lumped masses to the model!

2.0 DYNAMIC ANALYSIS PARAMETERS

To determine the fundamental frequencies and modes of vibration of the structural model click on ***Analyse / Extract frequencies.*** The following dialog box will be displayed:



2.1 Modes & Frequencies

Select the highest mode you require – all frequencies up to, and including, that mode will be calculated. Note the warning concerning the exclusion of vehicle loads! Normally all loads except dead loads should be removed from the model prior to extracting fundamental frequencies. Otherwise all loads from all load cases will be applied as lumped masses at the nodes.

2.2 Convergence

When extracting fundamental frequencies, ACES uses an iterative process controlled by the convergence factor specified in the dialog box. If calculations have not converged within a reasonable number of cycles, abort the process by pressing the ESC key and try a higher convergence factor (.00001 for example).

Then gradually decrease the convergence factor until the extracted frequencies seem independent of the chosen factor. Note that selecting a convergence factor less than the default (1.0E-12) will result in *reduced* accuracy, particularly in the higher modes.

2.3 Units

Units must be consistent. The length unit must be either: ***m, mm, ft,*** or ***in*** and the force unit used for the elastic constant E must be the same as the force unit used for the nodal loads and body-forces (densities). Dynamic analysis will terminate if the length unit specified is not one of the above. Units may be set via the ***Structure / Set units*** menu options.

Preset values for the gravity constant for these units are contained in the Dynamics Module. Consistency of units is required because the Dynamics Module generates a mass matrix by dividing the nodal loads and body-forces by the gravitational constant applicable for the defined length unit. This approach has been adopted so that the user is not required to mix force and mass units in the input.

The advantage of this system is that the actual force unit used becomes irrelevant. For example, the user could specify the E value in terms of kilograms per square metre, the nodal loads in kilograms and the body-forces in kilograms per cubic metre and get the same result as if the E value was in kiloNewtons per square metre, the nodal loads in kiloNewtons and the body-force in kiloNewtons per cubic metre.

Users should be aware that this approach *fails* if the structure is being designed for an environment where the gravitational constant is not the same as that on the surface of the earth (such as on the Moon). In such cases the nodal loads and body-forces used in the general solution module must be those applicable to the design environment, whereas the nodal loads and body-forces used in the dynamics module must be those applicable at the surface of the Earth.

Note that the ***vibrational*** characteristics of a structure are independent of the gravitational environment. For example a beam at the surface of the earth will vibrate at the same frequency if placed on the Moon, although its self-weight deflection will be roughly one sixth.

2.4 *Body-Forces* (Structural Mass)

The structural mass is applied at the nodes by calculating the member and element body-force contribution at each node (using the ***BODYFORCE*** or ***Density*** values for the members and/or elements ***BFX, BFY, BFZ***) and then dividing by the gravitational constant applicable for the defined length unit.

Body-forces (or densities) ***BFX, BFY*** and ***BFZ*** are defined in the member and element properties dialog boxes. Body forces applied under individual loadings as ***BODYFORCE*** are ignored.

The user may restrict the vibration analysis to one or more global degrees of freedom by specifying only the relevant body-force. For example only vibration in the *Global-Y* direction will be produced if only the ***BFY*** (Y-density) value is specified and the others are set to zero.

2.5 *Nodal Loads (Lumped Mass)*

Extra lumped forces not associated with the structural mass may be applied by specifying adding nodal loads (see *PART 3.4*). Nodal loads are converted to mass by dividing by the gravitational constant as described in previously.

Such things as the effect of machinery bases or the placement of extra mass to control vibration may be examined by this method. ***Note that all nodal loads in all loadings are included as lumped masses.***

The direction of the applied mass is assumed to act in the direction of the nodal load. If, for example, a machinery base is supported in the *Global-Y* direction but free to move in the *Global-X* direction, it would be modelled as a ***Nodal Load Force Y*** (refer to *EXAMPLE 5*).

3.0 THEORETICAL BASIS

Hookes Law states that the force required to extend a spring is given by:

$$F = ks \qquad (1.1)$$

where: F is the force
k is the spring constant
s is the extension.

Similarly, the classical elastic representation of a structure requires that it be described as a series of linked nodes which react to applied loads according to:

$$[Pd] = [K][x] \qquad (1.2)$$

where: [Pd] is the load vector
[K] is the stiffness matrix
[x] is the displacement vector

The load vector and displacement vector contain entries for each degree of freedom at each node and are single dimension arrays of order:

(Number of nodes) x (Number of degrees of freedom)

The stiffness matrix is a two dimensional array of the same order for both dimensions. It is now necessary to imagine the structure as consisting of masses (derived from the properties of the members and elements) attached to each node and being in some sort of motion relative to each other.

How this motion is initiated and the effect of damping is irrelevant to the calculation of the natural frequencies but it can be proved [1] that this motion is cyclic or vibratory in nature with respect to a definite time period.

Newton's second law of motion states that the force required to accelerate a mass is given by the product of the mass times the acceleration. This can be applied to the previously defined structure by expressing in matrix notation:

[Pa] = [M][ddx] ***(1.3)***

where: [Pa] is the load vector
[M] is the mass vector
[ddx] is the acceleration

Note that **ddx** is the double differential with respect to time of the nodal displacement **dx**. Note also that the mass vector is a single dimension array of the same order as the load and displacement vectors.

Given that the structure is in a state of dynamic equilibrium, (i.e., no external forces of a time-dependent nature are being applied), the load vector due to structural deformation **[Pd]** must be equal to the load vector due to nodal acceleration **[Pa]** and therefore:

[K][x] = -[M][ddx] ***(1.4)***

If it is assumed that deformations are a cosine function, (true for small deformations), use can be made of the fact that the double derivative of a cosine function is a linear multiple of the function itself and Equation 1.4 may be re-written as:

[K][A] = pp[M][A] ***(1.5)***

where: [A] is the nodal displacements
pp is the frequency squared.

The problem of calculating natural frequencies is therefore reduced to that of finding the displacement vector **[A]** which, when multiplied by the stiffness matrix **[K],** gives the same result as multiplying the frequency function **pp** by the mass matrix **[M]**. This is the classical eigenvalue problem, where the set of displacement vectors **[A]** satisfying Equation 1.5 are called the eigenvectors.

It can be shown that the number of possible eigenvectors (or modes of vibration) is equal to the order of the displacement vector i.e. equal to the total number of degrees of freedom in the structure.

For all but the most trivial of examples, it is impractical to solve for the eigenvectors directly. Consequently ACES uses the *Power Iteration Method* [see *Reference 1*] to determine them. Note that for numerical reasons, ACES operates on the flexibility matrix (the inverse of the stiffness matrix).

3.1 Analytical Process

The module uses the *Power Iteration* [see *Reference 1*] technique to determine the various modes of vibration. The iteration process for each mode will continue until the difference between successive iterations is less than the prescribed tolerance. By specifying a low tolerance (say 1.E-3) the iterative process will be faster but less accurate.

After every 100 iterations the module displays the current cumulative number of iterations. Pressing the ***ESC*** key will terminate the iterative process and initiate the graphical output module.

The analysis procedure generally takes the following course:

(a) Data Validation

(b) Assembly of mass matrix

From the member/element linkages and body-forces (and applied ***NODAL LOADS*** if any), ACES determines the force applied in each displacement direction at each node. The mass matrix is then generated by dividing by the gravity constant applicable for the specified length unit (eg 9.8 for metres).

(c) Assembly of stiffness matrix

(d) Assembly of unit force matrix

This is a unit matrix of order :

Number of Nodes x Number of degrees of Freedom

The matrix is treated as a set of loadings by the solution module and contains the flexibility matrix at the completion of the solution phase.

(e) Determination of the flexibilty matrix

Solve by using the modified *Gaussian Elimination* technique.

(f) Generation of the dynamic matrix

The mass matrix is multiplied by the flexibility matrix.

(g) Determination of the current vibration mode

The *Power Iteration* technique, [*Reference 1*], is used to accomplish this. ACES starts with a unit displacement vector and multiplies it by the dynamic matrix. This gives another displacement vector. The new displacement vector is checked against the original vector. If all values are within predefined tolerances the mode shape vector has been found and the frequency can be calculated. If not, then the new vector is substituted for the old and multiplied by the dynamic matrix and checked again.

The frequency is calculated from the mode shape vector and the deflected shape vector is determined by multiplying the mass vector by the flexibility matrix and the mode shape vector.

(h) Printing results

Frequencies and unitised forcing vectors and displaced shape vectors are printed.

(i) Reduction of the dynamic matrix

Knowing the frequency, the last determined mode shape is stripped from the dynamic matrix to produce a new dynamic matrix from which the next mode shape may be determined.

(j) Iterate if more modes required - repeat from step (g)

3.2 Reference

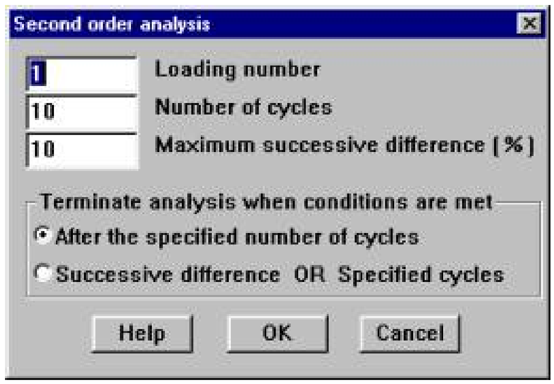
[1] WANG, C.K., "*Computer Methods in Advanced Structural Analysis*"

PART 4.6 Second Order Analysis

1.0 SECOND ORDER PARAMETERS

This option enables a Second Order Analysis to be performed on a *single load case* only. A moving vehicle load case cannot be selected since it incorporates many potential vehicle loading positions. However, a single vehicle *loading* (i.e. the vehicle(s) in that load case considered at a fixed position on the structure), can be investigated as a separate static load case.

To perform an analysis select ***Analyse / Second order analysis***. The following dialog box will appear:



Accuracy of the solution process can be controlled by specifying either the number of cycles (iterations) that ACES should perform or, alternatively, the convergence error of global structure deformation. The latter is expressed as a percentage difference between successive iterations.

WARNING: Second Order Analysis is not applicable to 2D slab, grillage and plate structures where all loads are normal to the model plane.

2.0 SOLUTION PROCESS

ACES uses an iterative process when performing a second order analysis. It analyses the selected load case then updates the node coordinates with the calculated deflections and re-analyses the structure. This process will continue for the specified number of cycles or until the difference between deflections of successive iterations is less than the specified tolerance. Therefore, as the loaded structure deforms, the loads applied to nodes and members are also displaced. Consequently the stiffness of the members and/or elements may change from cycle to cycle.

This procedure has been found to give results that are very close to the theoretical Euler buckling load for columns and frames and is able to preict if the model ultimately becomes unstable. It is not, however, suitable for automatically analysing moving vehicle loads. Each vehicle position would have to be investigated as a separate static load case.

Note carefully that the solution algorithm is based on the assumption that deflections have no effect on the member/ element stiffnesses. Hence great care should therefore be exercised in modelling structures where large deformations are occurring.

2.1 Summary of the Solution Process:

The solution process may be summarized as follows :

a) a) Validate the data.

b) b) Analyse the problem and calculate the deflection for the nominated load case. Modify the stiffness matrix on successive iterations as the size of the element/member changes when nodes move closer or further apart. If the distortion causes a rectangular element to become non-rectangular, the analysis will stop.

c) c) If a convergence factor has been specified, check the maximum current deflection with that of the previous cycle.

d) d) If the maximum deflection is within the specified convergence OR if the specified number of cycles has been reached then print the results and stop processing. (Otherwise continuing with the next cycle).

e) e) Validate the data as in (a) but modify the node coordinates by the deflections calculated in (b). The modified coordinates are display in the output file (*ACES.OUT*) along with the original node coordinates. Return to step (b).

The second order procedure is fully automated, the analysis iterating until either the nominated number of cycles has been reached or the convergence, (if given), has dropped below the specified limit. In models where buckling is occurring the deflections tend to increase, rather than decrease, from cycle to cycle.

Note that the deflections are not accumulated as the cycles proceed. In other words the base from which the node coordinates are modified remains the originally input values. Only the results from the final cycle are printed out and available for graphical display. The results for intermediate cycles are not saved nor printed.

This page intentionally left blank

PART 5

RESULTS & REPORTS

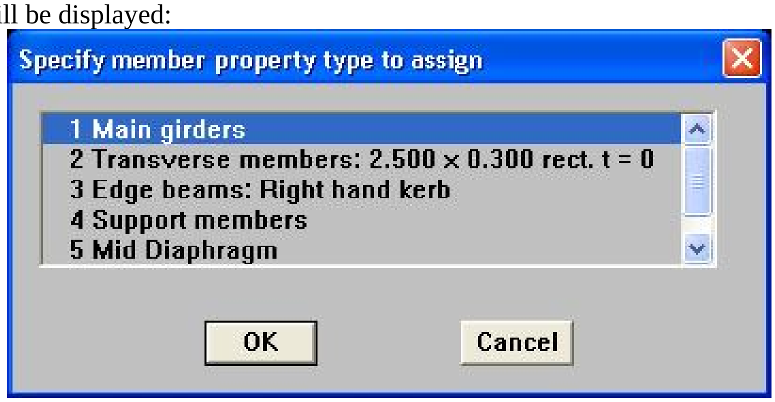
PART 5.1 Graphical Results – Frames & Grillages

1.0 DISPLAYING RESULTS DIAGRAMS

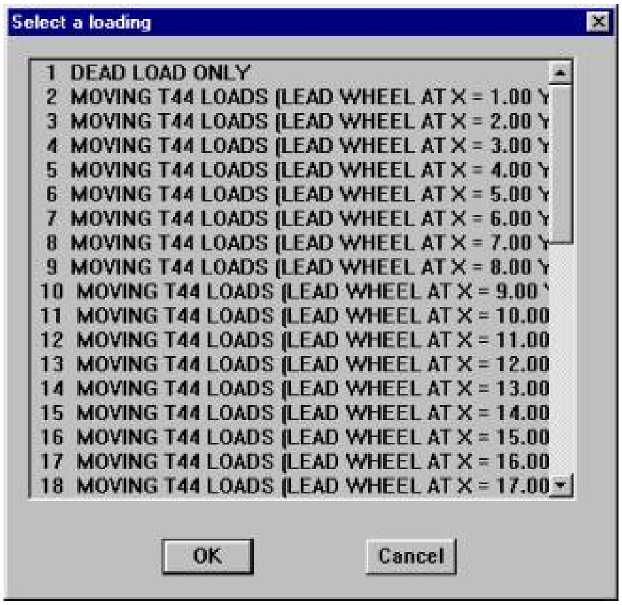
1.1 How to Display a Results Diagram

The procedure for displaying and printing any type of results diagram in graphical form is the same no matter what type of result vector is selected. It applies equally to moments, shears, torsions, reactions, displacements and curvatures and to all forms of output - envelopes, combinations and single load cases. The steps may be summarised as follows:

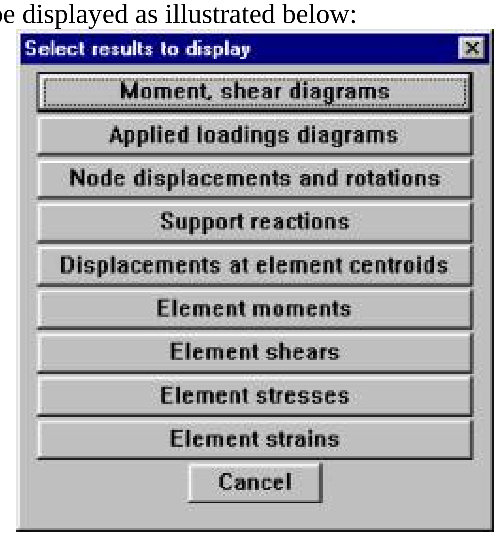
- • Click, in turn, on ***Results/Select a range of members/..*** then choose the range of members for which results are required from the pop-up menu e.g. for all main longitudinal girders select ***Members of a specified property type***. If this option is selected the following dialog box will be displayed:



- • Select the required member range (e.g. *Main Girders*). Click *OK*.
- • Now select the loading effect you require:
  - ⇒ ⇒ For an envelope click *Results / Envelope*
  - ⇒ ⇒ For single load case click *Results / Load case*
  - ⇒ ⇒ For a combination click *Results / Combination*
- • If you wish to graphically interrogate the results for a single load case select that option. The dialog box shown below will be displayed (refer to PART 5.3 for more information on envelopes).



- • Scroll down to the required load case, highlight it, then click *OK*. A submenu of results options will be displayed as illustrated below:



- • Select the type of results you require:
  - ⇒ ⇒ bending moments, shears & torsional moments
  - ⇒ ⇒ reactions
  - ⇒ ⇒ node displacements & rotations
  - ⇒ ⇒ loading diagrams
  - ⇒ ⇒ node contour diagrams

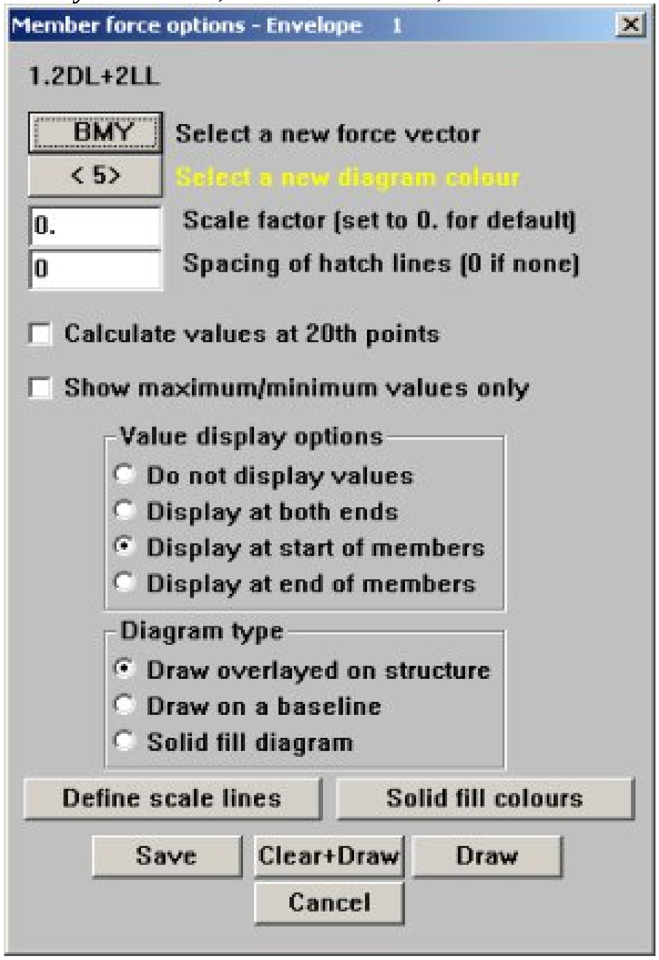
Selecting results type from the main menu

If a member range and load case/envelope have already been selected and you wish to view other result types, an alternative method for doing this is to click on the *Results* item in the main menu bar then select the required results option from the pop-up menu.

1.2 Moment, Torsion and Shear Diagrams

Refer to *Section 1.1* above for a description of the process in getting to this point. Once you have reached this stage, a results dialog panel as shown below will appear. Enter values as appropriate then click ***Draw***. To clear the current diagram before drawing the new diagram, select ***Clear***.

To change any of the diagram attributes after the diagram has been drawn click the icon to the right of the ***Analyse*** button or, from the menu bar, select ***Results / Moments, shear diagrams***.



Select a new force vector

This option enables you to select the required results vector by clicking onto the ***Select a new force vector*** button. For 2D grillage models there will only be three valid force vectors to choose from **-** bending moment *BMY*, shear force *SFZ* or torsional moment *TOR*. For 2D beams and frames there will also be three **-** bending moment**-***Z*, shear force**-***Y* or Axial force.

Colour and line style

Click this button to select the required results vector colour and line style (thick, thin, dashed).

Scale factor

Enter a scale factor (if set to zero ACES will calculate a value). To make the diagram larger decrease the value. For example, to double the height of the diagram, halve the current scale value.

Spacing of hatch lines

This parameter is used to create hatched force diagrams. If left zero the diagram will not be hatched. Otherwise, enter a hatch spacing (in *mm*).

Calculate values at 20th points

This parameter is used to create smooth results curves. Normally ACES plots the values at the nodes then joins those points with straight lines. This can lead to misleading results diagrams, particularly in situations where there might be static loads applied along individual members that have not been divided into sub-members.

If absolute accuracy is required in depicting the resultant vector diagrams, place a tick in this box. ACES will then calculate and plot the true value at 20th points along each member. Note that this is generally not required for moving vehicle loads, since individual wheel loads are always distributed to adjacent member *nodes.*

One of the disadvantages of displaying vector diagrams calculated at 20th points is that *numeric* values are only displayed at *nodes* (i.e. at member ends). So although the member force diagram itself will be correctly shown (such as a parabola, for example), any peak values occuring within the member will not be displayed (unless one or more nodes has been inserted along it).

Value display options

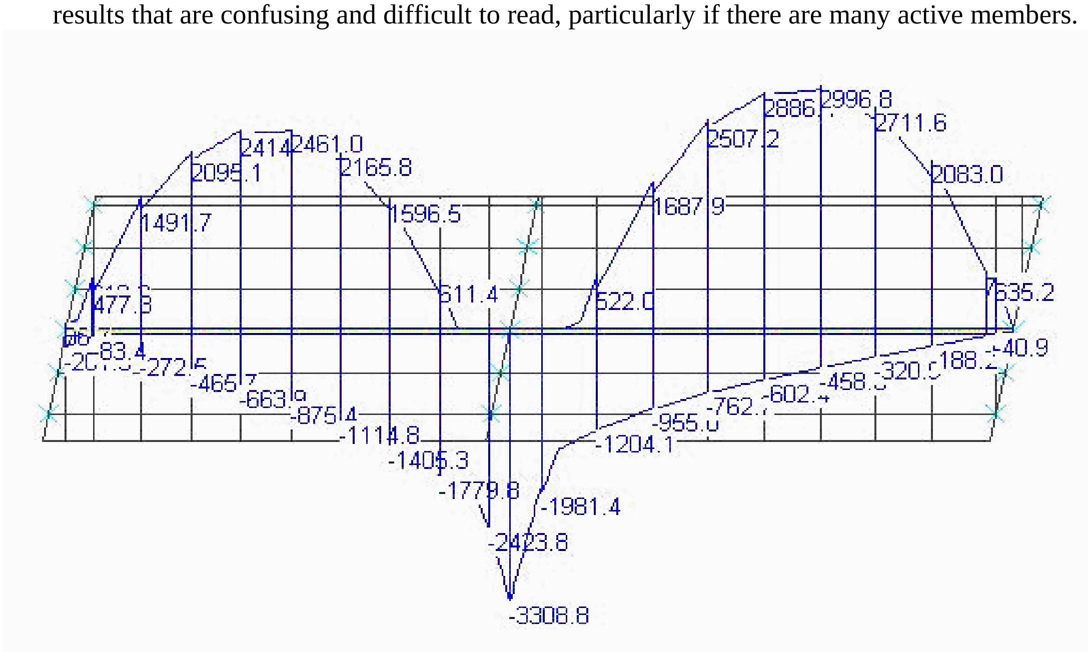
This option allows you to determine whether numeric results are to be displayed on the graphical diagram. Often you may not require the actual values to be displayed e.g., in situations where a simple comparison of the resultant force curves for a range of members. In this case, select the first option. The second option will display values for the selected vector at the beginning and end of every member in the currently active range. Depending on the complexity of the geometry and the selected scale this could lead to over-lap of values at each node, since the value at the end of one member may be superimposed onto the value at the beginning of the next. The last two options enable this problem to be circumvented.

Diagram type

This option enables one of three diagram types to be displayed:

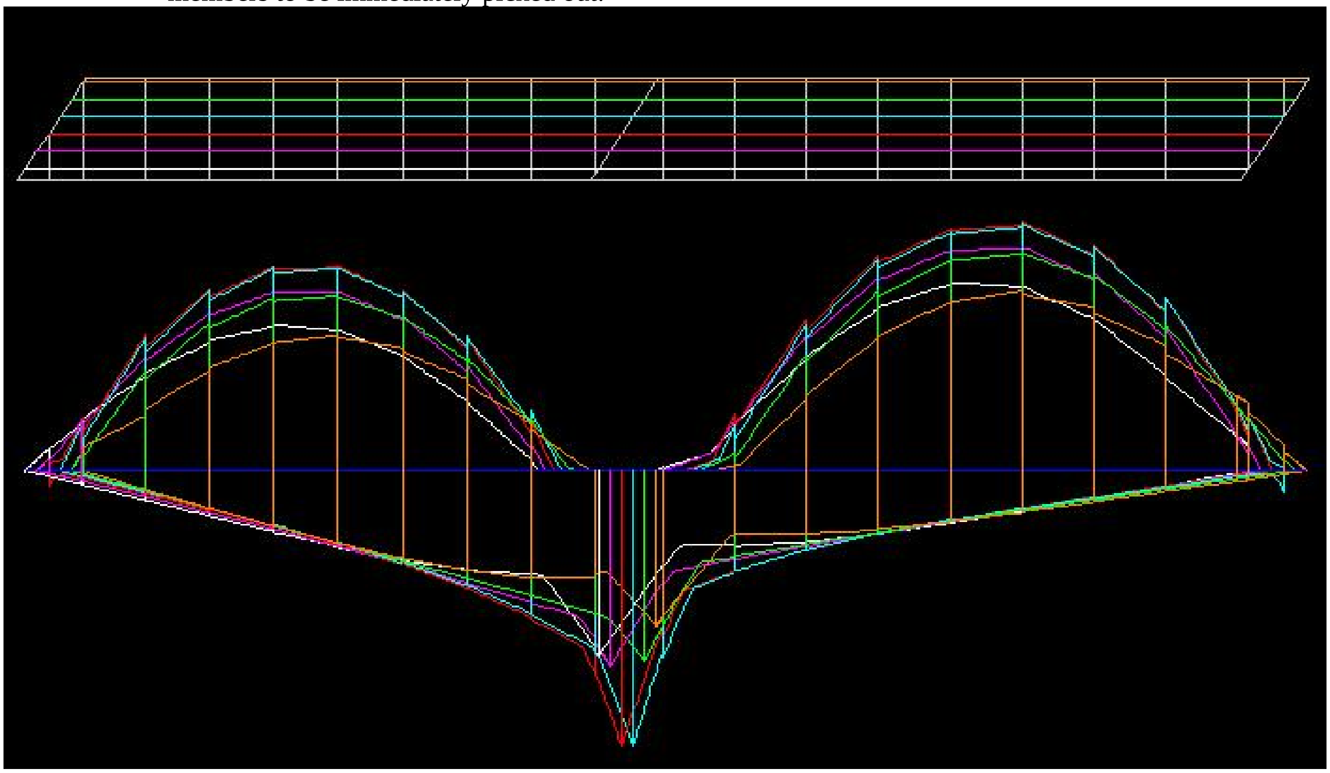
(a) Overlayed

The diagram is overlayed onto the base model as shown in the example below. This may produce results that are confusing and difficult to read, particularly if there are many active members.



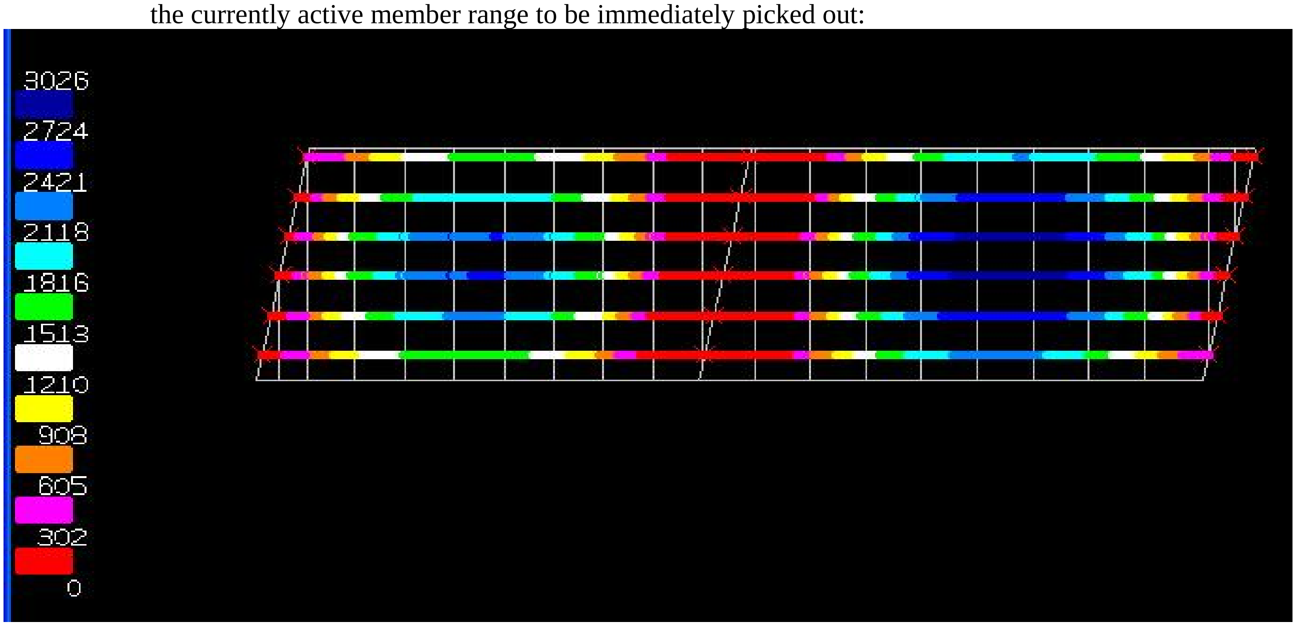
(b) Baseline

The vector diagram for every member in the currently active range is drawn on a common base-line. Each member is colour-coded to its respective diagram. This allows the worst affected members to be immediately picked out:



(c) Solid fill diagram

The maximum-minimum results spectrum for the load case (or envelope/combination) is sub-divided into 10 equal divisions. The results along each member are then drawn as solid fill lines that are colour-coded to their respective scaled division. This allows the worst affected areas in the currently active member range to be immediately picked out:



Solid Fill Colours

Click this button to change the solid fill colours used in the solid fill diagram above.

Define scale lines

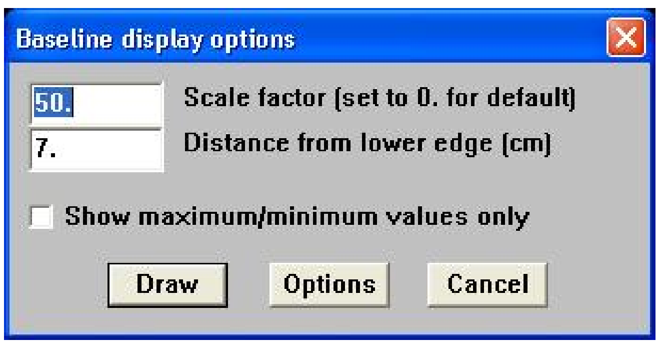
This option brings up a *Member force diagram scale lines* dialog box that enables up to 10 scale, or reference, lines to be superimposed onto either of the first two diagram types (overlay or baseline). These lines are purely optional and can represent a number of different diagram properties. As an example, they could be used as scale lines or as girder capacity levels. In either case you determine the number of lines that are applied to the diagram and their reference values. To include the lines in the diagram tick the box labelled *Include scale lines on diagrams?* Reference values can be positive or negative.

Save

This option enables the values associated with the selected vector, member range and diagram type to be saved to a text file and simultaneously displayed on the screen.

Changing Diagram Attributes

After the diagram has been draw, you can change any its attributes by clicking the icon located immedtiately to the right of the *Analyse* button or, from the main menu bar, select *Results / Moments, shear diagrams*. If the *Baseline* option is used, however, the following dialog box will appear when this button is clicked:

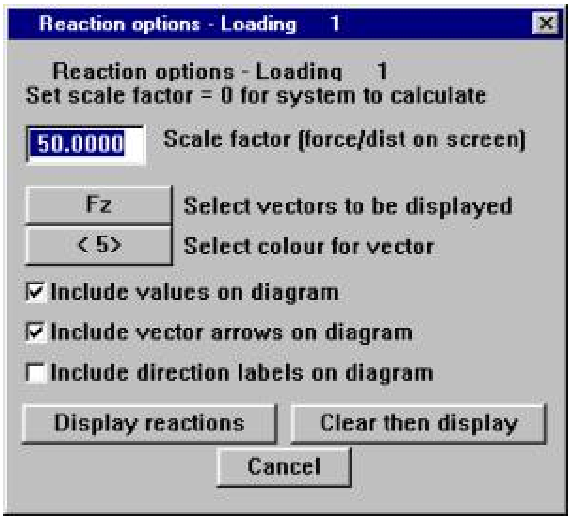


This will enable you to fine-tune the scale and location of the diagram. It will also allow you to display only the single maximum and minimum value for the entire member range rather than all values for every member. Note, however, that one of the value display options in the *Member forces* dialog box must be selected for this to work. The *Options* button will return you to the *Member forces* dialog box.

Once a load case, envelope or combination has been defined or selected for one result type it will remain active for all other types of results. After viewing, for example, the envelopes of moments and shears, the reactions could be displayed for that envelope by selecting *Results/Support reactions.*

1.3 Reaction Diagrams

Refer to *Section 1.1* above for a description of the process in getting to this point. Once you have reached this stage, a results dialog panel as shown below will appear. Enter values as appropriate then click ***Display reactions***. To clear the current diagram before drawing the new diagram, select ***Clear then display***. To change any of the diagram attributes after the diagram has been drawn click the icon to the right of the ***Analyse*** button or, from the menu bar, select ***Results / Support reactions***.



Scale factor

Enter a scale factor (if set to zero ACES will calculate a value). To *increase* the length of the reaction vectors *decrease* the scale value. For example, to double the length of the vectors as drawn, halve the current scale value. Note that if you are displaying reaction results for 2D grillages and slabs with the model shown in *plan* view you will not, in fact, be able to see the vector lines – you must change the viewing angle (e.g. by selecting an isometric view).

Select vectors to be displayed

This option enables you to select one or more reaction vectors that you would like to be displayed. For a 2D slab or grillage model lying in the X-Y plane with vertical pin supports and vertical loadings the only valid vectors would be the vertical reaction force ***Fz*** and the ***Mx*** and ***My*** bending moments. For pin supports, ***Mx*** and ***My*** are zero and would not, therefore, need to be displayed concurrently on the model – only ***Fz*** itself. If the structure is subjected to in-plane forces, or non-vertical forces, then support conditions will have to be modified appropriately. In this case the model must be converted into a 3D space frame and analysed accordingly.

Select colour for vector

This option enables you to select the colour in which the selected results vectors are to be displayed.

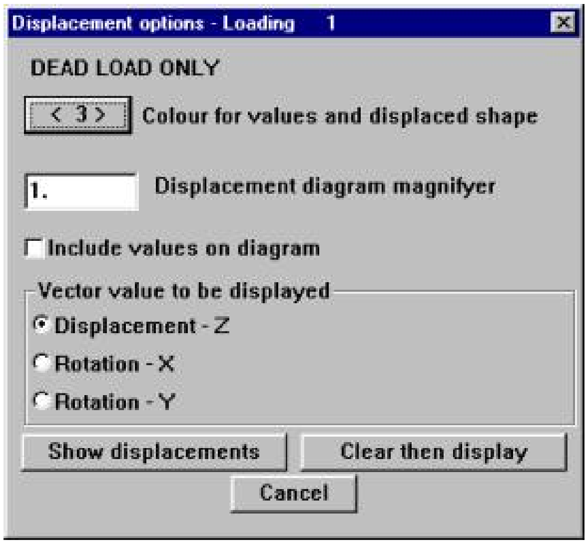
Set diagram display attributes

This option enables the following display attributes for the vectors to be either included, or excluded, from the diagram:

➢ ➢ *Values* (the actual numeric value of the vector (e.g. *150*)
➢ ➢ *Vector arrows* (the arrowed line representing the vector)
➢ ➢ *The direction label* to indicate the nature of the vector (e.g. *Fz*, *Mx*, etc)
➢

1.4 Displacements & Rotations

Refer to <u>*Section 1.1*</u> above for a description of the process in getting to this point. Once you have reached this stage, a results dialog panel as shown below will appear. Enter values as appropriate then click ***Show displacements***. To clear the current diagram before drawing the new diagram, select ***Clear then display***. To change any of the diagram attributes after the diagram has been drawn click the icon to the right of the ***Analyse*** button or, from the menu bar, select ***Results / Node displ+rotations***.



Colour for values and displaced shape

This option enables you to select the colour in which the selected displacement profiles and their associated numeric values are to be displayed.

Displacement diagram magnifier

Enter a magnification factor (the default is 1). To *increase* the magnitude of the drawn displacement vectors *increase* the magnification value. Note that if you have a 2D grillage or slab model and it is currently shown in *plan* view you will not, in fact, be able to see the displacement profile – you must change the viewing angle (e.g. by selecting an isometric view). If you do so and wish to view the previously specified displacement profile click on the ***Redraw last results diagram*** icon (second on the right from the ***Analyse*** icon).

Include values on diagram

Tick this box if you want the numeric displacement values to be displayed on the screen together with the displacement profile.

Vector value to be displayed

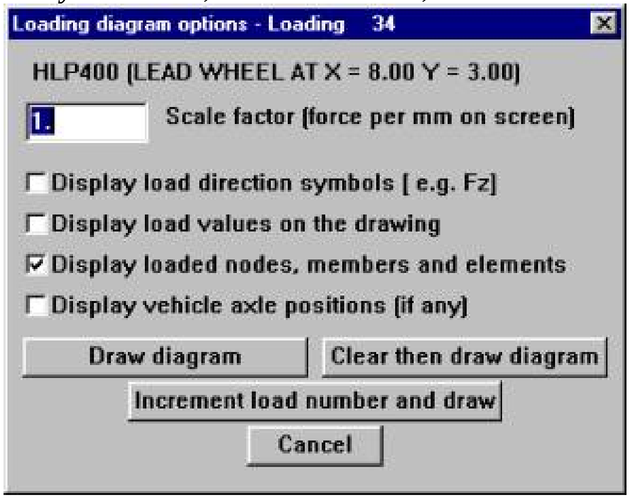
This option enables you to select the displacement/rotation vector that you would like to be displayed. For a 2D slab or grillage model lying in the X-Y plane with vertical pin supports and vertical loadings the only valid vectors would be the vertical displacement ***Z*** and the ***X*** and ***Y*** rotations. If the structure is subjected to in-plane forces, or non-vertical forces, then support conditions will have to be modified appropriately. In this case the model must be converted into a 3D space frame and analysed accordingly.

Rotations:
Rotations are measured around an appropriate axis. For example, if the main girders are parallel to the *Global X* axis then the main rotation would be a "*Rotation-z*" for a 2D frame model (*PLANE FRAME* type) , or "*Rotation-y*" for a 2D grillage model (*PLANE GRID* type).

1.5 Loading Diagrams

This option allows generated loading diagrams to be individually viewed. Refer to *Section 1.1* above for a description of the process in getting to this point (note, however, that it will not be necessary to select a member range). Once you have reached this stage, a loading diagram options panel as shown below will appear. Enter values as appropriate then click ***Draw diagram***. To clear the current diagram before drawing the new diagram, select ***Clear then draw diagram***.

To change any of the diagram attributes after the diagram has been drawn click the icon to the right of the ***Analyse*** button or, from the menu bar, select ***Results / Generated loadings***.



Scale factor

Enter a magnification factor (the default is 1). To *increase* the length of the reaction vectors *decrease* the scale value. For example, to double the length of the vectors as drawn, halve the current scale value. Note that if you have a 2D grillage or slab model and it is currently shown in *plan* view you will not, in fact, be able to see the loading vectors – you must change the viewing angle (e.g. by selecting an isometric view). If you do so and wish to view the previously specified loading diagram click on the ***Redraw last results diagram*** icon (second on the right from the ***Analyse*** icon).

Display load direction symbols

Tick this box if you want the load direction labels to be displayed on the drawing (e.g. *Fz, My* etc). This must be activated in conjunction with *Display loaded nodes*, *members and elements*.

Display load values on the drawing

Tick this box if you want the numeric load values to be displayed on the drawing (e.g. *Fz, My* etc). This must be activated in conjunction with *Display loaded nodes*, *members and elements*.

Display loaded nodes, members and elements

Tick this box if you want all nodal loads, members and elements to be displayed. If this is the only option ticked only arrowed lines representing the loads will be shown on the drawing. Numeric values will only be shown if *Display load values on the drawing* is toggled on.

Display vehicle axle positions

Tick this box if you want axle positions for the currently selected vehicle loading to be displayed on the drawing. This option can be toggled on with any of the other options and is useful for checking the distribution effect of the vehicle axle loads to nodes in the model.

Increment load number then draw

This option is useful for quickly checking the effect a moving vehicle has in terms of its load distribution to nodes. When used in conjunction with *Display vehicle axle positions* and *Display loaded nodes…* it shows the way in which wheel loads are distributed to adjacent nodes.

A typical diagram of nodal loadings for a particular position of an HLP400 vehicle is shown below.

1.6 Node Contouring Diagrams

This feature allows contours and shaded contours of the values represented by the nodal "Z" values to be produced. In effect this permits "topographical" views of data to be represented using ACES. It could, for example, be used to monitor the shape of an embankment, highway slope or distorted bridge deck. It is accessed via the *Results / Node Contours* menu options.

Creating a Contour Model

In order to use this feature the model must be created as a text file containing nodes with X,Y, Z coordinates, where the Z value represents either a real coordinate (such as an abutment face modelled in 3 dimensions) or a point on a contour line (such as an inclined slope). The file is initially read into ACES using the main opening menu option *"Start a new project*" followed by *"Reading 2D/3D node coordinates from a text file"*. (Refer to Section 2.3 for information regarding the layout and format of this text file).

Once the file has been read in, use *Structure/Other Options/Swap Axes* to ensure that the dependent variable (usually the Z coordinate) lies in the global Z-axis. Node contouring is performed in ACES using the inbuilt FE contouring engine. Because of this, all nodes must be joined with finite elements utilising the tools available in the *Structure/Finite Elements/Add element menu* menu option. Getting ACES to automatically generate a mesh of triangular elements is the easiest option, although rectangles produces the best contours.

Viewing Contours

Because node contouring is performed by the ACES contouring engine, the model must first be "analysed" to allow the contouring algorithms to work. You will therefore need to provide notional supports to the model and define at least one load case before performing the analysis. Turning on the auto calculation of self-weight in the first load case should be sufficient "loading". Ignore any error messages that may occur during the analysis.

After analysing the model with its single nominal loading, click *Results / Node Contours* to display values, save values, draw contour lines and produce shaded contours diagrams. Shaded contour diagrams are more effective than contour lines in representing this type of data. Note that contouring is based on values calculated at the centroid of each element, the centroidal value itself being based on averaging the Z values at the nodes of the elements. In order to obtain smooth and meaningful contours you should therefore provide a sufficiently dense mesh of nodal points.

2.0 SAVING & PRINTING THE RESULTS DIAGRAMS

2.1 Printing the Diagram

To obtain a printed copy of the current diagram select ***File / Print / Current diagram***.

2.2 Saving the Diagram as a Bitmap

To save the diagram as a bit-mapped image use ***File / Save diagram as bitmap***

3.0 OTHER INFORMATION

3.1 Sign Convention

For information on the sign convention refer to *PART 1, Section 6*.

3.2 Viewing Member Numbers

To view member numbers on the vector diagram select *Settings / Show symbols / Member numbers*. Note that only numbers for all *active* members in the currently selected range will be displayed.

3.3 Annotating the Diagram

To annotate the diagram, add text or to move any of the results values and labels, click on the ***T*** (text) icon in the tool bar. Refer also to *PART 1.3, User Interface*, for further instruction in using the text icons and menu options.

3.4 Viewing/Deleting Current Member Range

To view the currently active member range click on the *Display current member range* icon. To delete the range click the icon to the left of it.

3.5 Display Results in Tabular Form

To view the results in tabular form select ***Reports / Current graphical results***. See also *PART 5.2, Tabular Results*.

PART 5.2 Tabular Reports

1.0 DISPLAYING RESULTS IN TABULAR FORM

1.1 How to Display a Tabular Report

The procedure for displaying and printing any type of results in tabular (report) form is the same no matter what type of result vector is selected. It applies equally to moments, shears, element stresses, strains, torsions, reactions, displacements and curvatures and to all forms of output - envelopes, combinations and single load cases. The steps may be summarised as follows:

- • If you have just created a graphical view of the results in question and would now like to convert them to tabular form, select ***Reports / Current graphical results***. The results will be displayed in a scrollable window.

Note that cutting or copying of tabulated results directly from this window cannot be performed in this version of ACES. However, it can be copied to the clipboard using ***File / Copy report to clipboard***

- • If you have not created a graphical representation of the results and your model represents a frame, beam or grillage, click in turn on ***Reports/Select output range*** then choose the range of members for which tabulated results are required. A number of range options are available from the pop-up menu e.g. *Currently active items* or, if you wish to interrogate main longitudinal girders only, select ***By member property type***.

- • If ***By member property type*** is selected a small dialog box will open up and ask you to enter the required property type *number*. Enter the required member property type number e.g. *1* (*Main Girders*) and click *OK*.

- • For 3D finite element models you may wish to first activate one particular part of the model (such as a web or soffit slab in box girder structures) before creating a tabular report.

- • Now select the format in which you would like the report to be created by clicking ***Reports/Select format*** and choosing either *Standard* or *Spreadsheet*. The latter ensures that all headings, titles and other textual information in the report is enclosed in quotes for easy importing into spreadsheets.

- • Finally, select the type of results you require by again clicking on ***Reports/*** and choosing either:

  ⇒ ⇒ Moments and shears (& torsional moments)
  ⇒ ⇒ Displacements & rotations
  ⇒ ⇒ Reactions
  ⇒ ⇒ Element Moments
  ⇒ ⇒ Element Moments, Shears, Stresses and Strains

1.2 Moments, Torsions and Shears (Frames, Beams & Grillages)

Refer to *Section 1.1* above for a description of the process in getting to this point. Once you have reached this stage, a pop-up menu of loading types will be displayed. Select the type you require viz:

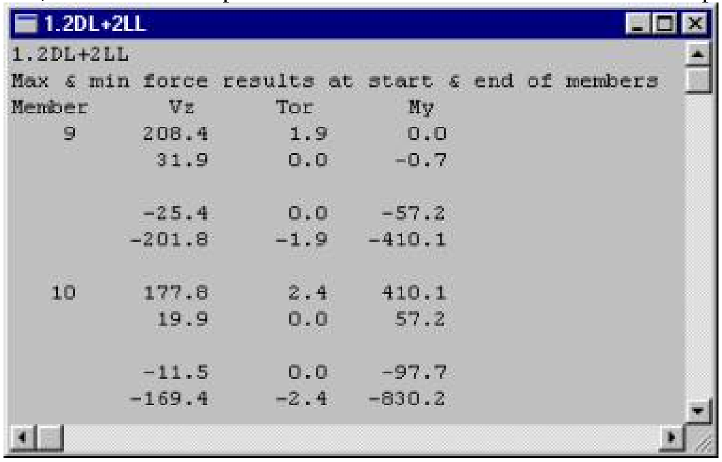
⇒ ⇒ Envelopes of maximum & minimum values
⇒ ⇒ Envelopes of maximum & minimum values of one vector with *corresponding values* of all others
⇒ ⇒ Results *sorted by load case number* for each member in the selected range
⇒ ⇒ Results for a *single loading* only

⇒ ⇒ Combinations

Envelopes of maximum & minimum values

If at least one envelope has already been created a dialog box will pop-up allowing you to either select one of the existing envelopes or to create a new envelope. In either case the envelope parameters window will appear. If you are unfamiliar with the envelope dialog box, *click here for details*.

Otherwise, the envelope dialog box will give you the opportunity to change any of the parameters (if you wish) or, alternatively, to simply click *OK* if no changes are required. Finally, another dialog box will allow you to enter the number of decimal places you require in the output report. Once done, click *OK*. A report similar to that shown below will be displayed.



This report type provides a summary of maximum and minimum values of all valid vectors for every member in the currently active range. Four lines of output are given for each member. Lines 1 and 2 show the maximum and minimum values respectively at the start of the member and lines 3 and 4 give the maximum and minimum values at the end of the member.

To print, save, or copy the report to the clipboard refer to *Section 2*.

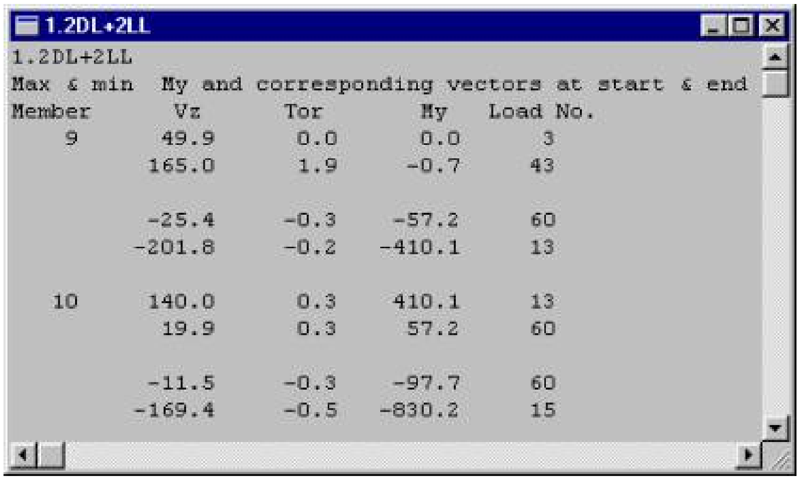
Envelopes of maximum & minimum values & corresponding values of all others

If at least one envelope has already been created a dialog box will pop-up allowing you to either select one of the existing envelopes or to create a new envelope. In either case the envelope parameters window will appear. If you are unfamiliar with the envelope dialog box, *click here for details*.

Otherwise, the envelope dialog box will give you the opportunity to change any of the parameters (if you wish) or, alternatively, to simply click *OK* if no changes are required.

The next dialog box will allow you to select the vector for which you require maximum/minimum values i.e., the vector to be enveloped. The corresponding values of all other force vectors will be shown with the enveloped vector.

The final dialog box will allow you to enter the number of decimal places you require in the output report. Once done, click *OK*. A report similar to that shown below will be displayed.



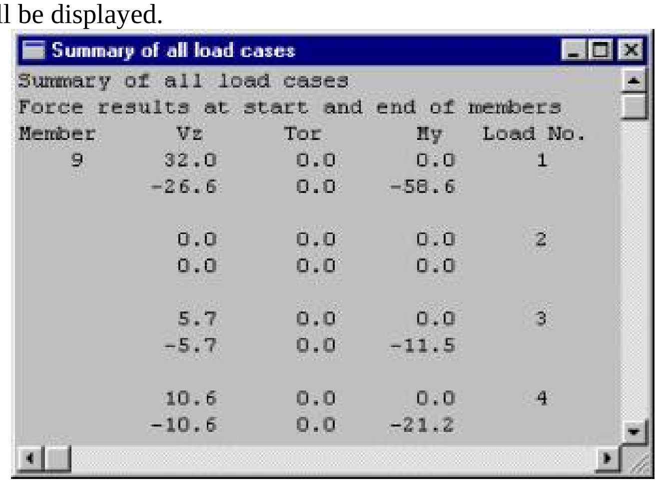
This report type provides a summary of maximum and minimum values of one nominated force vector for every member in the currently active range. Corresponding values of all other force vectors are also shown as well as the loading number producing the maximum/minimum effect.

Four lines of output are given for each member. Lines 1 and 2 show the maximum and minimum values respectively of the selected vector at the start of the member (in this case *My*) and lines 3 and 4 give the maximum and minimum values at the end of the member.

To print, save, or copy the report to the clipboard refer to *Section 2*.

Results sorted by load case number

This option will simply display a dialog box that will allow you to enter the number of decimal places you require in the output report. Once done, click *OK*. A report similar to that shown below will be displayed.



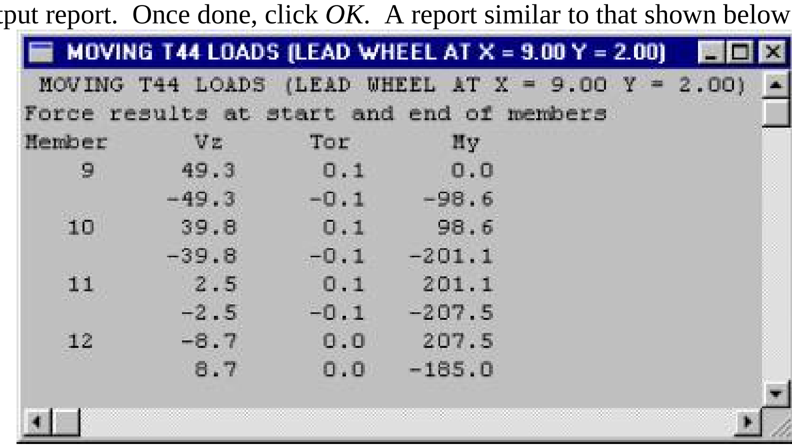
For every member in the currently active range this report displays all valid vectors ranked by loading number. Two lines of output are given for each loading. Line 1 shows the values at the start of the member and line 2 gives values at the end of the member.

To print, save, or copy the report to the clipboard refer to [Section 2](#).

Results for a single load case

When this option is selected a dialog box showing all directly defined and generated loading cases will be displayed. Scroll through the list and highlight the loading you wish to interrogate.

The next dialog box will allow you to enter the number of decimal places you require in the output report. Once done, click *OK*. A report similar to that shown below will be displayed.



For every member in the currently active range this report displays all valid force vectors. Two lines of output are given for each member. Line 1 shows the values at the start of the member and line 2 gives values at the end of the member.

To print, save, or copy the report to the clipboard refer to [Section 2](#).

1.3 Element Moments, Shears, Stresses, Strains

To create a report of element moments, shears, stresses and strains select ***Reports/Element..*** then choose the results range you wish to report on viz:

- Envelopes of maximum values
- Envelopes of minimum values
- Results sorted by loading case

- Single load case
- Combinations

Refer to *Section 1.1* above for a description of the process in getting to this point. For envelopes and combinations you will need to either create one first, (if none exists), or select one from the list. Alternatively, you may wish to create a new envelope. For additional information on sorting by load case, single load case and combinations refer to *PART 5.1* (Graphical Results). Comments made there-in apply equally to tabulated reports.

For help on creating envelopes refer to *PART 5.3*

For a more detailed discussion of Finite Element results refer to *PART 5.5*

1.4 Envelope of Envelopes

Refer to *PART 5.3, Envelopes*, for a discussion on enveloping envelopes.

1.5 Reaction Diagrams

To create a report of reactions select ***Reports/Reactions*** then choose the loading range you wish to report on viz:

- Envelopes of maximum values
- Envelopes of minimum values
- Results sorted by loading case
- Single load case
- Combinations

Refer to *Section 1.1* above for a description of the process in getting to this point. For envelopes and combinations you will need to either create one first, (if none exists), or select one from the list. Alternatively, you may wish to create a new envelope. For additional information on sorting by load case, single load case and combinations refer to *PART 5.1* (Graphical Results). Comments made there-in apply equally to tabulated reports.

If creating an envelope you will be asked to select the reaction (force) vector you wish to envelope (for 2D grillages it will generally be ***Fz***). The corresponding values of all other valid vectors will also be given in the report.

In all cases, you will be asked to specify the number of decimal places you would like results to be displayed in the output report.

1.6 Displacements & Rotations

To create a report of displacements or rotations select ***Reports/Displacements and rotations*** then choose the loading range you wish to report on viz:

- Envelopes of maximum values
- Envelopes of minimum values
- Results sorted by loading case
- Single load case
- Combinations

Refer to *Section 1.1* above for a description of the process in getting to this point. For envelopes and combinations you will need to either create one first, (if none exists), or select one from the list. Alternatively, you may wish to create a new envelope. For additional information on sorting by load case, single load case and combinations refer to *PART 5.1* (Graphical Results). Comments made there-in apply equally to tabulated reports.

2.0 SAVING, PRINTING & COPYING REPORTS

To print the current report select ***File / Print / Current report***. Note that the current report, (also known as the report last generated), can be printed at any time, even though it is not visible on the screen at the time. It will not be deleted until you exit out of ACES.

To save the report to an ASCII text file select ***File / Save current report***.

To copy the report to the clipboard select ***File / Copy report to clipboard***.

Note that cutting or copying of tabulated results directly from this window cannot be performed in this version of ACES. However, it can be copied to the clipboard using ***File / Copy report to clipboard***

3.0 OTHER INFORMATION

3.1 Sign Convention

For information on the sign convention refer to *PART 1, Section 6*, of the User Guide.

3.2 Viewing Member Numbers

To determine the numbering system used by ACES you will need to display member numbers for the currently selected range of members. To do so select the following menu options: *Settings / Show symbols / Member numbers*.

3.3 Viewing/Deleting Current Member Range

To view the currently active member range click on the *Display current member range* icon. To delete the range click the icon to the left of it.

PART 5.3 Envelopes

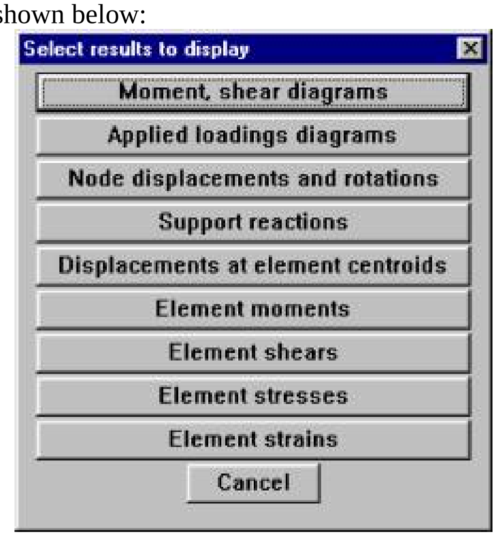
1.0 DISPLAYING ENVELOPE DIAGRAMS

1.1 How to Create & Display an Envelope

The procedure for displaying and printing any type of envelope diagram in graphical form is the same no matter what type of result vector is selected. It applies equally to moments, shears, torsions, reactions, displacements and curvatures. To interrogate output for combinations and single load cases refer to Parts 5.2 and 5.1 respectively. The steps in creating an envelope may be summarised as follows:

- • Click, in turn, on ***Results/Select a range of members/..*** then choose the range of members for which results are required from the pop-up menu e.g. for all main longitudinal girders select ***Members of a specified property type***. If this option is selected the following dialog box will be displayed:

- • Select the required member range (e.g. *Main Girders*). Click *OK*.
- • Now return to the ***Results*** menu and select ***Envelope*** from the options list.
- • An envelope parameters dialog window will be displayed. If you are unfamiliar with the envelope dialog box *click here for details*. Otherwise, enter parameters as appropriate and click *OK*.
- • Click *OK* once all parameters have been entered A submenu of results options will be displayed as shown below:



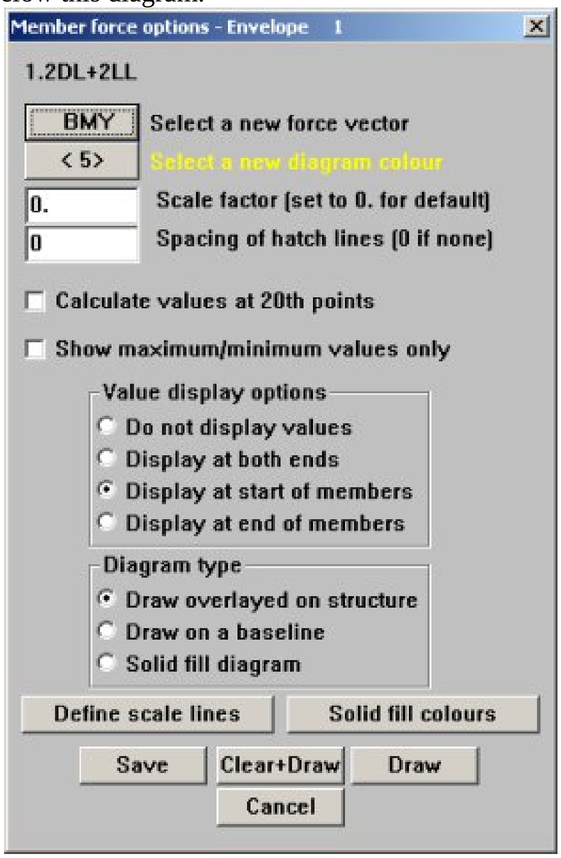
- • Select the type of results you require then use the browser's *Back* key to return to this same spot:

1.2 Envelope Parameters Dialog Box

Click here to view this dialog box and its parameters then use the browser's *Back* key to return to this spot.

1.3 Moment, Torsion and Shear Diagrams

Refer to Section 1.1 above for a description of the process in getting to this point. Once you have reached this stage, a results dialog panel as shown below will appear. Enter values as appropriate then click ***Draw***. To clear the current diagram before drawing the new diagram, select ***Clear+Draw***. To change any of the diagram attributes after the diagram has been drawn click the icon immediately to the right of the ***Analyse*** button or, from the menu bar, select ***Results / Moments, shear diagrams***. A description of the dialog box buttons and attibutes is given immediately below this diagram.



Select a new force vector

This option enables you to select the required results vector by clicking onto the ***Select a new force vector*** button. For 2D grillage models there will only be three valid force vectors to choose from - bending moment *BMY*, shear force *SFZ* or torsional moment *TOR*. For 2D beams and frames there will also be three - bending moment-*Z*, shear force-*Y* or Axial force.

Colour and line style

Click this button to select the required results vector colour and line style (thick, thin, dashed).

Scale factor

Enter a scale factor (if set to zero ACES will calculate a value). To make the diagram larger decrease the value. For example, to double the height of the diagram, halve the current scale value.

Spacing of hatch lines

This parameter is used to create hatched force diagrams. If left zero the diagram will not be hatched. Otherwise, enter a hatch spacing (in *mm*).

Calculate values at 20th points

This parameter is used to create smooth results curves. Normally ACES plots the values at the nodes then joins those points with straight lines. This can lead to misleading results diagrams, particularly in situations where there might be static loads applied along individual members that have not been divided into sub-members. If absolute accuracy is required in depicting the resultant vector diagrams, place a tick in this box. ACES will then calculate and plot the true value at 20th points along each member. Note that this is generally not required for moving vehicle loads, since individual wheel loads are always distributed to adjacent member *nodes*.

Show maximum/minimum values only

Tick this box if you want only the maximum and minimum (largest negative) values to be displayed on the diagram. Note that you must also switch on one of the *Display Values* options for this option to work.

Value display options

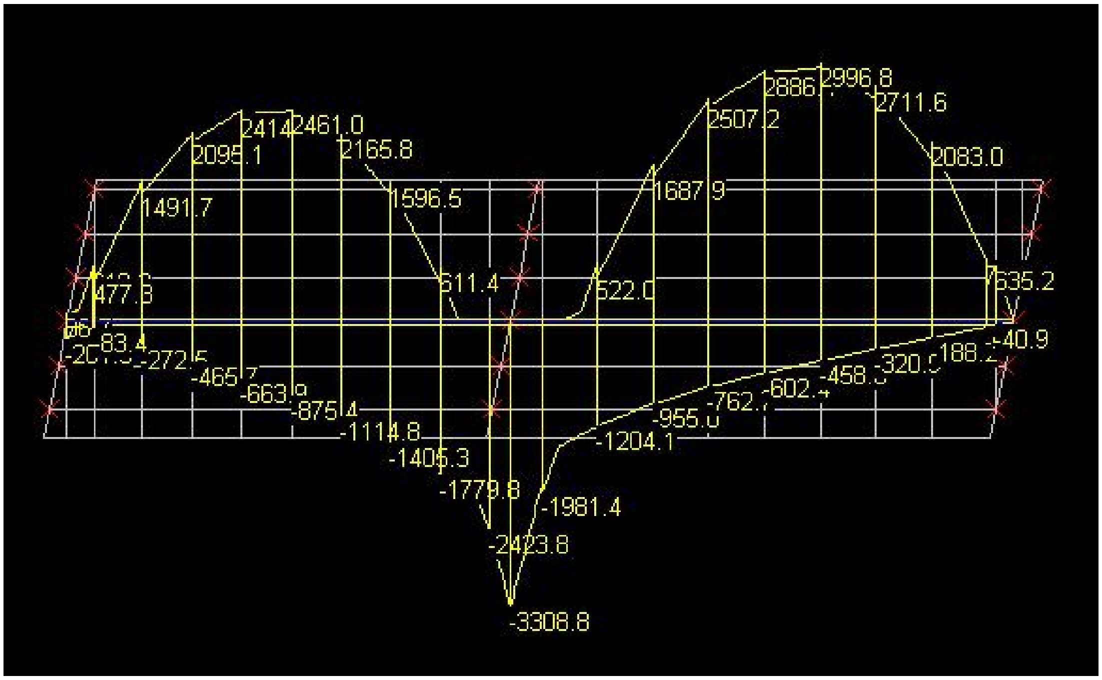
This option allows you to determine whether numeric results are to be displayed on the graphical diagram. Often you may not require the actual values to be displayed e.g., in situations where a simple comparison of the resultant force curves for a range of members. In this case, select the first option. The second option will display values for the selected vector at the beginning and end of every member in the currently active range. Depending on the complexity of the geometry and the selected scale this could lead to over-lap of values at each node, since the value at the end of one member may be superimposed onto the value at the beginning of the next. The last two options enable this problem to be circumvented.

Diagram type

This option enables one of three diagram types to be displayed:

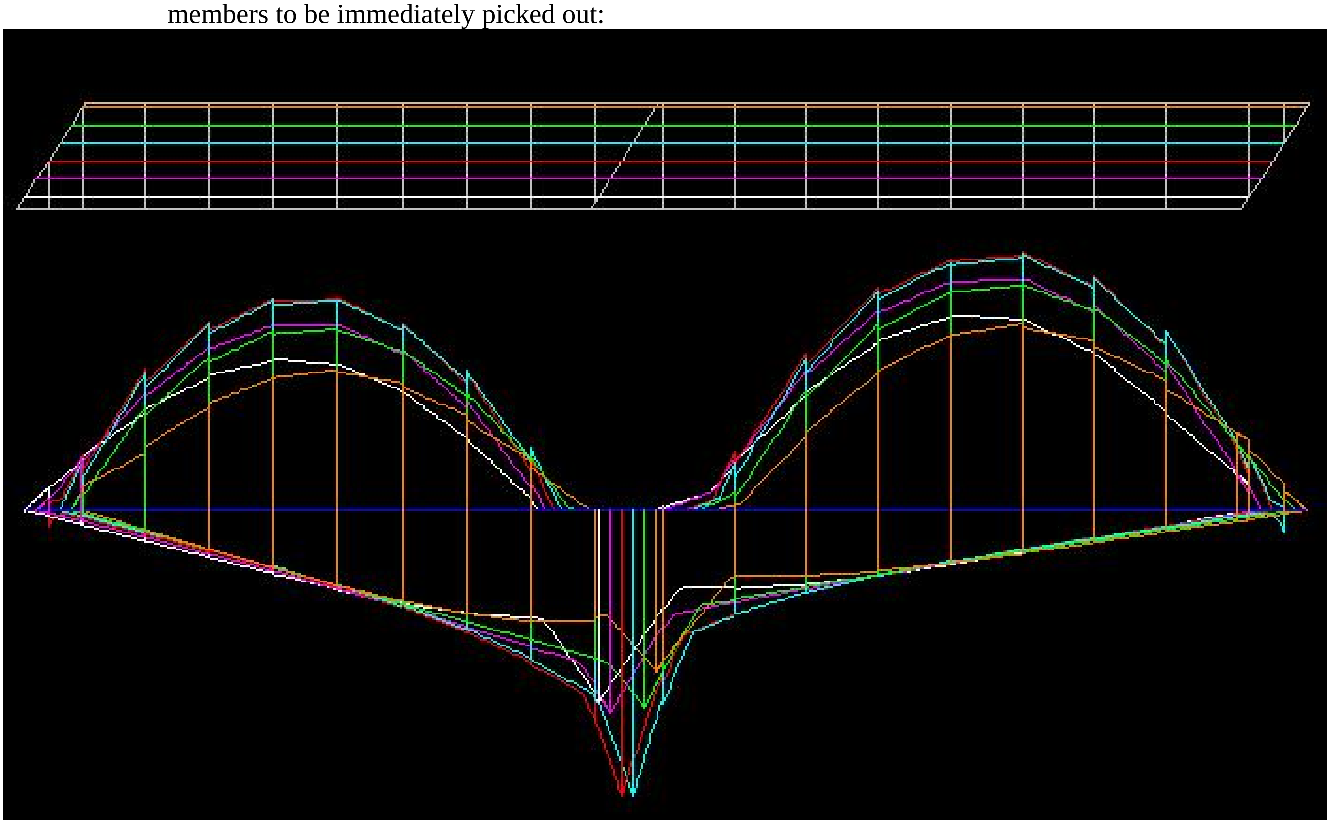
(a) Overlayed

The diagram is overlayed onto the base model as shown in the example below. This may produce results that are confusing and difficult to read, particularly if there are many active members.



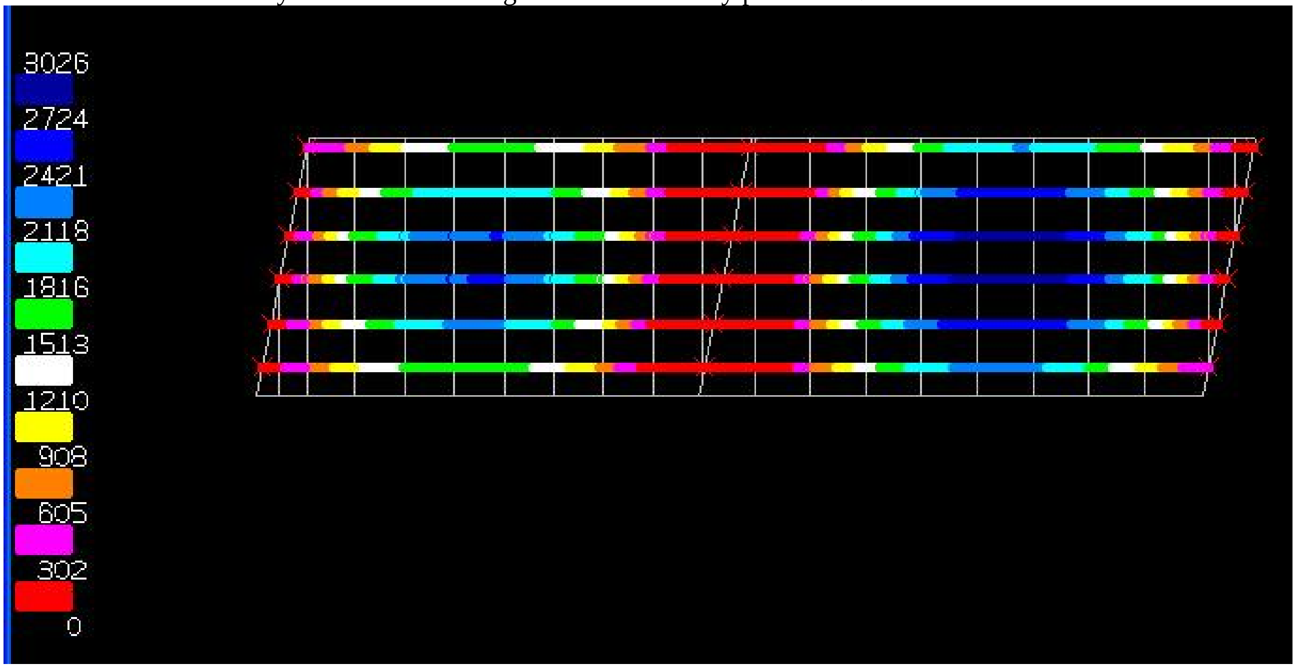
(b) Baseline

The vector diagram for every member in the currently active range is drawn on a common base-line. Each member is colour-coded to its respective diagram. This allows the worst affected members to be immediately picked out:



(c) Solid fill diagram

The maximum-minimum results spectrum for the load case (or envelope/combination) is sub-divided into 10 equal divisions. The results along each member are then drawn as solid fill lines that are colour-coded to their respective scaled division. This allows the worst affected areas in the currently active member range to be immediately picked out:



Define scale lines

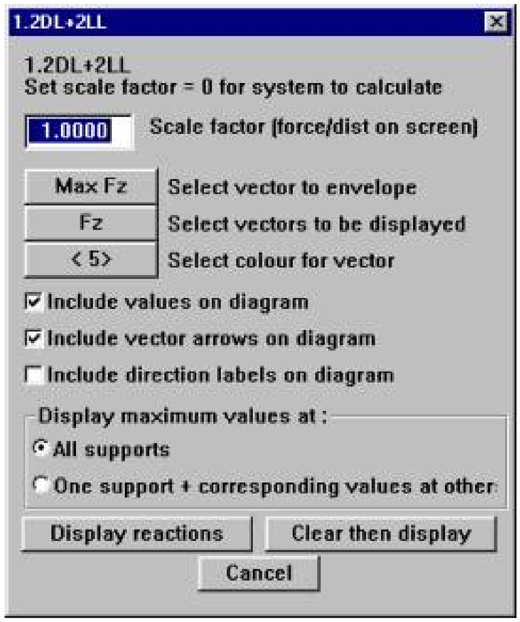
This option brings up a dialog box that enables up to 10 scale, or reference, lines to be superimposed onto either of the first two diagram types (overlay or baseline). These lines are purely optional and can represent a number of different diagram attributes. As an example, they could be used as scale lines or as girder capacity levels. In either case you determine the number of lines that are applied to the diagram and their reference values. To include the lines in the diagram tick the box labelled ***Include scale lines on diagram***. Reference values can be positive or negative.

To change any of the diagram attributes click the icon immediately to the right of the *Analyse* button or, from the menu bar, select *Results / Moments, shear diagrams*.

Once a load case, envelope or combination has been defined or selected for one result type it will remain active for all other types of results. After viewing, for example, the envelopes of moments and shears, the reactions could be displayed for that envelope by selecting *Results/Support reactions.*

1.4 Reaction Diagrams

Refer to *Section 1.1* above for a description of the process in getting to this point. Once you have reached this stage, a results dialog panel as shown below will appear. Enter values as appropriate then click ***Display reactions***. To clear the current diagram before drawing the new diagram, select ***Clear then display***. To change any of the diagram attributes after the diagram has been drawn click the icon to the right of the ***Analyse*** button or, from the menu bar, select ***Results / Support reactions***.



Scale factor

Enter a scale factor (if set to zero ACES will calculate a value). To *increase* the length of the reaction vectors *decrease* the scale value. For example, to double the length of the vectors as drawn, halve the current scale value. Note that if you are displaying reaction results for 2D grillages and slabs with the model shown in *plan* view you will not, in fact, be able to see the vector lines – you must change the viewing angle (e.g. by selecting an isometric view).

Select vector to envelope

This option enables you to select one of the available reaction vectors to envelope. For a 2D slab or grillage model lying in the X-Y plane with vertical pin supports and vertical loadings the only valid vectors would be the vertical reaction force ***Fz*** and the ***Mx*** and ***My*** bending moments. For pin supports, ***Mx*** and ***My*** are zero and would not, therefore, need to be enveloped – only ***Fz*** itself. If the structure is subjected to in-plane forces, or non-vertical forces, then support conditions will have to be modified appropriately. In this case the model must be converted into a 3D space frame and analysed accordingly.

Select vectors to be displayed

This option enables you to select one or more corresponding reaction vectors that you would like to be displayed concurrently with the enveloped vector. The vector to be enveloped is selected by clicking onto the preceeding button.

Select colour for vector

This option enables you to select the colour in which the selected results vectors are to be displayed.

Set diagram display attributes

This option enables the following display attributes for the vectors to be either included, or excluded, from the diagram:

- ➢ *Values* (the actual numeric value of the vector (e.g. *150*)
- ➢ *Vector arrows* (the arrowed line representing the vector)
- ➢ *The direction label* to indicate the nature of the vector (e.g. *Fz*, *Mx*, etc)

Display maximum values at:

This option enables the maximum enveloped reactions to be displayed at either:

- ➢ All valid and active support nodes in the model; *OR*
- ➢ At a single nominated support node only

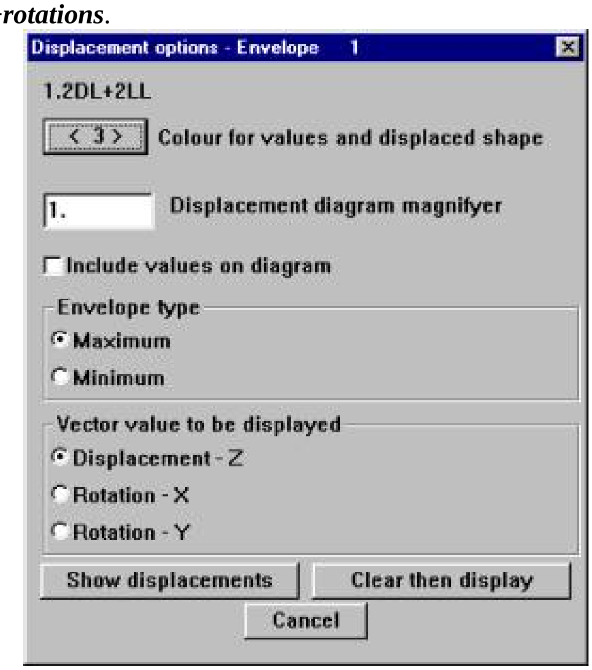
Maximum reaction plus corresponding vales

If you select the latter option, ACES will ask you to nominate the required support node number. Once this has been entered, the maximum (or minimum) reaction will be displayed at that node *together with the corresponding values* at all other active support nodes. To determine support node numbers click onto ***Settings / Show symbols*** and tick the *Node numbers* check box before you display the reactions dialog box.

Note also that the results title in the top left corner of the diagram will incorporate the loading number producing that maximum reaction value as well as a check sum value for the total reaction for that envelope. This will enable you to interrogate the results for that one single load case only (e.g. to obtain curvatures). Refer to *PART 5.1* for instructions in doing that.

1.5 Displacements & Rotations

Refer to *Section 1.1* above for a description of the process in getting to this point. Once you have reached this stage, a results dialog panel as shown below will appear. Enter values as appropriate then click ***Show displacements***. To clear the current diagram before drawing the new diagram, select ***Clear then display***. To change any of the diagram attributes after the diagram has been drawn click the icon to the right of the ***Analyse*** button or, from the menu bar, select ***Results / Node displ+rotations***.



Colour for values and displaced shape

This option enables you to select the colour in which the selected displacement profiles and their associated numeric values are to be displayed.

Displacement diagram magnifier

Enter a magnification factor (the default is 1). To *increase* the magnitude of the drawn displacement vectors *increase* the magnification value. Note that if you have a 2D grillage or slab model and it is currently shown in *plan* view you will not, in fact, be able to see the displacement profile – you must change the viewing angle (e.g. by selecting an isometric view).

If you do so and wish to view the previously specified displacement profile click on the ***Redraw last results diagram*** icon (second on the right from the ***Analyse*** icon).

Include values on diagram

Tick this box if you want the numeric displacement values to be displayed on the screen together with the displacement profile.

Envelope type

Click the required radio button.

Vector value to be displayed

This option enables you to select the displacement/rotation vector that you would like to be displayed. For a 2D slab or grillage model lying in the X-Y plane with vertical pin supports and vertical loadings the only valid vectors would be the vertical displacement ***Z*** and the ***X*** and ***Y*** rotations. If the structure is subjected to in-plane forces, or non-vertical forces, then support conditions will have to be modified appropriately. In this case the model must be converted into a 3D space frame and analysed accordingly.

1.6 Enveloping & Summing Envelopes

ACES provides a number of very powerful enveloping features. Once you have created two or more envelopes using the ***Results / Envelope*** or ***Reports*** menu options, the envelope dialog box will include an option to either envelope two or more envelopes that have already been created or to create the sum of a number of envelopes.

Clicking *Sum one or more envelopes* or *Envelope of envelopes* will display a dialog box that will enable you to select those envelopes that you wish to include in the new, combined, envelope.

Enveloping Envelopes:

At every node in the model, ACES will compare the maximum and minimum vector values in each selected envelope and extract the largest positive and negative values.

Summing Envelopes:

At every node in the model, ACES will separately add together the positive and negative vectors from each selected envelope then extract the largest maximum and minimum values. In effect, summation is performed using every possible combination of envelopes. It is important, therefore, that a particular load case (e.g. Dead Load, T44 etc) is not included more than once in the envelopes being summed (otherwise it will be added twice). If individual load cases, (such as Dead Load), are to be included in the enveloping process, they must either:

- already appear at least once in an existing envelope;
- or they must constitute their own unique envelope.

NOTE: A similar effect can be more easily achieved by using the Polarised feature found on the Envelope Factors dialog box (refer to Section 1.2 for details).

EXAMPLE - Summing Envelopes:

Assuming you have a 5 span structure, you might generate the following load cases:

Case 1 - Self Weight
Case 2 - Superimposed dead load
Case 3 - UDL span 1
Case 4 - UDL span 2
Case 5 - UDL span 3
Case 6 - UDL span 4
Case 7 - UDL span 5
Case 8 - Moving vehicle

Having solved the problem, you would need to create the following individual envelopes using appropriate factors for each load case within each envelope:

Envelope 1 - Load cases 1 and 2 as permanent, exclude all others
Envelope 2 - Load case 3, not permanent, exclude all others
Envelope 3 - Load case 4, not permanent, exclude all others
Envelope 4 - Load case 5, not permanent, exclude all others
Envelope 5 - Load case 6, not permanent, exclude all others
Envelope 6 - Load case 7, not permanent, exclude all others
Envelope 7 - Load case 8, not permanent, exclude all others

Note that in *Envelopes 2-6* it is not strictly necessary to have load cases 3, 4, 5, 6 and 7 designated as "*not permanent*" because they are the only loads included in their respective envelope. However, specifying them as non-permanent may help remind you how they will be treated in the "*Sum of Envelopes*" routine.

When you are ready to display your results or reports, select "*Sum of Envelopes*" from the envelopes panel and specify all 1 - 7 envelopes to be included in the summation process. The results and reports will then contain the dead load values specified in load cases 1 and 2 together with the summed values from all other load cases i.e., the summation process automatically accounts for alternately loaded spans together with the vehicle loads. Instead of looking at a series of load cases and selecting the maximum and minimum values this option in effect *adds* the results if they contribute to a maximum or minimum value or ignores them if they don't.

NOTE: A similar effect can be more easily achieved by using the Polarised feature found on the Envelope Factors dialog box (refer to Section 1.2 for details).

2.0 SAVING & PRINTING ENVELOPE DIAGRAMS

2.1 Printing the Diagram

To obtain a printed copy of the current diagram select ***File / Print / Current diagram***.

2.2 Saving the Diagram as a Bitmap

To save the diagram as a bit-mapped image use ***File / Save diagram as bitmap***

3.0 OTHER INFORMATION

3.1 Sign Convention

For information on the sign convention refer to *PART 1, Section 6*, of this help system.

3.2 Viewing Member Numbers

To view member numbers on the vector diagram (for the currently selected range of members) select *Settings / Show symbols / Member numbers*.

3.3 Annotating the Diagram

To annotate the diagram, add text or to move any of the results values and labels, click on the ***T*** (text) icon in the tool bar. Refer also to *PART 1.3, User Interface*, for further instruction in using the text icons and menu options.

3.4 Viewing/Deleting Current Member Range

To view the currently active member range click on the *Display current member range* icon. To delete the range click the icon to the left of it.

3.5 Display Results in Tabular Form

To view the results in tabular form select ***Reports / Current graphical results***. Refer also to *PART 5.2*, *Tabular Reports*, for further information.

PART 5.4 Combinations

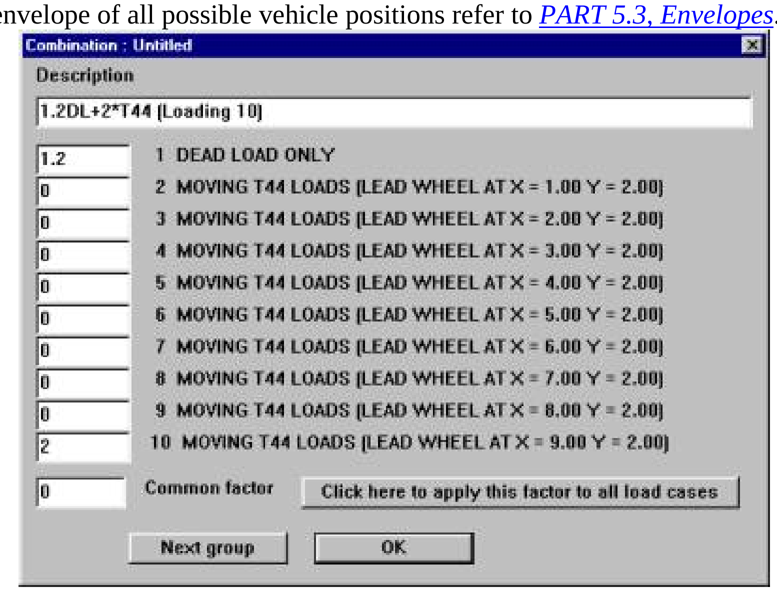
1.0 DISPLAYING COMBINATION DIAGRAMS

1.1 How to Create & Display a Combination

The procedure for displaying and printing any type of combination diagram in graphical form is the same no matter what type of result vector is selected. It applies equally to moments, shears, torsions, reactions, displacements and curvatures. To interrogate output for envelopes and single load cases refer to *PART 5.1, Frames & Grillages*, and *PART 5.3, Envelopes*, respectively. The steps in creating a combination may be summarised as follows:

- • Click, in turn, on ***Results/Select a range of members/..*** then choose the range of members for which results are required from the pop-up menu e.g. for all main longitudinal girders select ***Members of a specified property type***. If this option is selected the following dialog box will be displayed:

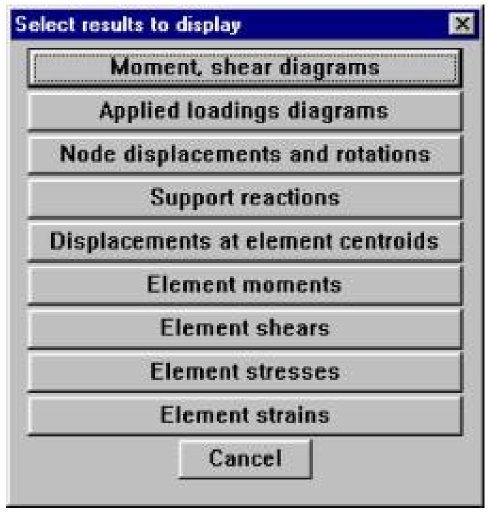
- • Select the required member range (e.g. *Main Girders*). Click *OK*.
- • Now return to the ***Results*** menu and select ***Combination*** from the options list.
- • The dialog box shown below will be displayed. Enter a combination description then select the loadings and factors required for the combination. Note that for moving vehicle load cases you should only select one specific location of a single given vehicle. To create an envelope of all possible vehicle positions refer to *PART 5.3, Envelopes*.



Common Factor

If factors for all load cases in the combination are all more or less the same, enter a value in the field labelled *Common factor* then click the button labelled *Click here to apply this factor to all load cases*. You can now change the factor for any load case to what it should really be. This is useful in situations where there are many loadings and you wish to set most of them to one particular value (e.g. zero). Any loading with a zero load factor will be excluded from the combination.

- • Click *OK* once all parameters have been entered A submenu of results options will be displayed as shown below:



- • Select the type of results you require then refer to the following sections of the help system for further information (comments made in the graphical results section pertain also to combinations):

2.0 SAVING & PRINTING COMBINATION DIAGRAMS

2.1 Printing the Diagram

To obtain a printed copy of the current diagram select ***File / Print / Current diagram***.

2.2 Saving the Diagram as a Bitmap

To save the diagram as a bit-mapped image use ***File / Save diagram as bitmap***

3.0 OTHER INFORMATION

3.1 Sign Convention

For information on the sign convention refer to *PART 1, Section 6*, of this help system.

3.2 Viewing Member Numbers

To view member numbers on the vector diagram (for the currently selected range of members) select *Settings / Show symbols / Member numbers*.

3.3 Annotating the Diagram

To annotate the diagram, add text or to move any of the results values and labels, click on the ***T*** (text) icon in the tool bar. Refer also to *PART 1.3, User Interface*, for further instruction in using the text icons and menu options.

3.4 Viewing/Deleting Current Member Range

To view the currently active member range click on the *Display current member range* icon. To delete the range click the icon to the left of it.

3.5 Display Results in Tabular Form

To view the results in tabular form select ***Reports / Current graphical results***. Refer also to *PART 5.2*, *Tabular Reports*, for further information.

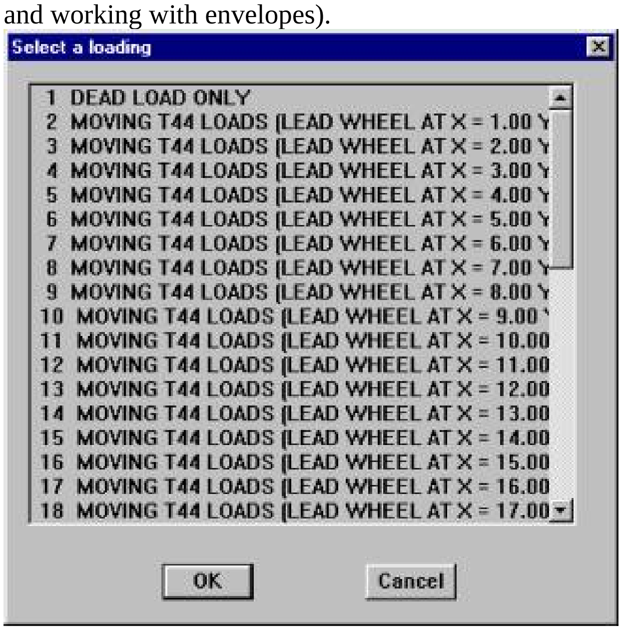
PART 5.5 Finite Elements

1.0 DISPLAYING RESULTS DIAGRAMS

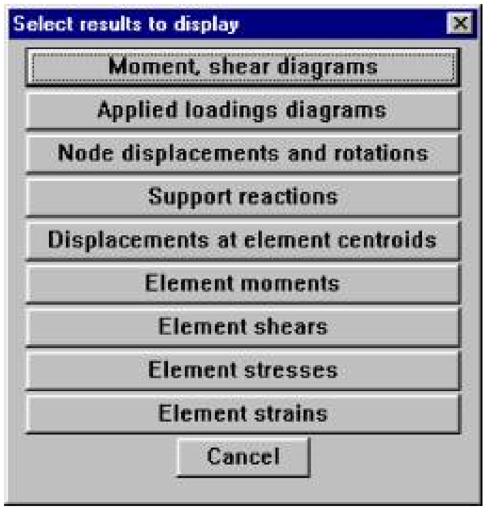
1.1 How to Display a Results Diagram

The procedure for displaying and printing any type of results diagram in graphical form is the same no matter what type of result vector is selected. It applies equally to element moments, shears, stresses, strains, reactions, displacements and curvatures and to all forms of output - envelopes, combinations and single load cases. The steps may be summarised as follows:

- • Select the loading results you require:
  - ⇒ ⇒ For an envelope click *Results / Envelope*
  - ⇒ ⇒ For single load case click *Results / Load case*
  - ⇒ ⇒ For a combination click *Results / Combination*
- • If you wish to graphically interrogate the results for a single load case select that option. The dialog box shown below will be displayed (refer to *PART 5.3* for further information on creating and working with envelopes).



- • Scroll down to the required load case, highlight it, then click *OK*. A submenu of results options will be displayed as illustrated below:



- • Select the type of results you require:

1.2 Element Moment Diagrams

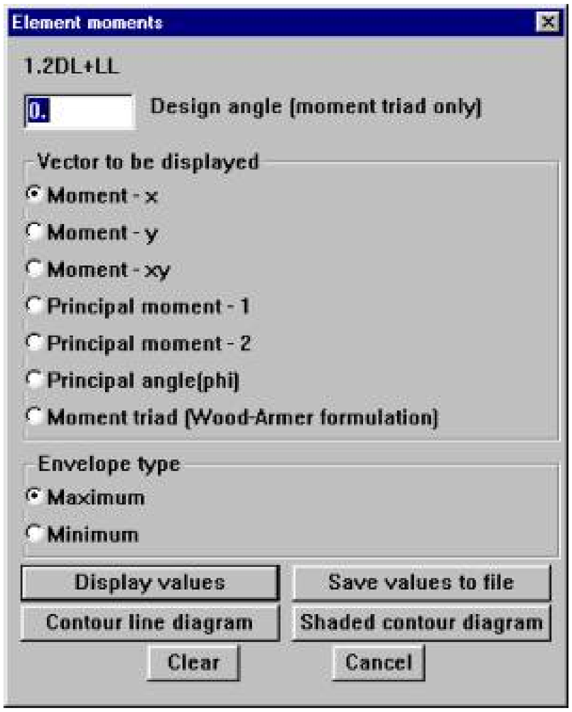
Element moments are only available for plate bending elements while stresses and strains are only available for shell elements.

Element moments are calculated and dispkayed by ACES in accordance with normal plate bending theory i.e. *'Moment X'* means the moment parallel to the local **x-**axis.

NOTE: When creating contour or solid shadediagrams it is very important that local axes of all elements are all orientedin the same direction, otherwise the diagrams will becomemeaningless.

T o check their orientation click ***Settings/Showsymbols*** from the top menu and tick the ***Element local axes*** box. If any element axis does not align itself with the corresponding global axes then such elements will need to be considered separately. Although ACES has routines for reversing and rotating local element axes and aligning the axes of groups of elements it may not always be possible to align them exactly, particularly if irregularly shaped triangles are used. Alternatively, use only the principal values rather than the basic ***x*** and ***y*** values.

Refer to *Section 1.1* above for a description of the process for displaying element moments. Once you have reached this stage, a results dialog panel as shown below will appear. Enter values as appropriate then click one of the four action buttons. To clear the current diagram before drawing the new diagram, select ***Clear***. To change any of the diagram attributes after the diagram has been drawn click the icon to the right of the ***Analyse*** button or, from the menu bar, select ***Results / Element moments***.



Vector to be displayed

Select the results vector you wish to examine. When analysing reinforced concrete structures it is generally recommended that the *Moment-Triad* (Wood-Armer formulation) be used, since it makes allowance for the "twisting" moment in the elements. In this instance you will need to enter a design angle (specified in degrees).

Note that element moments are only available for plate bending elements and stresses and strains are only available for shell elements.

Envelope type (only if viewing envelope results)

Indicate if you wish to view maximum positive values or minimum (maximum negative) values.

Save values to file

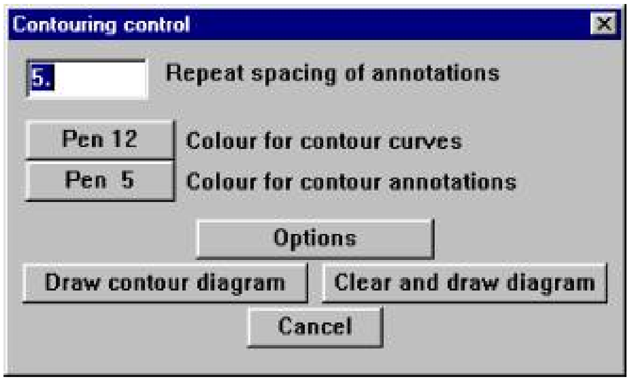
This button will save the tabular results values to a text file in the nominated directory. They can then be read into a spread-sheet etc.

Display Values

This option will display results in numeric form at element centroids. By default results are calculated and displayed by ACES only at element centroids, unless shears are required, in which case values are calculated at nodes.

Contour line diagram

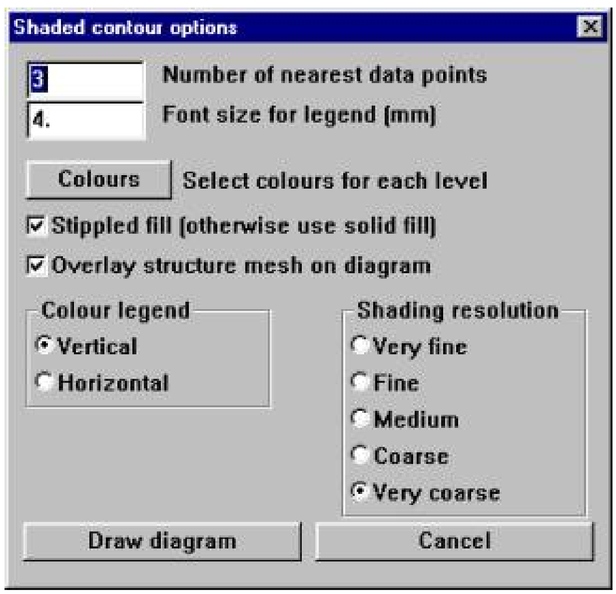
This option will display a diagram of results using contour lines. When this button is clicked a contour-control panel will first be displayed as shown below:



Enter the parameters as required then click one of the two draw buttons. The parameter *Repeat spacing of annotations* represents the distance (in centimetres) between annotations along any one contour line. The Options button contains features to assist in performing contour smoothing, interval specification and accuracy parameters

Shaded contour diagram

This button will display a diagram using either hatched lines, stippled shading or solid fill. When clicked, a shading-control panel will appear. Enter values as appropriate then click ***Draw diagram***.



Number of nearest data points

Specifies a parameter that controls contour smoothing and accuracy. For smoother curves, enter a higher value.

Font size for legend

Indicate the font size (in mm) in which to display legend scale values.

Stippled fill

Solid fill diagrams may take a long time to draw. If you require only a quick over-view of the worst affected areas of the model leave this box ticked and select a medium to very coarse shading resolution. You may need to experiment to determine the best combination of both of these values

Colour legend

Indicates if the scaled legend diagram is to be drawn vertically down the screen or horizontally across it. The maximum-minimum results spectrum for the load case (or envelope/combination) is sub-divided into 10 equal divisions.

Overlay structure mesh on diagram

This parameter indicates if you want the FE mesh to be overlayed on the final drawing.

Changing diagram attributes

To change any of the diagram attributes after it has been drawn click the icon to the right of the *Analyse* button or, from the menu bar, select *Results / Moments, shear diagrams*.

Once a load case, envelope or combination has been defined or selected for one result type it will remain active for all other types of results. After viewing, for example, the envelopes of moments and shears, the reactions could be displayed for that envelope by selecting *Results/Support reactions.*

1.3 Shears

1.3.1 Out-of-Plane Shear Forces

Out-of-plane shears will only be calculated by ACES if you click the *Yes* button in the dialog box that appears just prior to the beginning of the analysis. They are <u>*not*</u> given as *stresses* but as values in units of *Force/Unit element width* (e.g., in kN/m). However, they can easily be converted to stresses by dividing by the element thickness.

In effect they can be visualised as "punching" shear forces at the column/slab interface. They are calculated and displayed across an orthogonal plane drawn through the element centroid. You will be required to nominate the plane prior to the display - either the local ***x***-plane (i.e. parallel to the local ***x*** axis) or the local ***y***-plane (parallel to the local ***y*** axis). Refer also to the comments given in Section 1.3.2 regarding the interpretation of shear results.

Out-of-plane element shear forces are derived from the local ***z*** forces acting at element nodes and are calculated with respect to the *Local* element axis system.

In determining shear forces ACES first establishes a local reference system at the centroid of each element. For *SHELL* type elements the local ***x*** axis will always be aligned with the side joining the first two nodes (refer to Figure 2). In the case of plate bending elements the local axis system will correspond with the *Global X-Y* axis system which may not necessarily line up with any of the element sides (refer to Figure 1).

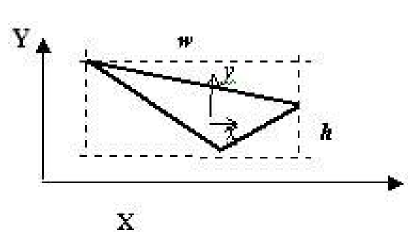


Figure 1: Skewed Triangular Plate Elements

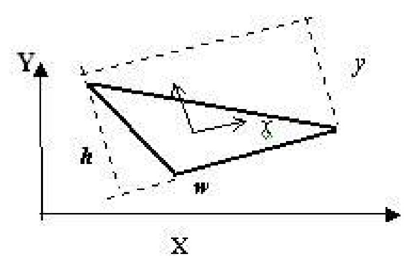


Figure 2: Skewed Triangular Shell Elements

The height, ***h***, and width, ***w***, of each element is assumed to be the vertical and horizontal distance, respectively, between the node extremities. Note that these distances may, in some circumstances, exceed the base length (or width) of the element as, for example, in highly skewed triangular SHELL elements or plate bending elements whose sides are not aligned with the global reference system.

The shear force ***Vx*** is calculated as the sum of all local ***z*** nodal forces to the *left* of the centroidal ***y*** axis divided by the element height, ***h***, and ***Vy*** as the sum of all local ***z*** nodal forces *below* the centroidal ***x*** axis divided by the element width, ***w***. If a node lies on one of the centroidal axes its contribution is ignored unless the value would tend to increase the shear force.

1.3.2 Interpreting Out-of-Plane Shears

It is important to be aware of the sign convention when interpreting out-of-plane shear results in FE models. For example, *Shear-x* acts in a plane that is parallel to the local element ***x***-axis while *Shear-y* acts in a plane parallel to the local element ***y***-axis (both planes intersect at the centroid of the element). *Shear-y* will generally behave in accordance with normal beam theory. This is not the case for *Shear-x.*

In some cases, such as highly skewed decks with irregular skewed triangular elements, it is possible for *Shear-x* to be greater at the centre of the span than that at the supports, particularly towards the outside edges of the deck. This is due to the fact that loads at the edge of the plate are distributed towards the plate centre by a shear acting across a plane parallel to the direction of the span.

The interpretation of shear values from these highly skewed triangular meshes will be affected by:
(a) The shape of the triangle (a symmetrically shaped triangle is preferred)
(b) Edge effects
(c) Skew effects
(d) Reversal of local x-axes to ensure that local z-axes are all in the same direction

The shape of the triangle is important because the displayed value is calculated at the centroid of the element. For skewed meshes the centroids of adjacent triangles may not be colinear in either the global-X or Y directions, so that shear values in adjacent triangular elements will not act in the same plane.

Generally speaking it is recommended that shear values from models containing skewed triangulated meshes not be used. Our advice would be to use reactions instead, particularly in conjunction with spring supports which have the tendency of evening out the load distributed between supports.

1.3.3 In-Plane Shear Stresses

In-plane shears are given as *stresses* and will only be calculated by ACES if the analysis type permits (e.g. *Plane Stress* and *Shell* type models) They may be visualised as web shear in a steel **I** girder. In-plane shear stresses are one of the three primary vectors available to be displayed, the other two being stresses in the local ***x*** and ***y*** directions. Refer also to the comments given in Section 1.3.3 regarding the interpretation of shear results.

1.3.4 Displaying Shear Forces

To display shear forces for individual load cases or envelopes refer first to *Section 1.1* above.

1.4 Stresses & Strains

It is important to be aware of the sign convention used in interpreting results of a FE analysis. In general, a result given relative to an axis acts in the direction of that axis. For example, *Stress-x* is in the direction of the local element ***x***-axis. For further information on the sign convention and related issues refer to *PART 1, Section 6*, of the User Guide.

NOTE: When creating contour or solid shadediagrams it is very important that local axes of all elements are all orientedin the same direction, otherwise the diagrams will becomemeaningless.

To check their orientation click ***Settings/Show*** symbols from the top menu and tick the ***Element local axes*** box. Refer also to *Section 1.6* for further comments relating to thisissue.

1.5 Envelopes of Moments, Shears, Stresses & Strains

When enveloping most FE vectors ACES will generally display the maximum or minimum values only. There is no facility for displaying maximum and minimum values together in the same diagram nor for reporting on maximum and minimum values of one force vector with corresponding values of all others.

Note: All references to “minimum” values in this context implies the maximum negative values of either element moments, shears, stresses or strains.

For additional information on creating envelopes refer to *PART 5.3* or click here for parameters associated with the *envelope specification dialog box*.

2.0 SAVING & PRINTING THE RESULTS DIAGRAMS

2.1 Printing the Diagram

To obtain a printed copy of the current diagram select ***File / Print / Current diagram***.

2.2 Saving the Diagram as a Bitmap

To save the diagram as a bit-mapped image use ***File / Save diagram as bitmap***

3.0 OTHER INFORMATION

3.1 Sign Convention

For information on the sign convention refer to *PART 1, Section 6*.

3.2 Viewing Element Numbers

To view element numbers on the vector diagram select *Settings / Show symbols / Element numbers*.

3.3 Annotating the Diagram

To annotate the diagram, add text or to move any of the results values and labels, click on the ***T*** (text) icon in the tool bar. Refer also to *PART 1.3, User Interface*, for further instruction in using the text icons and menu options.

3.4 Viewing the Current Element Range

To view all active elements in the model select *Structure/Display Elements*. If you wish to display only elements of a particular property type use the menu options *Structure/Finite Elements/Display same type*.

3.5 Display Results in Tabular Form

To view the results in tabular form select ***Reports / Current graphical results***. Refer also to *PART 5.2*, *Tabular Reports*, for further information.

3.6 Problems with Element Supports

In some situations if elements have supports on all four sides this can occasionally cause the solution process to fail. This is due to the fact that the calculated elemental values become infinitesimally small, a condition that will cause some of the older PC processors to crash.

In most cases when using finite elements it may be preferable to support the structure on spring supports in the major force direction. This reduces the tendency for fluctuations in the sign of adjacent support reactions and is particularly important when dealing with directly supported, highly skewed, slabs.

3.7 Wood Armer Moments (Moment Triads)

When analysing reinforced concrete structures it is generally recommended that the *Moment-Triad* (Wood-Armer formulation) be used, since it makes allowance for the "twisting" moment in the elements. In this instance you will need to enter a design angle (specified in degrees).

Note that the sign convention for finite elements is different to that for frame members. A *Moment-x* is actually a moment in the *direction* of the local ***x***-axis (i.e. it acts *about* the local ***y***-axis). Therefore, to design for reinforcement parallel to the local x-axis, enter an angle of 0 degrees. For reinforcement parallel to the local y-axis, enter an angle of 90 degrees. For example, in the case of a rectangular concrete plate oriented with the long edge in the x-direction and the short edge in the y-direction and reinforcement designed for moment about the ***y***-axis, the *Design Angle* would be 0.

PART 6

INCREMENTAL LAUNCHING MODULE

PART 6: ILB - Main Index

6.1 General Concepts & Features

6.1.1 General Launch Process
6.1.2 Limitations & Restrictions

6.2 Program Operation

6.2.1 Main Data Entry Panel
6.2.2 Entering Model & Launching Data
6.2.3 Viewing, Printing & Saving Results
6.2.4 Printing Input Data
6.2.5 Setting Colours & Line Styles
6.2.6 Loading & Saving Model Files

6.3 Input Data

6.3.1 General Launch Data
6.3.2 Casting Bed Data
6.3.3 Model Geometry
6.3.4 Nose Properties
6.3.5 Segment Properties
6.3.6 Assigning Segment Properties
6.3.7 Loadings
6.3.8 Printing Input Data

6.4 Analysis

6.4.1 Setting Analysis Options
6.4.2 Performing the Analysis

6.5 Output Results

6.5.1 Viewing Envelopes Graphically
6.5.2 Viewing Envelopes in Tabular Form
6.5.3 Viewing Reactions & Deformations
6.5.4 Printing Results
6.5.5 Saving Final Results
6.5.6 Saving Results at Intermediate Launch Positions

6.6 Example Problem

6.6.1 Input Data
6.6.2 Output Results

SECTION 6.1 - General Concepts & Features

6.1.1 General Launch Process

The Incrementally Launched Beam (ILB) module calculates and displays moments, shears, torsions, support reactions and deflections of an incrementally launched girder. An iterative linear-elastic analysis is performed for each launch increment, with the structure being modelled as a series of line elements.

Note that consistency of units must be maintained in all input data. All lengths, distances and section properties must be in the length unit specified and all loads and densities must be in the force unit specified.

The following steps would typically be involved in the incremental launch process:

1. A casting or assembly area some distance behind the first permanent support (called the left abutment in the ILB system) is established. The girder is cast or assembled in sections in this area and pushed into position in small increments.

2. A specially fabricated nose structure is attached at the start of the girder to allow the first segment to be launched before the second one is attached.

3. A series of supports are constructed for the girder in its final position. These may include temporary supports within one or more spans which reduce the launching stresses but are removed once the girder is in its final position.

4. A series of temporary supports between the left abutment and the casting bed may also be necessary.

6.1.2 Limitations & Restrictions

1. The girder may be straight or curved in plan. For a curved girder a three dimensional (SPACE FRAME) analysis is performed. For a straight girder a two dimensional (PLANE FRAME) analysis is performed.

2. The girder is made up of a number of segments of varying length which are attached to the girder during the launch process. Only those segments which are attached at the current stage of the launch are included in the structural model. Section properties must be uniform within a segment but different segments may have different properties.

3. ACES allows for temporary supports to be included between fixed piers and abutments (by specifiying "dummy" spans) as well as supports in the casting bed and supports between the abutment and the casting bed.

4. Spring supports as well as support settlements may be specified. However, both cannot be assigned to the same support node.

5. For thermal and prestress moments, the primary moments are analysed and the secondary effects are calculated by subtracting the primary moment in each segment.

6. The launching nose is assumed to be uniformly tapered between its ends.

6.2.1 Main Data Entry Panel

A screen image of the main data entry panel is shown below. Menu options are displayed in groups that reflect their function. A brief description of each button group is given in Sections 6.2.2 - 6.2.6 below. A detailed description of model input data is given in *Section 6.3*

To exit from the Incremental Launching Module click the ***Close*** button followed by the ***X*** icon at the top right hand corner of the screen (or use the conventional *File/Exit* menu commands). To minimise the ILB interface, first click *Close* followed by the Windows minimisation icon. Click the *Change model parameters* icon in the icon bar to redisplay the main control panel when re-maximising the screen.

Performing the Analysis

To analyse the model click the button labelled ***Begin Analysis***. ACES will display a launching screen that can dynamically show the progress of the analysis. If either of the last two options in the *Intermediate Results* group are checked, the moment and shear diagrams for the current increment will be superimposed onto the cumulative envelopes of maximum and minimum moments and shears.

Display all with pauses causes the analysis to stop after each launch increment has been processed, while ***Display all continuously*** automatically analyses each increment and displays the results. There are no pauses between increments. The option ***Do not display any*** will not produce a dynamic display of intermediate results - only the final moment and shear envelope diagrams will be shown together with the maximum tip deflection.

If you wish to save the results generated at every launch increment tick the box labelled ***Save results for every launch increment***. Otherwise, only the final enveloped results will be retained by the system and be available for viewing, saving and printing (refer to *Section 6.2.3* below).

Refer also to *Section 6.4* for details of the analysis process.

6.2.2 Entering Model & Launching Data

The structure model is created by sequentially accessing the options in the *Model Data* group of buttons and specifying key launch parameters via the *Other Launch Data* group. A detailed description of all input data required to create the model is given in *Section 6.3*). Replace the default values shown in the various dialog boxes with data to suit your structural model.

Note that when accessing the *Model Data* options ACES will only display a schematic representation of an incrementally launched structure. This will assist you in identifying more easily the key parameters and dimensions required. A scaled representation of your model will be displayed whenever you return to the main control panel shown in *Section 6.2.1* above.

To begin the analysis click the button labelled ***Begin Analysis***. Refer to *Section 6.2.1* for further details regarding model analysis.

To exit from the Incremental Launching Module click the ***Close*** button followed by the ***X*** icon at the top right hand corner of the screen (or use the conventional *File/Exit* menu options).

6.2.3 Viewing, Printing & Saving Results

Viewing, printing and saving results is accomplished using the options available in the ***View Results***, ***Print*** and ***Save Tabulated Results*** button groups (*Section 6.2.1*).

View Results

View Graphical Envelopes

Click this button to view (and optionally print) the previously generated envelopes of moments and shears and the maximum nose tip deflection. These diagrams will represent envelopes corresponding to either the fully launched position of the structure or to the final position of a partly launched structure (as defined by the start and end positons in *Other Launch Data*).

View Tabulated Results

Click this button to view final results in tabular form. These results will represent envelopes corresponding to either the fully launched position of the structure or to the final position of a partly launched structure (as defined by the start and end positons in *Other Launch Data*). The report will contain a summary of uplift conditions encountered during the launch as well as envelopes of maximum and minimum moment, shear and torsion (for curved structures only) together with corresponding values of the other vectors. The launch position at which the maxima occured is also given.

Print

Input and Envelope Data

Click this button to print both the input data as well as the final results (described under *View Tabulated Results* - see above).

Graphical Model

Click this button to print a scaled representation of the structural model.

Select Printing Colours

Click this button to select black and white printing only. This option should be selected if your printer cannot print in colour. If you fail to do so you may find that some of the detail in your diagram may either be lost or too faint to see.

Save Tabulated Results

Final Envelope Only

Click this button to save both the input data as well as the final results (as described under *View Tabulated Results* - see the *first part of this section* above). Data will be saved to your nominated file in Ascii (text) form so that it can be imported into a spread-sheet or other post-processing application.

Intermediate Results

Click this button to if you wish to save the results of every launch increment. Note, however, that the box labelled ***Save results for every launch increment*** must be ticked prior to the analysis (refer to *Section 6.2.1*).

6.2.4 Printing Input Data

To print a summary of all input data click the button labelled ***Input and Envelope Data*** within the *Print* options group. Note that final results of moment and shear envelopes will also be printed (refer to *Section 6.2.3*, *View Tabulated Results*, for details).

6.2.5 Setting Colours & Line Styles

To change the default colours in which the geometry and results diagrams are displayed on screen (and printed in hard copy) click the *Colours and Line Styles* button. Two separate windows will be shown together:

Drawing Colours & Line Styles

This panel lists all drawing attributes (*Segment Fill, Launching Nose* etc) together with their currently set line style/colour combination (in the form of a numbered button). The number on the button corresponds to a line style shown in the right hand window (*Current Line Styles* - see below). To change the colour, or style, of one of the attributes click the button adjacent to its decription. A line style selector window will pop up. Click the radio button of the number corresponding to the style you require followed by *OK*.

Current Line Styles

This panel shows all currently available line styles. It serves no other purpose than to help you to identify and match the default line style numbers with their corresponding colours and line thicknesses.

6.2.6 Loading & Saving Model Files

Save Model

Click this button to save structural model data using the conventional Windows saving routine.

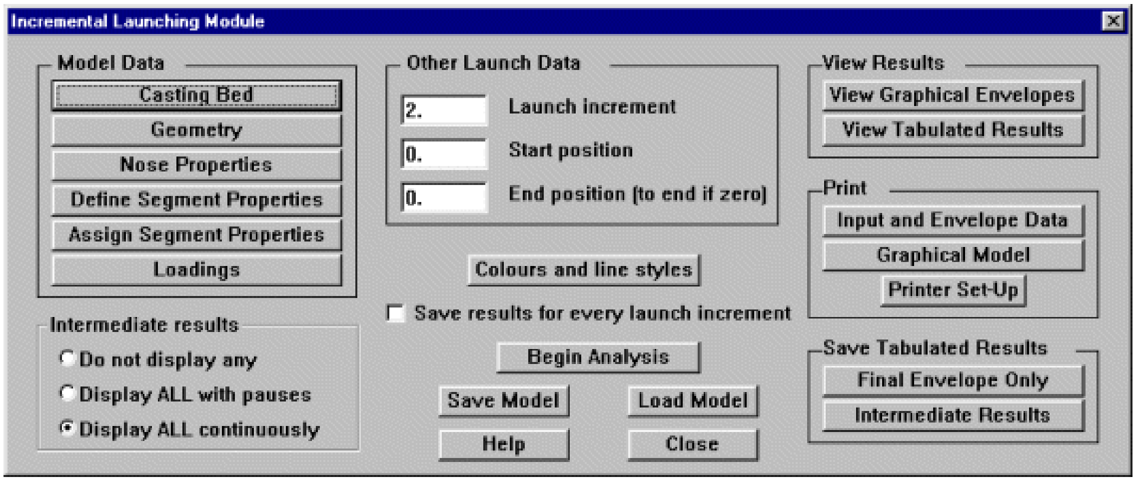
Load Model

To load another structural model into ACES you must first click the ***Close*** button then use the ***File/Open an existing file*** option in the top menu bar.

SECTION 6.3 - Input Data

6.3.1 General Launch Data

General launching data is specified in the ***Other Launch Data*** fields in the main ILB control panel (refer to the screen image shown below and in *Section 6.2*). Replace the default values to suit your structural model.



Launch Increment

The launch increment is the step interval at which the girder is to be analysed. It is also the approximate spacing at which the nodes defining the girder and nose members are placed. Note that the node coordinates may be modified slightly by the module during the course of the launch so that there is always a node over each support and at the nose/girder interface

Start & End Position

The start and end values designate the positions at which the launch will start and end. If both values are left zero then the girder will be launched over its full length. If the end position value is zero then the girder will be launched from the start value to the end of the girder. Note that values entered into these fields represent the distances in metres from the left abutment to the nose attachment.

Casting Bed Data

6.3.2 To enter casting bed data click on the button labelled ***Casting Bed*** in the main ILB control panel (refer to the screen image shown in *Section 6.3.1*). Replace the default values to suit your structural model.

Distance to Bed

This is the distance from the centre of the left abutment to the start of the casting bed (refer to *Figure 4*).

Length of Bed

This is the distance between the start and end supports in the casting bed (refer to *Figure 4*).

Number of Bed Supports

The number of notional support points to use in modelling the effect of the casting bed on the bottom soffit of the segment being cast (refer to *Figure 4*). Note that the casting bed would normally provide a continuous support, so this number represents an approximation to the actual support conditions.

The actual position of each point support within the casting bed must be specified by clicking the button labelled ***Bed Support Locations***. This will bring up a dialog box that will enable you to enter support distances.

All distances should be positive and given with respect to the start of the casting bed (assumed to be the edge nearest the left abutment). A negative value for a support distance will place that support to the *right* of the start of the casting bed. Bed supports need not be uniformly spaced.

Number of Intermediate Supports

Intermediate supports refers to supports which may be placed anywhere between the left abutment and the right hand edge of casting bed (see *Figure 4*). They are defined by clicking the button labelled ***Intermediate Support Locations*** and entering the distance in metres from the *left* abutment. If the distance entered is negative, the support will be placed to the *right* of the left abutment.

6.3.3 Model Geometry

To enter casting model geometry data click on the button labelled ***Model Geometry*** in the main ILB control panel (refer to the screen image shown in *Section 6.3.1*). Replace the default values to suit your structural model.

Radius

The radius of curvature of the final structure (refer to *Figure 1*). If the value is zero the girder is assumed to be straight. If the value is non-zero then a full three dimensional analysis will be performed. Note that for curved structures all dimensions, such as segment and span lengths, must be measured along the curved centreline of the structure. ***Support settlements are also not permitted for curved structures.***

Length of Nose

The nose length is measured from the right hand end of the girder (*Figure 2*). Note that the properties of the start (tip) of the nose correspond to the free end of the nose. Properties at the end of the nose correspond to the end attached to the girder (refer to *Figure 2*).

Overhang of Segment 1 from Left Abutment

The left overhang is the amount by which the girder end overhangs the left abutment bearing position when the girder is in its final position (refer to *Figure 4*).

Overhang of Last Segment Past Right Abutment

The right overhang is the amount by which the girder end overhangs the right abutment bearing position when the girder is in its final position (refer to *Figure 4*).

Number of Segments

Enter the number of segments then click the button labelled ***Segment Lengths*** to enter the length of each segment. Please note carefully that the total of the segment lengths *minus* the left and right overhangs must equal the distance between the abutments. *ACES* will warn you if these distances do not match (refer also to *Figure 3* and *Figure 4*). Each segment can have a different length.

Number of Spans

Enter the number of spans then click the button labelled ***Span Lengths*** to enter the length of each span. Note that span lengths are not necessarily the same as segment lengths. It is generally more usual to arrange the segments such that the final spans provide supports in the middle of a segment.

ACES inserts support points at the beginning and end of the structure and between each adjacent span. No distinction is made by the program between permanent supports and *temporary* supports. Temporary supports are often provided only during the launch phase and are subsequently removed once the entire girder is in its final position. Therefore, if you require temporary supports, spans must be defined in a way that includes all temporary supports.

6.3.4 Nose Properties

To enter nose property data click on the button labelled ***Nose Properties*** in the main ILB control panel (refer to the screen image shown in *Section 6.3.1*). Replace the default values to suit your structural model.

The nose is modelled using tapered members whose properties are calculated according to their longitudinal position between the start and end of the nose. If the nose properties are uniform then the start and end properties must be set to the same values. The depth of the nose is assumed to vary linearly along its length. Therefore ACES calculates the area, Ax, at any intermediate location on the basis of a linear variation along the length of the nose and calculates ***Iz*** and ***Ix*** values according to the cube of the section depth.

Ax at End and Start

These two parameters define the area of the nose at the start (the free end, or nose tip) and the end (connected to the first segment). The areas must be specified in metre units.

Iz at End and Start

The moment of inertia at the end and start (tip) of the nose. The inertias must be specified in metre units.

Ix at End and Start

Torsion at the end and start (tip) of the nose (these values are not used if the radius of curvature is zero).

E

Modulus of elasticity of the nose material.

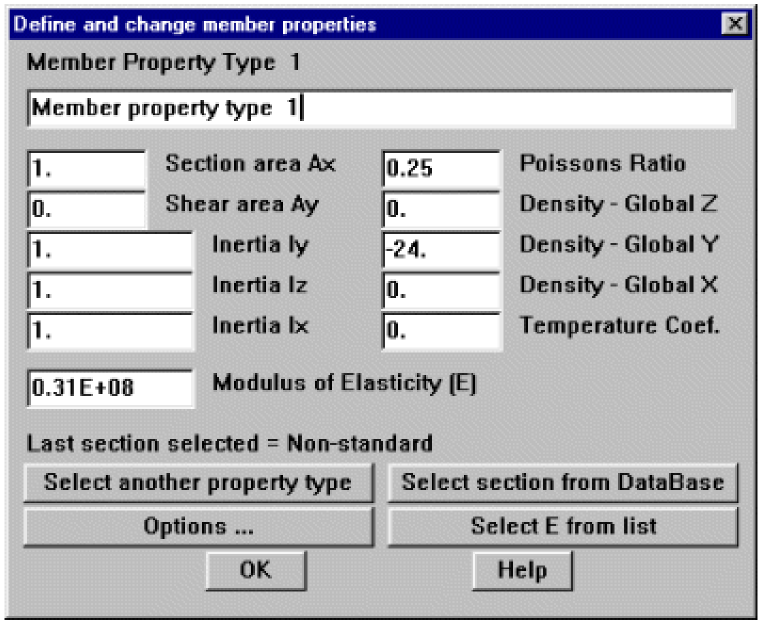
Density

The density of the nose material. This value *must* be negative. Note that ACES will calculate the self weight (dead load) effect by multiplying this value by the section area at the location concerned.

6.3.5 Segment Properties

To enter segment properties click on the button labelled ***Define Segment Properties*** in the main ILB control panel (refer to the screen image shown in *Section 6.3.1*). A *Member Properties Type* list will be displayed showing four default member property types already created by ACES.

Select the member property type you wish to edit then click the ***Edit*** button. If you wish to create a new member property type, click ***Add New***. In either case the member properties dialog box shown below will be displayed. Replace all default values to suit your structural model.



Description

Enter a member property type description into the field immediately below the label *Member Property Type X*.

Section Area Ax

Enter the area of the section in metre units. The *Shear Area*, *Ay,* is not required.

Inertias Ix, Iy, Iz

For a straight girder only the moment of inertia ***Iz*** is required. Set the other values to zero. For a curved structure, enter the torsional moment of inertia, ***Ix***. All inertias must be specified in metre units.

Poissons Ratio

Enter an appropriate value.

E

Modulus of elasticity of the segment material (metre units). Either enter a value or click the button labelled ***Select E from list*** to pick a value from a selected list of common material types.

Density - Global Y

The density of the nose material must be entered about the *Global Y* axis only and *must* be negative. Note that ACES will calculate the self weight (dead load) effect by multiplying this value by the section area at the location concerned.

Select Another Property Type

Click this button if you wish to edit or create another member property type.

Select Section from Database

Click this button if you wish to select a standard section from the database.

Options...

Click this button if you wish to use the ACES section properties calculator to calculate the section properties of the section. Refer to *Part 2* of the Manual for a description of this feature.

6.3.6 Assigning Segment Properties

To assign segment properties to segments click on the button labelled ***Assign Segment Properties*** in the main ILB control panel (refer to the screen image shown in *Section 6.3.1*). A *Member Properties Type* list will be displayed showing all currently defined member property types together with a dialog box that will allow you to specify the appropriate property type for each segment.

The quickest and easiest way to assign property types for a model with many segments is to first preset *all* segments to the most common property type, then individually change only those segments whose property type number is different.

SECTION 6.4 - Analysis

6.4.1 Setting Analysis Options

During analysis ACES will display a launching screen that can, if required, dynamically show the progress of the analysis. This mode can be set by selecting either one of the last two options in the *Intermediate Results* group (refer to *Section 6.2* for details). If one of these options is checked, the moment and shear diagrams are not only displayed for every increment but they are also superimposed onto the *cumulative* envelopes of maximum and minimum moments and shears.

Display all with pauses - this option causes the analysis to stop after each launch increment has been processed. You will be prompted to continue with the next increment.

Display all continuously - this option automatically analyses each increment and displays the results. There are no pauses between increments - the analysis proceeds continuously, without a break.

Do not display any - this option will *not* produce a dynamic display of intermediate results. Only the final moment and shear envelope diagrams will be shown together with the maximum tip deflection.

If you wish to save the results generated at every launch increment tick the box labelled ***Save results for every launch increment***. Otherwise, only the final enveloped results will be retained by the system..

6.4.2 Performing the Analysis

To analyse the model click the button labelled ***Begin Analysis*** (*Section 6.2*). For each launch increment the system creates an *ACES.INP* file in the standard *ACES Command Language* format, then analyses the current structural configuration, displays the moment and shear diagrams for that configuration and superimposes onto each diagram the cumulative moment and shear envelopes. The analysis may be aborted at any stage by hitting the *ESC* key.

Note that if a support experiences uplift at any stage, that support will be removed for that stage and the structure re-analysed. A listing of the supports experiencing uplift at each launch position will be included as part of the final results. Launching will terminate with a diagnostic error if the structure becomes unstable because of insufficient supports. Common causes for this condition may include:-

- Inappropriate support positions
- Inadvertent specification of duplicate support locations
- Support settlements specified at a launch position where the girder has become simply supported as a result of supports being removed due to uplift

SECTION 6.5 - Output Results

6.5.1 Viewing Envelopes Graphically

Click ***View Graphical Envelopes*** in the *View Results* options group to view (and optionally print) the previously generated envelopes of moments and shears and the maximum nose tip deflection. These diagrams will represent envelopes corresponding to either the fully launched position of the structure or to the final position of a partly launched structure (as defined by the start and end positons in *Other Launch Data*).

6.5.2 Viewing Envelopes in Tabular Form

Click ***View Tabulated Results*** in the *View Results* options group to view final results in tabular form. These results represent envelopes that correspond to either the fully launched position of the structure or to the final position of a partly launched structure (as defined by the start and end positions in *Other Launch Data*).

The report contains:

- a summary of uplift conditions encountered during the launch

- for each section along the structure (given by the parameter DIST as measured from the left end of the fully launched structure), the maximum and minimum moment, shear and torsion (the latter for curved structures only)

- corresponding values of all other vectors

- the launch position (measured from the tip of the nose) at which the maxima occured

6.5.3 Viewing Reactions & Deformations

To view reaction envelopes and the maximum tip deflection click ***View Tabulated Results*** in the *View Results* options group. Reaction maxima and minima are given for all permanent supports (piers and abutments) as well as temporary, intermediate and casting bed supports. The launch positions at which maximum values occur are also given.

6.5.4 Printing Results

To print a summary of all input data and results click the button labelled ***Input and Envelope Data*** within the *Print* options group. All results viewable in tabular form (see Sections 6.5.2 and 6.5.3 above) will be printed.

6.5.5 Saving Final Results

Click the button labelled ***Final Envelope Only*** within the *Save Tabulated Results* options group to save both the input data as well as the final results. Data will be saved to your nominated file in ASCII (text) form so that it can be imported into a spread-sheet or other post-processing application.

6.5.6 Saving Results at Intermediate Launch Positions

Click the button labelled ***Intermediate Results*** within the *Save Tabulated Results* options group if you wish to save the results of every launch increment. Note, however, that the box labelled ***Save results for every launch increment*** on the main control panel must be ticked prior to the analysis.

Data will be saved in standard ACES output format to a series of default files labelled *Interm_Launch_Env_0000.txt* (in Ascii text form). Although you may change the results file name, the last five characters (before the *.txt* extension) *must* end with an underscore and four zeros (viz: *filename _0000*).

SECTION 6.6 Example - 6 span structure

6.6.1 Input Data

An example of the input data for a six span incrementally launched girder is given below. It is written by ACES-ILB into the front portion of the output report and may be viewed and printed. In order to check that your version of the program is working properly enter the data into ILB manually and perform the analysis.

HEADING
Example Problem
JOB TITLE
6 span bridge
DESIGNER
P.C.
DATE RUN
10 Aug 2001
UNITS
m, kN
Plan curve radius (0 for straight +ve curve left -ve curve right)
0.000000
Length of nose (m)
14.0000
Launch increment (m)
1.65000
Launch Range from X1 to X2. If both zero then increment over full length. (m)
45.0000 60.0000
Distance to casting bed (behind abutment)
9.10000
Length of casting bed (m)
16.2000
Number of supports in casting bed
4
Distance from start of casting bed to casting bed support (m)
1 16.2000
2 10.8000
3 5.40000
4 0.000000
Number of intermediate supports between casting bed and abutment
2
Distance from abutment to Intermediate supports (m)
1 2.50000
2 5.50000
Left overhang of main girder from C/Line abutment bearing (final)
0.510000
Right overhang of main girder from C/Line abutment bearing (final)
0.510000
Number of segments and number of segment types (max 100)
7 4
Segment lengths (m)
1 15.2600 1
2 16.5000 1
3 16.5000 1
4 16.5000 1
5 16.5000 1
6 16.5000 1
7 15.2600 1
Number of spans (max 100)

6
Span lengths (m)
1 23.0000
2 16.5000
3 16.5000
4 16.5000
5 16.5000
6 23.0000
Ax for start (tip) of nose
0.100000
Ax for end of nose (at junction with Main Girder)
0.108400
Iz for start (tip) of nose
0.257000E-01
Iz for end of nose
0.393000E-01
Ix for start of nose (not used if curve radius is 0)
0.100000E-01
Ix for end of nose (not used if curve radius is 0)
0.100000E-01
E for nose
0.200000E+09
BFY for nose (MUST be negative)
-91.7000
Ax for girder segment type 1
6.24900
Iz
1.58100
Ix (not used if curve radius is 0)
0.000000
E
0.310000E+08
BFY for girder (MUST be negative)
-25.5000
Ax for girder segment type 2
1.00000
Iz
1.00000
Ix (not used if curve radius is 0)
1.00000
E
0.000000
BFY for girder (MUST be negative)
0.000000
Ax for girder segment type 3
1.00000
Iz
1.00000
Ix (not used if curve radius is 0)
1.00000
E
0.000000
BFY for girder (MUST be negative)
0.000000
Ax for girder segment type 4
1.00000
Iz
1.00000
Ix (not used if curve radius is 0)
1.00000
E
0.000000
BFY for girder (MUST be negative)

0.000000
Dead load factor (0 if DL is not required)
1.00000
Construction live load per segment (kN/m) NOTE: MUST BE NEGATIVE
1 -5.00000
2 -5.00000
3 0.000000
4 0.000000
5 0.000000
6 0.000000
7 0.000000
Thermal moment and prestress moment; + for hog; per segment
1 0.000000 0.000000
2 0.000000 0.000000
3 0.000000 0.000000
4 0.000000 0.000000
5 0.000000 0.000000
6 0.000000 0.000000
7 0.000000 0.000000
Support settlement (one for each abutment and piers, MUST be -ve.)
1 0.000000
2 0.000000
3 0.000000
4 0.000000
5 0.000000
6 0.000000
7 0.000000
Support springs (0 for no spring support)
1 0.000000
2 0.000000
3 0.000000
4 0.000000
5 0.000000
6 0.000000
7 0.000000
Number of applied point loads (max of 20)
0
Point loads (-P, Distance from nose attachment, Launch distance when applied)
0.000000 0.000000 0.000000
Printer Setting
1
Structure outline colour (DOS colour 0-15 [0 for black])
12
Segment fill colour (DOS colour 0-15 [0 for black])
9
Casting bed supports colour (DOS colour 0-15 [0 for black])
3
Top dimensions colour (DOS colour 0-15 [0 for black])
2
Bottom dimensions colour (DOS colour 0-15 [0 for black])
8
Launching nose colour (DOS colour 0-15 [0 for black])
4
Diagram border colour (DOS colour 0-15 [0 for black])
7
Bending moment diagram colour (DOS colour 0-15 [0 for black])
14
Shear force diagram colour (DOS colour 0-15 [0 for black])
12
Bending moment envelope max colour (DOS colour 0-15 [0 for black])
14
Bending moment envelope min colour (DOS colour 0-15 [0 for black])
13

Shear force envelope max colour (DOS colour 0-15 [0 for black])
12
Shear force envelope min colour (DOS colour 0-15 [0 for black])
11
Temporary envelope colour (DOS colour 0-15 [0 for black])
7

6.6.2 Results

Please note that the results shown below only represent a sub-set of the complete output report. They should prove sufficient in performing an audit check on the version of the program you are currently using.

SUMMARY OF NEGATIVE SUPPORT CONDITIONS

Negative Reactions Found at Position 45.0000
The following support nodes were removed
Stress Bed 5
Stress Bed 6

Negative Reactions Found at Position 46.6500
The following support nodes were removed
Stress Bed 5
Stress Bed 6

Negative Reactions Found at Position 48.3000
The following support nodes were removed
Stress Bed 5
Stress Bed 6

Negative Reactions Found at Position 49.9500
The following support nodes were removed
Stress Bed 5
Stress Bed 6

Negative Reactions Found at Position 51.6000
The following support nodes were removed
Stress Bed 5
Stress Bed 6

Negative Reactions Found at Position 53.5000
The following support nodes were removed
Stress Bed 5
Stress Bed 6

Negative Reactions Found at Position 54.9000
The following support nodes were removed
Stress Bed 5
Stress Bed 6

Negative Reactions Found at Position 56.5500
The following support nodes were removed
Stress Bed 5
Stress Bed 6

Negative Reactions Found at Position 58.2000
The following support nodes were removed
Stress Bed 5
Stress Bed 6

Negative Reactions Found at Position 59.8500
The following support nodes were removed
Stress Bed 4

Stress Bed 5
Stress Bed 6

FINAL RESULTS

Maximum downward nose tip deflection = -5.6mm
at launch position = 54.90

ENVELOPE OF SHEARS (kN)

V = Shear
Mc = Corresponding Moment
Lp = Launch Position

	Dist	Vmin	Mc	Lp	Vmax	Mc	Lp
33	48.26	0.0	0.0	0.00	288.6	0.0	58.20
40	58.07	-35.0	154.9	53.50	567.4	154.9	53.50
50	73.76	-1422.6	5761.8	53.50	1799.4	5761.8	53.50
60	88.76	-1614.9	3640.8	59.85	785.4	-2764.9	45.00
70	104.32	-1806.0	5532.8	45.00	1314.1	5532.8	45.00
80	119.24	-143.8	238.0	46.65	73.1	282.0	48.30
85	127.02	-38.8	0.0	56.55	0.0	0.0	0.00
Max		-1882.5	6549.5	48.30	1859.6	6148.7	45.00

ENVELOPE OF MOMENTS (kN.m)

M = Moment
Vc = Corresponding Shear
Lp = Launch Position

	Dist	Mmin	Vc	Lp	Mmax	Vc	Lp
33	48.26	0.0	0.0	0.00	0.0	0.0	0.00
40	58.07	-809.8	0.9	58.20	154.9	-35.0	53.50
50	73.76	-2652.6	775.8	59.85	5761.8	-1422.6	53.50
60	88.76	-4464.5	-75.3	49.95	3640.8	-1614.9	59.85
70	104.32	-1796.5	-106.1	56.55	5532.8	-1806.0	45.00
80	119.24	-219.3	-54.5	59.85	282.0	73.1	48.30
85	127.02	0.0	0.0	0.00	0.0	0.0	0.00
Max		-4685.6	68.2	45.00	6580.8	-1877.9	49.95

REACTIONS (kN)

SUPPORT	Minimum Reaction	Launch Position	Minimum Reaction	Launch Position
1	2895.4	48.30	3347.3	45.00
2	3120.2	45.00	3528.1	51.60
3	94.7	45.00	2139.7	59.85
4	38.8	56.55	127.5	59.85
5	0.0	0.00	0.0	0.00
6	0.0	0.00	0.0	0.00
7	0.0	0.00	0.0	0.00

REACTIONS ON STRESSING BED (kN)

SUPPORT	Minimum Reaction	Launch Position	Minimum Reaction	Launch Position
1	0.0	0.00	0.00	0.00
2	288.6	58.20	1579.80	53.50
3	602.4	53.50	1556.9	59.85
4	134.0	45.00	1112.5	48.30

REACTIONS ON INTERMEDIATE SUPPORTS (kN)

SUPPORT	Minimum Reaction	Launch Position	Minimum Reaction	Launch Position
1	0.00	0.00	0.00	0.00
2	0.00	0.00	0.00	0.00

PART 7

CONTINUOUS BEAM MODULE

ACES PART 7: Continuous Beam Module

Main Index

7.1 General Concepts & Features

7.1.1 General Modelling Process
7.1.2 Limitations & Restrictions

7.2 Model Data

7.2.1 Job Identification
7.2.2 Geometry
7.2.3 Supports
7.2.4 Section Properties
7.2.5 Loadings
7.2.6 User-Defined Vehicles

7.3 Analysis

7.3.1 Analysis Options
7.3.2 Performing the Analysis

7.4 Output Results

7.4.1 Individual Load Cases
7.4.2 Envelopes

7.5 Settings

7.5.1 Default Program Settings
7.5.2 Browser & User Model File Paths
7.5.3 Line Colours & Styles

CBM Section 7.1 - OVERVIEW

1. General Modelling Process

A continuous beam model is created as follows:

- Enter the requisite jod identification information on the *Job ID* panel or browse for an existing CBM file.
- Enter geometry data on the *Geometry* tab.
- Set the support conditions on the *Supports* tab.
- Enter the relevant section properties on the *Sec Props* tab.
- Define load cases on the *Loadings* tab (if you wish to use your own vehicle it must first be created on the *Vehicles* tab).
- Select the load cases you wish to analyse then perform the analysis via the *Analysis* tab.
- Interrogate the results on the *Results* tab.

2. Limitations & Restrictions

The following limitations and restrictions apply:

- Tapered section properties at intermediate points are based on averaging member start and end properties.
- Only 3 intermediate results points are allowed per span

CBM Section 7.2.1 - JOB IDENTIFICATION

This panel enables job ID data to be cleared, loaded, entered, edited, or saved to a user-specified directory.

Number of Spans

Any number of spans can be specified. However, it may be difficult to interpret some of the graphical diagrams if too many spans are defined. To generate a beam on an elastic foundation enter a large number of spans but set the span lengths and span divisions to a small value. An elastomeric support can then be assigned to each support point.

Units

Specification of units is purely for documentation purposes. The user must ensure that all units are consistent.

Bridge Design Code

This option allows you to automatically create and analyse vehicle load cases in accordance with a specified Bridge Design Code (subject to certain limitations). Only the Australian Standard 5100 (2004) catering for the MS1600 traffic loading has been implemented at this stage. Please refer to *Section 7.2.5* for further details and for restrictions and limitations on each bridge design code.

CBM Section 7.2.2 - GEOMETRY

This panel enables structure geometry data to be edited or entered. Note that support data is specified via the "*Supports*" tab. *Click here to return to the Main CBM Index*

No Divisions

During analysis ACES divides each span into the number of divisions specified in this column and creates a "node" between each segment. Results are reported, both graphically and in tabular form, at these nodes. Therefore, if you wish to interrogate results at various points along each span and you want smooth-looking moment and deflection diagrams, specify a reasonable number of divisions (e.g. 10, 15, 20 etc).

WARNING! *When ACES generates the continuous beam model and performs the analysis, nodes defined by the* ***No. Divns*** *(number of divisions) value in this table will be moved so that they line up with any "additional" span points that may have been inserted into the model. In other words, new nodes are* *not* *created but the position of some of the initial regularly spaced sections are simply adjusted.*

DLA

This field allows an optional span-based Dynamic Load Allowance (DLA) factor to be specified as a value between 0 and 1. It cannot exceed unity. ACES will multiply all vehicle axle loads in a particular span by one plus the relevant factor prior to performing the analysis (i.e. by 1+DLA).

Note that factors entered into this table will take precedence over any vehicle-specific DLA factors. They will also be applied to all vehicles in all load cases irrespective of whether the DLA check-box on the vehicle loadings form has been ticked on or off.

If all DLA factors are set to zero in this table it is still possible to apply a global vehicle DLA factor. Refer to *Section 7.2.5*, *Loadings*, for further details.

Point 1, Point 2, Point 3

ACES-BEAM allows you to specify up to three points on each span at which a unique result is required. Internally, the program does this by moving the closest defined nodes in the model to coincide with these new points. For long spans you may therefore wish to increase the number of span divisions in order to create sufficient nodes that will lie in close proximity to the added points. (This will create smoother and more accurate bending moment,deflection and shear force diagrams). Note that the distance to each node is measured from the left support of each span and *not* from the far left abutment.

Copy row data into all other spans

This option allows all data in the current row to be copied into all other remaining rows in the table. Place the cursor in any cell on the source row then click this button to perform the copying.

Print all geometry data

This button allows all geometry and support data to be viewed through a browser and optionally printed. Note that the location (path) for the browser must be specified through the *Settings* tab. (The default is *C:\Program Files\Internet Explorer\iexplorer.exe*).

CBM Section 7.2.3 - SUPPORTS

This panel enables support data to be entered and edited.

Type

ACES-BEAM allows six support types:-

Cantilever (C), *Hinge* (H), *Fixed* (F), *Pinned* (P, also referred to as a *knife edge* support), *Roller* (R) and *Spring* (S, also referred to as an *elastic* support).

Enter a letter corrresponding to the required support type (upper or lower case are both permitted) and press *ENTER*. The pin support has restraints in both the *X* and *Y* directions; the roller only a restraint in the vertical (*Y*) direction.

Elastic Constant 'k'

If spring (elastic) supports are specified enter an elastic constant 'k' for each spring support (in consistent units). The system will ignore elastic constant values if the associated support type is not set to '*S*'.

Other Buttons

Most of the buttons on this tab assist in quickly setting global values for support type, elastic constant and for selectively displaying diagram attributes.

Print geometry & support data

This button allows all geometry and support data to be viewed through a browser and optionally printed. Note that the location (path) for the browser must be specified through the *Settings* tab. (The default is *C:\Program Files\Internet Explorer\iexplorer.exe*).

CBM Section 7.2.4 - SECTION PROPERTIES

1. Introduction

The current version of ACES-BEAM allows both uniform (***U***) and variable (***V***) girders to be specified and analysed. Although all profiles can be created using the variable properties dialog box, the program will eventually be enhanced with wizards that will allow the quick generation of haunched (H), tapered (T) and plated (P) girder types.

Enter a letter corrresponding to the required structure type (upper or lower case are both permitted) and press *ENTER*. Refer to *Section 3* below for details regarding variable section properties.

2. Uniform Section Properties

Type

Enter the letter "***U***" in either upper or lower case and press *ENTER*. Refer to *Section 3* below for details regarding variable section properties.

E, Iz, Area, Density

Enter values as required (in consistent units). When analysing a structure uniform along its entire length, parameters ***E, Iz*** and ***Area*** can be arbitrarily set to 1. Density is only required if you want ACES-BEAM to calculate and apply self-weight (dead load) as a load case. (Refer also to the item at bottom regarding the diagramatic representation if ***Iz*** along the structure).

Ix (Torsion)

This parameter is only required for curved structures.

Copy current span data into all other spans

This option allows all data in the current span (or row) to be copied into all other remaining spans (rows) in the table. Place the cursor in any cell on the source row then click this button to perform the copying.

Other Buttons

Most of the other buttons on this tab assist in quickly setting global values for *E, Ix, Iz, Area* and *Density*.

Print section props

This button allows all section properties data to be viewed through a browser and optionally printed. Note that the location (path) for the browser must be specified through the *Settings* tab. (The default is *C:\Program Files\Internet Explorer\iexplorer.exe*).

Iz Diagram

The diagram of ***Iz*** only gives an indicative view of the stiffness distribution along the structure. Although uniform sections are accurately shown, linearly varying regions are represented as averaged blocks.

3. Variable Section Properties

To enter variable section properties for a particular span specify its property ***Type*** as ***V*** then click on the *Other Properties* button located just above *Help* (in the bottom right corner of the form).

Dist

This is the distance along the current span at which ***Iz*** changes. The first section *must* be defined at the left support (*Dist = 0*) and subsequent section distances are measured from the *left* support of the *current* span. Enter values as required (in consistent units).

WARNING! *When ACES generates the continuous beam model and performs the analysis, nodes defined by the* ***No. Divns*** *(number of divisions) value in the* ***Geometry*** *table will be moved to coincide with points that define a change in section properties. Therefore, the regularly spaced sections initially created in the* ***Sec Props*** *tab will, in the final ACES model, be adjusted to account for the locations at which the* ***Iz*** *profile changes.*

Note carefully the above warning when defining cut-off points that do *not* coincide with a standard sub-division node point. If you wish to preserve the exact sub-division spacing then you must adjust the ***Dist*** parameter so that it coincides with one of the generated nodes. Alternatively, use a larger number of subdivision points (for example 20 instead of the default value of 10).

Iz, Ix, Area

Enter values as required in consistent units. To set properties at the start of the span equal to those at the end of the previous span click the button labelled *Set start properties*. (Note: Ix, the torsional stiffness, will only be required for curved beam models or those in which a vehicle loading is transversely offset from the deck centreline).

The dialog box also has two other options for quickly setting properties for the current span. They are located immediately underneath the properties table. One allows you to set the properties of the current span to those of a previously defined span while the other allows you to perform the same task but with properties mirror-imaged. Type the appropriate span number into the relevant field and press the *ENTER* key.

To create a sharp "step" variation in ***Iz*** enter two identical values for ***Dist*** with two different ***Iz*** values. If adjacent ***Dist*** values differ (as well as their respective ***Iz*** values) the girder profile is assumed to vary linearly between the two sections. However, the analysis will be based on a *tapered* member properties profile i.e., ACES will vary ***Iz*** and ***Ix*** between node points using a cubic parabola distribution while ***Area*** will vary linearly.

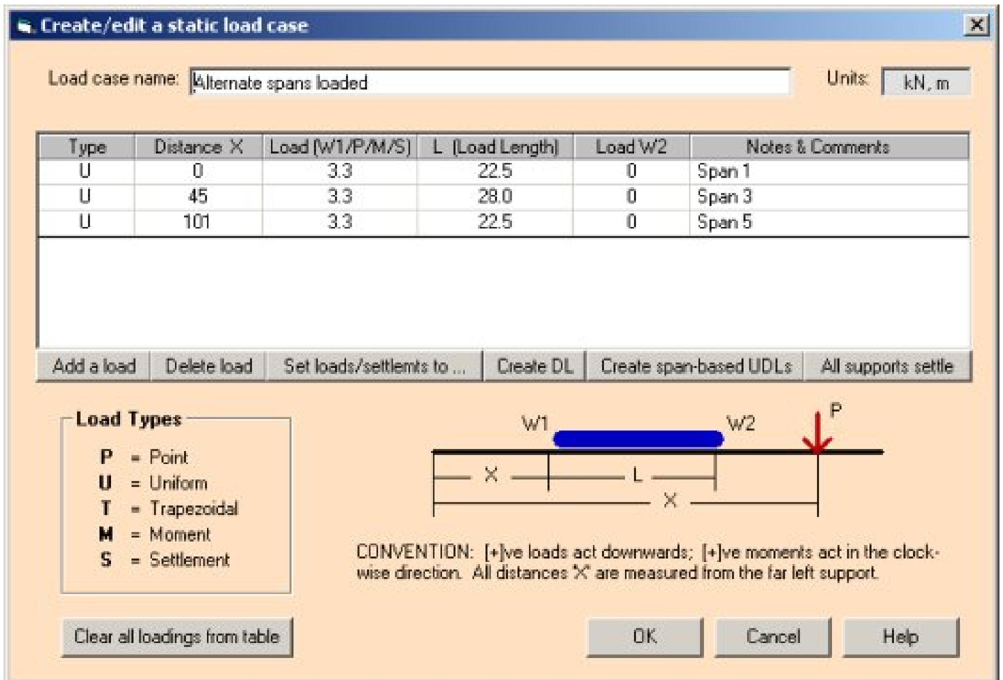
The last ***Iz*** value entered into the table will apply uniformly from that section through to the end of the *current* span.

Note that the ***Iz*** *diagram only gives a schematic view of the stiffness distribution along the structure. Although uniform sections will be accurately shown, linearly varying regions are only represented as averaged blocks.*

CBM Section 7.2.5 - LOADINGS

1. Add/Edit a Static Load Case

Click this button to create a <u>*new*</u> static load case i.e. one consisting of member loads or support settlements. To edit an existing load case, highlight it in the *Current Load Cases* window then click the *Edit load case* button or double click the load case you wish to edit. In either case the following dialog box will be displayed:



Enter a load case name then click one of the buttons immediately underneath the loading table to add an appropriate loading. Click *OK* to accept the values and close this window.

Add a load

Clicking this button will add a single row to the table. By default the *Type* will be set to "***U***" (uniform load), the distance ***X*** will be set to the full length of the structure and a loading value of unity will be inserted into the "*W1/P/M/S*" field. In effect, this constitutes a UDL distributed along the entire length of the structure and as such can easily represent dead load, super-imposed DL and so on. You may add as many rows as needed. Note that the load case can be a mixture of all valid loading types and that the loaded <u>*length*</u> must be provided for UDL and trapezoidal loadings.

When modelling settlements, distance ***X*** must be measured from the far left abutment to the required support.

Delete load

Clicking this button will delete the loading row in which the cursor is located. *Note: No warning is given if this button is clicked, so be sure that you do want to delete it!*

Set loads/settlements to..

Click this button to enter a global value for the load/moment/settlement *(W1/P/M/S)*. A small dialog box will pop up to enable you to do this.

Create DL

Click this button if you want *ACES-BEAM* to automatically create a self-weight load case. A separate loading row will be generated for each span, with the load value, ***W1***, calculated by multiplying the area of each span by the respective span density. For spans with variable section areas the system will take an average of the areas between adjacent sections when calculating ***W1***.

Create span-based UDLs

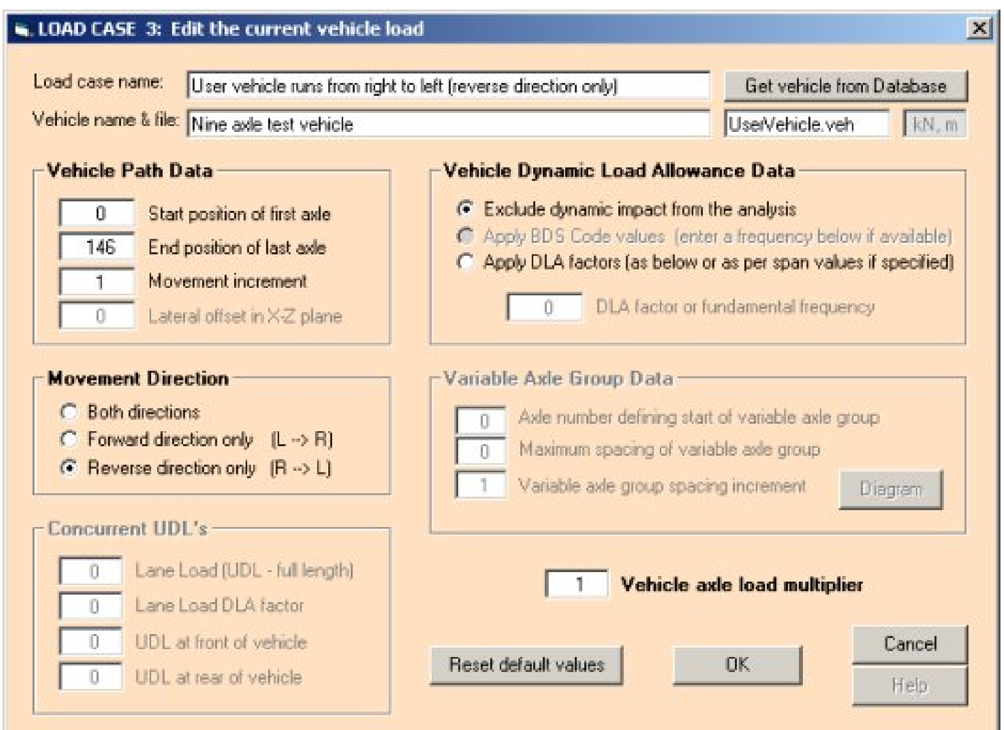
This is similar to the previous button, *Create DL*, with the exception that the separate loading rows generated for each span have a load value set to the default standard *Lane Load* as defined in the *Settings* tab (e.g. 12.5 kN/m). Alternating patch loads can be quickly created by deleting unwanted spans.

All supports settle

Click this button if you want *ACES-BEAM* to create a support settlement load case. A separate loading row will be generated for every support, with the settlement value for each set to 0.05. The distance of each support from the far left abutment will be entered into the *Distance X* column. In order to create the required support settlement configuration, delete the unwanted row(s).

2. Add/Edit a Vehicle Load Case

Click this button to create a *new* vehicle load case. To edit an existing load case highlight it in the *Current Load Cases* window then click the *Edit load case* button or double click the load case you wish to edit. In either case the following dialog box will be displayed:



Enter a load case name then click *Get vehicle from Database* to apply a vehicle to this load case. If you have defined your own vehicle in the *Vehicles* tab it will automatically appear in the *Vehicle name & file* fields (with *UserVehicle.veh* inserted in the file name field). In this case there is no need to get a vehicle from the database - your own vehicle will be applied.

2.1 Vehicle Path Data

Start position of first axle: This defines the position at which the leading axle of the vehicle will be located on the structure. (The default is zero, over the left-hand abutment). To move the vehicle in the *reverse* direction ensure that the start position is *less* than the end position and set the *Movement Direction* to *Reverse*.

End position of last axle: This defines the position at which the *last* axle of the vehicle will be located on the structure. (The default is set to the overall length of the structure i.e., over the right-hand abutment). If the vehicle is moved in *both* directions the leading axle will be placed in this position for the return trip. To move the vehicle in the reverse direction only ensure that the start position is less than the end position and set the *Movement Direction* to *Reverse*.

Movement increment: This defines the incremental distance by which the vehicle will be moved when stepping from the start position to the end position. The default is set to 1.

Lateral offset in the X-Z plane: This parameter allows the vehicle to be notionally offset from the centreline of the structure in order to calculate torsional effects. *This feature has not yet been implemented in this version.*

2.2 Movement Direction

Both directions: The leading axle will be moved from its start position until the last axle is at the designated end position. The first axle will then be placed at the end position and moved backwards until the last axle is at the start position. This option is disabled for MS1600 vehicles auto-generated using the AS5100 (2004) option (see *Clause 3* below).

Forward direction only (L-R): The leading axle will be moved from its start position at the left end of the structure until the last axle is at the designated end position (to the right of the start position).

Reverse direction only (R-L): The leading axle will be moved from the designated end position of the last axle (located to the right of the structure) until the last axle is at the designated start position of the first axle. This option is disabled for MS1600 vehicles auto-generated using the AS5100 (2004) option (see *Clause 3* below).

2.3 Vehicle Dynamic Load Allowance Data

Note! If DLA factors have been entered into the geometry data table they will take precedence over the vehicle-specific value on this form. Span-based DLA values will be applied to all vehicles in all load cases irrespective of whether the DLA flag described below is toggled on or off.

Exclude dynamic impact from the analysis: If this option is toggled *on* the Dynamic Load Allowance factor will *not* be applied to the vehicle, even if a value has been defined in the DLA field. Note, however, that even if this option is switched off and a DLA value exists, then DLA factors entered into the geometry data table they will take precedence over the vehicle-specific DLA value.

Apply DLA values: If the '*Exclude DLA...*' flag is off, vehicle axle loads will be multiplied by (*1+DLA*), where *DLA* is the value shown in the DLA field. If this is zero, then span-based factors entered into the geometry data table will be used. If they are also zero then DLA will not be applied. You may wish to include a reference to the DLA factor in the load case title in order to quickly identify that it has been applied to the vehicle.

DLA factor or fundamental frequency: If this is a DLA value it must be less than 1 and greater than zero. Vehicle axle loads will be multiplied by (*1+DLA*) prior to the analysis being performed.

2.4 Vehicle Axle Load Multiplier

Multiplier factor: Vehicle axle loads will be multiplied by this factor prior to the analysis being performed. You may wish to include a reference to this factor in the load case title in order to quickly identify that it has been applied to the vehicle. (Note also that any DLA factor specified for this vehicle will also be applied to the axle loads).

2.5 Vehicle Axle Group Data

Axle number defining start of axle group: This represents the number of the last axle immediately before the gap defining the variable spacing (numbering begins from the leading axle).

Maximum spacing of variable axle group: This is the maximum allowable spacing of the variable axle group. It must be an integer multiplier of the spacing increment. If not, the increment distance will be internally changed during the analysis so that it is.

Variable axle group spacing increment: The spacing increment. Note that the maximum variable axle group spacing must be an integer multiplier of this increment. If it's not, the increment distance will be internally changed during the analysis so that it is. However, this calculated value will not be reflected back into this field.

3. AS 5100 (MS1600) Loading Option

This option allows the M1600 and S1600 traffic loadings is AS5100 (2004) to be automatically analysed. No other load cases (apart from two pre-generated static loadings to cater for dead load and superimposed DL) are permitted and any that do exist will be deleted. The following restrictions and limitations apply:

1. Each vehicle (M1600 and S1600) must be analysed separately and the resulting envelopes combined manually (or the results saved then imported into EXCEL).
2. Both vehicles can only travel in one direction (from left to right).
3. The *Polarise* option on the envelope dialog box is permanently set ON. This forces ACES to create all possible envelopes and sum them automatically to determine the worst effects. (Refer to Section 7.4.0 for further details).
4. There are no restrictions on the maximum axle group spacing or the spacing increment. Note, however, that if the axle group spacing is not an exact multiple of the increment, the latter will be changed in order to accomplish this.
5. Default vehicle parameters (such as DLA values, axle group spacing and increment etc) may need to be changed to suit local conditions.
6. Apart from two pre-generated static loadings (to cater for dead load and superimposed DL) and a single vehicle load case, no other loadings are permitted in the run.

4. Print Load Case Data

The two top buttons in the right-hand button group allow load case data to be viewed through a browser then optionally printed. The top button will display data for the current load case. The second button will display data for all load cases.

Note that the location (path) for the browser must be specified through the *Settings* tab. (The default is *C:\Program Files\Internet Explorer\iexplorer.exe*).

5. Copy & Delete Load Cases

The *Copy* and *Delete* buttons allow load cases to be copied and deleted. Copies can be made of any of the currently defined load cases (including moving vehicle loadings).

CBM Section 7.2.6 - USER VEHICLES

Vehicle Name

If you are creating a new vehicle enter its name/title or description. If you wish to modify an existing vehicle that already exists in the vehicle database click the button labelled *Browse for vehicle*.

No. of Axles

Enter an integer value and hit *ENTER*. A maximum of 99 axles is allowed.

Vehicle Axle Data

For each axle enter the axle load and axle spacing. Spacing is relative to the previous axle, so the first axle must have a spacing of zero. To clear axle data only, click the button labelled *Clear axle data*. To clear *all* data on the form click *Create a new vehicle*.

Replicate Rows

This option allows all data for the current axle (or row) to be copied into all other remaining axles (rows) in the table. Place the cursor in any cell on the source row then click this button to perform the copying.

Create a New Vehicle

All vehicle data on the form will be cleared (including the vehicle name, number of axles and axle data table). To clear axle data only, click the button labelled *Clear axle data*..

Print Vehicle Details

This button allows vehicle data to be viewed through a browser and optionally printed. Note that the location (path) for the browser must be specified through the *Settings* tab. (The default is *C:\Program Files\Internet Explorer\iexplorer.exe*).

Save Vehicle to Database

This option allows all vehicle data to be saved to the ACES vehicle database. When clicked a dialog box will appear that asks you to nominate an axle width (the default is 2 units) and vehicle file name. A file extension of ***.veh*** will be automatically appended to the name you specify. This must *not* be changed.

Note that if you only wish to perform a one-off analysis it is *not* essential to save the vehicle to this database. ACES-BEAM will still allow you to apply this vehicle to one or more load cases even if it has not been saved.

CBM Section 7.3 - ANALYSIS

You must perform an analysis before the RESULTS tab can be used to interrogate the output!

Analysis Options

Select the load cases you would like to be included in the analysis by ticking the appropriate check boxes. Alternatively, use the *Select all load cases* or *Deselect all load cases* buttons to quickly initialise the list to one of the two required states.

Performing the Analysis

Once the required load cases have been selected click the large ***ANALYSE*** button to begin the solution process. If an error is encountered during the data validation process ACES will display a report showing the generated input file together with an error message at the end of the file. This will normally indicate the problem encountered (usually some inadequacy in defining member properties or attributes).

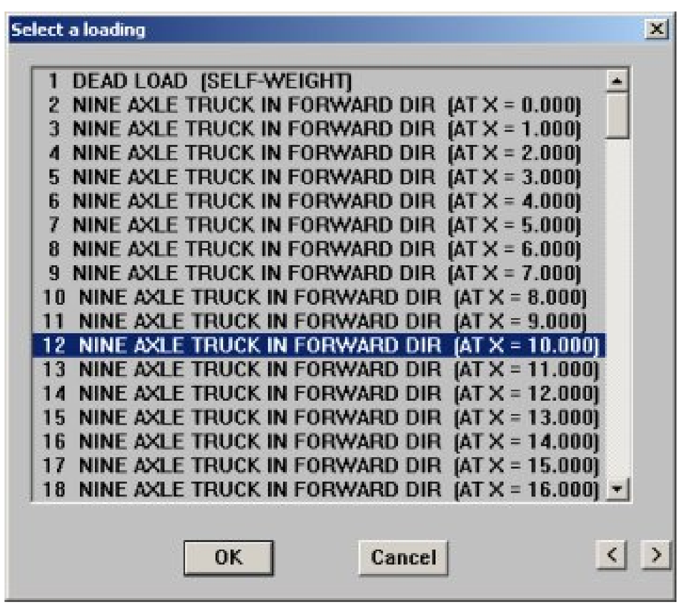
CBM Section 7.4 - RESULTS

1. LOAD CASES

1.1 Selecting & Viewing a Load Case

To select, view and print results for an individual load case click the large button located in the *Results for Individual Load Cases* frame. Before you do, however, first select the number of decimal points you want your results displayed to on the graphical diagram.

Once the button "*View & print results for all load cases"* is pressed the system will display a dialog box of all static load cases and generated vehicle loadings (see below). A separate loading will be shown for each vehicle position.



Any one of these loadings can be selected for interrogation. Highlight the required load case and click *OK*. The bending moment diagram for the selected load case will be displayed together with a floating menu bar at the top left corner of the drawing window (refer to item 1.2 below). The options in the menu bar will allow you to display other relevant vectors, to print the diagram or to change various diagram attributes (the drawing scale, diagram colour etc).

WARNING! *When ACES generates the continuous beam model and performs the analysis, nodes defined by the* ***No. Divns*** *(number of divisions) value in the* ***Geometry*** *table will be moved to coincide with points that define a change in section properties. Therefore, the regularly spaced sections initially created in the* ***Sec Props*** *tab will, in the final ACES model, be adjusted to line up with the locations at which the* ***Iz*** *profile changes. The same holds true for additional, inserted, points. (Refer to* *Section 7.2.2* *for details).*

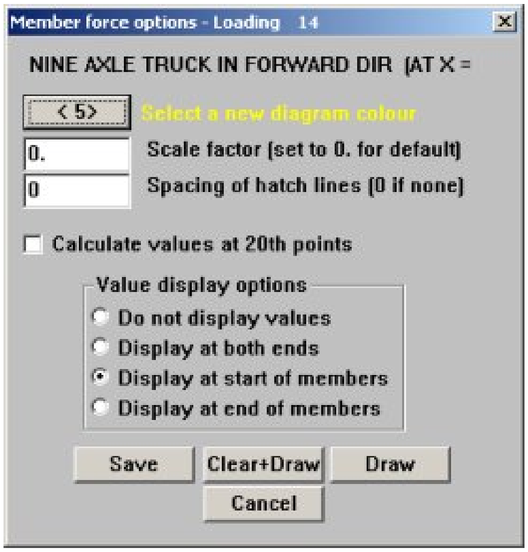
Because of this, results diagrams may look a little strange.

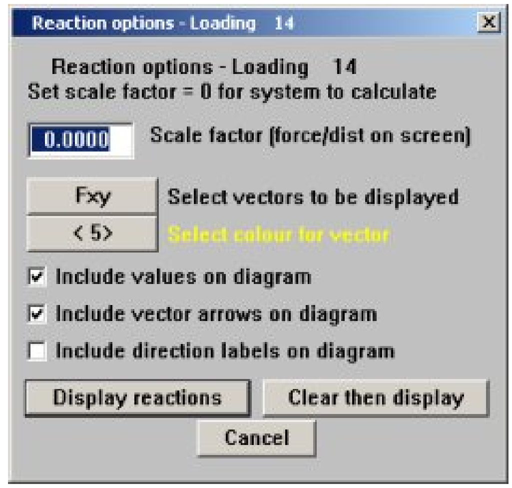
1.2 Diagram Menu Options

Load Case	Display the load case dialog box (to select another loading).
Prev. Load	View the results diagram for the previous loading and show the vehicle position.
Next Load	View the results diagram for the next loading and show the vehicle position.
Animate	Automatically step through all loading diagrams. Press *ESC* to stop the animation.
Options	Display the diagram attributes dialog box (refer to item 1.3 below).
Print	Print the current graphical diagram.
Table	Display the current graphical results in summarised tabular form.
Moment	Display the bending moment diagram for the current loading.
Shear	Display the shear force diagram for the current loading.
Reaction	Display the reaction diagram for the current loading.
Deflection	Display the deflection diagram for the current loading. (Note: the number of decimal places must be selected from the main *Results* form).
Torsion	Display the torsional moments diagram for the current loading (this option will only apply if the structure is curved or if one of the load cases has a vehicle loading that has been applied off-centre). A warning message will be given if this button is clicked and the model has no torsional loading.
Veh Posn	Display the position of the vehicle on the current diagram.
Return	Return to the main *Results* form.

1.3 Changing Diagram Options & Attributes

Click on the *Options* button in the menu bar. One of the following member force attributes dialog boxes will be displayed, depending on the type of results vector currently being displayed. The left hand window (shown below) appears if moments, shears or torsions are selected and the right-hand dialog box for reactions. Deflections have a similar dialog box.





If displaying a diagram for the first time leave the *Scale factor* zero and let the system select an appropriate scale for you. There-after you can adjust it to suit your requirements. Click either *Draw* or *Clear+Draw* to display the results diagram. (Note that the number of decimal points used in the display must first be selected from the *Results* tab in the data entry form).

1.4 Displaying Vehicle Position

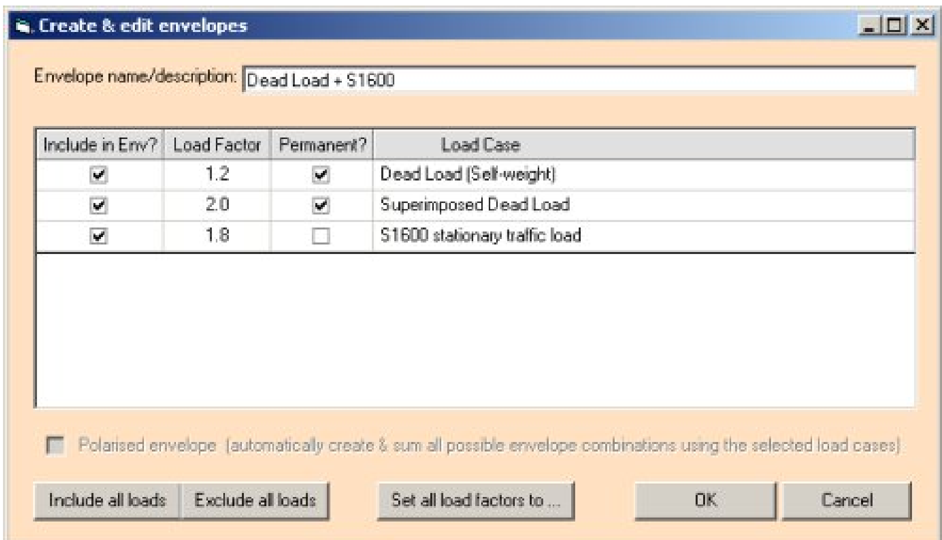
To display the vehicle position for a particular load case, select the load case first (click *Load Case*) then click the button labelled ***Veh Posn***. Alternatively, click either *Next Load* or *Prev Load* and step through the load cases until you get to the one you require.

2. ENVELOPES

2.1 Creating an Envelope

Click *New Envelope* to create a new load case envelope. The dialog box shown below will be displayed. Tick the load cases you want to include in the envelope, enter suitable load factors and select the loads that are permanent.

Note that the result vectors for *permanent* loads will be separately *added* to each of the transient load results prior to the extraction of maximum / minimum values. Refer to [Section 2.4](#) for a detailed explanation of the *Polarised Envelope* option.



Now select the following attributes from the *Results* tab:

- The number of decimal points to be used (enter an integer number)
- The type of envelope (maximum + minimum values or maximums plus corresponding values)
- The type of values you would like displayed in the graphical diagram (all values or maximum and minimums values only)

To view the envelope in **graphical** (diagram) form click *View & print envelope graphically*
To view the envelope in **tabular** (report) form click *Tabulated envelope results*.

If the graphical option is selected the bending moment envelope will be displayed together with a floating menu bar at the top left corner of the drawing window (see item 2.1 below). This menu will allow you to view all results vectors and to print, save and tabulate their maximum and minimum values. Refer also to [Section 2.3](#) for details regarding maximum/minimum results with their corresponding values.

WARNING! *When ACES generates the continuous beam model and performs the analysis, nodes defined by the **No. Divns** (number of divisions) value in the **Geometry** table will be moved to coincide with points that define a change in section properties. Therefore, the regularly spaced sections initially created in the **Sec Props** tab will, in the final ACES model, be adjusted to line up with the locations at which the **Iz** profile changes. The same holds true for additional, inserted, points. (Refer to [Section 7.2.2](#) for details).*

Because of this, results diagrams may look a little strange.

2.2 Diagram Menu Options

The options in the menu bar will allow you to display other relevant vectors, to print the diagram or to change various diagram attributes (the drawing scale, diagram colour etc).

Node No On	Turn node numbers on.
Node No Off	Turn node numbers off.
Decimals	Select the number of decimal points to be used in the graphical display and tabular reports.
Options	Display the diagram attributes dialog box (the exact form of the dialog box will depend on the type of results vector currently being displayed and will be identical to those described in *Item 1.3* above).
Print	Print the current graphical diagram.
Table	Display the current graphical results in summarised tabular form (not yet implemented).
Moment	Display the bending moment diagram for the current loading.
Shear	Display the shear force diagram for the current loading.
Reaction	Display the reaction diagram for the current loading.
Deflection	Display the deflection diagram for the current loading.
Torsion	Display the torsional moments diagram for the current loading (only if the structure is curved or if one of the load cases has a vehicle loading that has been applied off-centre). A warning message will be given if this button is clicked and the model has no torsional loading.
Return	Return to the main *Results* form.

2.3 Maximum + Corresponding Values

Two envelope types are available for viewing and printing maximum/minimum results vectors together with their corresponding values. Both can only be viewed in tabular form. If a *summary* envelope is required click *View & print envelope graphically* on the *Results* tab followed by the *Table* option on the floating menu at the top of the screen. The report contains tables for all valid force vectors - scroll down the list to view all results.

If a *detailed* report is required click the button labelled *Tabulated envelope results* on the *Results* tab. A small dialog box will appear that will enable you to select the vector for which the maximum values is required (e.g. either ***Mz***, ***Vy*** or ***Axf***). Once this has been done the tabular report will be displayed.

It will show the maximum positive and negative values of the selected vector at the start and end of each member together with the corresponding values of all other vectors, together with the vehicle loading number at which the maximums were found.

2.3.1 Reactions

To view the maximum reaction at one support with corresponding values at the others first display maximum/minimum reactions then click the *Options* button. A reaction dialog box similar to the one shown in *Section 1.3* above will be displayed, the difference being that it will have an additional frame labelled "*Display maximum values at*".

Click the second option, "*One support + corresponding values at others*" and enter a support number (you will need to turn node numbers on and set decimal points before accessing this option). Note that the vehicle loading number producing that reaction will also be echoed in the title.

To view maximum values at other supports simply click the *Reactions* button again and enter another support node number. To view minimum reactions click *Options* followed by the ***Max Fy*** button then repeat the above steps.

2.4 Polarised Envelopes

Polarised Envelopes are, in effect, a series of envelopes that are automatically and transparently generated by ACES then internally summed to produce the worst effects.

At every node in the model, ACES will separately add together the positive and negative vectors from each load case and extract the largest maximum and minimum values. In effect, summation is performed using every possible combination of load cases that you select for inclusion in the Polarised envelope. It is important, therefore, that a particular loading (e.g. Dead load) is not included more than once in the load cases being summed (otherwise it will be added twice).

EXAMPLE:

Assuming you have a 5 span structure, a single static load case (representing Dead Load) and a single M1600 moving vehicle load case, ACES will generate the following loadings:

Case 1 - Self Weight (permanent effect)
Case 2 - UDL in span 1
Case 3 - UDL in span 2
Case 4 - UDL in span 3
Case 5 - UDL in span 4
Case 6 - UDL in span 5
Case 7 - M1600 moving vehicle

If the *Polarised* envelope option is switched on ACES will internally create the following individual envelopes using the load factors specified against each load case:

Envelope 1 - Load case 1 only with all others excluded
Envelope 2 - Load case 2 only with all others excluded
Envelope 3 - Load case 3 only with all others excluded
Envelope 4 - Load case 4 only with all others excluded
Envelope 5 - Load case 5 only with all others excluded
Envelope 6 - Load case 6 only with all others excluded
Envelope 7 - Load case 7 only with all others excluded

All seven envelopes will then be summed. The results and reports will contain the dead load values specified in load case 1 together with the summed values from all other load cases i.e., the summation process will automatically account for all combinations of loaded spans together with the superimposed moving vehicle loads. Instead of looking at a series of separate envelopes and selecting the maximum and minimum values (as in a normal envelope) the Polarise option in effect adds the results if they contribute to a maximum or minimum value or ignores them if they don't.

Polarised Envelopes are equivalent to the summing envelopes option found in the main ACES program. (Refer to *Section 5.3, Item 1.6*, for further details).

PART 8

MODEL TEMPLATES

Index To Model Templates

8.1 Beams

8.1.1 Continuous Beams
8.1.2 Beams on Elastic Foundations

8.2 Frames

8.2.1 Culverts
8.2.2 Frames With Inclined Legs
8.2.3 Pitched Portal Frames
8.2.4 Multibay Portals (Types 4, 5)

8.3 Grillages

8.3.1 Parallel Main Girders (Types 1, 2, 3)
8.3.2 Non-Parallel Main Girders (Types 4, 5, 6)
8.3.3 Main Girders on a Circular Curve (Type 8)
8.3.4 Grillages Having an Arbitrary Shape (Types 7, 9)

8.4 Slabs

8.4.1 Parallel-Sided Slabs (Types 1, 2, 3)
8.4.2 Non-Parallel Sided Slabs (Types 4, 5, 6)
8.4.3 General Multispan Shape (Type 7)
8.4.4 Slabs on a Circular Curve (Types 8, 9)

8.5 Grillage Slabs

8.5.1 Parallel Main Girders (Types 1, 2, 3)
8.5.2 Non-Parallel Main Girders (Types 4, 5, 6)
8.5.3 General Multispan Shape (Type 7)
8.5.4 Girders & Slabs on a Circular Curve (Types 8, 9)

8.6 Trusses

8.6.1 Pitched Truss (Type 1)
8.6.2 Pratt Truss (Type 2)
8.6.3 Modified Pratt Truss (Type 3)

8.7 Walls, Silos, Tanks

8.7.1 Wingwalls & Retaining Walls (Types 1, 2)
8.7.2 Silos & Tanks - Cylindrical
8.7.3 Silos & Tanks - Tapered
8.7.4 Silos & Tanks - Segmented
8.7.5 Rectangular Tanks (Types 1, 2)

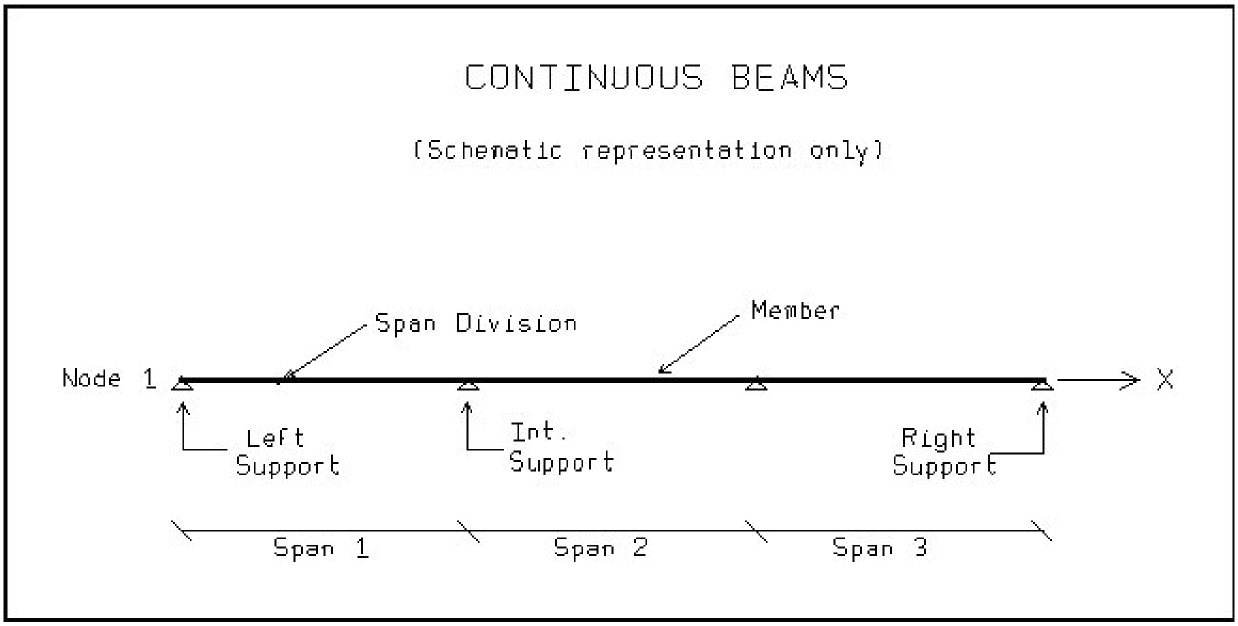
8.8 Box Girders, 3D Sections

8.8.1 Box Girders (Types 1, 2)
8.8.2 Box Culverts & Crown Units (Types 1, 2, 3)
8.8.3 Culvert Headwall

Section 8.1.1 - Continuous Beams Template

Multispan Continuous Beams Parameters

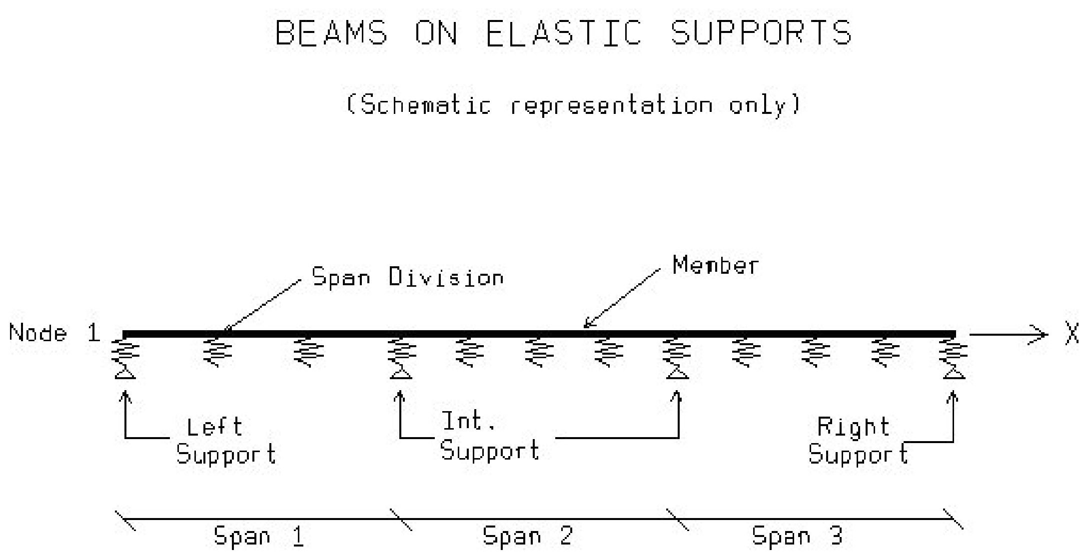
No. Spans	Number of spans (maximum of 99). ACES will generate a pin restraint in the *X,Y* direction under the left-most support and roller (*Y*) restraints under all other supports.
Y Offset	Offset of the beam in the global-Y direction. This will enable the linear model to be converted to a 2D frame structure. Refer to *Section 2.4* for details.
Span Lengths	List all span lengths.
Span Divisions	Enter the number of segments, (or "members"), into which each span is to be divided. Output results are only given at nodes separating each member. Therefore to obtain smooth vector diagrams (particularly animation of dynamic modes of vibration) a sufficient number of divisions should be specified. The default is 10 divisions.



Section 8.1.2 - Foundation Beam Template

Parameters for Beams on Elastomeric Foundations

No. Spans	Number of spans (maximum of 99). ACES will generate a pin restraint in the *X,Y* direction under the left-most support and roller (*Y*) restraints under all other supports. After the model has been generated you will need to change the support property types to elastomeric and specify appropriate elastic constants.
Y Offset	Offset of the beam in the global-Y direction. This will enable the linear model to be converted to a 2D frame structure. Refer to *Section 2.4* for details.
Span Lengths	List all span lengths. This allows models to be created of beams (or 1D-rafts) where the characteristics of spring supports (i.e. of soil elasticity) varies between spans.
Span Divisions	Number of segments, (or "members"), into which each span is to be divided. Output results are only given at nodes separating each member. Therefore to obtain smooth vector diagrams (particularly animation of dynamic modes of vibration) a sufficient number of divisions should be specified. The default is 10 divisions.



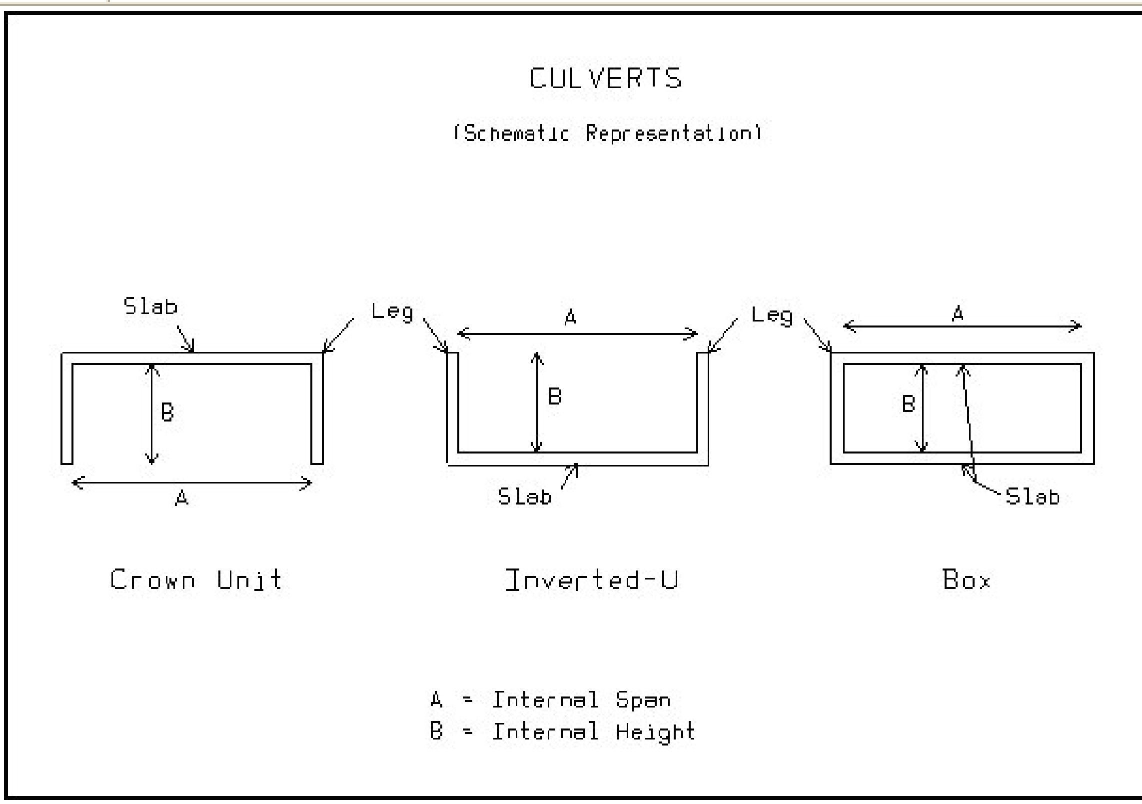
Section 8.2.1 - Culverts

Parameters for 2D Culverts

This template allows three types of 2D culvert unit to be modelled: *Crown, Inverted-U* and *Closed Box* sections. Note that the structural analysis will be based on section properties defined after the base model has been generated. These properties must incorporate the culvert width if it is less than or greater than unity (for example, a 1.2 metre wide unit).

To create 3D culvert and box section models refer to *Section 8.8.2* (note that *Section 8.8.3* also includes the special case of a 3D box culvert with monolithic headwalls, wingwalls and apron).

Dimension A	Internal span.
Dimension B	Internal height.
Slab Thickness	Slab thickness.
Leg Thickness	Leg thickness.
Slab Divisions	Number of divisions ("nodes") along the top (and bottom) slabs. Output results are only given at these nodes. Therefore to obtain smooth vector diagrams (particularly if animation of dynamic modes of vibration is required) a sufficient number of divisions should be specified.
Leg Divisions	Number of divisions along each leg. Output results are only given at the nodes. Therefore to obtain smooth vector diagrams (particularly animation of dynamic modes of vibration) a sufficient number of divisions should be specified.

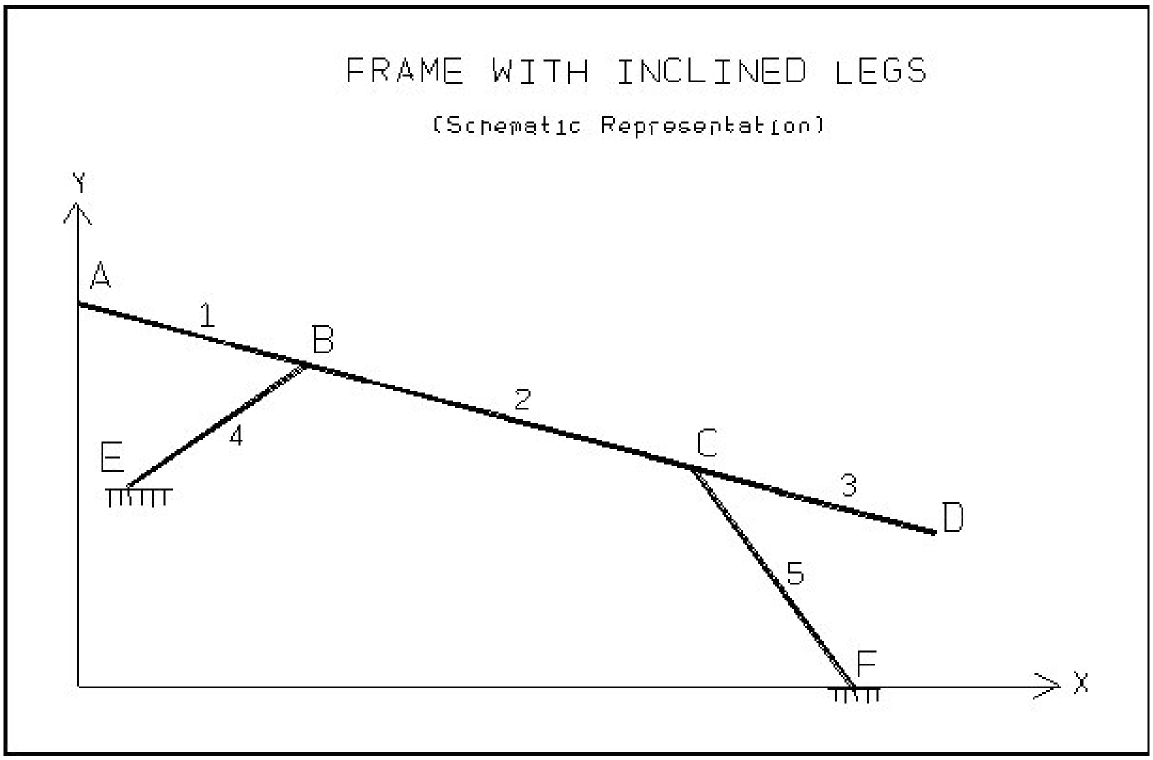


Section 8.2.2 - Frames With Inclined Legs

Parameters for Frames With Inclined Legs

ACES will generate a model in which the left and right supports are *X,Y* pin restraints and the inclided pier legs are fixed at the base.

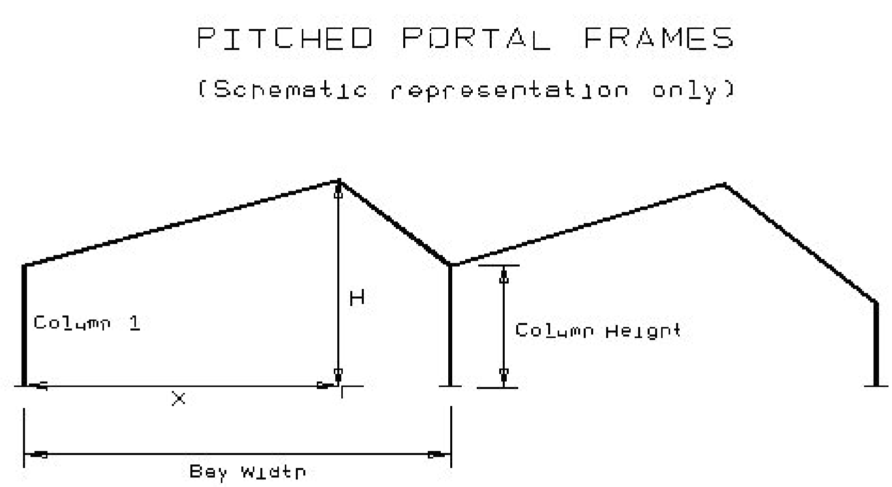
Number of Frames	Number of parallel frames to be created. Note that if a vehicle is run over a multi-framed structure it will only be applied to the first frame.
Transverse Divisions	Number of divisions into which the transverse distance between adjacent frames is to be divided. The area between adjacent frames will be subdivided into rectangular FE plate elements whose total number will depend on this spacing and the number of divisions given by the *Segment Divisions* parameter below. The resultant 3D structure will be analysed as a combined 3D space frame/plate bending problem.
X Coordinates ..	Specify X-coordinates of points A, B, C, D, E
Y Coordinates ..	Specify Y-coordinates of points A, B, C, D, E.
Segment Divisions ..	Specify the number of divisions in segments 1, 2, 3, 4, 5. Output results are only given at the nodes. Therefore to obtain smooth vector diagrams (particularly animation of dynamic modes of vibration) a sufficient number of divisions should be specified.
Frame Spacing ..	Specify spacing between adjacent parallel frames.



Section 8.2.3 - Pitched Portal Frames

Parameters for Portal Frames

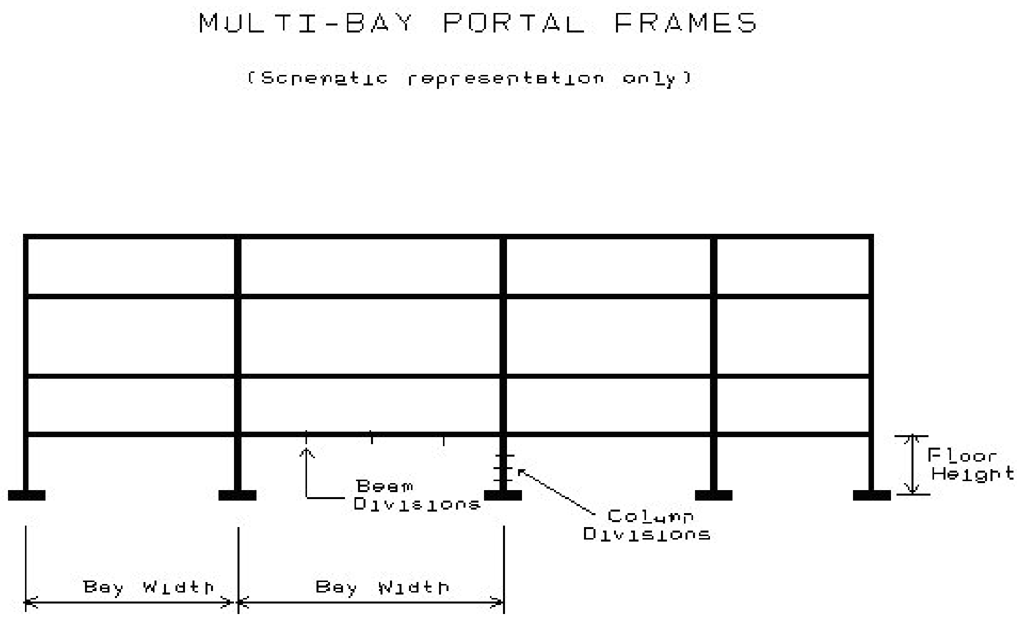
Number of Bays	Number of bays.
Column Divisions	Number of segments into which columns are to be divided. The larger the number the more accurate the results diagrams. Output results are only given at the nodes. There-fore to obtain smooth vector diagrams (particularly animation of dynamic modes of vibration) a sufficient number of divisions should be specified.
Beam Divisions	Number of segments into which beam members are to be divided. The larger the number the more accurate the results diagrams. Output results are only given at the nodes. Therefore to obtain smooth vector diagrams (particularly animation of dynamic modes of vibration) a sufficient number of divisions should be specified.
Number of Frames	Number of parallel frames to be created. Horizontal purlins will be inserted between all frames running through the nodes created by the beam and column division points. You will need to assign each of these transverse members a separate member property type after the full model has been generated.
Transverse Divisions	Number of divisions into which the transverse members between adjacent frames are to be divided.
Bay Widths ...	Column bay widths.
Column Heights ...	Column heights.
Heights (H) ...	Portal peak heights.
Distance (X) ...	Distance of the roof peak from the left column in each bay.
Frame Spacing ...	Spacing between adjacent parallel frames.



Section 8.2.4 - Mulitbay Portal Frames (Types 4, 5)

Parameters for Multi-bay Portal Frames

Number of Bays	Number of bays in the frame.
Column Divisions	Number of segments into which columns are to be divided. Output results are only given at the nodes. Therefore to obtain smooth vector diagrams (particularly animation of dynamic modes of vibration) a sufficient number of divisions should be specified.
Beam Divisions	Number of segments into which beam members are to be divided. Output results are only given at the nodes. Therefore, to obtain smooth vector diagrams, (particularly animation of dynamic modes of vibration), a sufficient number of divisions should be specified. If a flat slab is to be included at each floor level, (see *Include Floor Slab* parameter below), this value will determine the FE mesh size.
Number of Floors	Number of floors in the structure.
Number of Frames	Number of parallel frames to be created.
Transverse Divisions	Number of divisions into which the transverse members between adjacent frames are to be divided. They will be assigned a separate member property type. If the *Floor Slab* parameter is toggled to *Yes* this will determine the number of FE mesh divisions between adjacent frames.
Include Floor Slab?	Include a floor slab in the model? If *Yes*, ACES will model all floors using rectangular FE plate bending elements. Mesh density will be determined by the number of *Transverse Divisions* and the number of *Beam Divisions* and the model will be analysed as a combined 3D space frame + FE plate bending problem. (Note that for complex multi-bay, multi-storey models the resultant total number of elements can become very large). If this parameter is toggled to *No* the model will be analysed as a 3D space frame structure. Finite elements will not be generated.
Bay Widths ...	Bay widths (distance between columns).
Floor Heights ...	Floor heights.
Frame Spacing ...	Spacing between adjacent frames. This will only be required if the number of frames is larger than 1.


MULTI-BAY PORTAL FRAMES
(Schematic representation only)
Beam
Divisions
Column
Divisions
Floor
Height
Bay Width
Bay Width

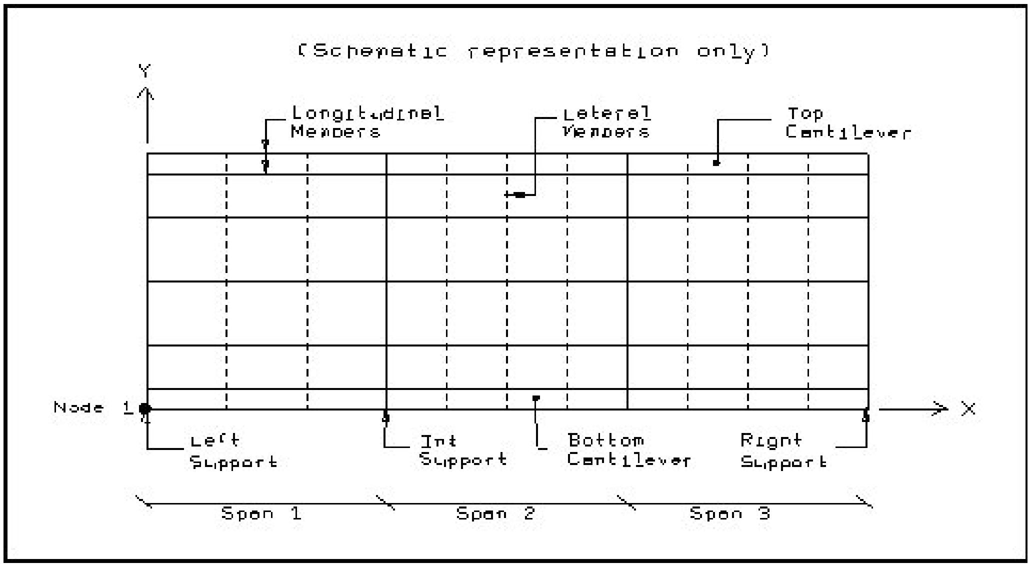
Section 8.3.1 - Grillage Types 1, 2, 3

Parameters for Parallel-Sided Grillages

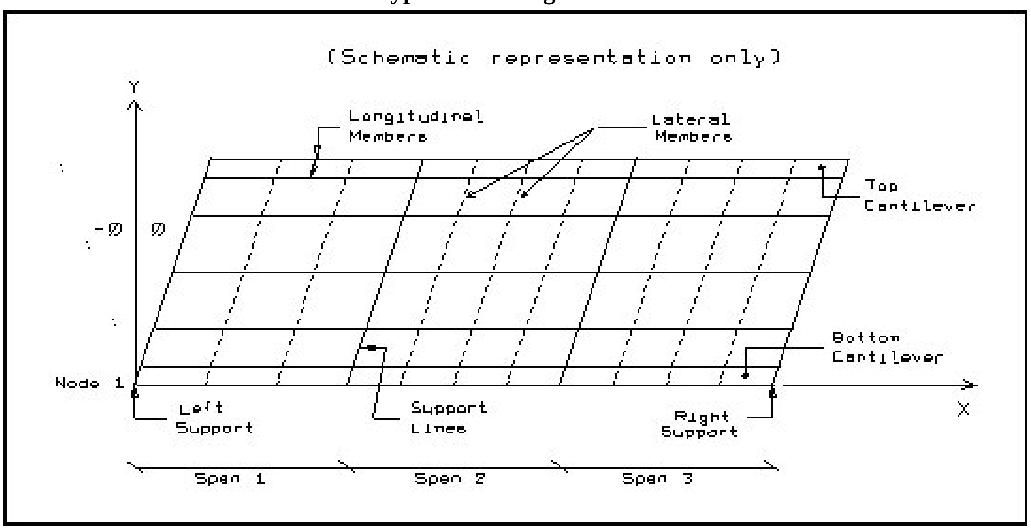
Grillage Types 1,2 and 3 represent a category of grillage models in which all main longitudinal girders are parallel. Three types of grillage templates are available (refer to the diagrams at the bottom of the table):

Type 1	All support lines in the structure are orthogonal to the main girders
Type 2	All support lines in the structure are uniformly skewed
Type 3	Each support line in the structure can have a different skew

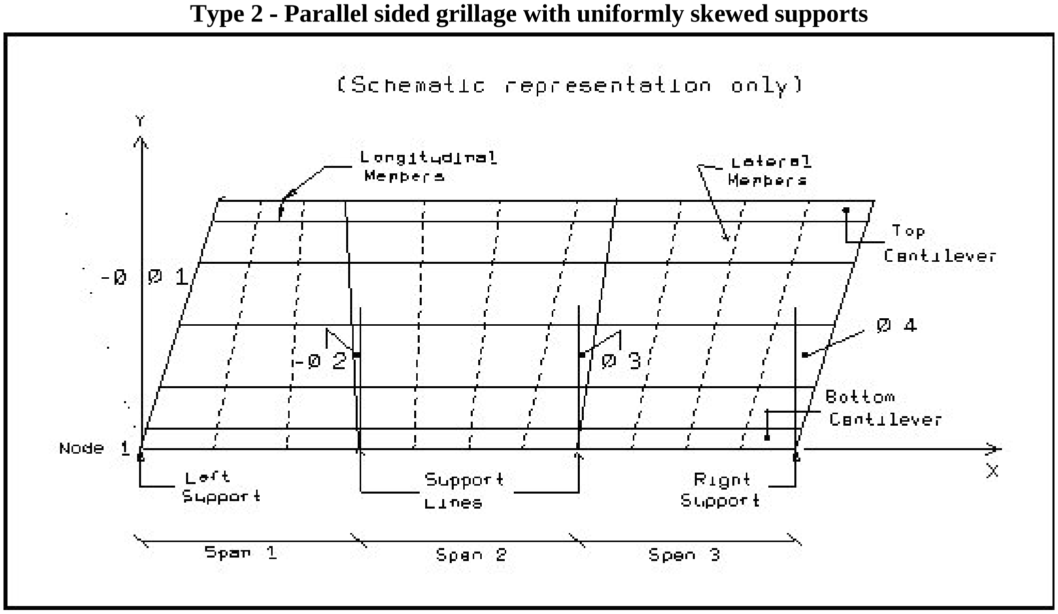
Number of Spans	Total number of spans, or support *lines*, in the structure. This should not be confused with individual points of support (otherwise referred to as *supports*).
Span Length	Span lengths measured along the X axis (with respect to the bottom cantilever edge beam).
X Divisions	Number of X divisions in each span. This is a measure of mesh density. It defines the number of transverse division lines, (and hence transverse members), within each span of the grillage.
Number of Girders	Number of main longitudinal girders (excluding cantilever edge beams). Girder 1 is assumed to lie closest to the bottom cantilever. Note that the CAD Modeller will allow you to define these girders as composite members (refer to *Section 2.8* for details).
Girder Spacings	Spacing of longitudinal girders. Girder 1 represents the first longitudinal main girder nearest to Node 1 (it is *not* the cantilever edge beam).
Top Cantilever	Width of top cantilever (measured at right angles to the main girder). Set to zero if a top cantilever is not required.
Bottom Cantilever	Width of bottom cantilever (measured at right angles to the main girder). Set to zero if a bottom cantilever is not required.
Skew Angle	Skew angle of supports (measured in *Degrees*). It is not required for *Type 1* grillages. A data entry dialog box will appear for *Type 3* models to allow skew angles to be entered for all lines of support.
Z Offset	Z coordinate offset of the *X-Y* plane to convert the 2D structure into a 3D model. Refer to page *Section 2.2* for details.
Mesh Type	Select the required mesh - *Rectangular* or *Skewed*. If *Rectangular*, the transverse grillage members are created at right angles to the bottom edge of each span. Otherwise they are uniformly skewed.
Girder Width	Nominal girder width. The transverse grillage members will be modelled using two separate member property types. One type, assigned to members on each side of the girder centreline, will have a length equal to half this dimension. The other type will be assigned to the remainder of the transverse girder-girder spacing. This will enable *Super-T* sections and similar girders to be modelled.



Type 1 - Rectangular deck



Type 2 - Parallel sided grillage with uniformly skewed supports



Type 3 - Parallel sided grillage with

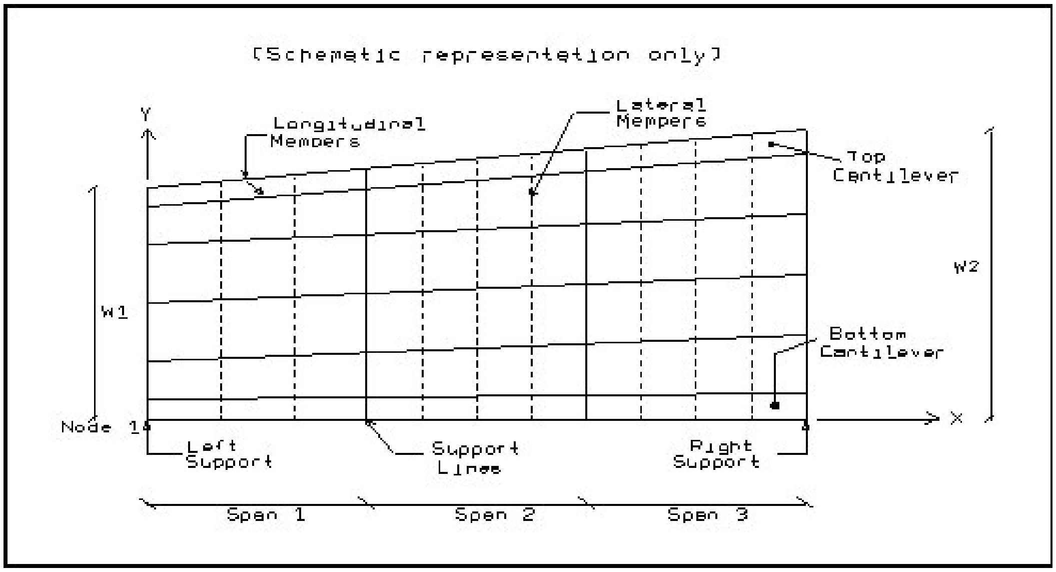
Section 8.3.2 - Grillage Types 4, 5, 6

Parameters for Grillages With Non-Parallel Main Girders

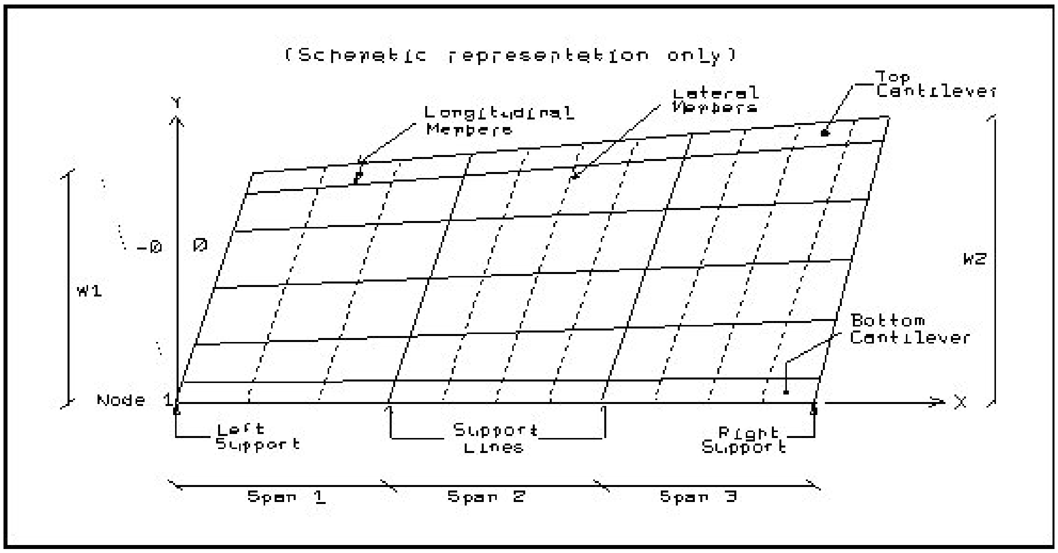
Grillage Types 4, 5 and 6 represent a category of grillage models in which all main longitudinal girders are non-parallel but straight throughout the length of the structure. Three types of grillage templates are available (refer to the diagrams at the bottom of the table):

Type 4	All support lines in the structure are orthogonal to the first (bottom) main girder
Type 5	All support lines in the structure are uniformly skewed
Type 6	Each support line in the structure can have a different skew

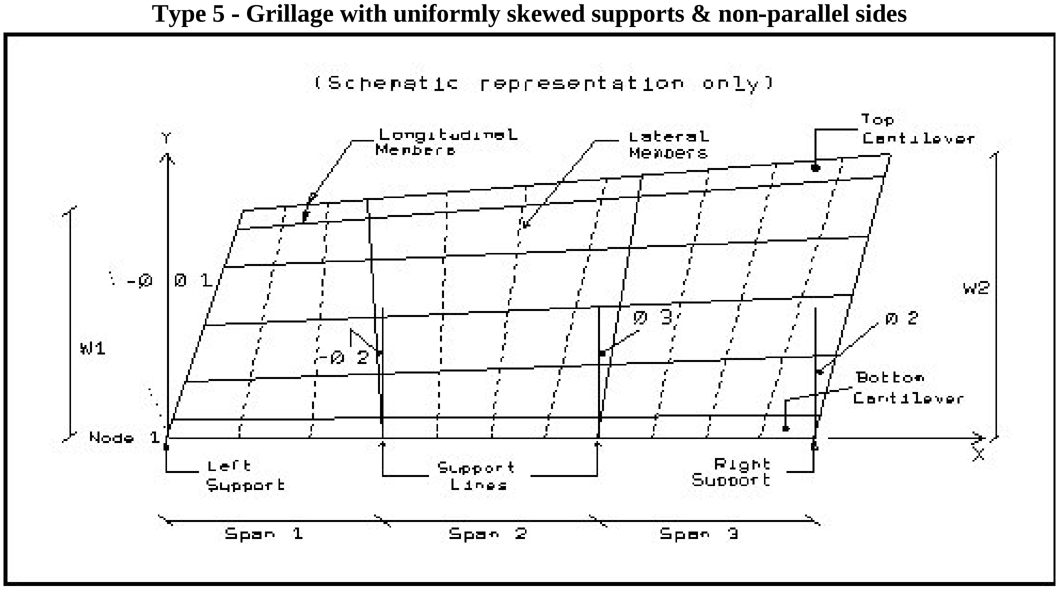
Number of Spans	Total number of spans, or support *lines*, in the structure. This should not be confused with individual points of support (otherwise referred to as *supports*).
Span Length	Span lengths measured along the X axis (i.e. with respect to the bottom cantilever edge beam).
X Divisions	Number of X divisions in each span. This is a measure of mesh density. It defines the number of transverse division lines, (and hence transverse members), within each span of the grillage.
Number of Girders	Number of main longitudinal girders (excluding cantilever edge beams). Girder 1 is assumed to lie closest to the bottom cantilever. Note that the CAD Modeller will allow you to define these girders as composite members (refer to *Section 2.8* for details).
Girder Spacings	Girder spacing as measured along the *left* support line. Girder 1 represents the first longitudinal main girder nearest to Node 1 (it is *not* the bottom cantilever edge beam).
Top Cantilever	Width of top cantilever (measured at right angles to the top edge of the first span at the left support line). It will then splay out to the far support in the same ratio as the main girders. Set to zero if a top cantilever is not required.
Bottom Cantilever	Width of bottom cantilever (measured at right angles to the bottom edge of the first span at the left support line). It will then splay out to the far support in the same ratio as the main girders. Set to zero if a bottom cantilever is not required.
Skew Angle	Skew angle of supports (measured in *Degrees*). It is not required for *Type 4* grillages. A data entry dialog box will appear for *Type 6* models to allow skew angles to be entered for all lines of support.
Z Offset	Z coordinate offset of the *X-Y* plane to convert the 2D structure into a 3D model. Refer to page *Section 2.2* for details.
Mesh Type	Select the required mesh - *Rectangular* or *Skewed*. If *Rectangular*, the transverse grillage members are created at right angles to the bottom edge of each span. Otherwise they are uniformly skewed.
Right Width (W2)	Width (*W2*) of right support normal to the ***X*** axis. *W1* is calculated by the system from the parameters given above.



Type 4 - Grillage with non-skewed supports & non-parallel sides



Type 5 - Grillage with uniformly skewed supports & non-parallel sides



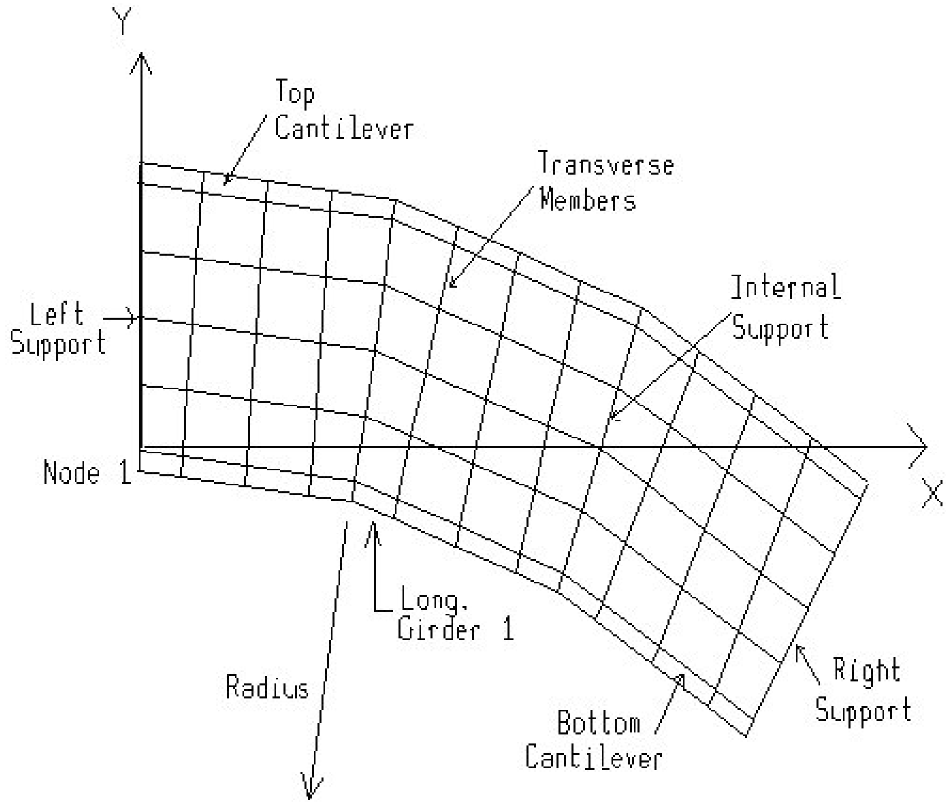
Type 6 - Grillage with non uniformly

Section 8.3.3 - Grillage Type 8

Parameters for Main Girders Lying on a Circular Curve

Grillage Types 8 represents a model in which all main longitudinal girders are straight within each span but lie on a circular curve when measured along the length of the structure.

Number of Supports	Total number of supports (or lines of support) in the structure.
Span Length	Span lengths measured along the radius of curvature.
Longitudinal Divisions	Number of divisions in each span. This is a measure of mesh density. It defines the number of transverse division lines, (and hence transverse members), within each span of the grillage.
Number of Girders	Number of main longitudinal girders (excluding cantilever edge beams). Girder 1 is assumed to lie closest to the bottom cantilever. Note that the CAD Modeller will allow you to define these girders as composite members (refer to *Section 2.8* for details).
Girder Spacings	Girder spacing measured along the *left*-most support line.
Top Cantilever	Width of top cantilever, measured at right angles to the top edge of the first span. This will be uniform along the full length of the deck. Set to zero if a top cantilever is not required.
Bottom Cantilever	Width of bottom cantilever, measured at right angles to the bottom edge of the first span. This will be uniform along the full length of the deck. Set to zero if a bottom cantilever is not required.
Z Offset	Z coordinate offset of the *X-Y* plane to convert the 2D structure into a 3D model. Refer to page *Section 2.2* for details.
Mesh Type	Select the required mesh - *Rectangular* or *Skewed*. If *Rectangular*, the transverse grillage members are created at right angles to the bottom edge of each span. Otherwise they are uniformly skewed.
Radius	Radius of curvature. The radius is measured with respect to the *centreline* of the deck and not the bottom edge.

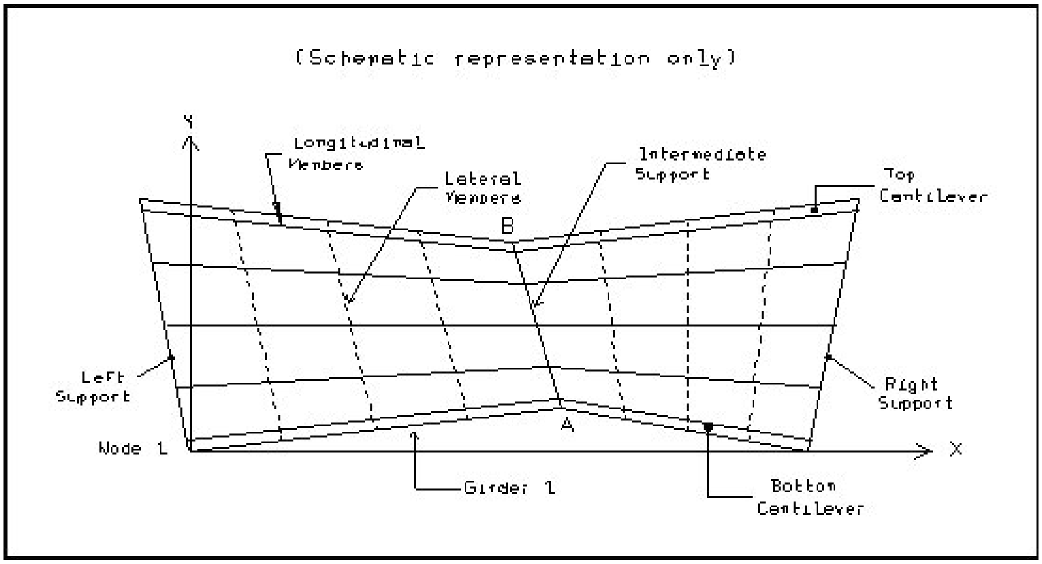

Y
Top
Cantilever
Transverse
Members
Internal
Support
Left
Support
Node 1
X
Long.
Girder 1
Radius
Bottom
Cantilever
Right
Support

Section 8.3.4 - Grillage Types 7, 9

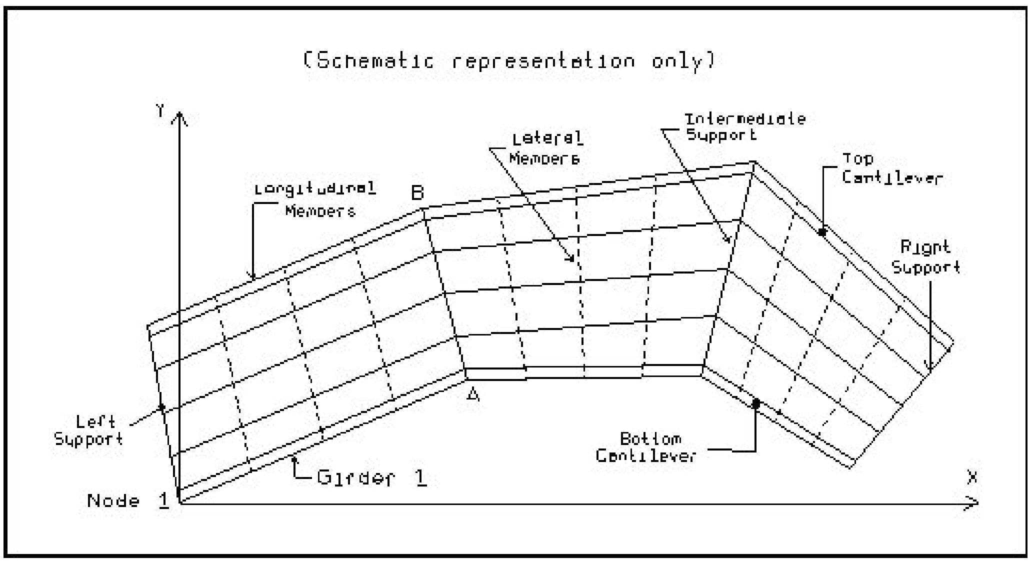
Parameters for Grillages Having an Arbitrary Shape

Grillage Types 7 and 9 represent a category of grillage models of a general, arbitrary, shape i.e., the main longitudinal girders may be non-parallel to one another and may not lie in a straight line along the full length of the deck. Two types of grillage templates are available as illustrated in the diagrams at the bottom of the table:

Number of Supports	Total number of support *lines* in the structure. This should not be confused with individual points of support (otherwise referred to as *supports*).
Support Geometry	Enter the (*X,Y*) start and end coordinates of all support lines in the model. For each support line *A*-*B* as shown in the schematic diagram, ***A*** refers to the end closest to the bottom edge of the bottom cantilever (if any) and ***B*** to the end closest to the top edge of the top cantilever (if any).
Long. Divisions	Number of longitudional divisions in each span. This is a measure of mesh density. It defines the number of transverse division lines, (and hence transverse members), within each span of the grillage.
Number of Girders	Number of main longitudinal girders (excluding cantilever edge beams). Girder 1 is assumed to lie closest to the bottom cantilever. Note that the CAD Modeller will allow you to define these girders as composite members (refer to *Section 2.8* for details).
Girder Spacings	Girder spacing along the far left support line. Girder 1 represents the first longitudinal main girder nearest to Node 1 (and not the cantilever edge beam). If the girders are splayed, the spacing at the right support line in that span will vary in direct proportion to the spacing along the left support line. ACES will check that the length *A*-*B* at the first support line is compatible with the cantilever widths and girder spacings. If not, the ***B*** coordinate will be moved until it is.
Top Cantilever	Width of top cantilever (measured at right angles to the top edge of the first span at the left support line). It will then splay out to the far support in the same ratio as the main girders. Set to zero if a top cantilever is not required.
Bottom Cantilever	Width of bottom cantilever (measured at right angles to the bottom edge of the first span at the left support line). It will then splay out to the far support in the same ratio as the main girders. Set to zero if a bottom cantilever is not required.
Z Offset	Z coordinate offset of the *X*-*Y* plane to convert the 2D structure into a 3D model. Refer to page *Section 2.2* for details.
Mesh Type	Select the required mesh - *Rectangular* or *Skewed*. If *Rectangular*, the transverse grillage members are created at right angles to the bottom edge of each span. Otherwise they are uniformly skewed.



Type 7 - Grillages of arbitrary shape



Type 9 - Grillage of arbitrary shape

Section 8.4.1 - Slab Types 1, 2, 3

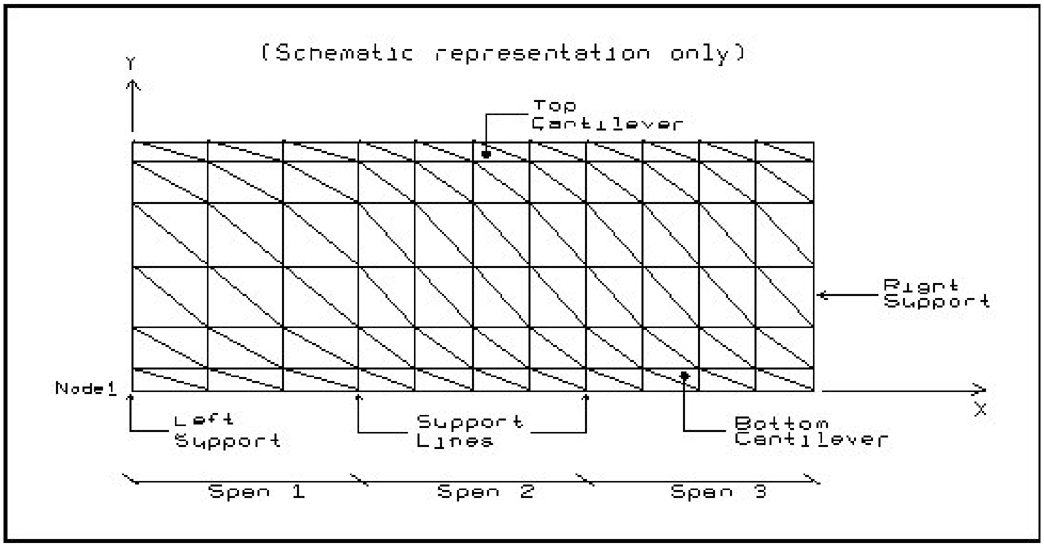
Parameters for Parallel-Sided Slabs

The slab (or bridge deck) is modelled using rectangular or triangular plate bending elements and is based on the theory developed by O.C. Zienkiewicz. For details refer to: "*The Finite Element Method in Engineering Science*".

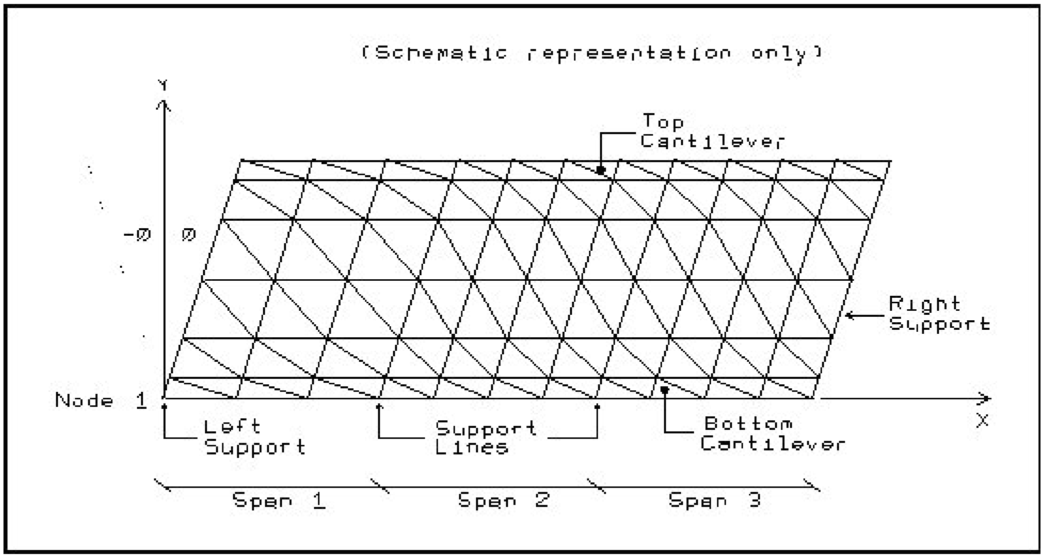
Slabs are generated using either rectangular plate bending elements or a combination of triangular and rectangular elements. Slabs *Type 1,2* and *3* represent FE models in which all lines of elements are straight and parallel. Three types of slab meshing templates are available (refer to the diagrams at the bottom of the table):

Type 1	All support lines in the structure are orthogonal to the global *X-axis*
Type 2	All support lines in the structure are uniformly skewed
Type 3	Each support line in the structure can have a different skew

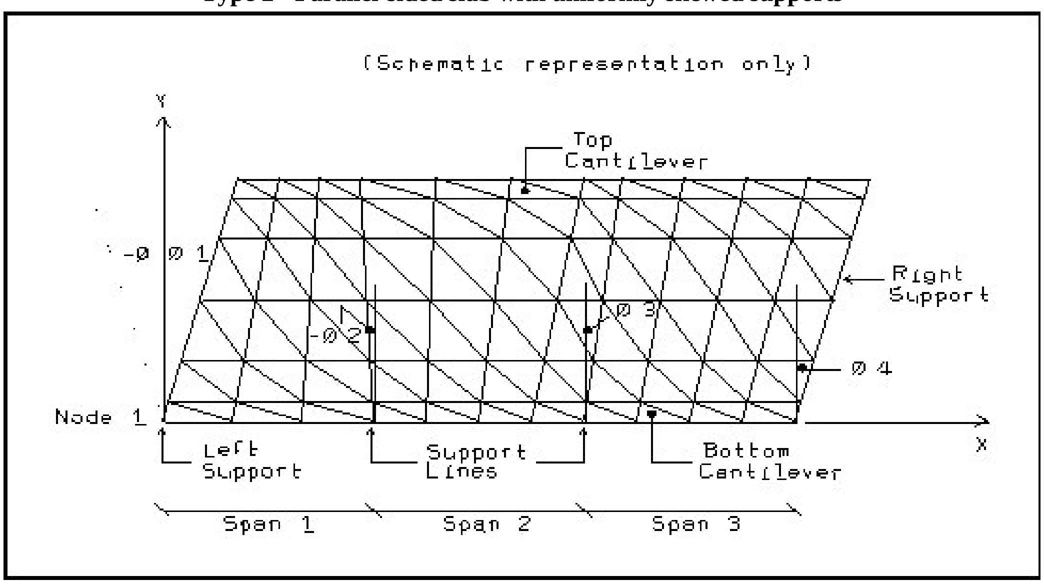
Number of Spans	Total number of spans, or support *lines*, in the structure. This should not be confused with individual points of support (otherwise referred to as *support points*) located at selected nodes.
Span Length	Span lengths measured along the X axis (refer to individual diagrams for concise definition).
X Divisions	Number of longitudinal mesh divisions in each span. The slab is modelled using rectangular and/or triangular plate bending elements (refer to *Part 2.2* for details of the modelling process).
Y Divisions	Number of transverse mesh divisions (excluding top and bottom cantilevers). The slab is modelled using rectangular and/or triangular plate bending elements (refer to *Part 2.2* for details of the modelling process).
Deck Width	Width of deck - excluding top and bottom cantilevers (if any).
Top Cantilever	Width of top cantilever (measured at right angles to the top edge). Set to zero if a top cantilever is not required.
Bottom Cantilever	Width of bottom cantilever (measured at right angles to the bottom edge). Set to zero if a bottom cantilever is not required.
Skew Angle	Skew angle of supports (measured in *Degrees*). It is not required for *Type 1* slabs. A data entry dialog box will appear for *Type 3* models to allow skew angles to be entered for all lines of support.
FE Type	Type of Finite Element with which to model the deck slab. This parameter is only available for *Type 1* slab structures (in all other cases triangular elements used). See diagram below.
Z Offset	Z coordinate offset of the *X-Y* plane to convert the 2D structure into a 3D model. Refer to page *Section 2.2* for details.



Type 1 - Rectangular deck



Type 2 - Parallel-sided slab with uniformly skewed supports



Type 3 - Parallel sided slab with non uniformly skewed supports

Section 8.4.2 - Slab Types 4, 5, 6

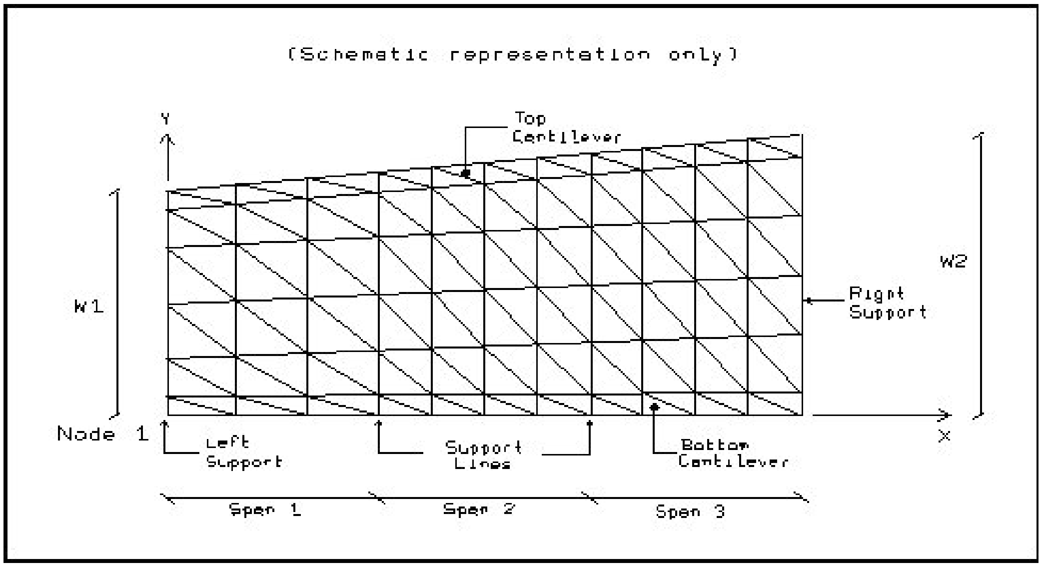
Parameters for Nonparallel-Sided Slabs

The slab (or bridge deck) is modelled using triangular plate bending elements and is based on the theory developed by O.C. Zienkiewicz. For details refer to: "*The Finite Element Method in Engineering Science*".

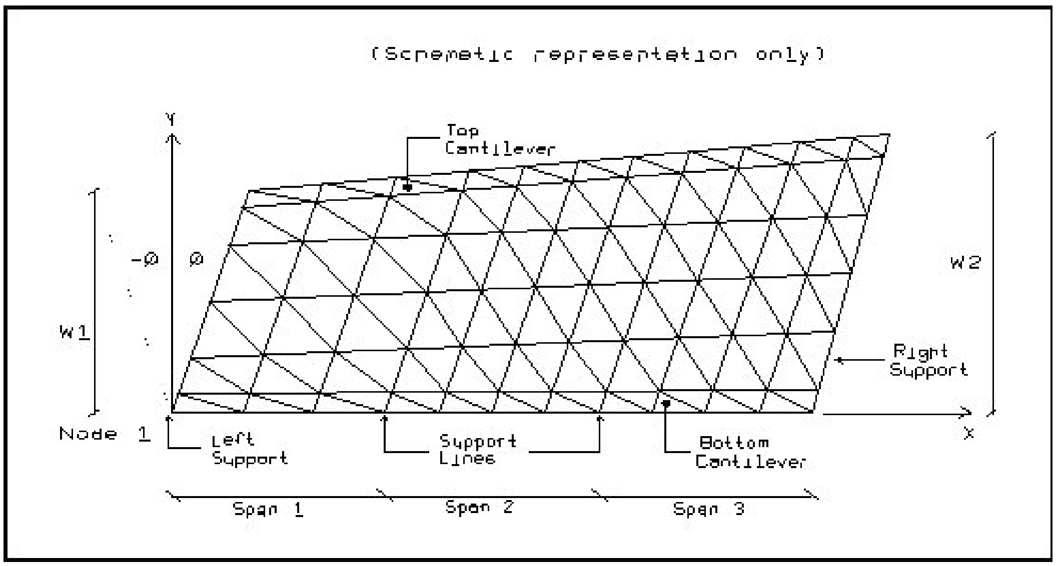
Slabs *Type 4, 5* and *6* represent FE models in which all lines of elements are straight but not necessarily parallel. Three types of slab meshing templates are available (refer to the diagrams at the bottom of the table):

Type 4	All support lines in the structure are orthogonal to the global *X-axis*
Type 5	All support lines in the structure are uniformly skewed
Type 6	Each support line in the structure can have a different skew

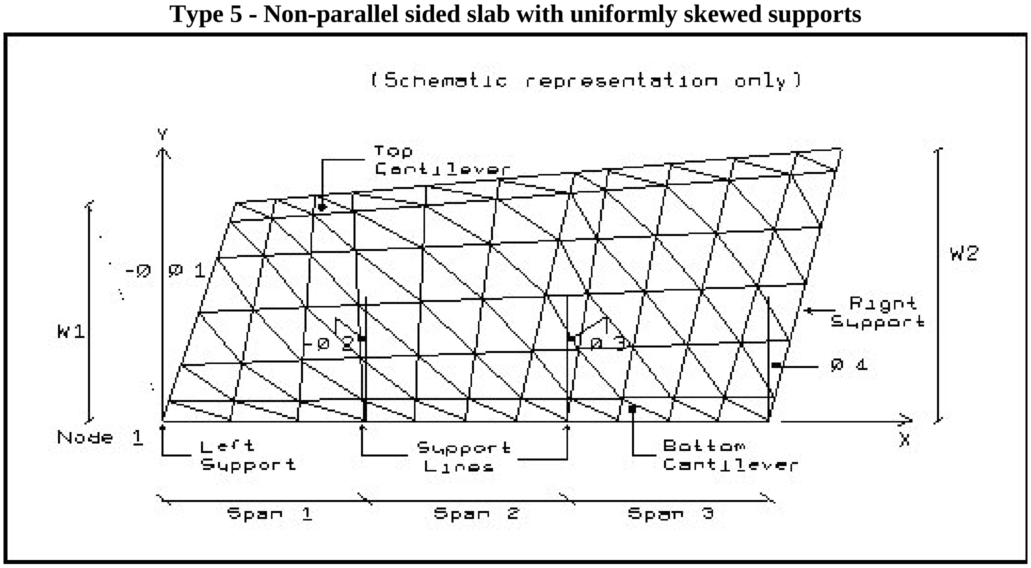
Number of Spans	Total number of spans, or support *lines*, in the structure. This should not be confused with individual points of support (otherwise referred to as *support points*) located at selected nodes.
Span Length	Span lengths measured along the X axis (refer to individual diagrams for concise definition).
X Divisions	Number of longitudinal mesh divisions in each span. The slab is modelled using triangular plate bending elements (refer to *Part 2.2* for details of the modelling process).
Y Divisions	Number of transverse mesh divisions (excluding top and bottom cantilevers). The slab is modelled using triangular plate bending elements (refer to *Part 2.2* for details of the modelling process).
Deck Width (W1)	Width of deck normal to the X axis (*excluding* any top and bottom cantilevers).
Top Cantilever	Width of top cantilever (measured at right angles to the top edge). Set to zero if a top cantilever is not required.
Bottom Cantilever	Width of bottom cantilever (measured at right angles to the bottom edge). Set to zero if a bottom cantilever is not required.
Skew Angle	Skew angle of supports (measured in *Degrees*). It is not required for *Type 4* slabs. A data entry dialog box will appear for *Type 6* models to allow skew angles to be entered for all lines of support.
Z Offset	Z coordinate offset of the *X-Y* plane to convert the 2D structure into a 3D model. Refer to page *Section 2.2* for details.
Right Width (W2)	Width (*W2*) of the far right support measured normal to the X axis.



Type 4 - Non-parallel sided slab with non-skewed supports



Type 5 - Non-parallel sided slab with uniformly skewed supports



Type 6 - Non-parallel sided slab with non-uniformly skewed supports

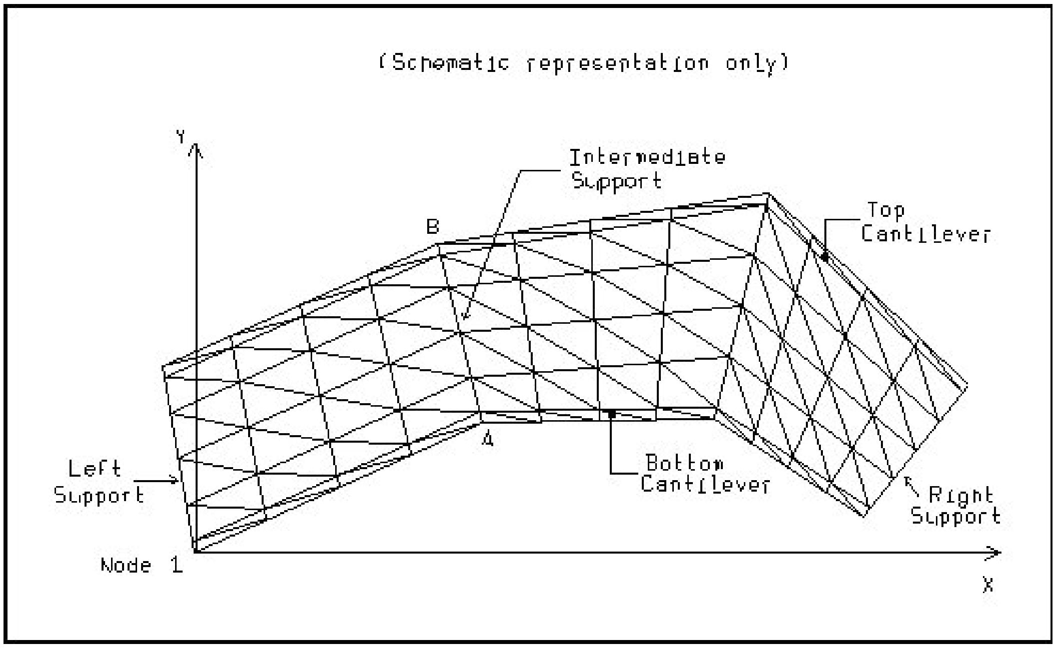
Section 8.4.3 - Slab Type 7

Parameters for Slabs of General Shape

The slab (or bridge deck) is modelled using triangular plate bending elements and is based on the theory developed by O.C. Zienkiewicz. For details refer to: "*The Finite Element Method in Engineering Science*".

Slab *Type 7* represents a FE model in which all lines of elements within individual spans are straight but not necessarily parallel (refer to the diagram at the bottom of the table).

Number of Supports	Total number of spans, or support *lines*, in the structure. This should not be confused with individual points of support (otherwise referred to as *support points*) located at selected nodes.
Support Geometry	Select a support and enter its (X, Y) start and end coordinates. When activated, a list of support lines will be displayed in the panel. For each support line *A*-*B* as shown in the schematic diagram, ***A*** refers to the end closest to the bottom edge of the bottom cantilever (if any) and ***B*** to the end closest to the top edge of the top cantilever (if any).
Long. Divisions	The number of longitudinal mesh divisions in each span. When activated, a list of spans with default span divisions will be displayed. This parameter is essentially used to create the number of longitudinal slab elements in the mesh. The slab is modelled using triangular plate bending elements (refer to *Part 2.2* for details of the modelling process).
Y Divisions	Number of transverse mesh divisions (excluding top and bottom cantilevers). The slab is modelled using triangular plate bending elements (refer to *Part 2.2* for details of the modelling process).
Top Cantilever	Width of top cantilever. Enter zero if no top cantilever is required. Note that the outside edge of the cantilever will be parallel to the outside top main girder only for the first span. There-after the width will vary in the same proportion as the splay of the main girders.
Bottom Cantilever	Width of bottom cantilever. Enter zero if no bottom cantilever is required. Note that the outside edge of the cantilever will be parallel to the first main girder only for the first span. There-after the width will vary in the same proportion as the splay of the main girders.
Z Offset	Z coordinate offset of the *X-Y* plane to convert the 2D structure into a 3D model. Refer to page *Section 2.2* for details.


(Schematic representation only)
Y
Intermediate Support
B
Top Cantilever
Left Support
A
Bottom Cantilever
Right Support
Node 1
X

Section 8.4.4 - Slab Types 8, 9

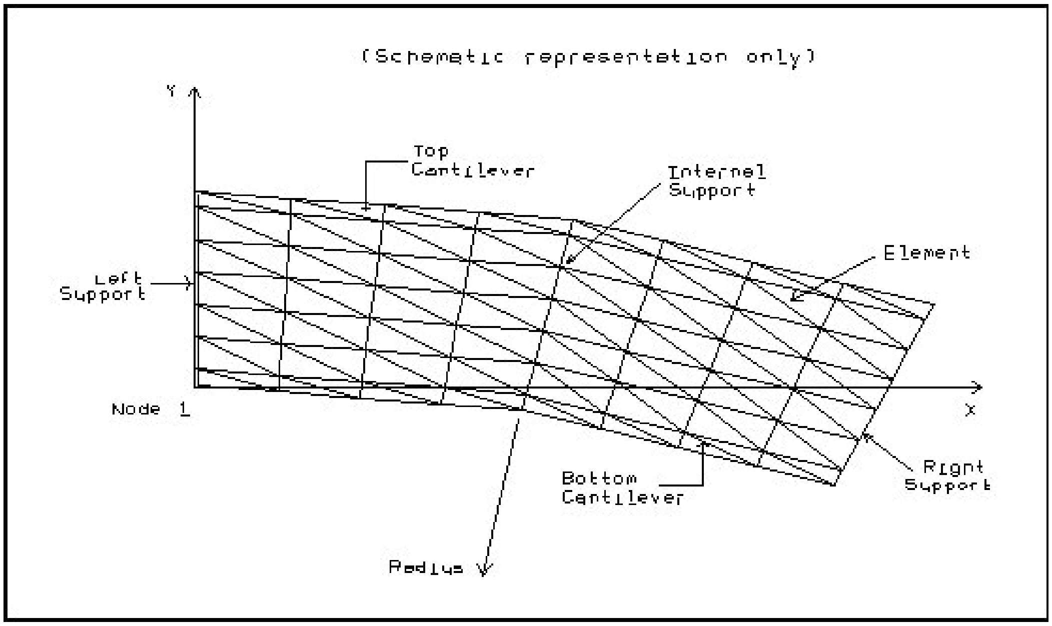
Parameters for Slabs on a Circular Curve

The slab (or bridge deck) is modelled using triangular plate bending elements and is based on the theory developed by O.C. Zienkiewicz. For details refer to: "*The Finite Element Method in Engineering Science*".

Slabs *Type 8* and 9 represent FE models in which the span geometry follows a circular curve. Two types of slab meshing templates are available (refer to the diagrams at the bottom of the table):

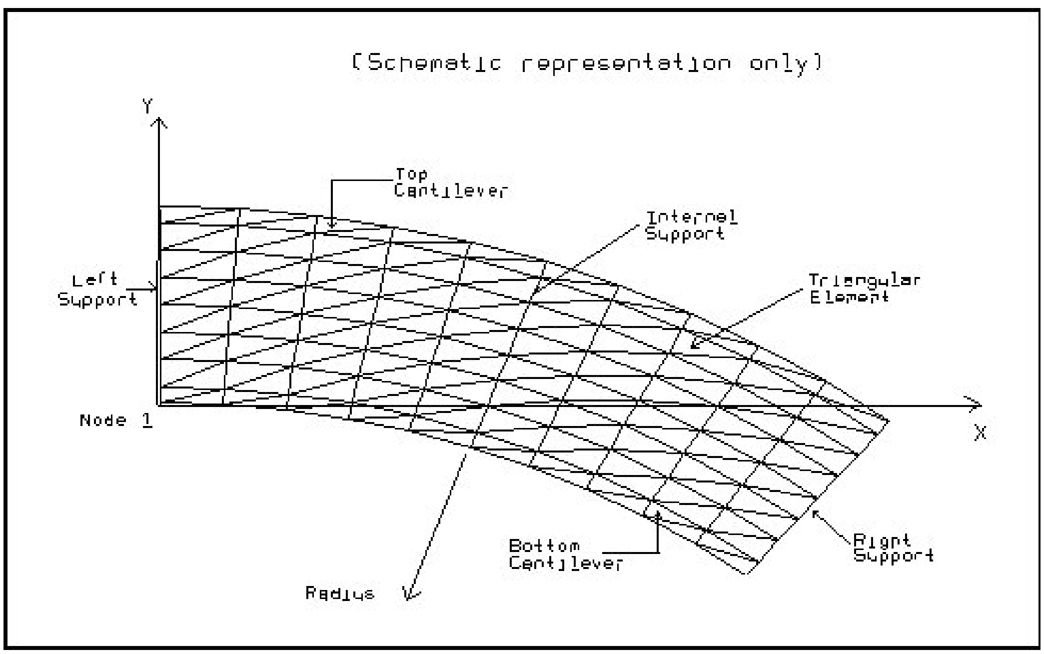
Type 8	Each span is straight but the support lines are oriented on a circular curve
Type 9	All spans are curved and all support lines lie on a circular curve

Number of Supports	Total number of support *lines* in the structure. This should not be confused with individual points of support (otherwise referred to as *support points*) located at selected nodes.
Span Length	Span lengths measured along the radius of curvature at the centre-line of the deck.
Longitudinal Divisions	Number of longitudinal mesh divisions in each span. When activated, a list of spans with default span divisions will be displayed. This parameter is essentially used to create the number of longitudinal triangular plate bending elements in the mesh (refer to *Part 2.2* for details of the modelling process).
Y Divisions	Number of transverse mesh divisions (excluding top and bottom cantilevers). The slab is modelled using triangular plate bending elements (refer to *Part 2.2* for details of the modelling process).
Top Cantilever	Width of top cantilever (measured at right angles to the top edge). Set to zero if a top cantilever is not required.
Bottom Cantilever	Width of bottom cantilever (measured at right angles to the bottom edge). Set to zero if a bottom cantilever is not required.
Z Offset	Z coordinate offset of the *X-Y* plane to convert the 2D structure into a 3D model. Refer to page *Section 2.2* for details.
Width	Overall width of deck.
Radius	Radius of curvature. The radius is measured with respect to the centreline of the deck and not the bottom left corner.



Type 8 - Parallel sided deck along circular curve

▲



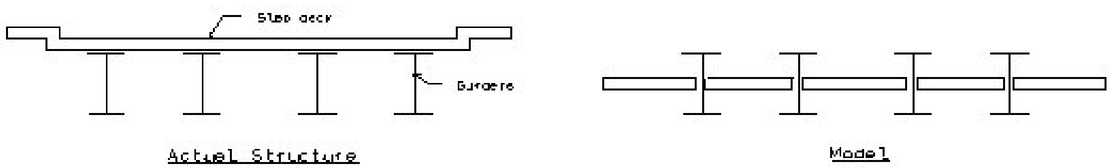
Type 9 - Curved deck on circular curve

Section 8.5.1 - Grillage Slab Types 1, 2, 3

Grillage Slabs With Parallel Main Girders

In these model types the girders are represented as straight grillage members and the top deck slab as a mesh of plate bending elements. Finite Element analysis is based on the theory developed by O.C. Zienkiewicz. (For details refer to the publication: "*The Finite Element Method in Engineering Science*").

Warning! The offset between the slab and girder centroids is not accurately represented in the final generated model. Geometrically, the slab is assumed to be located at the centroid of the girder, as shown in the model diagram at right.



Note, however, that once the mesh has been generated the program does have a facility for automatically calculating the composite section properties of the main longitudinal girders, where proper allowance is made for the slab-girder centroidal offset distance.

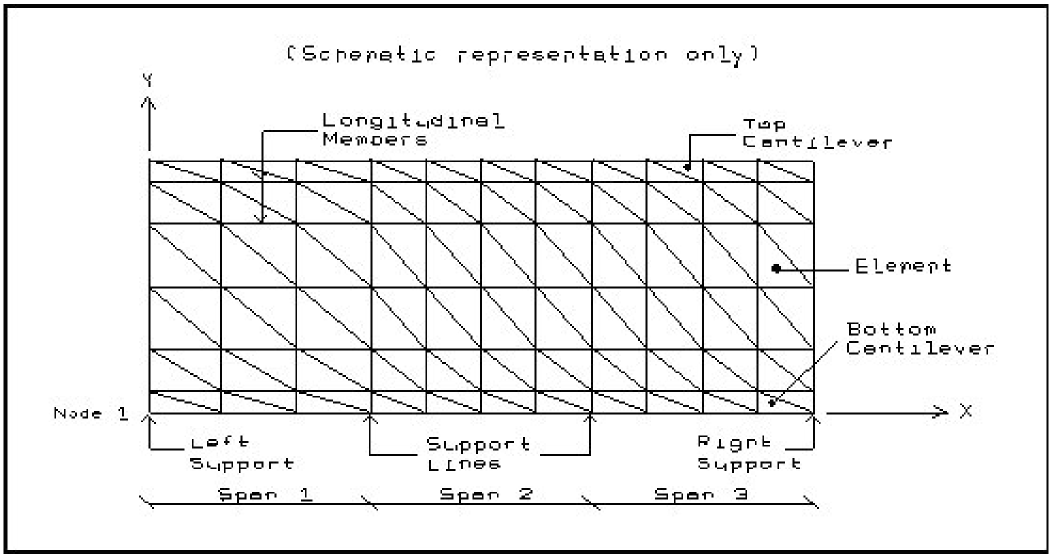
The slab mesh is generated using either rectangular plate bending elements or a combination of triangular and rectangular elements. Grillage-slabs *Type 1,2* and *3* represent FE models in which all lines of elements are straight and parallel. Three types of grillage-slab meshing templates are available (refer to the diagrams at the bottom of the table):

Type 1	All support lines in the structure are orthogonal to the global *X-axis*
Type 2	All support lines in the structure are uniformly skewed
Type 3	Each support line in the structure can have a different skew

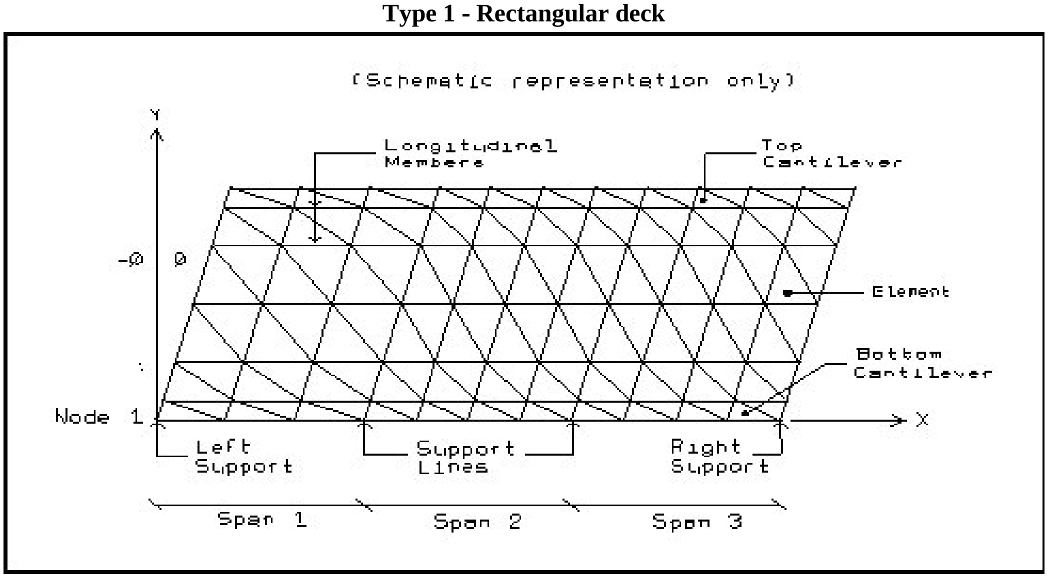
Parameters:

Number of Spans	Total number of spans in the structure.
No. Girders	Number of main longitudinal girders. Cantilever edge beams are *not* generated for this structure type.
Top Cantilever	Width of top cantilever (measured at right angles to the top edge). Set to zero if a top cantilever is not required.
Bottom Cantilever	Width of bottom cantilever (measured at right angles to the bottom edge). Set to zero if a bottom cantilever is not required.
FE Type	Type of Finite Element with which to model the deck slab. This parameter is only available for *Type 1* slab structures (in all other cases triangular elements used). See diagram below.
Z Offset	Z coordinate offset of the *X-Y* plane to convert the 2D structure into a 3D model. Refer to page *Part 2.2* for details.
Transverse Divisions	The number of transverse subdivisions of the slab between the main girders. This parameter is only used to subdivide the slab mesh transversely between the main girders.

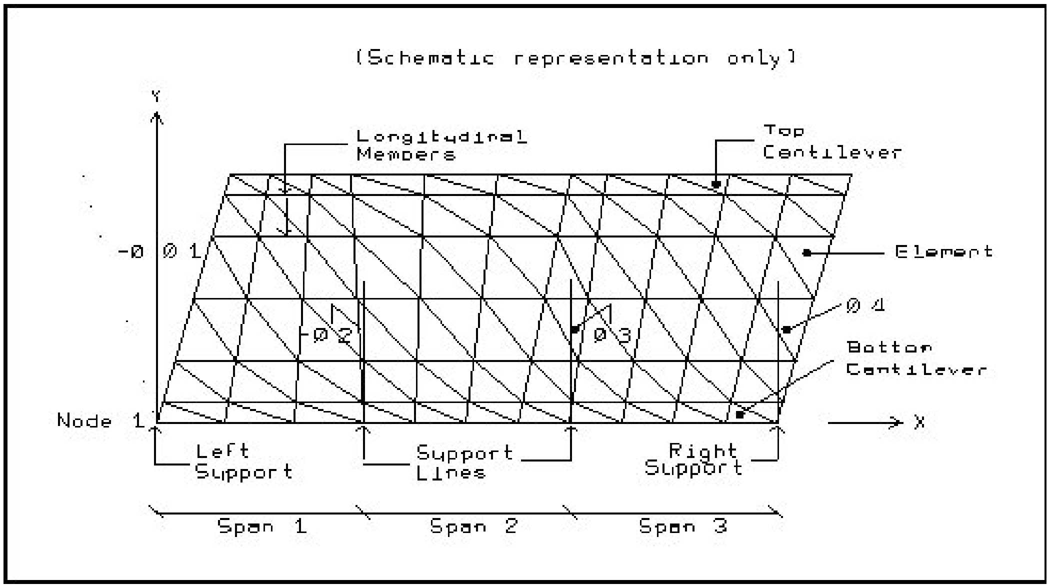
Cantilever Divisions	The number of transverse subdivisions of the cantilever deck slabs. This parameter is only relevant if top or bottom cantilevers are present.
Span Lengths ...	Span lengths measured along the X axis with respect to the bottom edge of the bridge deck (refer to individual diagrams for concise definition).
Mesh Divisions ...	Number of longitudinal mesh divisions in each span. Note that transverse *grillage members* are only created at the support lines and _not_ between supports. The slab is modelled using rectangular and/or triangular plate bending elements (refer to *Part 2.2* for details of the FE modelling process).
Girder Spacing ...	Spacing of main longitudinal girders. Edge beams are not generated for these structure types. Girder 1 is closest to Node 1 (i.e to the bottom edge of the deck).
Skew Angle ...	Skew angle of supports (measured in *Degrees*). It is _not_ required for *Type 1* slabs. A data entry dialog box will appear for *Type 3* models to allow skew angles to be entered for all lines of support.
Verify Geometry	This button enables the generated FE mesh to be viewed from the three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the geometry prior to clicking *OK*.



Type 1 - Rectangular deck



Type 2 - Parallel-sided deck slab with uniformly skewed supports



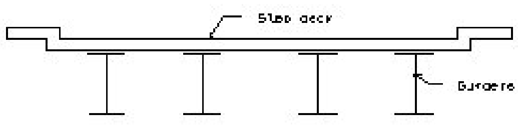
Type 3 - Parallel sided deck with non uniformly skewed supports

Section 8.5.2 - Grillage Slab Types 4, 5, 6

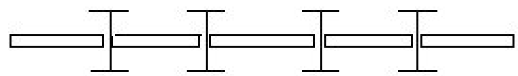
Grillage-Slabs With Non-Parallel Main Girders

In these model types the girders are represented as straight grillage members and the top deck slab as a mesh of triangular plate bending elements. Finite Element analysis is based on the theory developed by O.C. Zienkiewicz. (For details refer to the publication: "*The Finite Element Method in Engineering Science*").

Warning! The offset between the slab and girder centroids is not accurately represented in the final generated model. Geometrically, the slab is assumed to be located at the centroid of the girder, as shown in the model diagram at right.



Actual Structure



Model

Note, however, that once the mesh has been generated the program does have a facility for automatically calculating the composite section properties of the main longitudinal girders, where proper allowance is made for the slab-girder centroidal offset distance.

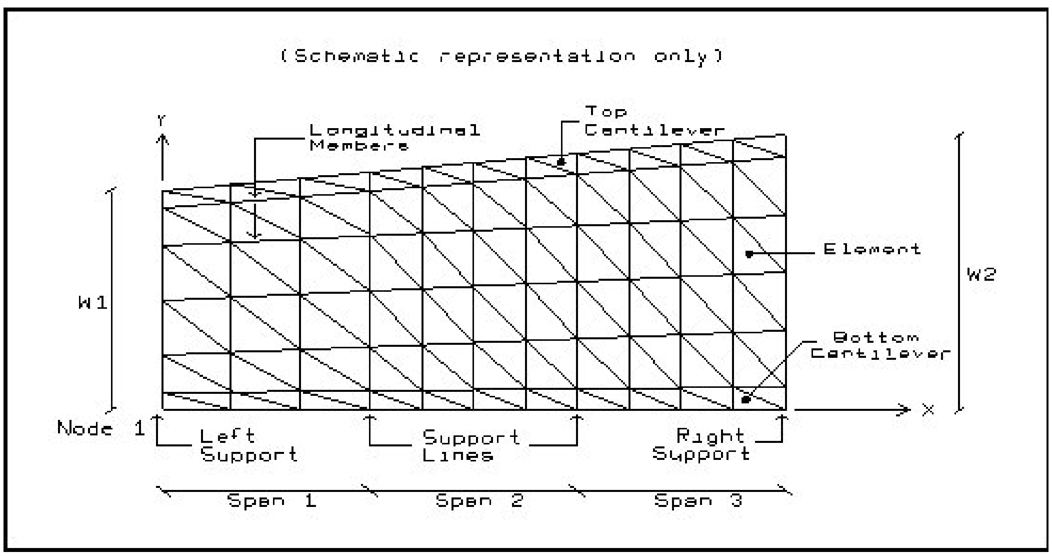
The slab mesh is generated using triangular plate bending elements. Grillage-slabs *Type 4,5* and 6 represent FE models in which main longitudinal beam members and lines of elements are not necessarily parallel. Three types of grillage-slab meshing templates are available (refer to the diagrams at the bottom of the table):

Type 4	All support lines in the structure are orthogonal to the global *X-axis*
Type 5	All support lines in the structure are uniformly skewed
Type 6	Each support line in the structure can have a different skew

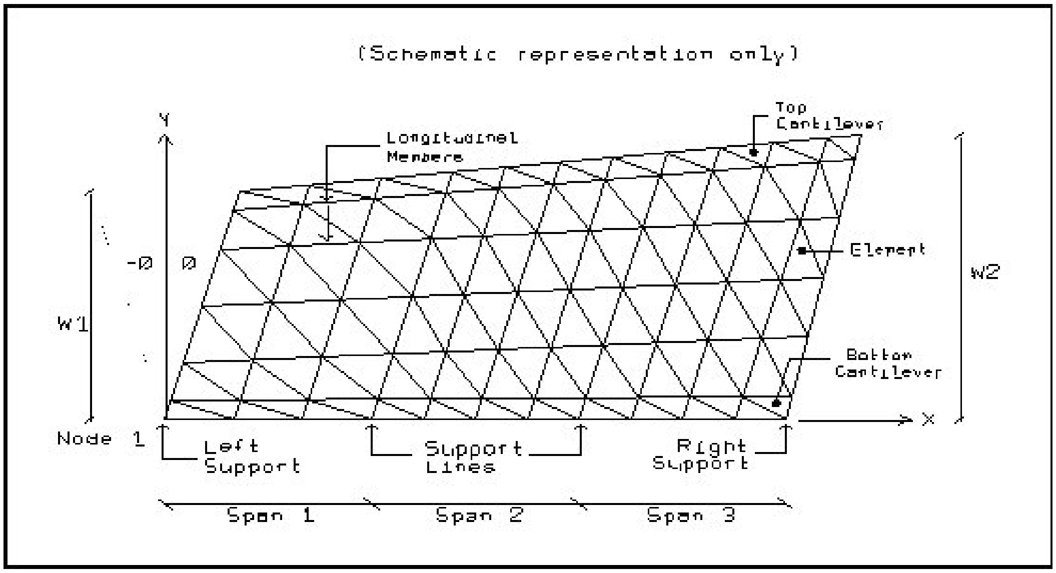
Model Parameters:

Number of Spans	Total number of spans in the model.
Main Girders	Number of main longitudinal girders. Cantilever edge beams are *not* generated.
Top Cantilever	Width of top cantilever (measured at right angles to the top edge). It will then splay out to the far support in the same ratio as the main girders. Set to zero if a top cantilever is not required.
Bottom Cantilever	Width of bottom cantilever (measured at right angles to the bottom edge). It will then splay out to the far support in the same ratio as the main girders. Set to zero if a bottom cantilever is not required.
Right Width (W2)	Width of the far *right* support measured normal to the X-axis. (Note that ***W1*** will be calculated by the system from the other parameters given in this table).
Transverse Divisions	The number of transverse subdivisions of the slab between the main girders. This parameter is only used to subdivide the slab mesh transversely between the main girders.

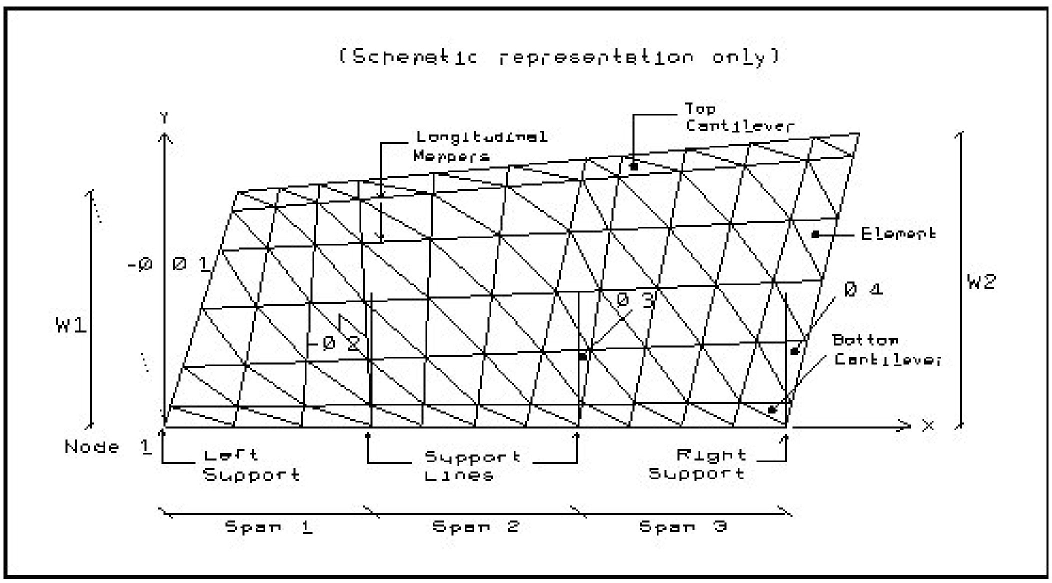
Cantilever Divisions	The number of transverse subdivisions of the cantilever deck slabs. This parameter is only relevant if top or bottom cantilevers are present.
Span Lengths ...	Span lengths measured along the X axis with respect to the bottom edge of the bridge deck (refer to individual diagrams for concise definition).
Mesh Divisions ...	Number of longitudinal mesh divisions in each span. Note that transverse *grillage members* are only created at the support lines and *not* between supports. The slab is modelled using rectangular and/or triangular plate bending elements (refer to *Part 2.2* for details of the FE modelling process).
Girder Spacing ...	Spacing of main longitudinal girders. Edge beams are not generated for these structure types. Girder 1 is closest to *Node 1* (i.e to the bottom edge of the deck).
Skew Angle ...	Skew angle of supports (measured in *Degrees*). It is not required for *Type 4* grillage slabs. A data entry dialog box will appear for *Type 6* models to allow skew angles to be entered for all lines of support.
Verify Geometry	This button enables the generated FE mesh to be viewed from the three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the geometry prior to clicking *OK*.



Type 4: Non-parallel sided deck with no-skewed supports



Type 5: Non-parallel sided deck with uniformly skewed supports



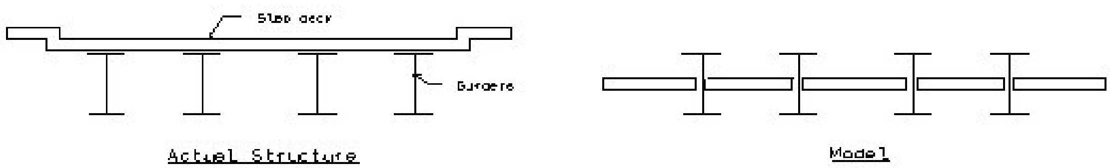
Type 6: Non-parrallel sided deck with non-uniformly skewed supports

Section 8.5.3 - Grillage Slab Type 7

General Grillage-Slab Shapes

In these model types the girders in each span are represented as straight grillage members and the top deck slab as a mesh of triangular plate bending elements. Finite Element analysis is based on the theory developed by O.C. Zienkiewicz. (For details refer to the publication: "*The Finite Element Method in Engineering Science*").

Warning! The offset between the slab and girder centroids is not accurately represented in the final generated model. Geometrically, the slab is assumed to be located at the centroid of the girder, as shown in the model diagram at right.

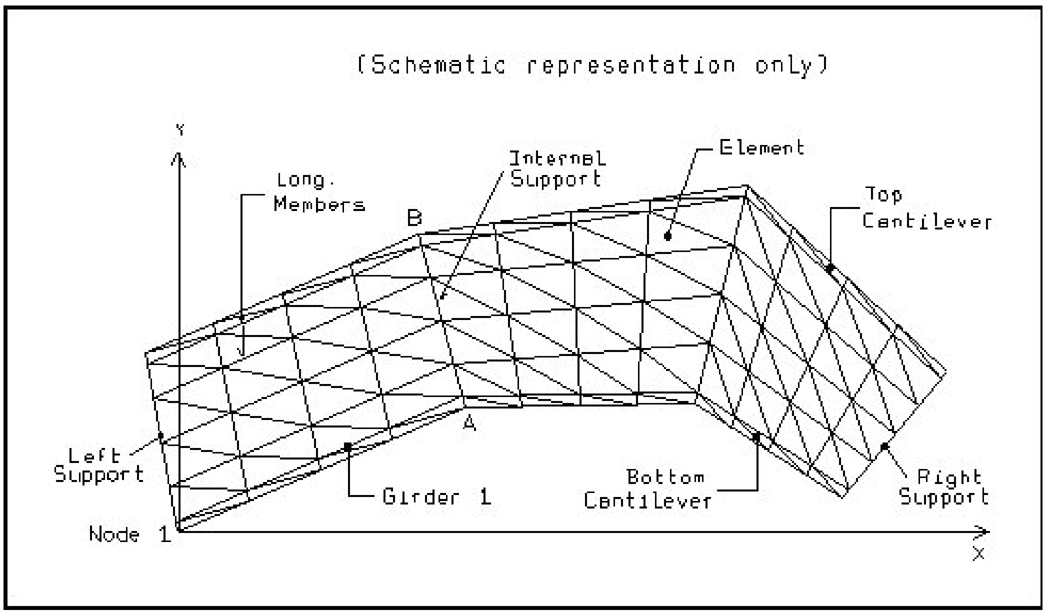


Note, however, that once the mesh has been generated the program does have a facility for automatically calculating the composite section properties of the main longitudinal girders, where proper allowance is made for the slab-girder centroidal offset distance.

Model Parameters:

Number of Supports	Total number of support *lines* in the structure. This is not to be confused with vertical support *points* that are created by ACES under each main longitudinal girder where they intersect the support lines.
No. Girders	Number of main longitudinal girders. Cantilever edge beams are *not* generated.
Top Cantilever	Width of top cantilever (measured at right angles to the top edge at the *left* support line). It will then splay out to the far support in the same ratio as the main girders. Set to zero if a top cantilever is not required.
Bottom Cantilever	Width of bottom cantilever (measured at right angles to the bottom edge at the *left* support line). It will then splay out to the far support in the same ratio as the main girders. Set to zero if a bottom cantilever is not required.
Z Offset	Z coordinate offset of the *X-Y* plane to convert the 2D structure into a 3D model. Refer to page *Part 2.2* for details.
Girder Divisions	The number of transverse subdivisions of the slab between the main girders. This parameter is only used to subdivide the slab mesh transversely between the main girders.
Cantilever Divisions	Number of transverse FE mesh subdivisions of the slab cantilever slab (only relevant if top or bottom cantilevers are present). This is used to subdivide the slab mesh transversely within the cantilevers. Both top and bottom cantilevers will be subdivided using this number.
Support Coordinates ...	Span lengths measured along the X axis with respect to the bottom edge of the bridge deck (refer to individual diagrams for concise definition).

Mesh Divisions ...	Number of longitudinal mesh divisions in each span. Note that transverse *grillage members* are only created at the support lines and *not* between supports. The slab is modelled using rectangular and/or triangular plate bending elements (refer to *Part 2.2* for details of the FE modelling process).
Girder Spacing ...	Girder spacing along the far left support line. When activated a list of all girder-to-girder spacings will be displayed with default values preset by ACES. You may change these to suit. Edge beams are not generated for this structure type and Girder 1 is closest to the bottom edge of the deck.. Girder spacings at all other support lines will be calculated by ACES in proportion to the splay of the main girders. ACES will check that the length ***A-B*** at the first support line is compatible with the cantilever widths and girder spacings. If not, the ***B*** coordinate will be moved until it is.
Verify Geometry	This button enables the generated FE mesh to be viewed from the three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the geometry prior to clicking *OK*.

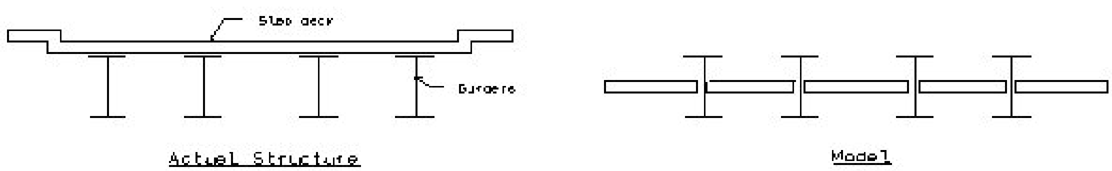


Section 8.5.4 - Grillage Slab Types 8, 9

Main Girders and Deck on a Circular Curve

In these two model types the girders are represented as straight grillage members and the top deck slab as a mesh of triangular plate bending elements. Finite Element analysis is based on the theory developed by O.C. Zienkiewicz. (For details refer to the publication: "*The Finite Element Method in Engineering Science*").

Warning! The offset between the slab and girder centroids is not accurately represented in the final generated model. Geometrically, the slab is assumed to be located at the centroid of the girder, as shown in the model diagram at right.



Note, however, that once the mesh has been generated the program does have a facility for automatically calculating the composite section properties of the main longitudinal girders, where proper allowance is made for the slab-girder centroidal offset distance.

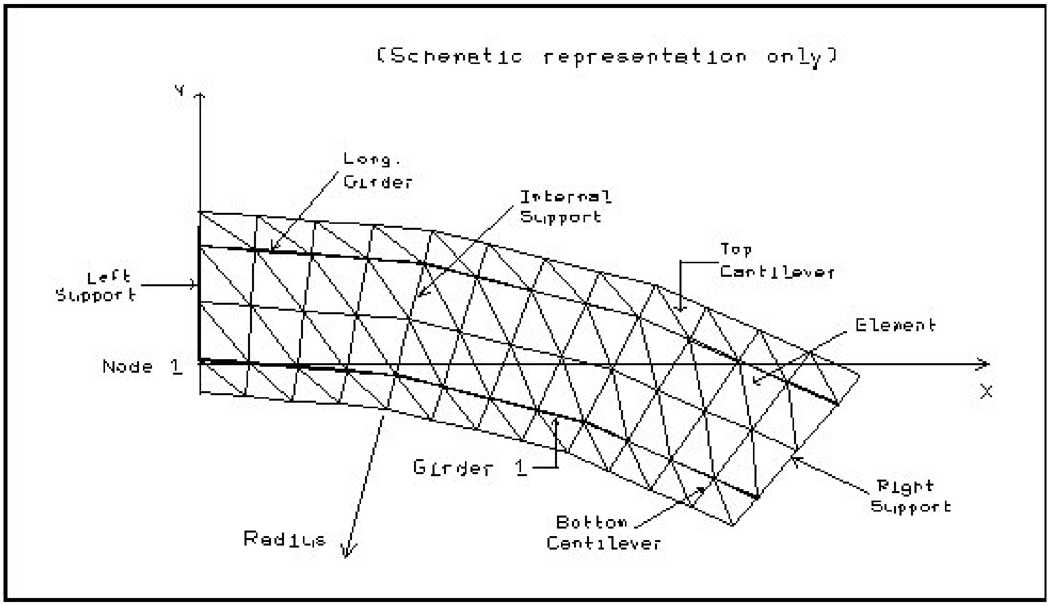
The slab mesh is generated using triangular plate bending elements. Grillage-slabs *Type 8* and *9* represent FE models in which main longitudinal beam members and lines of elements lie on a circular curve. Two types of grillage-slab meshing templates are available (refer to the diagrams at the bottom of the table):

- *Type 8* All girders and elements in each span are straight but the support lines lie on a circular curve.
- *Type 9* All girders in each span are straight but the deck is curve in plan and support lines lie on a circular curve.

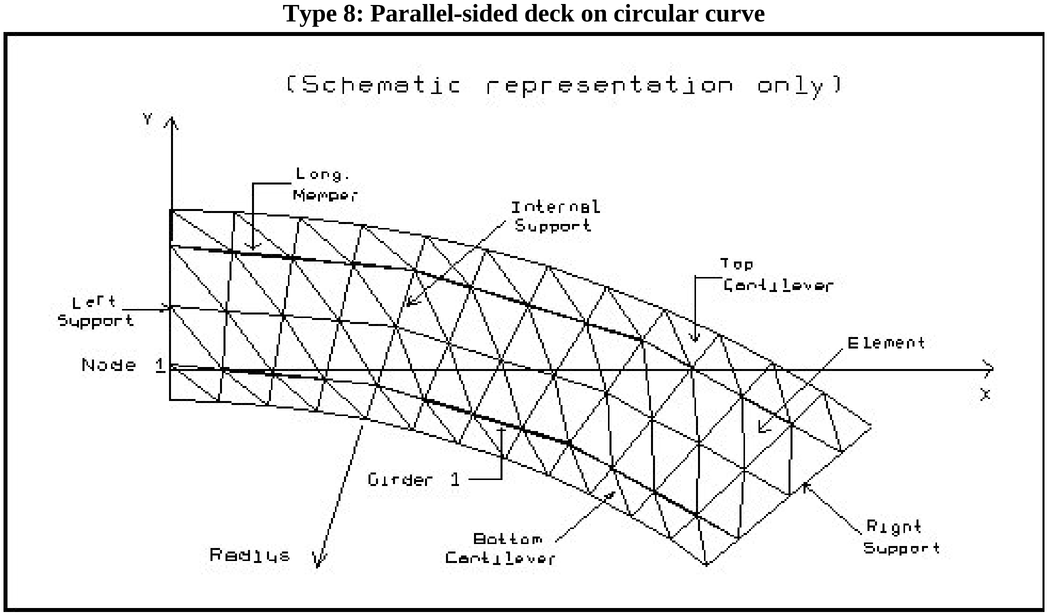
Model Parameters:

Number of Supports	Total number of support *lines* in the structure. This is not to be confused with vertical support *points* that are created by ACES under each main longitudinal girder where they intersect the support lines.
No. Girders	Number of main longitudinal girders. Cantilever edge beams are *not* generated.
Girder Spacing	Spacing of main longitudinal girders at the left support. Note that all girders have the same spacing and edge beams are not generated for these structure types. Girder 1 is closest to *Node 1* (i.e to the bottom edge of the deck).
Top Cantilever	Width of top cantilever (measured at right angles to the top edge). Set to zero if a top cantilever is not required.
Bottom Cantilever	Width of bottom cantilever (measured at right angles to the bottom edge). Set to zero if a bottom cantilever is not required.
Z Offset	Z coordinate offset of the *X-Y* plane to convert the 2D structure into a 3D model. Refer to page *Part 2.2* for details.
Transverse Divisions	The number of transverse subdivisions of the slab between the main girders. This parameter is only used to subdivide the slab mesh transversely between the main girders.

Cantilever Divisions	The number of transverse subdivisions of the cantilever deck slabs. This parameter is only relevant if top or bottom cantilevers are present.
Radius	Radius of curvature. The radius is measured with respect to the centre line of the deck and _not_ the bottom edge.
Span Lengths ...	Span lengths measured along the radius of curvature of the deck at the centre line..
Mesh Divisions ...	Number of longitudinal mesh divisions in each span. Note that transverse *grillage members* are only created at the support lines and _not_ between supports. The slab is modelled using rectangular and/or triangular plate bending elements (refer to Part 2.2 for details of the FE modelling process).
Verify Geometry	This button enables the generated FE mesh to be viewed from the three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the geometry prior to clicking *OK*.



Type 8: Parallel-sided deck on circular curve

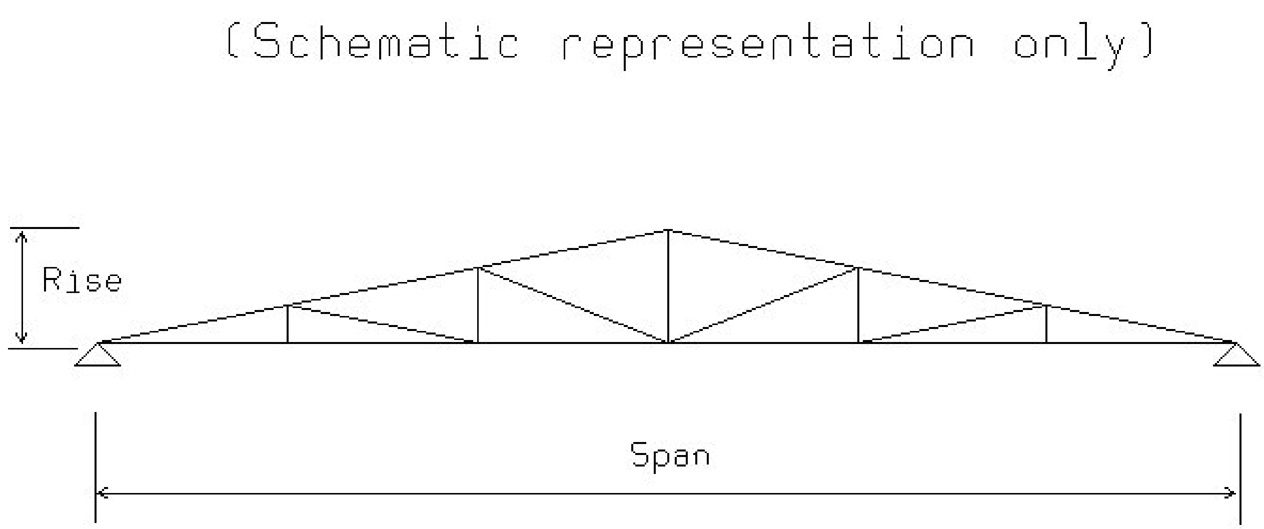


Type 9: Curved deck on circular curve

Section 8.6.1 - Truss Type 1

Parameters for Pitched Truss

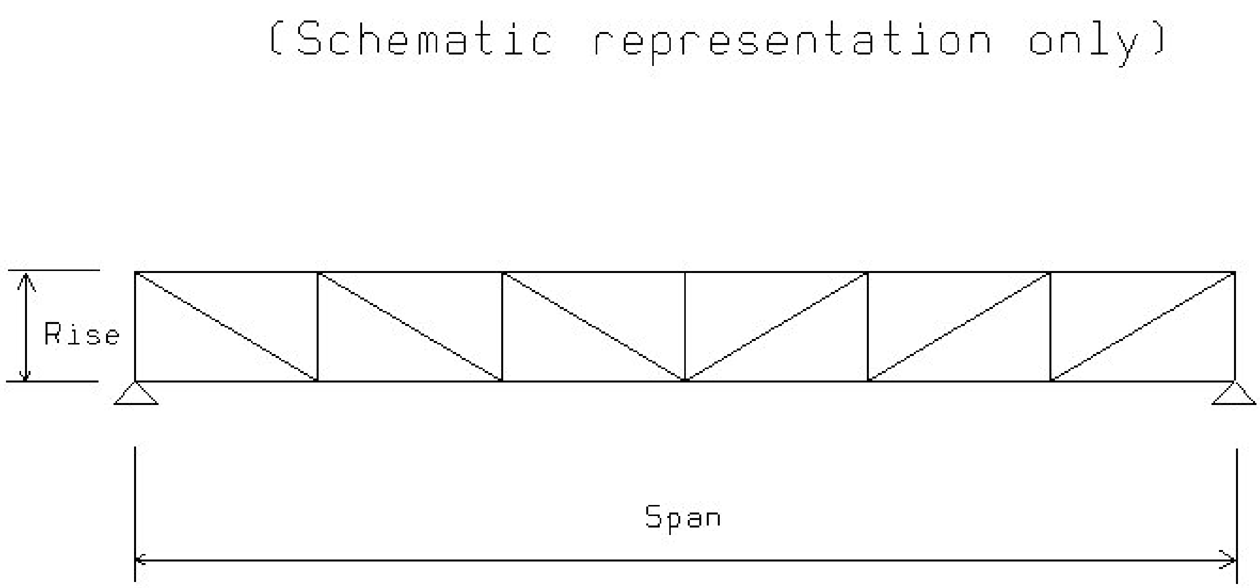
Span Length	Span length
Rise	Rise
Chord Divisions	Number of bottom chord divisions (it must be an even number).
Number of Trusses	Number of parallel trusses.
Y Offset	Offset of the beam in the Y direction (e.g. to allow for integral piers). Refer to *Section 2.4* for details.
Truss Spacing ...	Spacing between adjacent trusses.
Verify Geometry	This button enables the generated truss geometry to be viewed from three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the model prior to clicking *OK*.



Section 8.6.2 - Truss Type 2

Parameters for Pratt Truss

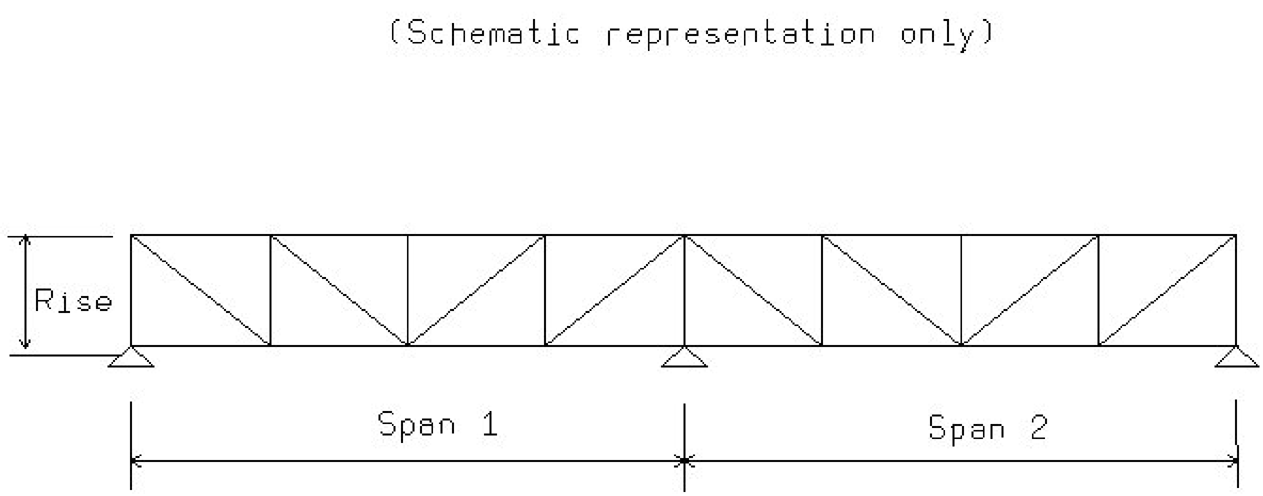
Span Length	Span length
Rise	Rise
Chord Divisions	Number of bottom chord divisions (must be an even number).
Number of Trusses	Number of parallel trusses.
Y Offset	Offset of the beam in the Y direction (e.g. to allow for integral piers). Refer to *Section 2.4* for details.
Truss Spacing ...	Spacing between adjacent trusses.
Verify Geometry	This button enables the generated truss geometry to be viewed from three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the model prior to clicking *OK*.



Section 8.6.3 - Truss Type 3

Parameters for Modified Pratt Truss

Number of Spans	Number of spans
Span Lengths	Span length
Rise	Rise
Chord Divisions	Number of bottom chord divisions (must be an even number).
Number of Trusses	Number of parallel trusses.
Y Offset	Offset of the beam in the Y direction (e.g. to allow for integral piers). Refer to *Section 2.4* for details.
Truss Spacing ...	Spacing between adjacent trusses.
Verify Geometry	This button enables the generated truss geometry to be viewed from three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the model prior to clicking *OK*.



Section 8.7.1 - Walls Types 1, 2

Wingwalls & Retaining Walls

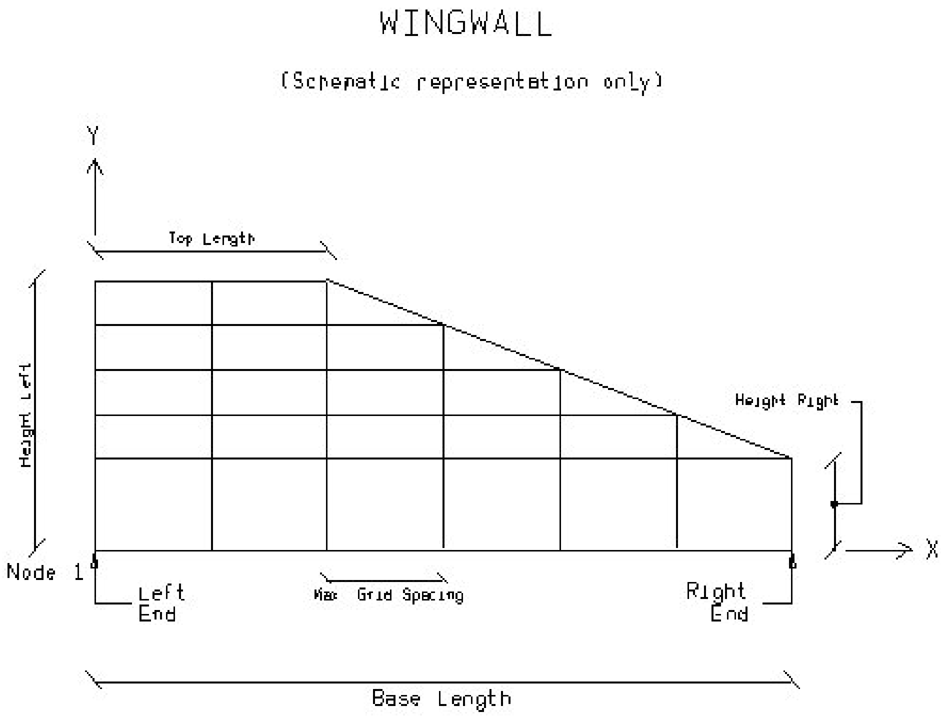
These two structure types enable wingwalls and retaining walls to be modelled using rectangular and triangular plate bending elements. Finite Element analysis is based on the theory developed by O.C. Zienkiewicz. (For details refer to the publication: "*The Finite Element Method in Engineering Science*").

Two types of wall meshing templates are available (refer to the diagrams at the bottom of the table):

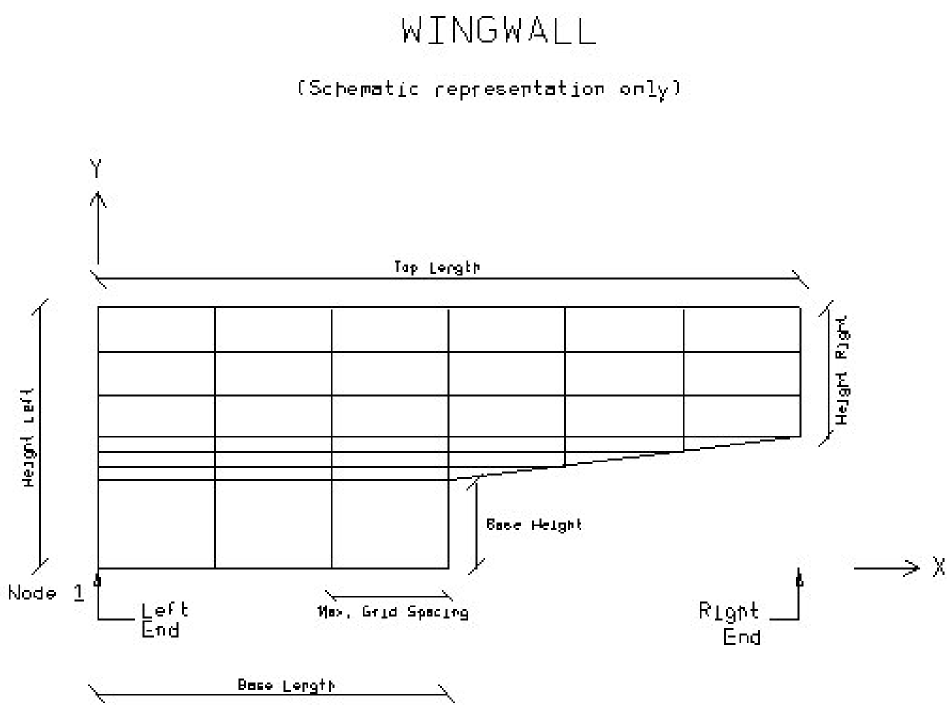
Type 1	Conventional wingwalls
Type 2	Elephant-ear walls

Parameters:

Top Length	Length of top of wall.
Base Length	Base length.
Base Height	Height at base of wall. (Ignore this parameter for *TYPE 1* wingwalls.)
Height Left	Height at left of wall.
Height Right	Height at right of wall.
Grid Spacing	Maximum allowable horizontal grid spacing. ACES will determine the appropriate number of mesh divisions.
Wall Thickness	Thickness of wall slab.
Triangular Pressure	Slope of the triangular earth pressure acting on the wall (applied as a pressure *per unit depth* e.g. *20 kPa/m*). ACES will determine the vertical distribution to each element using the assumption that the surface of the soil follows the top free edge of the wall. The resultant trapezoidal earth pressure loading on the elements is converted into uniformly distributed loads, with appropriate allowance being made for triangular elements. (Tests have shown that if reasonable mesh subdivisions are used the reduction in accuracy is negligible).
Uniform Pressure	Uniform horizontal pressure acting on the wall. This will be applied to all elements as a uniformly distributed load.
Left Supported?	Is the left end of the model supported?
Right Supported?	Is the right end of the model supported? (Ignore this parameter for *TYPE 2* walls).
Base Supported?	Is the base of the model supported?
Verify Geometry	This button enables the generated FE mesh to be viewed from the three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the geometry prior to clicking *OK*.



Type 1: Wingwall



Type 2: Elephant-ear wall

Section 8.7.2 - Cylindrical Silos & Tanks

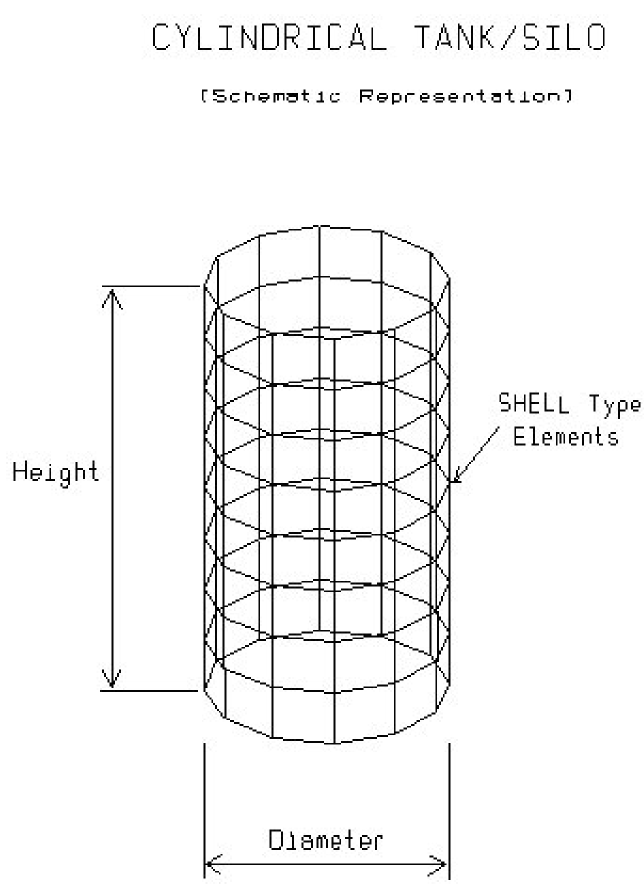
Cylindrical Silos & Tanks

This 3-dimensional structure type is modelled using rectangular *shell* elements. Finite Element analysis is based on the theory developed by O.C. Zienkiewicz. (For details refer to the publication: "*The Finite Element Method in Engineering Science*").

Refer also to *Part 3.7.1 (Creating a Hydrostatic Load)* for details regarding the application of hydrostatic pressure loads to the internal walls of the structure.

Parameters:

Diameter	Diameter of silo.
Height	Height of silo.
Height Divisions	Number of mesh divisions vertically. Rectangular *SHELL* type elements will be generated.
Radial Divisions	Number of radial mesh divisions (i.e., around the circumference of the cylinder).
Verify Geometry	This button enables the generated FE mesh to be viewed from the three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the geometry prior to clicking *OK*.



Section 8.7.3 - Tapered Silos & Tanks

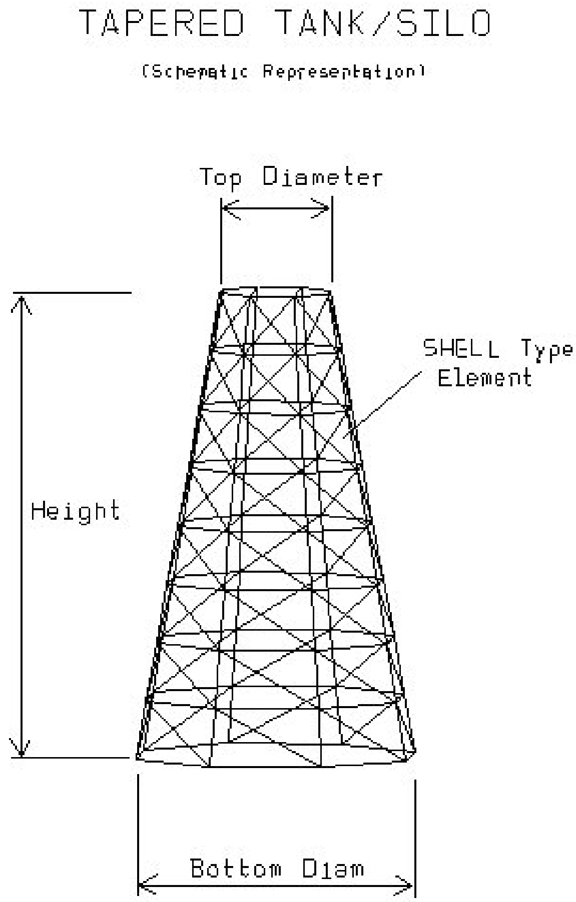
Tapered Silos & Tanks

This 3-dimensional structure type is modelled using rectangular *shell* elements. Finite Element analysis is based on the theory developed by O.C. Zienkiewicz. (For details refer to the publication: "*The Finite Element Method in Engineering Science*").

Refer also to *Part 3.7.1 (Creating a Hydrostatic Load)* for details regarding the application of hydrostatic pressure loads to the internal walls of the structure.

Parameters:

Diameter at Top	Diameter of silo at the top of the structure.
Diameter at Bottom	Diameter of silo at the bottom of the structure.
Height	Height of silo.
Height Divisions	Number of mesh divisions vertically. Rectangular SHELL type elements will be generated.
Radial Divisions	Number of radial mesh divisions (i.e., around the circumference of the cylinder).
Verify Geometry	This button enables the generated FE mesh to be viewed from the three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the geometry prior to clicking *OK*.


TAPERED TANK/SILO
(Schematic Representation)
Top Diameter
SHELL Type
Element
Height
Bottom Diam

Section 8.7.4 - Segmented Silos & Tanks

Segmented Silos & Tanks

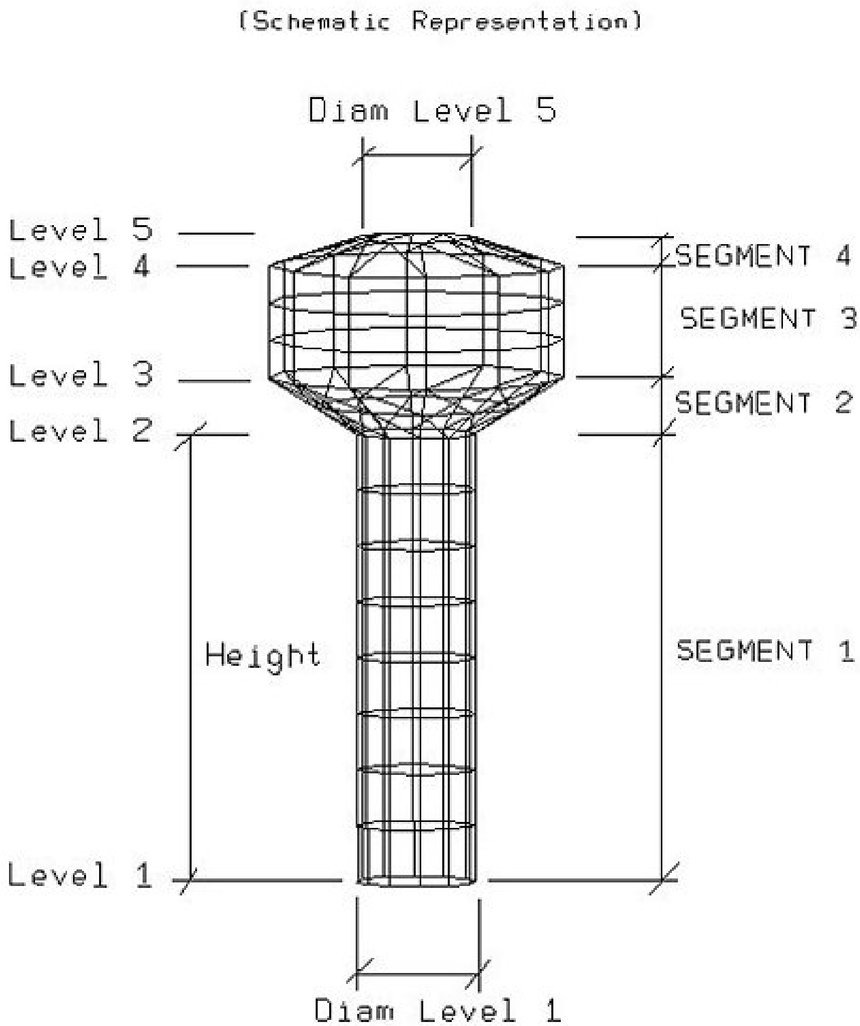
This 3-dimensional structure type is modelled using rectangular *shell* elements. Finite Element analysis is based on the theory developed by O.C. Zienkiewicz. (For details refer to the publication: "*The Finite Element Method in Engineering Science*").

Note that only one element property type is created and assigned to all elements in the model!

Refer also to *Part 3.7.1 (Creating a Hydrostatic Load)* for details regarding the application of hydrostatic pressure loads to other parts of the structural model.

Parameters:

Number of Segments	Number of segments in silo structure.
Radial Divisions	Number of radial mesh divisions (i.e., around the circumference of the cylinder). This value is constant for all segments in the model.
Segment Diameters ...	Diameter of each segment.
Segment Heights ...	Height of each segment.
Height Divisions	Vertical mesh divisions in each segment. A mixture of triangular and rectangular SHELL type elements will be generated as shown in the diagram.
Verify Geometry	This button enables the generated FE mesh to be viewed from the three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the geometry prior to clicking *OK*.


(Schematic Representation)
Diam Level 5
Level 5
Level 4
Level 3
Level 2
Level 1
SEGMENT 4
SEGMENT 3
SEGMENT 2
SEGMENT 1
Height
Diam Level 1

Section 8.7.5 - Rectangular Tanks Types 1, 2

Rectangular Tanks

These 3-dimensional structure types are modelled using rectangular *shell* elements. Finite Element analysis is based on the theory developed by O.C. Zienkiewicz. (For details refer to the publication: "*The Finite Element Method in Engineering Science*").

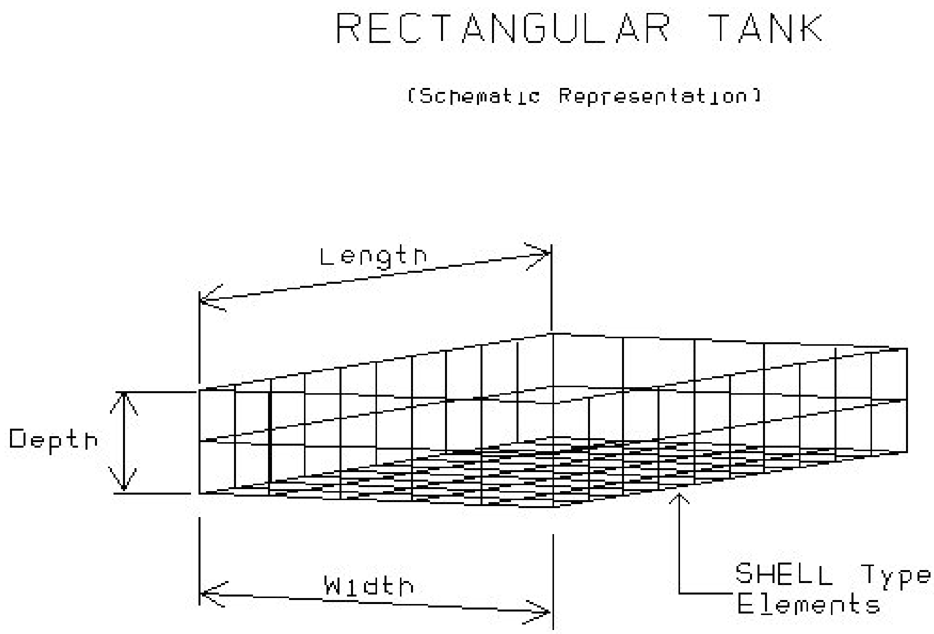
Two types of rectangular tank meshing templates are available (refer to the diagrams at the bottom of the table):

Type 1	Tanks with no ring beams
Type 2	Tanks with a ring beam

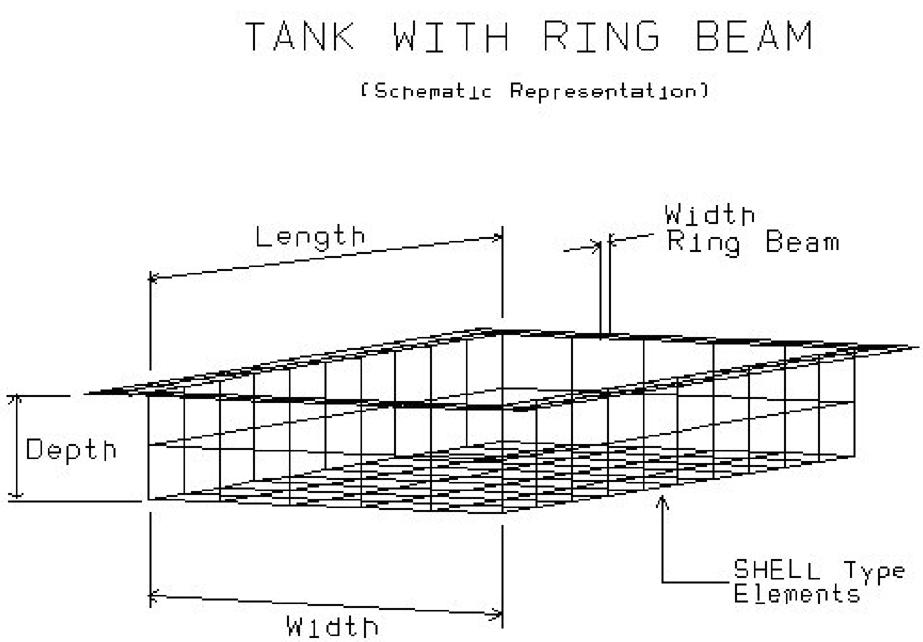
Refer also to *Part 3.7.1 (Creating a Hydrostatic Load)* for details regarding the application of hydrostatic pressure loads to other parts of the structural model.

Parameters:

Length	Length of tank.
Width	Width of tank.
Depth	Depth of tank.
Width Ring Beam	Width of ring beam. (Ignore this parameter for tank *Type 1*).
Length Divisions	Number of divisions along the length of the tank.
Width Divisions	Number of divisions along the width of the tank.
Depth Divisions	Number of divisions along the depth of the tank.
Ring Divisions	Number of divisions transversely within the ring beam. (Ignore for tank *Type 1*).
Verify Geometry	This button enables the generated FE mesh to be viewed from the three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the geometry prior to clicking *OK*.



Type 1 - No Ring Beam



Type 2 - With Ring Beam

Section 8.8.1 - Box Girders Types 1, 2

Box Girder Structures

These 3-dimensional structures are modelled using *shell* elements. Finite Element analysis is based on the theory developed by O.C. Zienkiewicz. (For details refer to the publication: "*The Finite Element Method in Engineering Science*").

ACES will generate meshes consisting of rectangular SHELL elements for straight structures and triangular SHELL elements for curved structures. Note that triangular elements are inherently less accurate than rectangular elements and should be used with caution (or a finer mesh should be employed when modelling the structure).

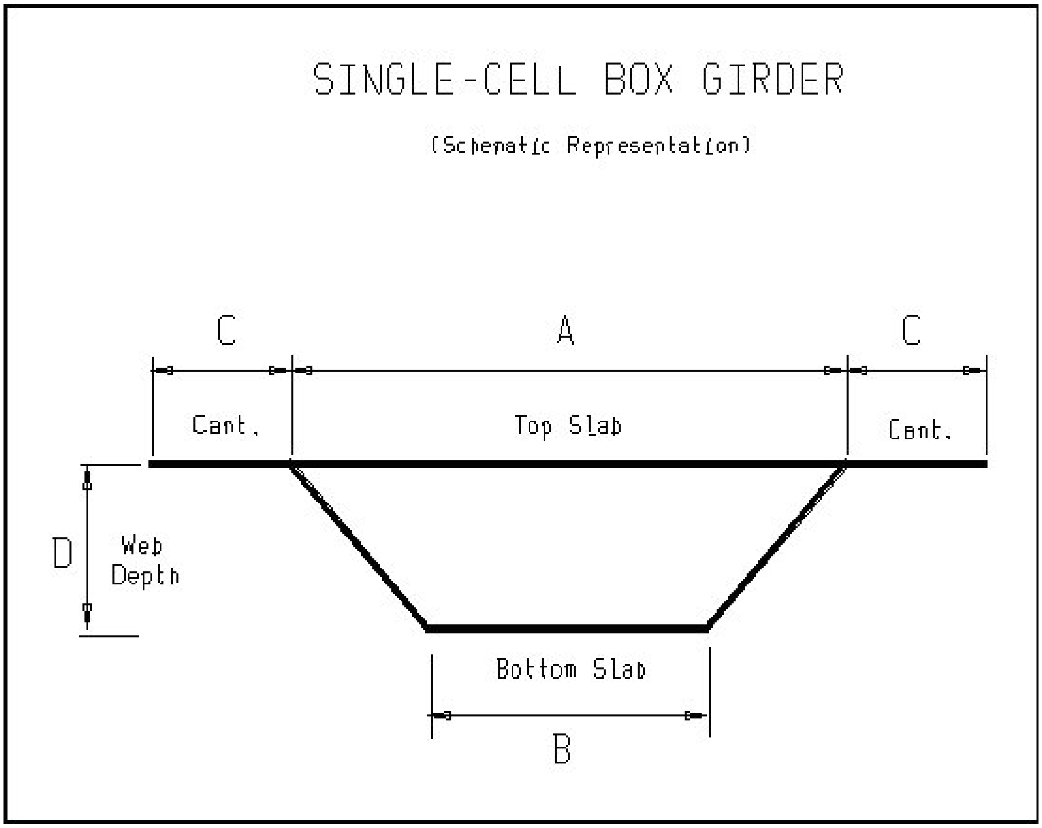
Use caution when selecting the number of mesh divisions, since very large models may result if a fine mesh is used.

Two types of box girder meshing templates are available (refer to the diagrams at the bottom of the table):

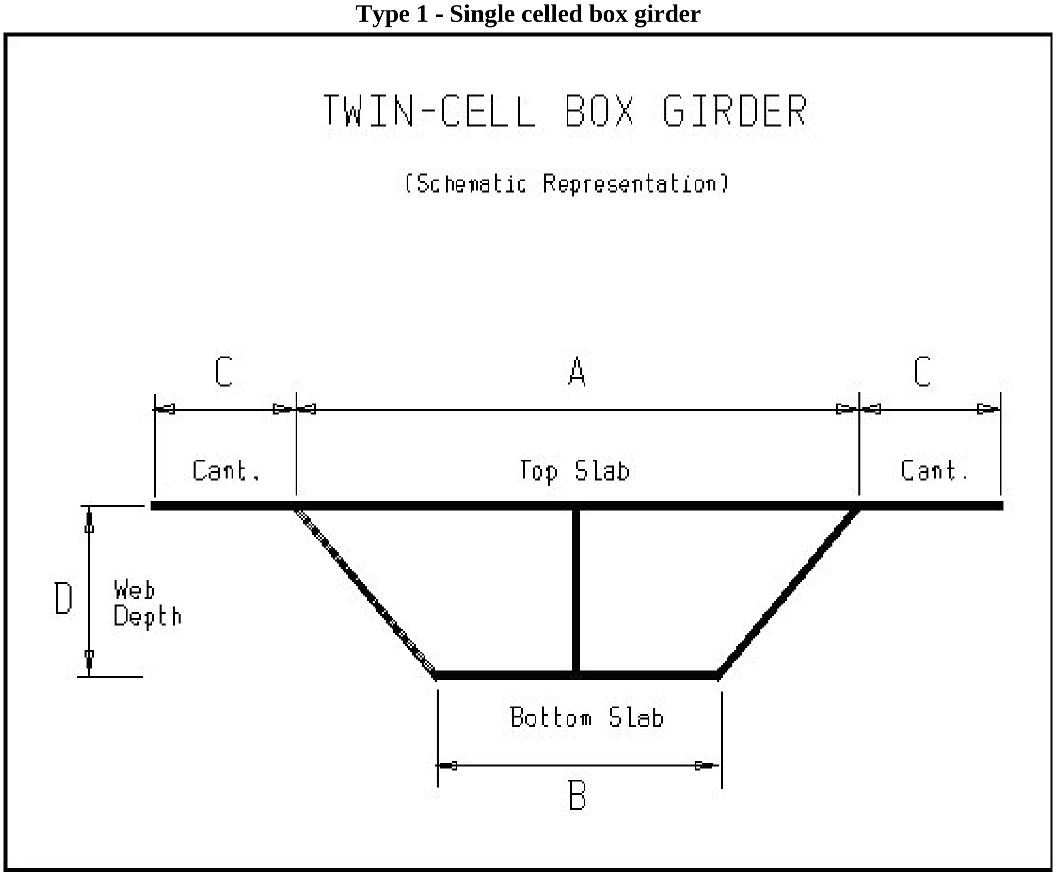
Type 1 Single celled box girder
Type 2 Twin-celled box girder

Parameters:

Number of Spans	Number of spans in structure.
Deck Radius	Radius of curvature of deck (this value should be ***+ve*** if the structure curves down when moving from left to right and ***-ve*** if it curves up). The radius is measured from the centreline of the deck.
Span Lengths ...	Span lengths.
Section Dimensions ...	Section dimensions.
Support Diaphragms ...	Indicate if support diaphragms are present. For each support line toggle the flag to *Yes* if a diaphragm is to be included or *No* if not.
Mesh Divisions ...	Mesh divisions in deck, webs and cantilevers. Use caution when selecting the number of mesh divisions, since very large models may result if fine meshes are used. ACES will generate meshes consisting of rectangular SHELL type elements for straight structures and triangular SHELL type elements for curved structures. Note that triangular elements are inherently less accurate than rectangular elements and should be used with caution (or finer meshes should be employed in modelling the structure).
Verify Geometry	This button enables the generated FE mesh to be viewed from the three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the geometry prior to clicking *OK*.



Type 1 - Single celled box girder



Type 2 - Twin celled box girder

Section 8.8.2 - Box Culverts Types 1, 2, 3

Box Culverts & Crown Units

These 3-dimensional structures are modelled using _shell_ elements. Finite Element analysis is based on the theory developed by O.C. Zienkiewicz. (For details refer to the publication: "*The Finite Element Method in Engineering Science*").

Use caution when selecting the number of mesh divisions, since very large models may result if a fine mesh is used.

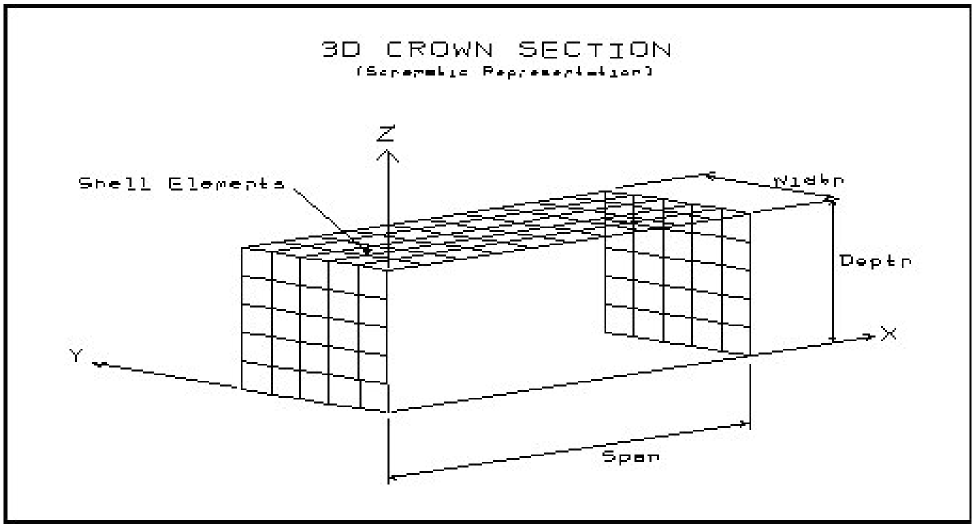
Three types of box culvert templates are available (refer to the diagrams at the bottom of the table):

Type 1 Crown units
Type 2 Crown units with a rectangular hole in the top slab
Type 3 Open-ended box

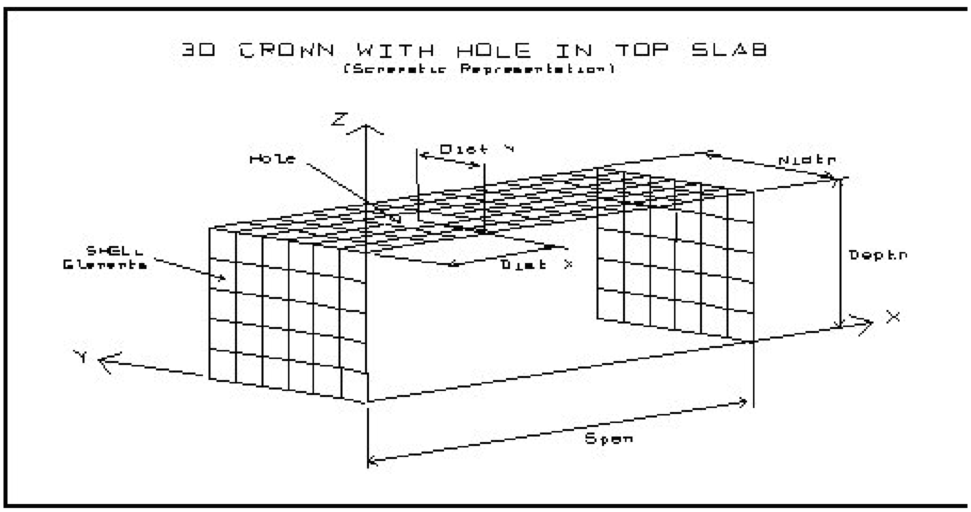
Refer also to *Part 3.7.1 (Creating a Hydrostatic Load)* for details regarding the application of hydrostatic pressure loads to other parts of the structural model.

Parameters:

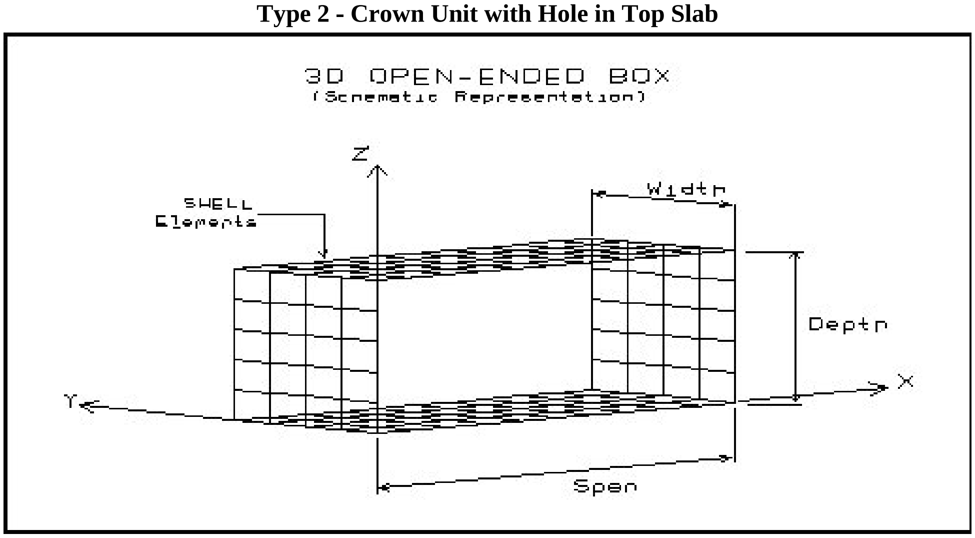
Span	Length of culvert unit.
Width	Width of culvert unit.
Height	Height of culvert unit.
Span Divisions	Number of mesh divisions along span.
Width Divisions	Number of mesh divisions along span width.
Depth Divisions	Number of mesh divisions down unit leg..
Hole Diameter	Diameter of hole (leave zero if there is no hole). Ignore for crown unit *(Type 1)* and box shell (*Type 3*) structures. Note that the hole may not appear circular on the screen drawing.
Hole Distance X	Hole distance ***X***. Ignore for crown unit *(Type 1)* and box shell structures (*Type 3).*
Hole Distance Y	Hole distance ***Y***. Ignore for crown unit (*Type 1)* and box shell structures (*Type 3).*
Verify Geometry	This button enables the generated FE mesh to be viewed from the three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the geometry prior to clicking *OK*.



Type 1 - Crown Unit



Type 2 - Crown Unit with Hole in Top Slab



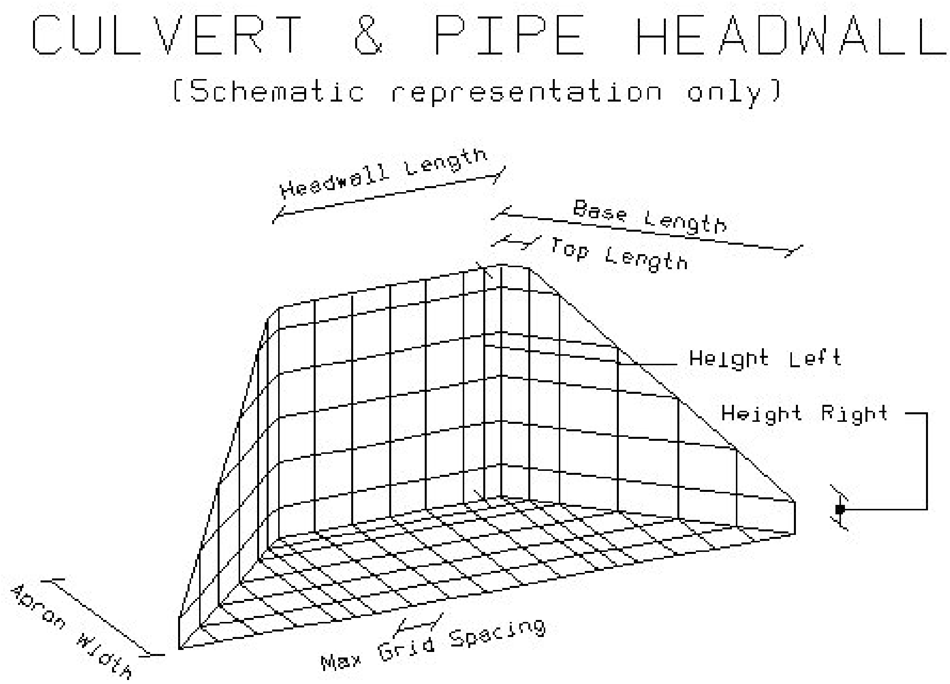
Type 3 - Open Ended Box

Section 8.8.3 - Culvert Headwall

Parameters For Culvert Headwall

This model type enables a culvert headwall structure with integral wingwalls and apron to be modelled using rectangular and triangular plate bending elements. Finite Element analysis is based on the theory developed by O.C. Zienkiewicz. (For details refer to the publication: "*The Finite Element Method in Engineering Science*").

Top Length	Length at the top of the wingwall part of the structure.
Base Length	Length at the bottom of the wingwall part of the structure.
Height Left	Height at the headwall/wingwall junction.
Height Right	Height at the right (free end) of the wingwall.
Grid Spacing	Maximum allowable horizontal grid spacing. ACES will determine the appropriate number of mesh divisions based on this dimension.
Apron Width	Width of apron.
Length Headwall	Length of headwall.
Triangular Pressure	Slope of the triangular earth pressure acting on the wall (applied as a *pressure per unit depth* e.g. 20 kPa/m). ACES will determine the vertical distribution to each element using the assumption that the surface of the soil follows the top free edge of the wall. The resultant trapezoidal earth pressure loading on the elements is converted into uniformly distributed loads, with appropriate allowance being made for triangular elements. (Tests have shown that if reasonable mesh subdivisions are used the reduction in accuracy is negligible). Refer also to *Part 3.7.1 (Creating a Hydrostatic Load)* for details regarding the application of hydrostatic pressure loads to other parts of this structural model.
Uniform Pressure	Uniform horizontal pressure acting on the wall (e.g. 10 kPa). This will be applied to all elements as a uniformly distributed load.
Verify Geometry	This button enables the generated FE mesh to be viewed from the three different orthogonal directions (*X-Y, X-Z* and the *Y-Z* planes). Use it to verify the geometry prior to clicking *OK*.


CULVERT & PIPE HEADWALL
(Schematic representation only)
Headwall Length
Base Length
Top Length
Height Left
Height Right
Apron Width
Max Grid Spacing