*C*oncept
*A*nalyst

USER GUIDE

Version 3.0 (beta)

Concept Analyst Ltd.
Durham University
Department of Engineering
South Road
Durham DH1 3LE

June 2020

Contents

# DISCLAIMER

This software is provided as a tool for an engineer. While it has undergone a series of tests, it is important to recognise that it is performing numerical simulations of physical phenomena, and that these involve approximations. An engineer performing these simulations with Concept Analyst does so at his/her own risk. Neither Concept Analyst, Ltd. nor Durham University shall be held responsible for the results obtained, nor for any consequential loss. Confirming the accuracy and/or usefulness of all the solutions is the responsibility of the licensee or user.

1
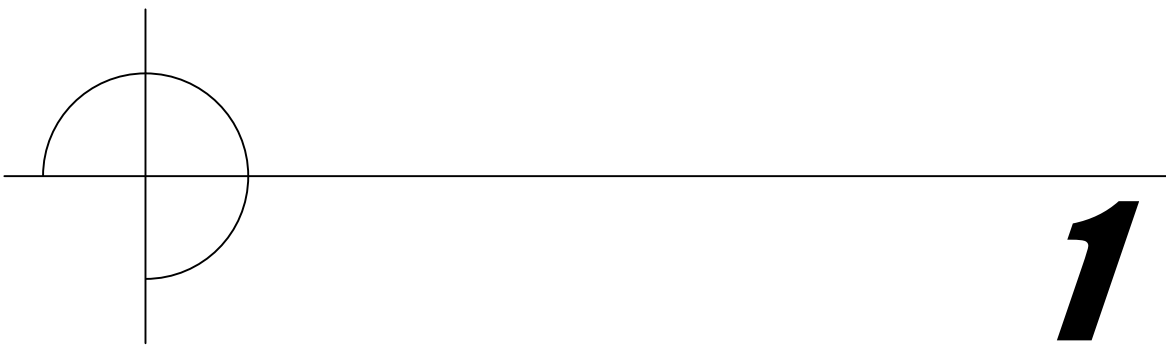
Introduction

Concept Analyst is a Windows-based software system for the calculation of information that allows engineers to assess the durability of mechanical and structural components. It performs these calculations in a sketching environment that allows very rapid solution of the stress distributions through general, two-dimensional shapes. A re-analysis capability allows many design changes to be made while the program automatically updates the stresses that are displayed on the screen.

The program performs numerical analysis calculations that have long been available in commercial software systems. Namely, it performs linear static stress analysis. The advantage that Concept Analyst offers over these well-established packages is the simplicity and speed of usage. The decisions that have traditionally been taken by experienced analysts have, to a large extent, been automated within the software.

1.1 Overview of usage

The software is typically used in the following fashion, which is similar to the basic procedure in any finite element of boundary element system:

- Define geometry by sketching or more precise coordinate definition

- Define applied loads

- Define constraints

- Start the analysis

- View the results

Other operations are available, such as defining the material to be used, the refinement of the model, the view states, re-analysis, etc. However, the above scheme is sufficient as a minimum.

A key feature of Concept Analyst is its ease of use. The geometry is defined in terms of plane shapes, which may be deformed to allow the construction of complex parts. Almost all of the above steps are performed using only mouse input. Indeed, it is possible to create an entire model, run the analysis and view the results without once requiring keyboard input. This enhances the speed of the analysis.

1.2 Usage of the mouse – defaults and definitions

Most mouse operations are performed in the usual way, by positioning the cursor at some location and clicking the **left** mouse button. In the Concept Analyst documentation, including this User Guide, usage of the left mouse button will be assumed unless otherwise stated.

Throughout this documentation, the mouse operations will have their usual meanings, as follows:

Click: press and release the left mouse button

Right-click: press and release the right mouse button

Double-right-click: press and release the right mouse button rapidly twice

Click-and-drag: hold down the left mouse button, drag to a new location and release.

1.3 Layout of the Concept Analyst window

The screen display on starting up the application will include details of the version of the software being used. On clicking in the main window, or on giving any command to start using the software, the initial screen fades and the main screen layout appears as shown in figure 1-1.
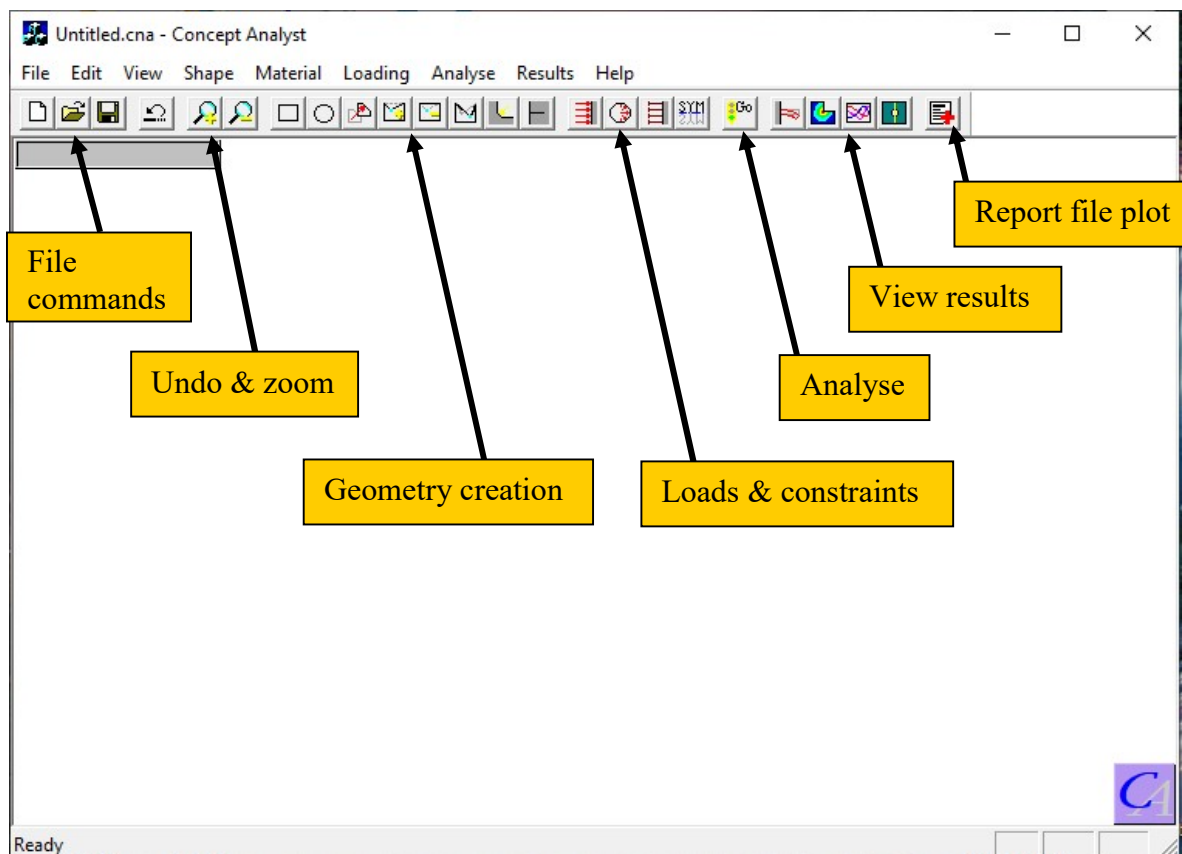
Figure 1-1. Concept Analyst screen layout

The commands to perform all operations are located in the pull-down menus ('File', 'Edit', etc.). The most commonly used commands from these menus are also represented on the toolbar using graphical icons. This greatly enhances the speed of using the program.

2

Installation

2.1 Files supplied

The Concept Analyst software is supplied in the form of three files, as follows:

- concept.exe : the main executable file for the application

- CAMaterials : materials database

- CAconfig : coded file containing licence information

Note that the file 'CAMaterials' is designed to be edited by the user community to build a database of material properties used in that community. See section 4.3 for the formatting information relating to this file.

2.2 Installation steps

The installation of Concept Analyst is very simple. The installation script extracts the required files, places them in the required location, and creates a desktop icon to activate the program.

Registry entries are created to store certain parameters used to control execution of the program and to keep track of personal preferences of users, for example default unit sets and display options.

3
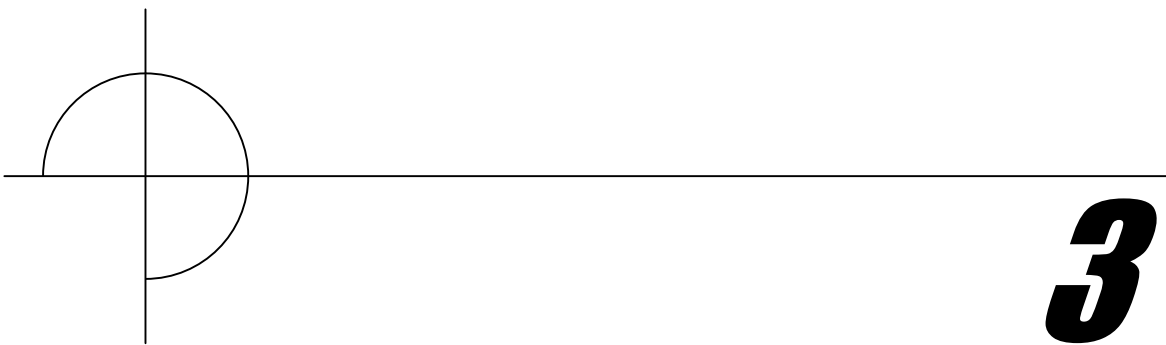
Geometry creation

3.1 Shapes

The basic geometric entity in Concept Analyst is the 'shape'. This is a two-dimensional closed-loop, which may take one of two forms:

- General deformable polygon shape

- Circle shape

Models may contain up to 15 shapes. Because each shape represents a closed loop, one shape will define the outer boundary of the object being analysed, and all other shapes will represent holes of various shapes and sizes in the object.

Circle shapes largely remain circular throughout, though their diameter may be modified later. The exception is that a crack may be generated that emanates from a circle shape as part of a fracture mechanics simulation.

A general shape (deformable polygon) is defined initially as a rectangle. This may later be deformed by adding or removing polygon vertices, by dragging vertices to new locations, by adding fillet radii, etc.

3.2 Units

The units used by Concept Analyst include the following three options:

- Newtons, metres, Pascals

- Newtons, millimetres, megaPascals

- Pounds, inches, psi

The *default* unit set on entering Concept Analyst is (Newtons, millimetres, megaPascals). Note that 1 MPa = 1 N/mm^2. Units are saved along with general model information when a model is saved to file. Units may be changed by giving the command Edit – Units.

3.3 Definition and deformation of shapes

To define a *rectangle*, which may later be deformed, give the command Shape – Rectangle, or select the rectangle icon

Next click-and-drag in the main window area and the rectangle will be drawn. Release the mouse button when the rectangle has reached the required geometry.

Notice that the current (x,y) location of the cursor is displayed in the top left hand corner of the window. There is a default on entering Concept Analyst, that the window spans a certain horizontal dimension and a certain vertical dimension depending on the screen size. If this size is inappropriate for a model, then it may be modified using the command Edit – Width of Unzoomed Screen. This allows any dimension (in the current unit system) to be specified as the dimension that fills the horizontal window size.

To define a *circle* shape, give the command Shape – Circle, or select the Circle icon

Next click-and-drag, starting at the centre of the circle and dragging the mouse until a circle of the required diameter is reached. The radius of the circle is displayed throughout the click-and-drag operation.

3.3.1 Deforming shapes

Circular shapes may be deformed only by changing their radius. This may be done using the command Shape – Resize Circle, followed by a click-and-drag operation on the circle, releasing the mouse button when the desired radius has been achieved and is displayed in the box shown on the screen.

Other shapes, i.e. those initially defined as rectangles, may be deformed in various ways. In general, they will be deformed into general polygons by various operations, as described in sections 3.3.2 to 3.3.11, including:

- Addition of polygon vertices
- Removal of polygon vertices
- Moving polygon vertices
- Moving shapes
- Fillets and other arcs
- Splines
- Rotation of shapes
- Notches
- Moving of lines and line groups

3.3.2 Addition of polygon vertices

Give the command Shape – Add Points, or the add points icon ![icon], and then place the cursor on the shape edge on which the new vertex is to be created. Use a click-and-drag to define the new vertex and drag it to the desired location. For example, after the add points command is given, place the cursor at 'A' (in figure 3-1) and click-and-drag to 'B' to generate the new shape.
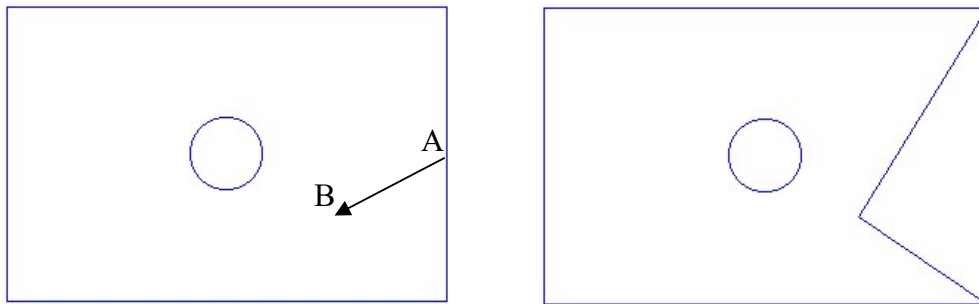
Figure 3-1. Add points example

The point A may be anywhere on (or close to) the right hand edge.

The Shape – Add points command (or add points icon) may be used to create points on existing lines without dragging to new locations.
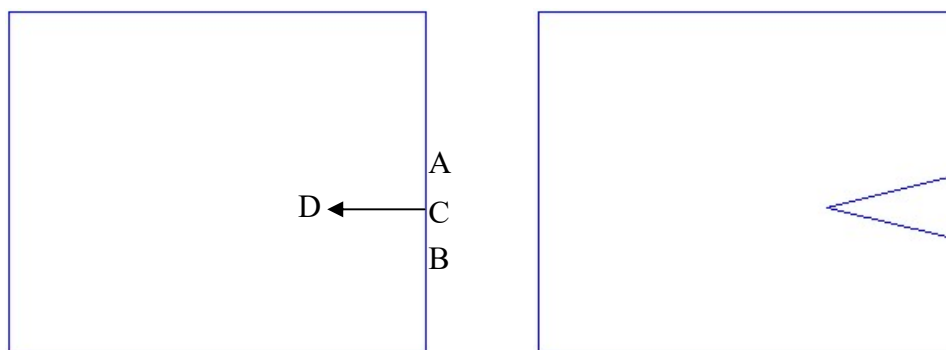
Figure 3-2. Use of Shape – Add points to create a sharp notch

In order to snap new points onto an existing line or arc, the shift key should be pressed when the mouse is clicked to define the point.

At first sight it seems unnecessary to define a new point on an existing line as the geometry appears unchanged, but it is useful for the creation of a wide variety of geometries. For example, figure 3-2 shows a method for creation of a notch. After giving the Shape – Add points command, click and release (with the Shift key pressed and without dragging) at both A and B, and then click-and-drag point C to D.

Points may be desired to be snapped directly onto an existing line in order to define a boundary condition that extends only over part of a line.

3.3.3 Removal of polygon vertices

Unwanted polygon vertices may be removed. Give the command Shape – Delete points or the icon , and click on the polygon vertex to be deleted. The point will be deleted and the new geometry shown.

For example, figure 3-3 shows the removal of a rectangle corner to leave a triangular shape.
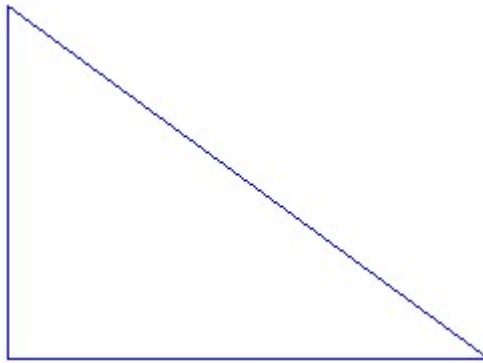
Figure 3-3. Example use of Shape – Delete points

3.3.4 Moving polygon vertices

Any vertex of a general (i.e. non-circular) polygon may be moved to a new location. Simply give the command Shape – Move points or the icon [icon], and then click-and-drag the existing point to a new location.

For example, in figure 3-4, clicking at 'A' and dragging to 'B' has moved the vertex concerned.
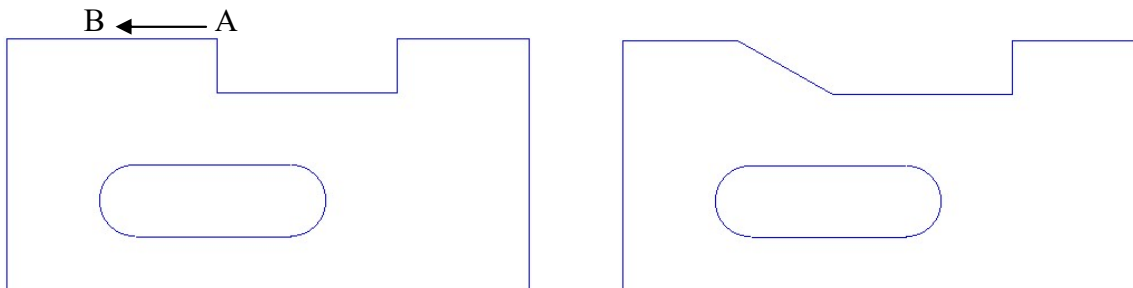
Figure 3-4. Shape – move points example for shape deformation

3.3.5 Moving shapes

Any shape may be moved to a new location. Give the command Shape – Move shape or the icon [icon], place the cursor near an edge on the shape to be moved, and then click-and-drag the shape to a new location. In the example shown in figure 3-5, a circular shape is moved to a new location by clicking at 'A' and dragging to 'B'.
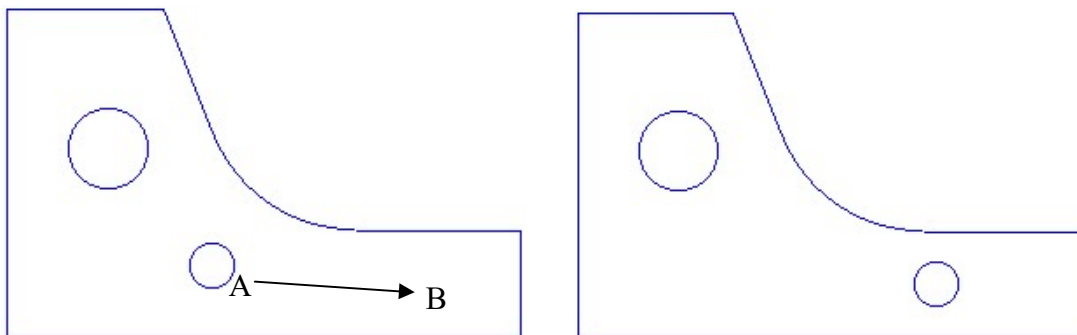
Figure 3-5. Shape – Move shape example, moving a hole

In order to ensure that the (x, y) coordinates displayed in the top left hand corner of the screen relate directly to a well-defined location on the shape, the cursor must be positioned close to one of the shape vertices on the 'click' operation. The shape immediately snaps to the cursor location, and the shape may then be positioned accurately. For circular shapes, the cursor must be close to one of the four points that define the shape location. Normally these will be top, left, bottom and right, but may be different if the circular shape has been rotated.

The display of shape points may be toggled on and off in the Edit – Preferences menu.

3.3.6 Fillets and other arcs

Edges of general (i.e. non-circular) polygons may be given a curvature, e.g. the radius in figure 3-5. This may be done in two ways:

- Any polygon vertex that is the intersection of two ***straight*** lines may be given a fillet radius. Give the command Shape – Fillet, or the fillet icon ▣, place the cursor on the vertex and click-and-drag diagonally to vary the fillet radius (which is displayed in a box in the corner of the screen). Dragging the cursor horizontally or vertically has no effect – it is diagonal dragging that affects the radius. It is irrelevant, however, whether the diagonal is towards the top-right, bottom-right, top-left or bottom-left. For example, in figure 3-6, click-and-drag from 'A' to 'B' to define a fillet.
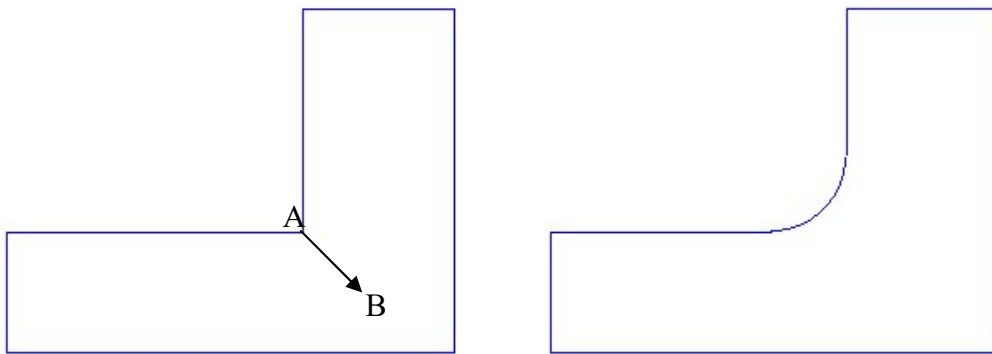
Figure 3-6. Creation of a fillet

- Any straight line may be converted into a circular arc. Give the command Shape – Line to arc (there is no toolbar icon for this command) and click the line in question. A dialog box opens to allow the radius to be defined. Finally, since there are two possible arcs of this radius that pass through the end points of the selected line, the arc definition becomes complete by clicking the mouse button anywhere to the side of the line on which the centre of curvature lies. For example, figure 3-7 shows a model in which a line has been selected, and is shown highlighted. The radius of the required arc is defined by the user in the dialog box, and then (after pressing OK) the point 'A' is clicked to indicate that the centre of curvature lies on that side of the line.
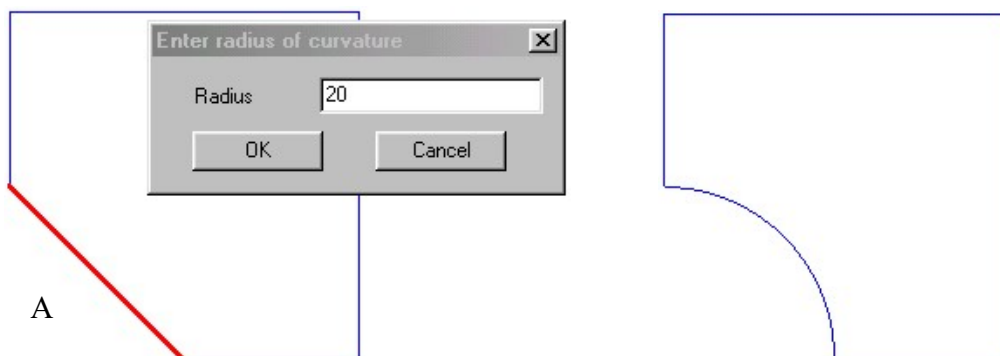
Figure 3-7. Illustration of Shape – Line to arc command

There are differences between the behaviour of these two types of arc. In particular, wherever possible, Concept Analyst will aim to preserve the tangency conditions at the ends of a fillet arc. This means that if one of these tangent points is moved (using Shape – Move points, as described in section 3.3.4) the entire fillet and adjacent tangent line are both also moved to preserve both tangencies and the fillet radius.

Arcs may be 'flipped' using the command Shape – Flip Arc. This has the effect of replacing an existing arc with the other possible arc that has the same end points and the same radius of curvature, as in the example given in figure 3-8.
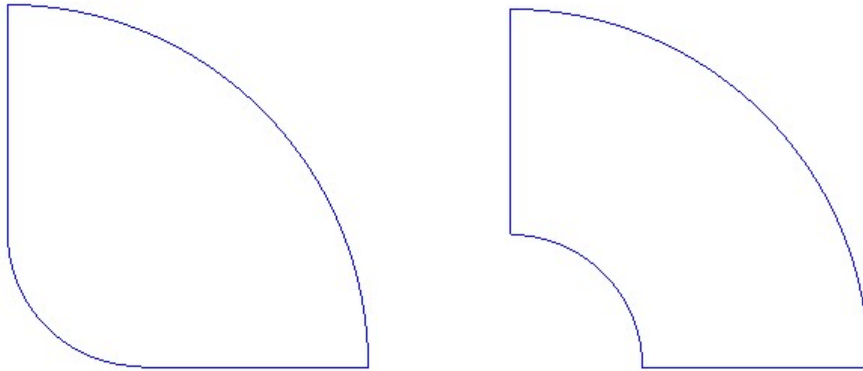
Figure 3-8. Arc before and after using the command Shape – Flip Arc

Clearly, any tangency conditions that might have existed between the arc and its neighbouring lines will be lost in the process of the flip arc operation.

Any arc, however it was created, may be converted back to a straight line by giving the command Shape – Arc to line and selecting the arc in question.

3.3.7 Re-sizing fillets and circles

Once defined, the radius of a circle or fillet may be modified. These operations require a different set of commands to those used for the initial creation of the circle or fillet.

To resize a circle:

> Give the command Shape – Resize circle, place the cursor on the circle and click-and-drag to resize.

To resize a fillet:

> Give the command Shape – Resize fillet, place the cursor on the fillet and click-and-drag to resize.

3.3.8 Splines

Curves that cannot be defined in terms of straight lines and circular arcs may be using B-splines. These are convenient descriptors of generally curved lines. They are defined in terms of a sequence of three or more 'control points'. The two control points at the beginning and end of the sequence are located at the two ends of the spline. The remaining control points guide the path taken by the spline, but the spline will not pass directly through these points (figure 3-9). Instead, the control points act rather like magnets, pulling the spline towards them.
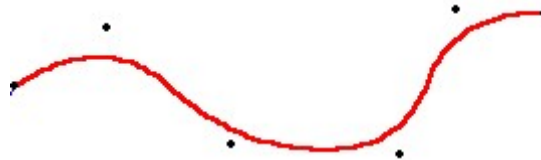
Figure 3-9. B-spline and its control points

Any straight edge of a general (i.e. non-circular) shape may be converted into a B-spline using the following simple procedure.

1. Give the command Shape – Line to spline
2. Click on the straight line to be converted to a spline (it is highlighted)
3. Click the screen location of the control points in order from one end of the line to the other
4. When defining the last control point (not the one at the end of the line, since it is already there, but the last point that is not at the end of the line) double-click

Splines may be modified using the command Shape – Modify spline. After this command is given, select a spline (which becomes highlighted) and click-and-drag one of the control points that have also become displayed. Notice that the ends of the spline cannot be moved in this way. Engineers new to B-splines may find it useful to experiment with the Shape – Modify spline command to gain insight into the way B-spline curves are defined in terms of the control points.

Any spline may be converted back to a straight line using the command Shape – Spline to line and then selecting the spline in question with the cursor.

3.3.9 Rotation of shapes

Any shape may be rotated in the plane of the model by giving the command Shape – Rotate shape. When the shape to be rotated is selected using the cursor, the shape becomes highlighted and a rotate bar is displayed. Click-and-drag the circle at the right hand end of the rotate bar, and the shape will be rotated. Release the mouse when the desired rotation has been achieved.
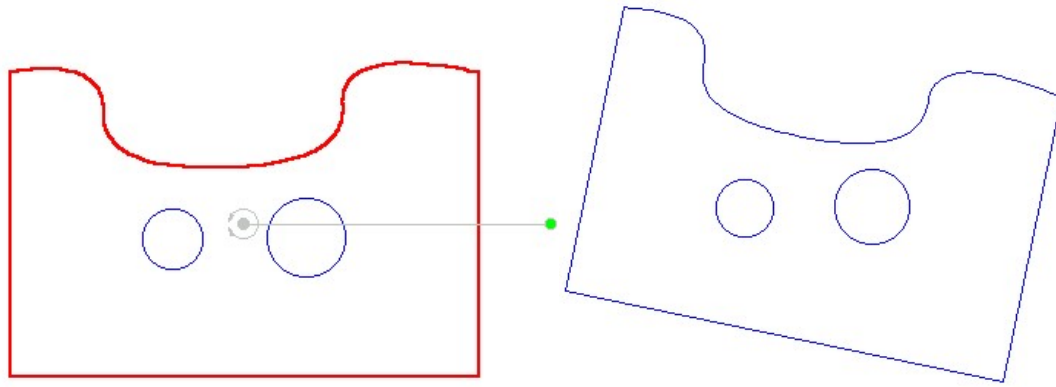
Figure 3-10. The rotate bar in a shape rotation example

The angle of rotation is continuously displayed during the click-and-drag rotation. This angle can be forced to change in 1° increments by holding the Shift key during the click-and-drag operation.

By default, the centre of rotation is assumed to be the approximate centroid of the shape, and this is displayed using a grey circle at the left hand end of the rotate bar. If another centre of rotation is required, click-and-drag this grey circle to a new location.

This method of rotating shapes is applicable when a sketched geometry is sufficient. For greater precision, the double-right click operation may be used (see section 3.5).

3.3.10 Notches

A notch may be defined on any straight line boundary, as illustrated in Figure 3-11.

(a)
(b)
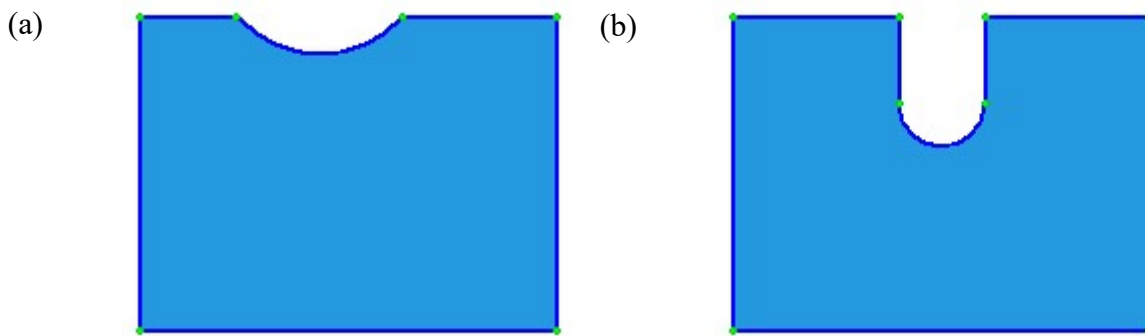
Figure 3-11. Notches; (a) depth < radius, (b) depth > radius

The notch is defined in terms of the following:

- Centre point on the existing boundary line
- Notch depth
- Notch radius

If the notch radius exceeds the notch depth, the notch will take the form of a circular arc defining material cut away from the component (fig. 3-11(a)). If the depth exceeds the radius, the notch

will take the form of a deep notch, having straight sides and a 180° arc at the end (fig. 3-11(b)). Notches cannot be defined outside the material, i.e. they are always deemed to be removing material from the component. However, the command 'Shape – Flip arc' may be used to convert the internal arc to an external arc if required. Users should note that the notch is defined as a circular arc, or two straight lines and a circular arc, and that afterwards these lines are treated as any other lines on a shape. In other words, the concept of a notch is used only for definition of this type of geometry, and the associated lines are not subsequently considered to be part of a special notch feature.

Notches may be defined graphically with a cursor click-and-drag operation, or by typed entry of data.

Cursor definition of notches:
Give the command 'Shape – Notch', and then place the cursor over the point on a straight line that will be the 'centre point' of the notch. Use a click-and-drag operation, moving the cursor inside the material. Cursor movement perpendicular to the boundary line changes the depth of the notch, and cursor movement parallel to the boundary line changes the radius of the notch. Both quantities are displayed numerically on the screen as the cursor is moved. Release the mouse button when the notch geometry is satisfactory.

Type data entry definition of notches:
Give the command 'Shape – Notch', and then place the cursor over the point on a straight line that will be the 'centre point' of the notch. Use a double-right-click of the mouse to display a dialog box in which the notch definition parameters may be entered. The centre point will take the default value being the cursor location when the double-right-click was issued.

3.3.11 Moving lines and line groups

A line may be moved to a new location, while retaining its length and orientation. Figure 3-12 illustrates a model before and after the vertical line on the right hand side has been moved.
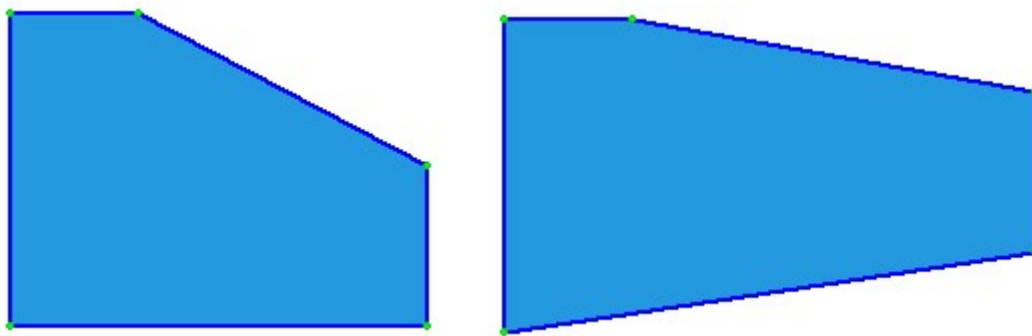

Figure 3-12. Line move command applied to a straight line

The line on the right hand side of the model has been dragged to a new location, and the straight lines on either side of the moved line distort to accommodate the movement. To move a line, give the command 'Shape – Move line' and click-and-drag the line to the required location. If the Shift

key is pressed during the dragging operation, the line is constrained to move only vertically or horizontally in the screen.

This operation may be applied to a straight line, circular arc or spline. However, it is only permissible to move lines for which both ends are connected to straight lines. A line that joins an arc at one or both ends may not be moved (this is because the arc definition would become unclear), Users wishing to move a line connected to an arc should first convert the arc to a straight line ('Shape – Arc to line'), then move the required line, and then convert the newly distorted adjacent line back to the required arc.

In order to move a group of lines, which may contain straight lines and circular arcs, give the command 'Shape – Move line' and then press the Shift key while using a click-and-drag operation to highlight a section of boundary. The Concept Analyst default is that groups of lines are defined in a counter-clockwise orientation around the outer boundary and clockwise around a hole. Release the mouse button and then the Shift key, and the group of lines remains highlighted. Now the highlighted line group may be moved using click-and-drag in exactly the same manner as a single line is moved. Both lines adjacent to the line group must be straight.

3.3.12 Cracks

Concept Analyst offers the capability to insert cracks in the model and to predict the stress intensity factors K_I and K_{II} at all crack tips. Cracks may be edge cracks or internal cracks. They may be defined using the command "Crack" in the Shape menu or clicking the crack icon

There are three different ways of creating a crack:

- Give the Shape – Crack command and use a click-and-drag operation to sketch a crack. If either end is sufficiently close to the edge of the component the crack will be deemed to be an edge crack, i.e. users do not have to use very delicate mouse control to click exactly over the problem boundary. If both the start and end of the crack are snapped to the boundary an error message is given. If the crack is an edge crack, the crack tip may be at either end of the line, i.e. it does not matter which direction the line is defined in.

- Give the Shape – Crack command and then double-right-click the mouse to type the coordinates of the start and end of the crack line. The dialog box that allows users to type coordinates will take its initial values of start location from the location of the mouse. If the crack is an edge crack, the crack tip may be at either end of the line, i.e. it does not matter which direction the line is defined in.

- Display a contour plot of results and give the Shape – Crack command or click the crack icon. A crack will automatically be created at the location of peak maximum principal stress. The user is prompted for the required length of the crack, which is assumed to propagate perpendicular to the local boundary surface. The analysis of the cracked geometry is carried out automatically.

There are certain restrictions on using models with cracks. These are:

- Only single zone models are permitted

- Symmetry/rollers boundary conditions are not permitted anywhere in the model. All displacement boundary conditions must be specified in terms of displacements in the *x* and *y* directions.
- In the automatic meshing process, element subdivision is not activated. (For non-crack models, the final pass in the automeshing process is the consideration of elements that face each other across a narrow gap, e.g. where a hole approaches the edge of a plate, and automatic subdivision of elements identified as being too large in relation to the gap). Users should be careful in interpreting results in these regions.
- Contour plots are not active.

It is anticipated that some or all of these restrictions will be addressed in later versions of the software.

3.4 The moveable origin

Concept Analyst uses a conventional (*x*, *y*) Cartesian coordinate system to define the geometry of shapes and the subsequent stress analysis. Each shape vertex, etc., is defined in terms of its (*x*, *y*) coordinates. During many operations the coordinates of the current cursor location are displayed in the coordinates box in the top left hand corner of the screen. This is useful in determining the location of any new points created, points being moved to new positions, etc.

If the geometry creation is started by creating a rectangle, as is common, the origin is taken to be the lower left corner of the rectangle. It is possible to redefine the origin of the coordinate system. This may be done as required during the building of a model. It is a useful way of defining new geometric features, since it allows a simple method of offsetting geometric features from others.

As has been seen in the preceding discussion, the geometry is built predominantly by graphical sketching. This may easily be done without regard to the precise location of the origin. Of course, it is important to analyse a model of the correct dimensions but, as long as the object is defined to be the right shape and size, the location of the origin is irrelevant.
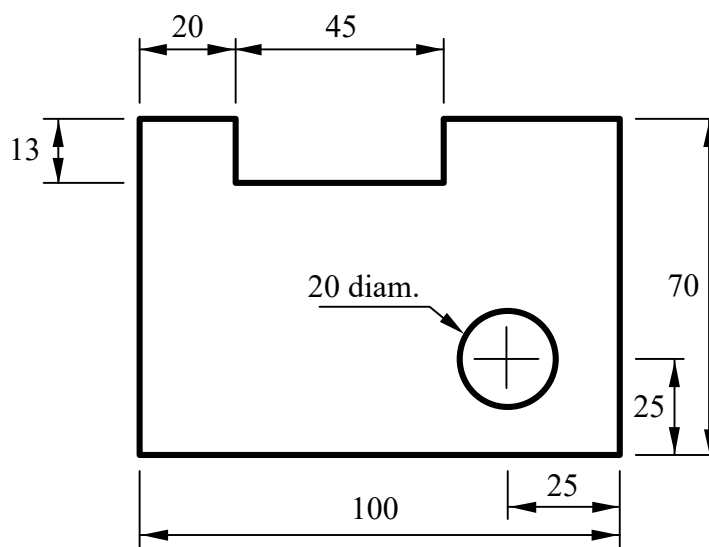

Figure 3-13. Plate to be constructed in Concept Analyst

This point is best illustrated by the use of an example. Consider the construction of the plate model shown in figure 3-13. As usual in working from an engineering drawing, there are a number of features dimensioned from a nearby datum surface.

The model may be conveniently built by following the steps:

1. Define a rectangle of dimensions 100 x 70 mm (either by careful positioning of the mouse in relation to the (x,y) coordinates displayed or by using the double right click option as described in section 3.5).

2. Give the command Shape – New Origin and click on the top left hand corner. This snaps the origin to that polygon vertex, and it may be noted that the cursor coordinates as displayed reflect this change.

3. Give the command Shape – Add points, or the toolbar icon , and define new shape points, as described in section 3.3.2, at positions (20,0), (20,-13), (65, -13) and (65,0) to define the rectangular slot in the top surface.

4. Give the command Shape – New origin and click on the bottom right hand corner of the shape.

5. Give the command Shape – Circle, or the Circle icon and define a new circular shape by positioning the mouse first at coordinates (-25,25) as the centre, and dragging a circle of radius 10mm.

The origin may be moved to any point in the screen area. If there is a shape vertex near the cursor location then that vertex will be used as the origin. If the cursor is clicked away from a shape, then the screen location picked will become the origin, though a warning message will be given.

3.5 The double right click operation

The above discussion about the definition and deformation of shapes is based largely around the use of sketched geometry. This enables an engineer working in conceptual design to evaluate different options very rapidly. However, it is important to recognise that the software can almost as easily be used for more detailed design work, in which the precise geometry of a component will be known, and it is required to obtain accurate stress information.

In these cases, Concept Analyst may be used to enter geometric data more precisely. This is done through the use of the double-right-click, i.e. clicking the right mouse button twice rapidly in succession, which allows keyboard entry of coordinate information, etc.

The effect of the double right click depends on the location of the cursor at the time the mouse button is clicked. This context-sensitive command works in the following way:

1. A variety of commands allow typed entry of data. If one of these commands is active then the user is prompted to enter the required data.

2. Otherwise, if the cursor is near a circle, the user is prompted to enter the coordinates of the centre of the circle and the radius of the circle.

3. Otherwise, if the cursor is near a (non-rotated) rectangle, the user is prompted to enter the horizontal and vertical dimensions of the rectangle. This is the case even if the cursor is near one of the rectangle corners).

4. Otherwise, if the cursor is near a fillet, the user is prompted to enter the new fillet radius. The fillet is updated, and new tangent points created at the intersection with the adjacent lines.

5. Otherwise, if the cursor is near a general polygon vertex, the user is prompted for the (x,y) coordinates of the vertex.

This algorithm allows simple and rapid building of precise geometries, in particular when combined with the flexibility offered by the moveable origin.

4

Materials

4.1 Linear elastostatic properties required

Concept Analyst performs its analysis using the assumption that the material stays in the linear elastic regime, i.e. that Hooke's Law applies.

In this case, there are few material properties that are needed to perform the analysis. Concept Analyst requires that the following properties are defined:

- Young's Modulus

- Poisson's ratio

- Yield stress

- Fatigue limit

- Mass density

- Fracture toughness

Of these, only the Young's modulus and Poisson's ratio are essential; without knowledge of these the analysis cannot run. The mass density is required if self-weight loading is to be included. The shear modulus, G, may be found from Young's modulus, E, and Poisson's ratio, ν, using the expression $G = E / 2(1 + \nu)$.

The yield stress is used only to display contour plots of failure criteria (section 8.2); it is emphasised that Concept Analyst performs linear elastic analysis only. The fatigue limit is used to display contour plots showing the percentage of fatigue limit that has been reached. The fracture toughness is used as a stopping criterion for crack growth analysis.

4.2 The CAMaterials file

Instead of requiring the user to enter these mechanical properties, Concept Analyst keeps a database of material types. The file 'CAMaterials' is available for users to keep such a database. This editable file contains a list of material names and their properties, and at run time the user may select any material by name, and the correct mechanical properties will be taken automatically from the file. Properties are converted automatically for any unit set is activated, so selecting by name is sufficient.

On a normal installation the file 'CAMaterials' will be saved in the same directory as the main 'concept.exe' executable file for the program. The file is readable and may be edited to modify existing properties, add new materials, etc.

The format of the file is as follows

Line 1:   A number indicating the units in which properties will be specified in the file
         0: MPa, kg/m^3, MPa-$m^{0.5}$ 1: psi, lb/in^3, ksi-$in^{0.5}$
Line 2:   A number indicating the number of materials listed in the file
Line 3:   The material name (including a dash '–' to replace any blank spaces)
Line 4:   Young's modulus (in N/mm^2 or MPa)
Line 5:   Poisson's ratio
Line 6:   Yield stress (in N/mm^2 or MPa)
Line 7:   Fatigue limit (in N/mm^2 or MPa)
Line 8:   Mass density (in kg/mm^3 or lb/in^3)
Line 9:   Fracture toughness (in MPa-$m^{0.5}$ or ksi-$in^{0.5}$)
Lines 10 to 16:   As lines 3 to 9, but for the second material
Lines 17 to 23:   As lines 3 to 9, but for the third material
etc.

4.3 Editing material properties

The mechanical properties of a material may be edited by selecting the command 'Material – Edit material'. The user is prompted for the material whose properties are to be edited, and a dialog box opens up to allow the editing of the properties themselves.

The new material properties are active only for the current modelling session. The properties in the CAMaterials file are unaltered. However, the edited material properties are retained if the model file is saved and later reopened.

4.4 Assigning material properties to shapes

Material properties may be assigned to shapes. All shapes have assigned material properties, even shapes that are used to model holes, though in this latter case the properties are not used in the stress analysis. The default material property set on entering Concept Analyst is material number 1 in the CAMaterials file, which (unless the CAMaterials file has been edited) is mild steel.

Any new shapes created using the circle or rectangle commands are assigned the current material property set, and the shape is colour-filled with the corresponding colour. The colour definition is as presented in Table 4-1.

Material id number	Material	Colour
1	Mild steel*	Grey
2	Aluminium*	Blue
3	Carbon steel*	Green
4	User-defined	Red
5	User-defined	Purple
6	User-defined	Orange
7	User-defined	Pink
8	User-defined	Olive
9	User-defined	Brown

Table 4-1. Material identification

*The table assumes the materials 1, 2 and 3 to be unaltered from the CAMaterials file as delivered with the software.

In order to define the current material property set, to be used for all subsequently created shapes, give the command 'Material – Select material'.

To change the material property set assigned to an existing shape, give the command 'Material – Select material', select the required material property set and then simply position the cursor inside the shape in question and click the left mouse button. Please note that this cannot be done while a shape is currently used to define a hole. Such a shape would have to be moved outside the containing shape before changing its material properties, and then moved back inside the containing shape. (Material property sets assigned to shapes modelling holes become important if a pin is defined in the hole (see section 5.3)).

5

Zones

5.1 Introduction and reason for zones

A zone is a substructure of a boundary element model. Most models consist of only one zone. Two types of multizone modelling are available. Firstly, different zones may be mated together along a single line interface, as shown in figure 5-1. Secondly, as shown in figure 5-2, a pin zone may be created in a circular hole.
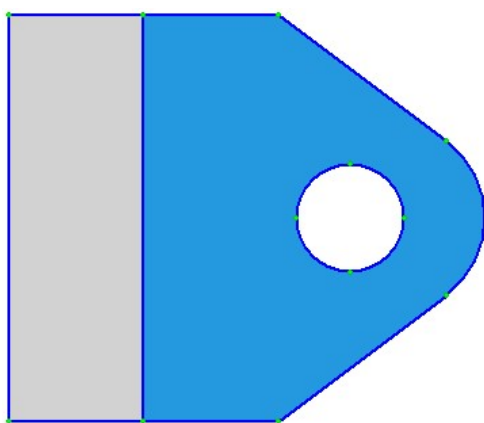
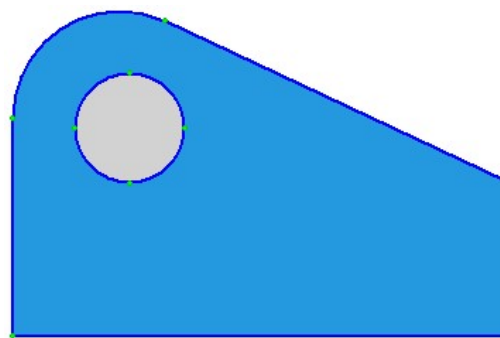Figure 5-1. Mated zones   Figure 5-2. Pin zone in hole

Recall the colour shading for materials given in Table 4-1. Grey is the colour used to depict the first material in the CAMaterials file (by default this is mild steel), and blue is the colour used to depict the second material in the CAMaterials file (by default this is aluminium).

The most obvious reasons for building models containing more than one zone are that there are different regions having different material properties and in order to apply a pin load.

5.2 Defining mated zones

Mated zones are created simply by 'bonding' together two shapes along a common line. Let us say we wish to mate shape A to shape B. Shape A contains a line *a* that is in the same orientation as, and has the same length as a line *b* on shape B, and these are the lines to be merged.

Mated zones may be created using the following procedure:

1. Define the individual shapes in separate locations on the screen, i.e. without attempting to draw them in their correct locations
2. Ensure that the lines to be mated, one on each shape, are in the same orientation and have the same length
3. Give the command 'Shape – Mate parts'
4. Click on the line *a*; it will become highlighted
5. Click on the line *b*

The shapes will be mated by moving shape B, and all shapes (holes) contained within shape B so that lines *a* and *b* become coincident.

5.3 Defining pin zones

A pin may be defined to fill a circular hole. This can be done by clicking the command Shape – Pin in hole. Give this command and click inside a circular hole. A prompt will be given allowing the user to enter a coefficient of friction. The pin will become shaded with the colour corresponding to the current material.

In this version:
- the pin must be accompanied by a pin load (command Loading – Pin load)
- there can be no other loads applied to the model (if other loads are present, one workaround is to not define a pin at all, and to applying a bearing load instead. This will be a good approximation.
- as for all multizone models, there can be no rollers boundary conditions

When started, the analysis of the model will solve the non-linear contact problem to find the correct contact region and contact stress distribution.

5.4 Limitations

There are a number of limitations associated with zones in Concept Analyst.

Maximum number of zones

The current version is limited to two zones.

Boundary conditions

For models containing two zones, all displacement boundary conditions must be defined in terms of x and y constraints. Symmetry/rollers conditions are not available.

Some or all of these limitations may be removed in later versions.

Loading and constraints

6.1 Introduction

In order for an analysis to be run successfully to provide stress and displacement information, a combination of loads and constraints needs to be applied.

The loads are all applied as distributed loads. This is perhaps unusual for users familiar with finite element analysis, in which the application of point loads is commonplace. Point loads give rise to problems in the boundary element analysis used by Concept Analyst; a good way of thinking of this problem is that a load acting at a point will generate infinite stress since a point has no surface area over which the load may be distributed.

6.2 Loads

Distributed loads may be applied in the following directions:

- x-direction (horizontal)
- y-direction (vertical)
- normal direction (perpendicular to the surface)

Distributed loads may be applied in terms of 'traction' or force. Traction is a surface stress (having units of stress); see section 6.2.1 for details. If distributed loads are defined in terms of force then at run time the force is converted into an equivalent traction, at time of definition, by dividing by the length of the appropriate line (Concept Analyst uses a default thickness of 1, which cannot be altered).

Distributed loads are applied by giving the Loading – pressure command, or icon ![icon]. Next click the line (shape edge) on which the load is to be applied. A dialog box appears (figure 6-1) in which the direction and value of the distributed load may be specified. On selecting 'OK' the load is displayed graphically using red arrows for those loads specified as tractions, or blue arrows for those loads specified as forces. Notice that, if a line on which a load has been applied is somehow changed by a subsequent geometric change, e.g. by moving one of the end points of the line to change its length and/or orientation, then:

- if the load was originally defined in terms of traction (and is displayed using red arrows) then the traction will be maintained on the geometric change, causing the total force transmitted over the line to change.

- if the load was originally defined in terms of force (and is displayed using blue arrows) then the force will be maintained on the geometric change, causing the traction applied over the line to change.
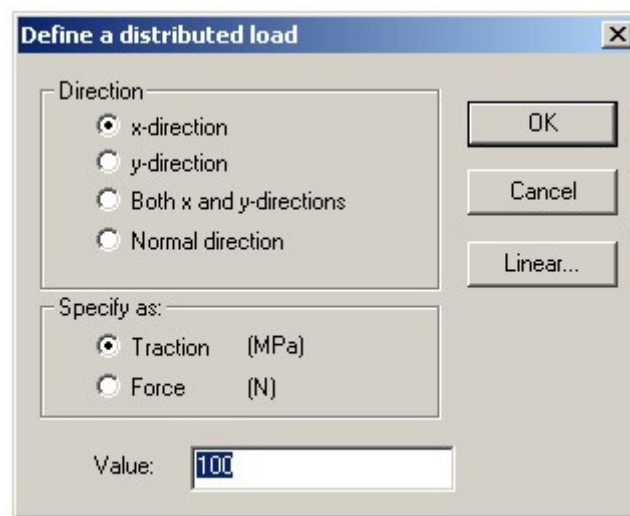
Figure 6-1. Dialog box for specifying distributed pressure loads

The default value of 100 may be overwritten by any chosen value. The current units are shown. Positive and negative values of load have the following meaning:

- For x-direction loads, positive values indicate loads in the positive x-direction (left to right) and negative values indicate loads acting in the negative x-direction.

- For y-direction loads, positive values indicate loads in the positive y-direction (upwards) and negative values indicate loads acting in the negative y-direction (downwards).

- For normal direction loads, positive values indicate pressure loads and negative values indicate tension acting perpendicular to the surface.

The 'Both x and y-directions' option allows identical conditions to be applied in both directions in a single command. Different loading values may be applied in x and y-directions to any line segment, but this cannot be defined in a single operation. Instead, the line would have to be selected twice – on one occasion the x-direction distributed load can be defined, and on the other the y-direction load.

The direction that is selected by default on entering this dialog box is as follows:

- x-direction (as shown in figure 6-1) if the line selected is vertical

- y-direction if the line selected is horizontal

- normal direction if the line selected is inclined or curved

Normal direction loads may not be mixed with x and/or y-direction loads on the same line.

The distributed load will have units of **stress** – either MPa (equivalent to N/mm^2), Pa or psi – as defined by the current unit set (see section 3.2) and will be applied to the entire highlighted line segment.

If a load is required to be applied over part of a line segment, it will be necessary first to create a new point (or points) on the shape so that the loaded part of boundary becomes defined by a whole shape line segment. In this way, if a short line segment is defined, a load approximating a point load may be considered.

6.2.1 Traction

Distributed loads are applied by Concept Analyst in the form of 'tractions'. Traction has the units of stress, but it is not always the same as a stress. In fact, the definition of traction in the x and y-directions is as follows:

$$t_x = \sigma_x n_x + \tau_{xy} n_y$$
$$t_y = \sigma_y n_y + \tau_{xy} n_x$$

where t_x and t_y are the traction components, σ_x and σ_y are the direct stress components, τ_{xy} is the shear stress and (n_x, n_y) is the unit vector pointing in the normal direction away from the material.

Traction relates to the stress components in the following ways (the first two of which are simple and apply in almost all practical cases):

- Tractions of a certain value, applied perpendicular to the surface, are simply uniformly distributed loads of the same value in the perpendicular direction.

- Tractions of a certain value applied tangentially to the surface are simply uniform shear stresses of the same value.

- On inclined lines the formulae above should be used to determine the relationship between traction value and the corresponding stress components. However, care should be taken not to confuse, say, the traction t_x with the stress σ_x on inclined lines. In practice, *if there is no shear stress on a boundary line*, the traction in the *x*- or *y*-direction will be numerically equal to the force in that direction divided by the length of the boundary line.

6.2.2 Non-uniform loads

If a non-uniform traction is required, e.g. to define a classical bending load or combined bending and axial load on a section, then select the button marked 'Linear…' in the dialog box specifying loads (figure 6-1). This causes the opening of a new dialog box allowing the nature of the non-uniform load to be defined, as shown in figure 6-2.
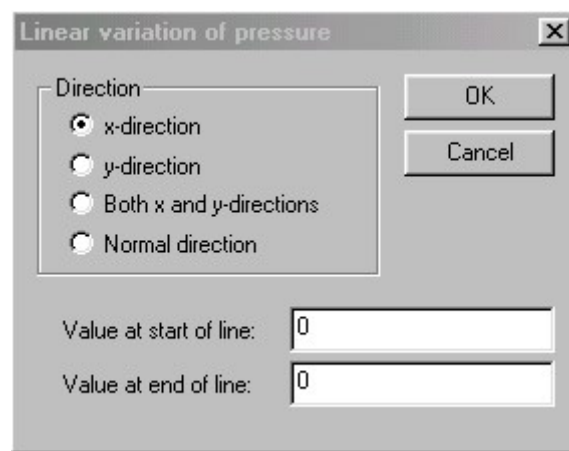

Figure 6-2. Dialog box for specifying non-uniformly distributed loads

This acts identically to the standard loading dialog box (figure 6-1) but allows values to be specified for pressure at the start and end of the line. Locations part way along the line are assigned pressure values by interpolating linearly between the end values.

This raises the question of which end of a line is the start and which is the end. This has (deliberately) not been addressed in this documentation. In fact, the outer (first defined) shape is defined in a counter-clockwise direction while shapes modelling holes are defined in a clockwise direction. However, in spite of the fact that there is a sign convention for line directions, it is not expected that users should have to remember this. As a result, when the 'OK' button in this dialog box is selected, the linearly varying load is displayed and a new menu appears (figure 6-3).

Figure 6-3. Menu for confirming non-uniformly distributed loads

If the loads have been displayed correctly, i.e. the user has 'guessed' correctly which is the start and which is the end of the line, the 'Accept' button confirms the load. If the start and end values are to be swapped, the 'Reverse' button may be selected. The 'Abort' button cancels any load defined by this command.

6.2.3 Pin loads

If a pin has been defined in a circular hole, a pin load needs to be applied. This is done through the command Loading – Pin load (there is no toolbar icon). Users can then input the *x* and *y* components of the pin load, which is given as a force. A subsequent analysis will solve the non-linear problem of finding the correct contact region and contact stress.

One limitation in the current version is that the pin load cannot be combined with loads on other lines in the model. For users wishing to create such a combination, a workaround would be to not create a pin in the hole, but instead to define a bearing load on the hole, as described in the next subsection.

6.2.4 Bearing loads on circular holes

The final method of defining loads considers circular holes used to model bearings through which shafts or bolts apply in-plane loads to the object under analysis. In this type of loading, it is conventional to apply the load as a sinusoidally distributed pressure in the normal direction around 180° of arc. This provides a reasonable approximation to the correct contact pressure distribution around a well-fitting bearing without specifying a full contact problem. This may provide a workaround for users wishing to combine pin loads with other loads on other lines.

To define a bearing load, give the command Loading – Bearing load, or the icon , and then click the circular hole shape to which the load is to be applied. A dialog box allows the definition of the load (figure 6-4).
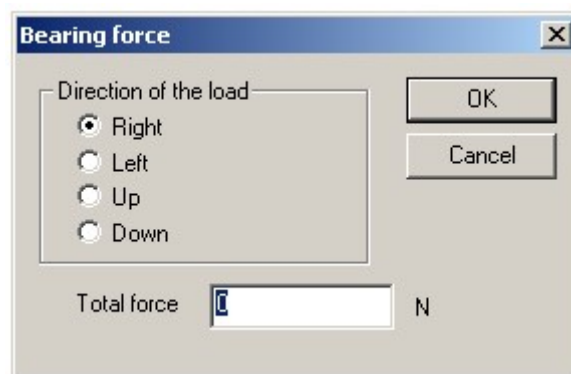

Figure 6-4. Dialog box for defining bearing loads

The load is now specified as a force (not a stress/pressure) that may be applied in any of the four directions up, down, left or right. The program automatically calculates the stress distribution, based on Concept Analyst's assumption that the model has unit thickness. As for distributed loads specified in terms of force, the program stores only the force applied to the hole. This means that

if the diameter is subsequently changed, the traction distribution will be changed to maintain the specified value of force.

Bearing loads may be applied in only one direction on any one hole.

If a bearing load is required to act in a direction other than the four available, it is possible that construction of an appropriate model may still be possible by making use of shape rotation (section 3.3.9). The definition of the directions selected in the dialog box, i.e. "right", "left", "up" and "down", may not be intuitive for shapes that have been rotated through large angles, especially if they have been rotated through 90º.

6.2.5 Checking the load value

The value of distributed load is not normally displayed on the model, so that after a load has been defined all that remains on the screen is the graphical depiction (by red or blue arrows) that a load of some value has been defined on a line or lines. The value can be easily displayed by giving the command Loading – Check. This causes the value of any applied load to be displayed over the line concerned. The current unit set for force and traction (i.e. pressure) is displayed in the top left hand corner of the screen and, following the colour convention for loads, those loads specified in terms of traction are shown in red lettering and those specified in terms of force are shown in blue lettering. Figure 6-5 shows an example of this display.
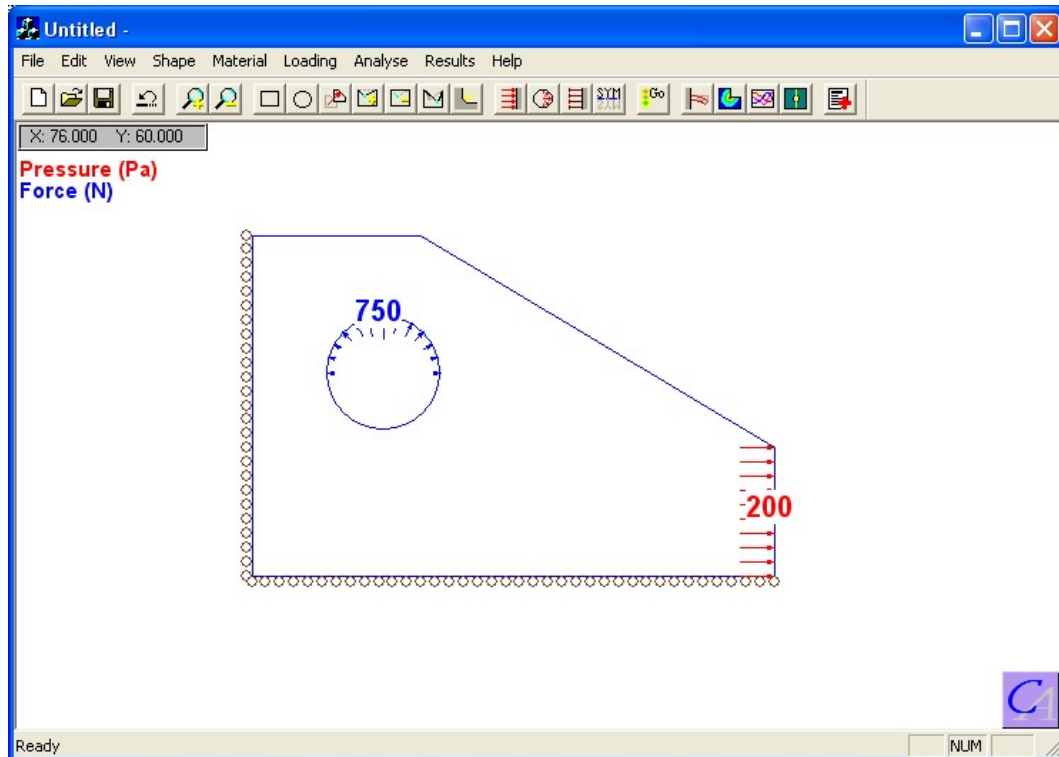

Figure 6-5. Loading – Check display

Remove the loading check display by giving another plot command, e.g. single right click to refresh the geometry view, a results plot of any type, or alternatively give the Loading – Check command again to toggle the feature on/off

6.2.6 Self-weight

Self-weight may be considered as part of the loading of a model, and this facility may be used to define any body loads resulting from uniform acceleration of the object under analysis. The command Loading – Self weight allows the user to switch self-weight on/off. If the facility is turned on, then a dialog box appears in which the user may prescribe the *x* and *y* components of the acceleration vector. Default acceleration is 1g in the negative *y*-direction (downwards).

6.3 Constraints

All Concept Analyst models need to be sufficiently constrained. In general, this means that a set of constraints needs to be defined that together prevent rigid body motion in any direction.

Constraints are displacement boundary conditions, and may be applied in two ways: the general displacement constraint and the symmetry/rollers condition.

6.3.1 General constraints

Any edge of any shape may be constrained such that its displacement takes a given value. Most commonly, this value will be zero.

To apply a constraint, give the Loading – Constraint command, or click the icon [icon], then click on the line to which the constraint is to apply. The line becomes highlighted, and a dialog box (figure 6-6) appears in which the direction and value of the constraint may be specified.
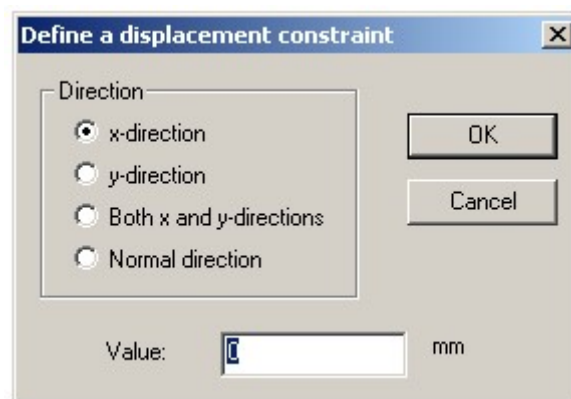

Figure 6-6. Constraint dialog box

The value defaults to zero to save typing since this is overwhelmingly the most common value for a displacement constraint. However, it is possible to enter a different value to prescribe that the displacement in some direction will be, for example, 2mm.

The displacement constraint is applied uniformly to the whole line. It is not currently possible to define uniformly varying displacement constraints along a line in the same way as is possible for loading (section 6.2.2).

If the direction is set to 'Both x and y-directions' the same displacement constraint is defined for both these directions. This is commonly used to fix a line completely.

If the direction is set to 'Normal direction' the command has the same effect as the use of the symmetry/rollers condition (section 6.3.2).

A general displacement constraint is displayed graphically on a line using brown 'T' symbols. The direction of the main axis of the 'T' indicates the direction of the constraint.

6.3.2 Symmetry/rollers conditions

Any straight edge of a shape may be given a symmetry/rollers condition. This prevents displacement in a direction perpendicular to the line. As the name of the condition suggests, this can be used to model two situations, i.e. a plane of symmetry or a boundary condition normally displayed graphically using roller symbols.

To define this condition, give the Loading – Symmetry Line command, or icon , and simply click on the line(s) concerned. No dialog box is required since the command assumes zero displacement in the normal direction.

Concept Analyst uses roller symbols to display these conditions graphically.

It should be noted that for horizontal and vertical lines, it may be quicker to define symmetry/rollers conditions than the corresponding general constraint. This is because a single mouse click per line is sufficient, whereas a second mouse click (to close the dialog box) is required when specifying general constraints.

Symmetry/rollers conditions may not be applied to any edge on which loading is also defined.

Symmetry/rollers conditions may be generally mixed with x and y-direction constraints in the same model, though not on the same line. However, where a symmetry/rollers condition is applied to a line adjacent to a line carrying a general constraint in the x and/or y-direction, the general constraint must be zero valued.

Symmetry/rollers conditions are currently available only in single zone models.

6.3.3 Incompletely constrained models

If a model is not fully constrained, then in general the analysis will be unable to cope with the infinite number of possible solutions. Such a case will often be identified automatically and, depending on the degree of constraint omitted, the analysis aborted.

Examples of incompletely constrained models include those shown in figure 6-7.
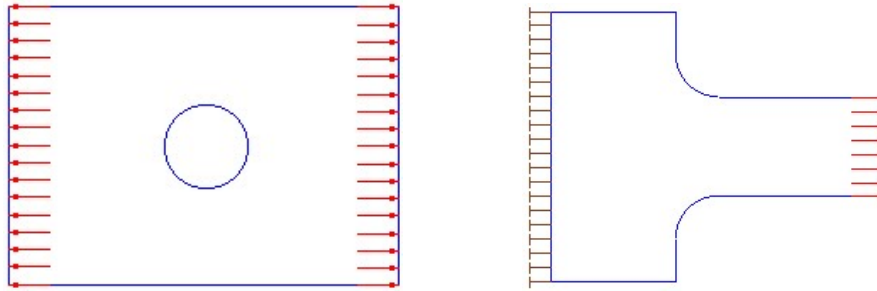

Figure 6-7. Example models having incomplete constraints

Both models are intuitively acceptable in an engineering sense since equilibrium is maintained, and engineers are familiar with drawing free body diagrams such as this on a daily basis. Let us explore, then, the reason for the problem. It can be seen that the model on the left has no constraints at all. The loads are exactly balanced to the left and right, so the object is in equilibrium. The problem is that there are still an infinite number of possible solutions to this problem, each involving rigid body motion of the object. While an engineer will know that the problem remains the same if the plate is picked up, moved ten metres to the left and the same loads applied, a computational FEM or BEM simulation will see this as a problem which is incompletely defined. (Mathematically, the governing system matrix A (see the Concept Analyst Theoretical Guide) becomes singular, i.e. has no inverse, and the analysis execution stops with what is reported to the user as a 'Zero pivot error').

The model on the right of figure 6-7 has a constraint in the *x*-direction on the left hand edge of the shape but is unconstrained in the *y*-direction. Therefore there exist an infinite number of solutions to this problem, each being located somewhere up the *y*-axis. However, there are no loads applied in the *y*-direction.

Problems of this type are handled effectively by Concept Analyst by its automatic inclusion of a soft spring. If an object is incompletely constrained, but no loading is defined in the unconstrained direction, the program automatically attaches a weak spring to one of the elements (see the Theoretical Guide for a discussion of the boundary elements), normally the element in the lower left corner of the model. This is normally sufficient to allow the analysis to proceed and to enable an accurate set of results to be obtained. However, if the loading in such a case is very small, the very low value of the spring stiffness starts to become significant compared to the general loading, then in some (comparatively rare) cases the results can be seen to be influenced by the presence of this spring. This will be characterised by unexpected stress results in the lower left part of the model.

Soft spring constraints are only ever applied in a single direction, so models such as that on the left of figure 6-7 will not be permitted to run, and a message will be displayed asking the user to provide suitable constraints.

As a final comment on the soft spring constraint, it has been found that the stress concentration results, in comparison with handbook solutions, are slightly more accurate when soft spring constraints are used than when some artificial constraint is defined explicitly as described in sections 6.3.1 and 6.3.2. This is not unexpected since the classical analysis on which the handbook solutions are based do not have an artificial constraint.

6.3.4 Constraining to avoid stress concentration

In general, unless the boundary conditions of the particular problem demand it, it is good practice to avoid fixing lines (shape edges) in both the *x* and *y*-directions. This is because of the Poisson effect. When loaded in uniaxial tension in the *x*-direction, it is well known that the positive strain in that direction will give rise to a negative strain in the *y*-direction as defined by the Poisson's ratio of the material. However, where the model is fully fixed by a constraint in both *x* and *y*, the material will be unable to contract in this way. This is normally manifested by some quite sharp stress concentrations at the ends of the constrained line.

As an example, figure 6-8 shows a model of a simple rectangular plate under uniaxial tension, having its left hand edge fully constrained in both x and y-directions. The contour plot of von Mises stress shows the stress concentrations that result.
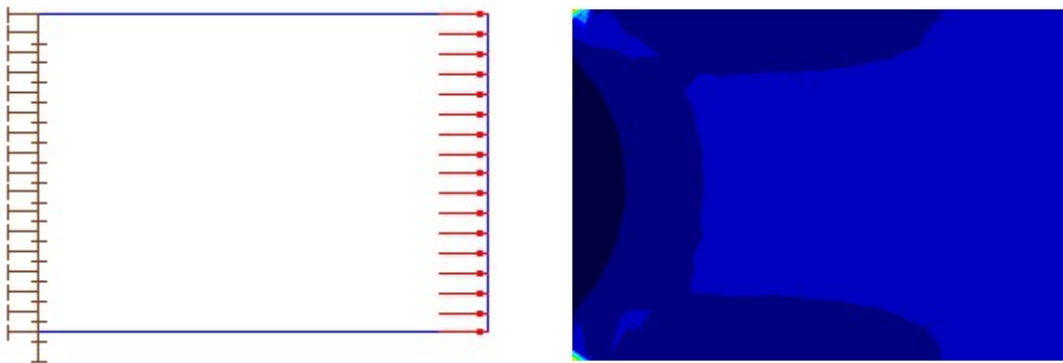

Figure 6-8. Stress concentrations at the ends of a fully constrained line

The stress concentrations are so sharp that the contours shown in yellow and red, marking the highest valued von Mises stresses, are in this case so localised at the top left and bottom left corners that they become hard to see visually.

It would be better practice, if a uniform stress distribution is required for this model, to remove the vertical constraint from the left edge and then proceed according to any of the following:

1. Constrain a horizontal edge against vertical motion (easy but may not produce the required deformed shape)

2. Create a new point on the shape, creating a short line segment, and constrain vertically over that short line segment.

3. Omit the vertical constraint entirely and allow Concept Analyst's soft spring constraint to be automatically applied to prevent rigid body motion in the vertical direction.

Of these, option 3 is the simplest and for most cases is very effective.

6.4 Application of pure shear

It is often desirable to determine the stress concentrations that arise around holes and other features as the result of application of a pure shear load case. This may be applied in two ways in Concept Analyst:

6.4.1 Pure shear by application of tangential tractions

Define a rectangular plate aligned with the Cartesian (x,y) axis system.

Apply tractions using the icon in the usual way, applying distributed loads to all four edges of the plate. Care should be taken to ensure that the loads defined are complementary so that the plate is in equilibrium of both forces and moments. This is most easily ensured by starting with a square plate and applying equal tractions in the tangential directions to all four edges in the complementary shear stress sense depicted in Figure 6-9.

No displacement constraints or rollers need be specified. Indeed, specifying constraints would interfere with the pure shear nature of the load case. Instead, if the model is in force equilibrium, soft springs will be applied automatically in **both** the x- and y-directions.

This will allow the analysis to proceed to a solution. However, the model is very weakly constrained, and it has been found that the solution is stable only for the first analysis and errors can occur in the re-analysis. For this reason, if such a model is changed while results contours are displayed a full analysis will be performed instead of a re-analysis, leading to a slight degradation in performance. For this reason the second method of applying pure shear, described in section 6.4.2, is recommended.
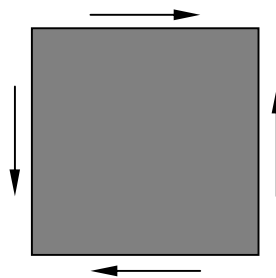

Figure 6-9. Complementary shear loads

6.4 2 Pure shear by application of normal tractions

A piece of material which is subjected to a biaxial stress field defined by

$$\sigma_x = -\sigma_y$$
$$\tau_{xy} = 0$$

will be in a state of pure shear. This is effectively the same stress state as the condition described in section 6.4.1, but rotated through 45°. This is the recommended method of applying pure shear. The simplest method of defining this is to define a rectangular plate and apply tensile and compressive distributed loads of equal magnitude to perpendicular edges, as shown in figure 6-10.
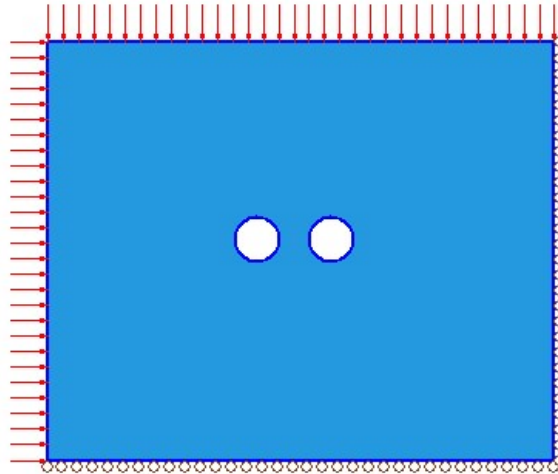

Figure 6-10. Simplest approach for pure shear

If it is required to define a state of pure shear in (x,y) such that

$$\sigma_x = \sigma_y = 0$$
$$\tau_{xy} = \tau_0$$

where τ_0 is some non-zero value, simply rotate the rectangular boundary shape through 45° as shown in figure 6-11.

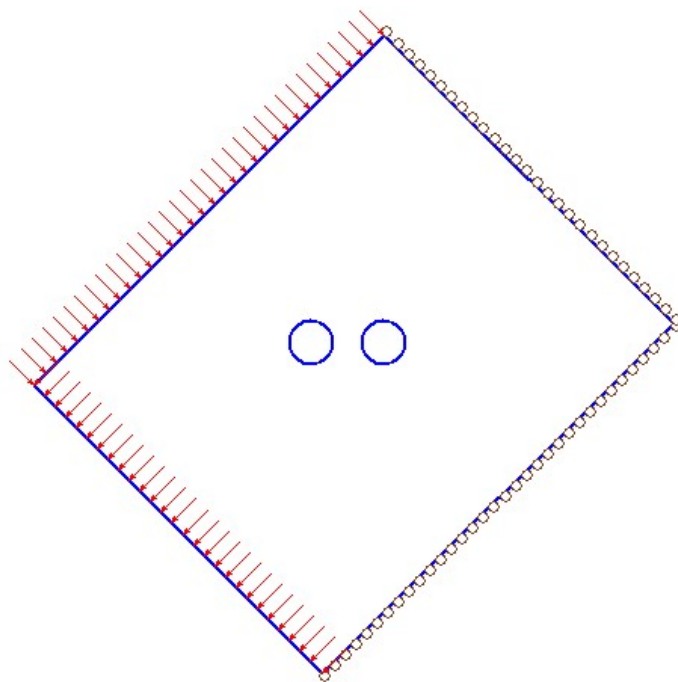Figure 6-11. Pure shear case with $\sigma_x = \sigma_y = 0$

7

General viewing features

7.1 The screen area

The Concept Analyst screen area, in which the model is defined, has the following approximate default sizes on starting the program.

$$-20 < x < 100$$
$$-20 < y < 60$$

This is the case whichever unit set has been selected, but will vary depending on the screen size and default window size being used.

Clearly, there are going to be cases in which this screen area is inappropriate for the model in question, which may be much larger or smaller than this size. In cases like this, the model size that can fit in the display area may be changed in a variety of ways:

Zooming out using the command View – Zoom out, or the zoom out icon ![icon] will automatically fill the screen with the entire model. This is useful for models that are physically much smaller than the default dimensions given above.

Double right clicking on a polygon vertex allows keyboard entry of a set of (x,y) coordinates for the point. If this should be outside the current maximum screen area, the screen area is rescaled so the model fits within the screen area.

Giving the command Edit – Width of unzoomed screen… causes the user to be prompted for the size of model that would fill the screen area in the horizontal direction. This may be changed to any desired value.

7.2 Zooming and panning

Zoomed views of localised parts of a model are useful for detailed geometric modelling and for detailed interpretation of results.

To zoom in, give the command View – Zoom in, or click the zoom in icon ![icon]. Next either:
- give a single mouse click and the view will be doubled in scale with the cursor location in the centre of the view

or:
- use a click-and-drag operation to define a rectangular window through which to view the model

Notice that the resulting zoomed-in view may not exactly match the dimensions of the rectangle as drawn on the screen. This is for two reasons:

1. The dimensions of the Concept Analyst window do not change, and this enforces a certain horizontal/vertical aspect ratio of the view area that is unlikely to be matched by a user-defined window.

2. It is important for the sketching and re-analysis features that each pixel represents a 'round' number in the coordinates being used. Concept Analyst makes extensive use of graphical positioning of geometric details, such as shape vertices, etc. Therefore, when a zoomed view is drawn, the view displayed is carefully selected to be the closest to that required that allows each screen location to represent a 'round' set of coordinates. As a counter example, if this were not the case, then after zooming the user would find it impossible to use the mouse to drag a point to the location (10,10), but instead would be seeing coordinates such as (9.87219, 10.04881) displayed.

Zoom out operations, using the View – Zoom out command or the icon ![icon], always return the view to the entire model. It is not currently possible to zoom out only partially. Undo commands do not affect the current view, but instead undo the most recent changes to the model.

In order to pan, i.e. to move the view, at the same scale, to an adjacent part of the model, give the command View – Pan and simply click-and-drag the model in any desired direction.

Notice that geometric views are displayed dynamically as the pan operation proceeds, but (for reasons of speed) contour plots of results are refreshed only when the mouse button is released.

## 7.3 Refreshing the geometric display

Following a plot of results, it may be that the user would like to return to the standard view of the model geometry in order to continue making design changes without re-analysis being initiated.

Clicking the right mouse button once in the screen area will cause the results plot to be erased and the standard geometry view to be refreshed.
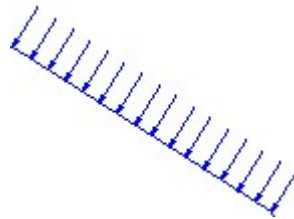
7.4 Display of loads, elements and points

The default view of the geometry of an object includes only the shapes and the loading. Shapes are displayed using blue lines tracing the perimeter of the object being modelled. Loads and constraints are displayed in the following ways:
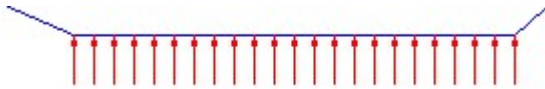
Pressures   *Forces*

Displacement constraints

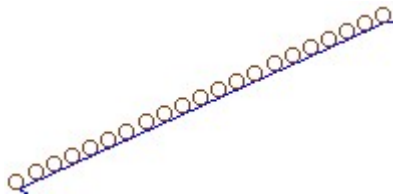   *x*-direction constraint   *y*-direction constraint

Symmetry/rollers constraint

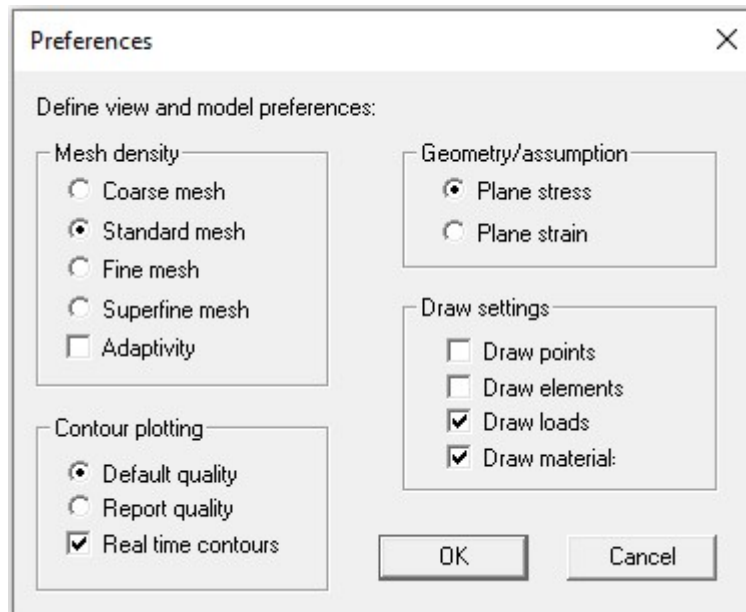These entities may be suppressed in the drawing by use of the Edit – Preferences… dialog box (figure 7-1).

Figure 7-1. The Edit – Preferences… dialog box

The 'Draw settings' menu in the bottom right corner of this box allows the user to select whether or not the display will include the geometry points (i.e. shape polygon vertices), the boundary elements and/or the loads. The 'Draw material' setting controls the colour-shading of regions to distinguish materials (section 4.4).

Notice that elements are created only on initiation of an analysis. Therefore, the 'Draw elements' check box will have no effect until the analysis has been performed, then the elements will be displayed. Each element is displayed using a small red circle to mark each end of the element. Although Concept Analyst uses quadratic elements which have a node in the middle of the element, this is never displayed.

7.5 Screen grid

A faint grid may be displayed on geometry plots. The command View – Grid… allows this grid to be turned on and off. The same command also allows a grid spacing to be specified by the user, and this is in the same units as the model.

The spacing may also be displayed on the screen to give an indication of the physical size of the model being analysed.

The grid is currently unavailable for the snapping of geometry.

8

The first analysis

8.1 The first analysis

An analysis is initiated by giving the command Analyse, or clicking the Go icon [Go]. This involves the following steps, all of which are followed automatically:

- A 'mesh' of boundary elements is defined on the geometry lines bounding each shape

- Internal points are defined in rings around fillets and holes, and also scattered throughout the material

- The boundary element stress analysis is performed over the elements and internal points

- The 'derived' stress components (i.e. the principal stresses and von Mises stress) are calculated over the elements and internal points.

For a full discussion on the procedures involved, consult the Concept Analyst Theoretical Guide.

Thus the first analysis, initiated in this way, is always a full boundary element analysis. No re-use of earlier analysis information is made here, and the entire analysis is performed from the start.

8.2 Mesh density

Concept Analyst includes some sophisticated automatic meshing, creating for most problems a set of elements that will provide reasonable accuracy in results in a reasonable time. However, as with all numerical simulation methods, the use of more detail in the model will generally provide for improved solutions.

Concept Analyst allows the level of detail to be varied in terms of the mesh density. Three settings are available in the program for mesh density. This allows for the mesh to be improved where extra accuracy is required, or to coarsen the mesh for more rapid solution where accuracy is less important than run time.

This setting is accessed through the Edit – Preferences… command, which generates a dialog box as shown in figure 8-1.
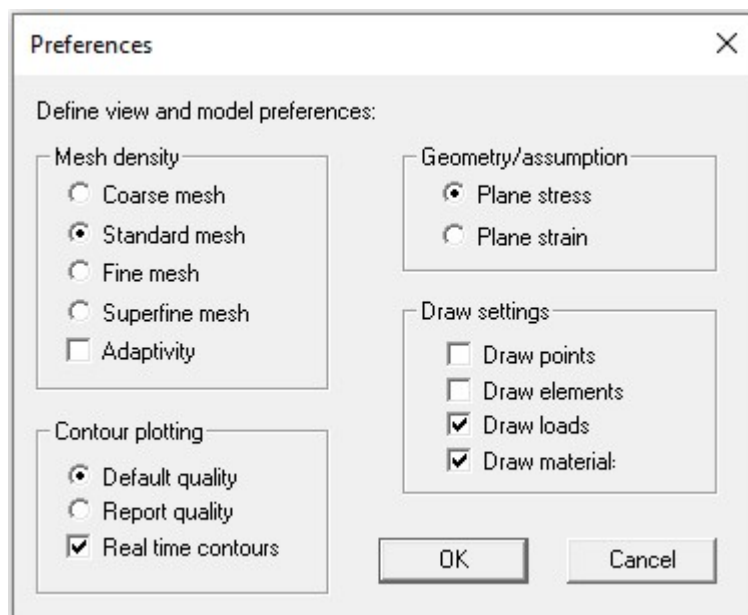

Figure 8-1. The Edit – Preferences… dialog box

The figure shows the default settings, but these can be reset using the user's personal settings file (section 1.4). The 'Adaptivity' option referred to in this section of the dialog box is explained in section 8.4.

8.3 Geometry/assumption

The Edit – Preferences… dialog box (figure 8-1) also allows for the user to set the assumption used for the analysis, the options being

- Plane stress (default)
- Plane strain

8.4 Adaptive mesh refinement

The Edit – Preferences… dialog box (figure 8-1) also allows for the user to set the adaptivity option. This causes a subsequent analysis run ('Go') to do an adaptive analysis.

Adaptivity refers to the program's ability to assess the likely level of error in a set of displacement and stress results. Following this error calculation, the mesh of elements is automatically improved in the required areas and the analysis restarted. This type of mesh improvement and restarted analysis continues until the program deems the results to be satisfactory. Details of the technical algorithm behind the error indicator and refinement strategy are given in the Concept Analyst Theoretical Guide.
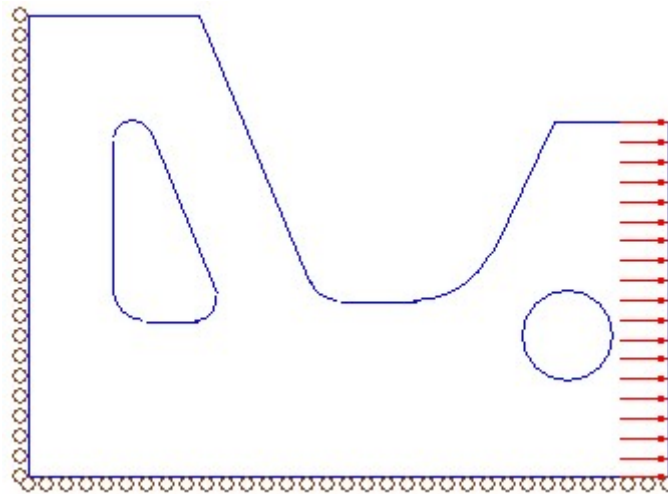
Figure 8-2. Example model

If the example in figure 8-2 is analysed using a conventional analysis the mesh would depend, naturally, on the mesh density selected by the user. Figures 8-3 to 8-5 show the three meshes that would result from coarse, standard and fine mesh settings. Note that there is also a 'superfine' mesh density also.
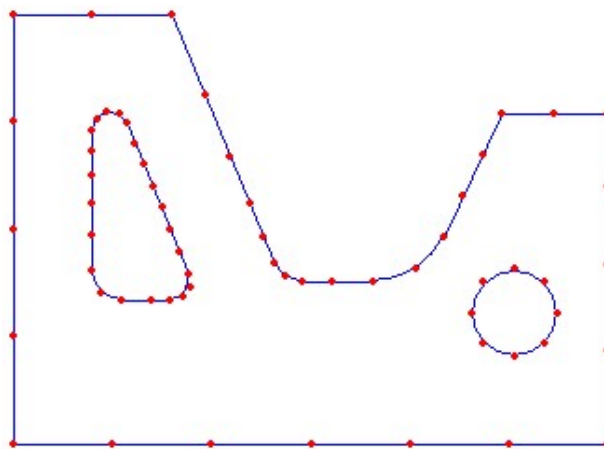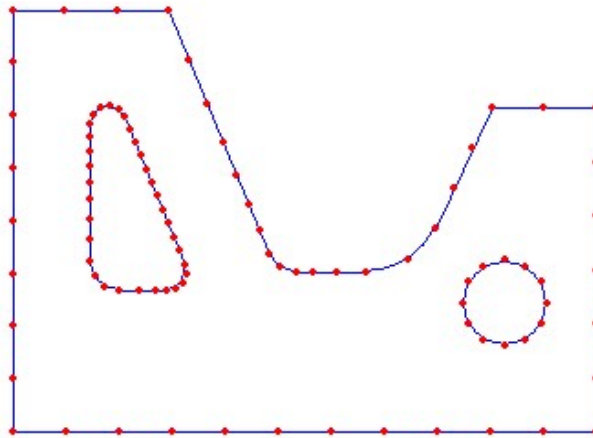
Figure 8-3 Coarse mesh
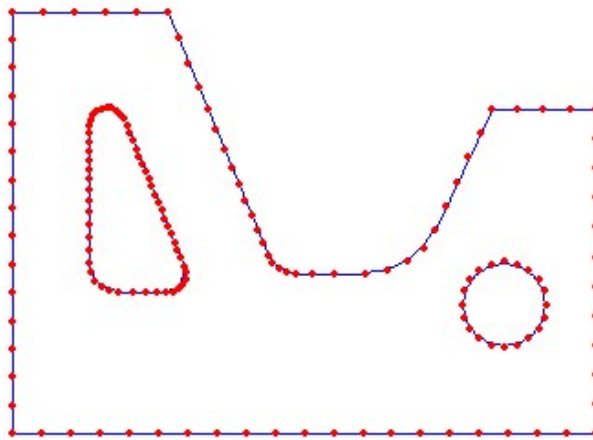
Figure 8-4 Standard mesh

Figure 8-5 Fine mesh

The adaptive analysis may use any one of these meshes as its starting point. It is recommended that the coarse mesh be selected.

The adaptive procedure then normally results in a mesh that is more efficient in its use of elements than any of the meshes shown in the examples in figures 8-3 to 8-5, since the elements are tailored to the particular problem being solved, and the particular stress distribution that occurs.

The final mesh for an adaptive analysis starting from a coarse mesh is shown in figure 8-6. It is clear that the mesh has become refined around certain areas of the mesh, notably around fillets and other stress risers.
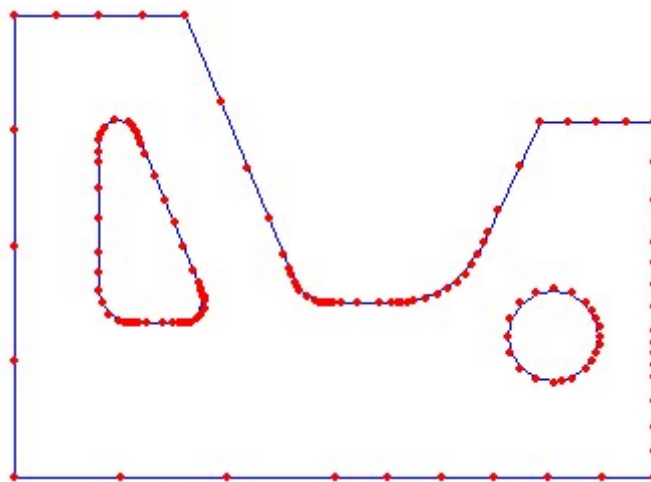
Figure 8-6. Final mesh from adaptive run starting from a coarse mesh

The automatic convergence to a set of results that has a satisfactory result from the error indicator is a useful method of establishing increasing confidence in the result.

The trade-off between accuracy and run-time varies in a problem dependent way, but in general the coarse-adaptive analysis will give rise to results close to the fine mesh but in a generally shorter time and with greater confidence to the inexperienced analyst. Some results for different mesh densities and with the adaptive options are given in the Verification Guide.

9

Results plotting and interpretation

9.1 Deformed geometry plots

An animated deformed geometry plot is initiated with the icon [icon] or the command Results – Deformation (figure 9-1). The deformation is shown greatly exaggerated so that the maximum displacement on the screen is always approximately 10mm.
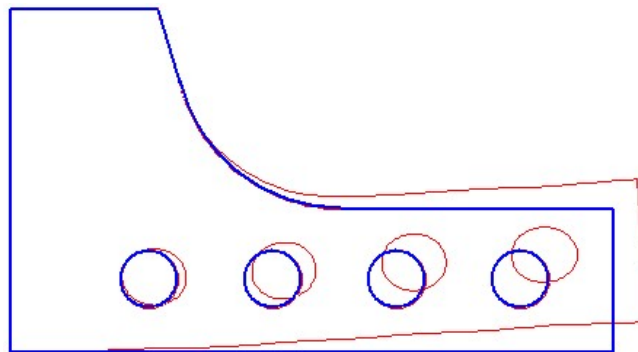
Figure 9-1. Deformed geometry plot after animation

9.2 Contour plots

The most common way of displaying the results of a stress analysis is the contour plot, in which different colours are used to represent areas of the model that have any desired stress or displacement component within a certain band.

On giving the command Results – Contour plot…, or the icon , the program displays a dialog box asking the user to select the results quantity to display.
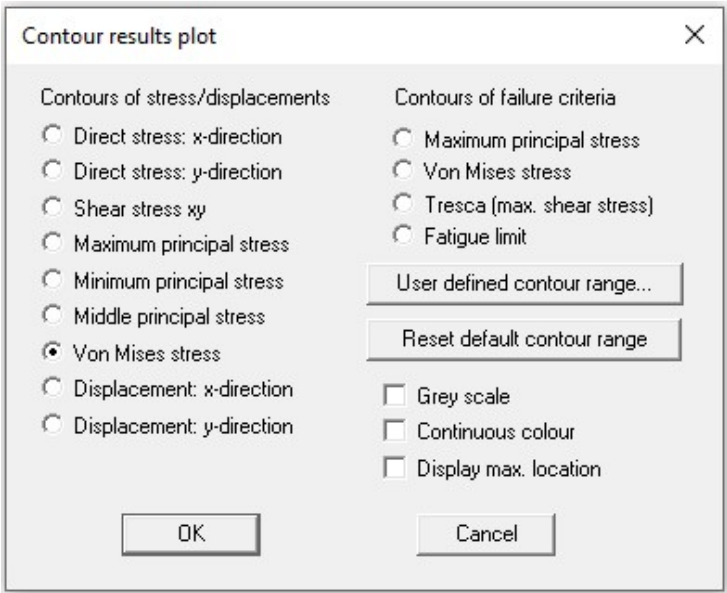

Figure 9-2. Contour plot dialog box

The columns on the left relate to specific stress and displacement components (the Concept Analyst Theoretical Guide provides definitions of these terms). Selecting one of these options will cause the appropriate contour plot to be displayed (figure 9-3).
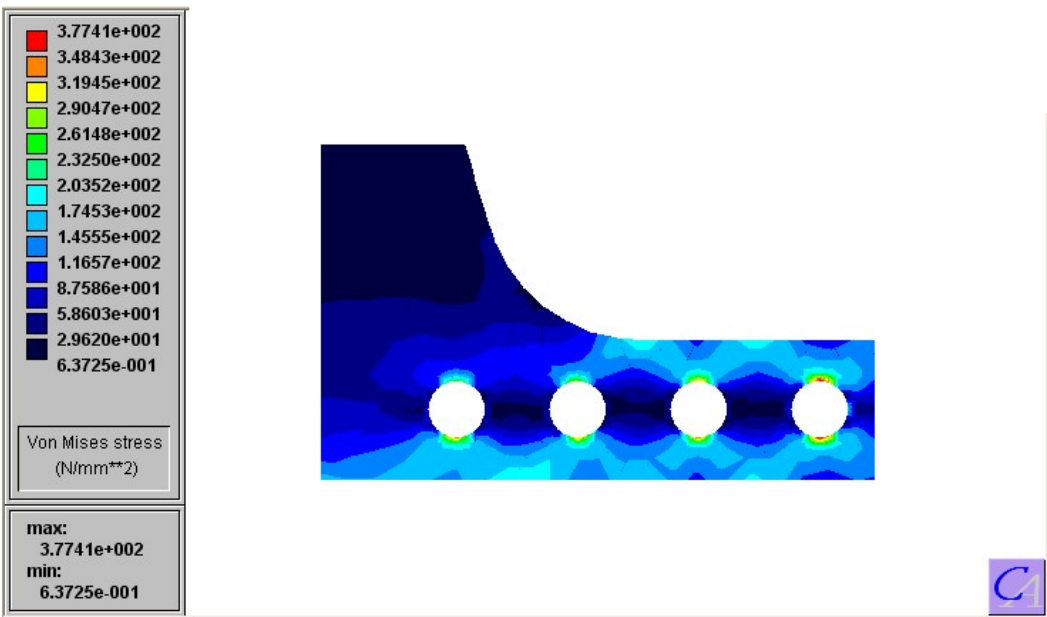

Figure 9-3. Typical contour plot of results

The contour plot is accompanied by a legend indicating the values of stress or displacement that are indicated by each colour. The plot type and maximum/minimum values are also displayed.

The dialog box (figure 9-2) also contains options for displaying 'Failure criteria' contour plots. The last of the options, fatigue limit, causes a contour plot to be displayed showing the percentage of the material's fatigue limit to have been reached. For materials that do not exhibit a fatigue limit, this option is not available. The first three options under 'Failure criteria', i.e. maximum principal stress, Tresca and von Mises, cause plots to be displayed showing how any one of these criteria may (or may not) be violated (figure 9-4).
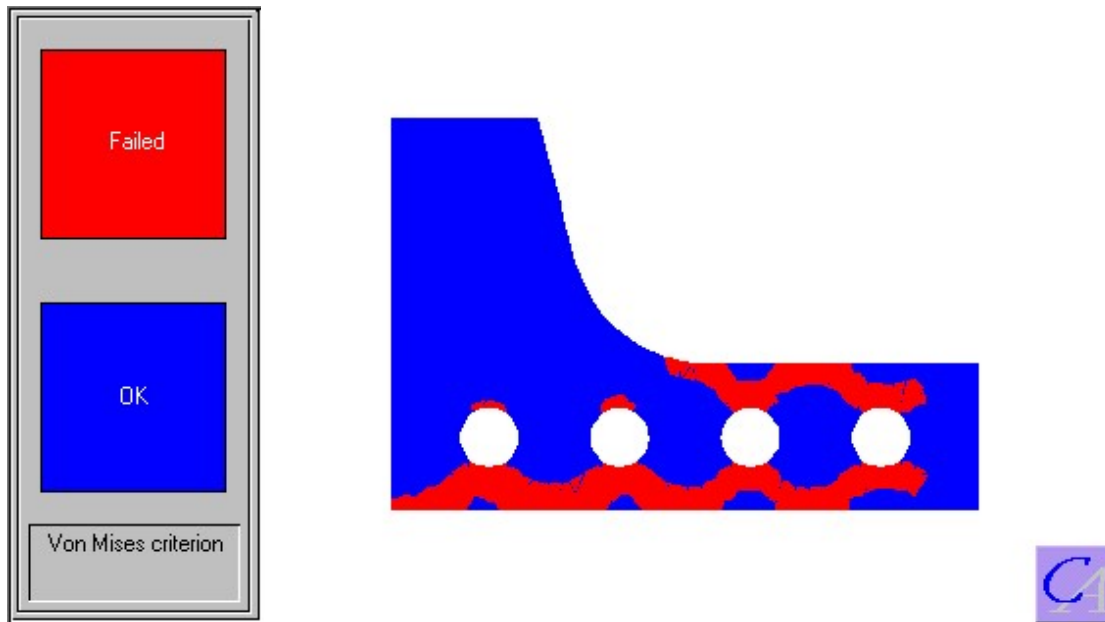

Figure 9-4. Failure criterion contour plot

It is important that we are aware of the significance of a plot like this because there is a danger of the information being misleading. In particular, it must be recognised that:

- The areas of the model displayed in red are those areas for which Concept Analyst calculates the stress conditions to cause failure according to the particular failure criterion.

- The analysis performed by Concept Analyst is linear-elastic only and does not consider material yielding, or any non-linearity in material behaviour.

- Redistribution of stress following localised yielding will change the stress distribution in other parts of the component. This effect is not considered in Concept Analyst, and instead all materials are allowed to withstand stress levels in excess of yield.

- In general, the above points suggest that the true yielded portion of material will be larger than that predicted by a plot of the type shown in figure 9-4.

Failure criterion contour plots such as this are therefore to be used only to assess the onset of yield, and it is not safe to predict post-yield behaviour using this approach.

9.2.1 User defined contour ranges

Figure 9-2 shows the dialog box that is displayed in order to allow the user to select the results quantity to display in contour fashion. Two buttons also appear on this box, allowing the selection of a user-defined range for the contours, i.e. the minimum and maximum levels to appear in the legend box, and allowing the return to the default setting. The default setting will always be the complete range of values for the model.

On selecting the button labelled "User defined contour range…" a new dialog box appears, as shown in figure 9-5.

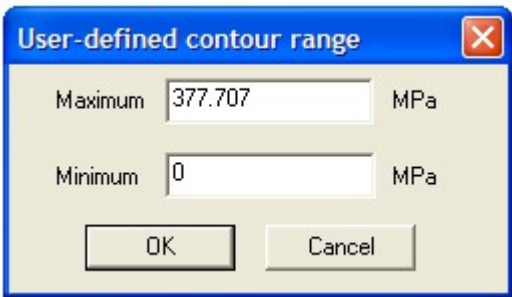Figure 9-5. User defined contour range dialog box

The current minimum and maximum levels are displayed, which will on first use be the default settings, and these values may be edited. Subsequent contour plots will include dark and light grey contour bands to represent those areas of the model outside the user-defined contour ranges (figure 9-6).
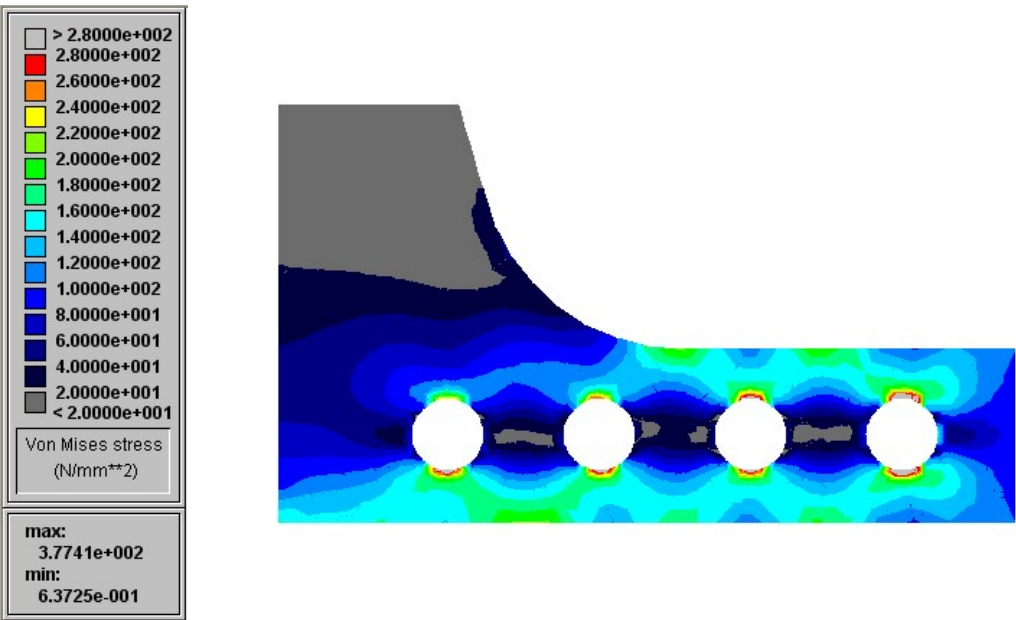

Figure 9-6 Contour plot with user-defined ranges

The contour levels are reset to the default settings (full range) on the following:

- A different results quantity is selected to that active when the user-defined ranges were set

- The button "Reset default contour range" is pressed

It is envisaged that the user-defined contour ranges will be particularly useful when performing multiple re-analysis runs, so that the stress distributions in the different geometries considered can be visually compared more easily.

9.2.2 Report quality contours

The contour plots are generated by Concept Analyst by making use of a set of 'internal points' that are defined automatically and invisibly by the program. (For a detailed discussion about the positioning of these points and the calculation of results there, consult the Concept Analyst Theoretical Guide). However, without going into unnecessary detail here, it is sufficient to note that internal points add to the analysis effort, and typically account for around one third of the run time.

Contours are interpolated between internal points over a triangulated grid between these points and the points on the boundary. It is clear, then, that the greater the number of internal points, the greater the clarity of definition of the contour results plots.
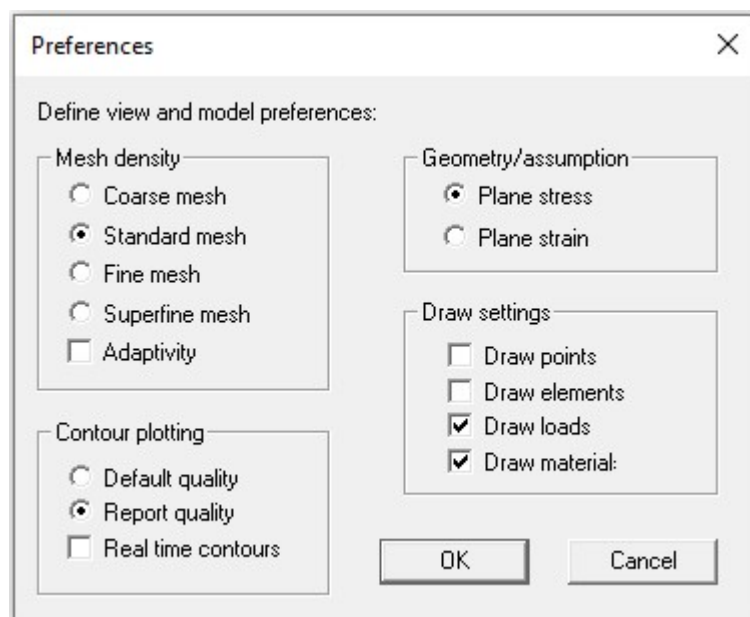
Figure 9-7 Report quality contours may be set in Edit – Preferences … dialog box

For most cases, the automatically defined internal points provide a reasonable balance between run time and contour plot clarity. However, the contours can appear rather angular in areas in which few internal points have been defined. For this reason, a feature has been added to allow 'report quality' contour plots. When this feature is set, Concept Analyst defines a much larger

number of internal points so a smoother contour plot is produced at the expense of longer run time.

In order to produce a report quality contour plot, enter the Edit – Preferences… command dialog box (figure 9-7) and select the 'Report quality' button at the bottom left. On leaving the dialog box with 'OK':

- If contours are already being displayed, the program will automatically perform a re-analysis and display the refined contour plot.

- If contours are not being displayed, the program will wait until the next analysis is performed, and will define a large number of internal points at that time.

The report quality contours will then be active for subsequent contour plots of different quantities, e.g. von Mises stress, y-displacement, etc. However, the feature will be turned off automatically for the next full analysis or re-analysis. There is no facility to have report quality contours active permanently; they must be activated as required. This is because the standard quality contours provide reasonable definition in a shorter time, and report quality contours are intended for use only when the final plot is ready to be displayed.

As an illustration, figure 9-8 compares standard and report quality contours.
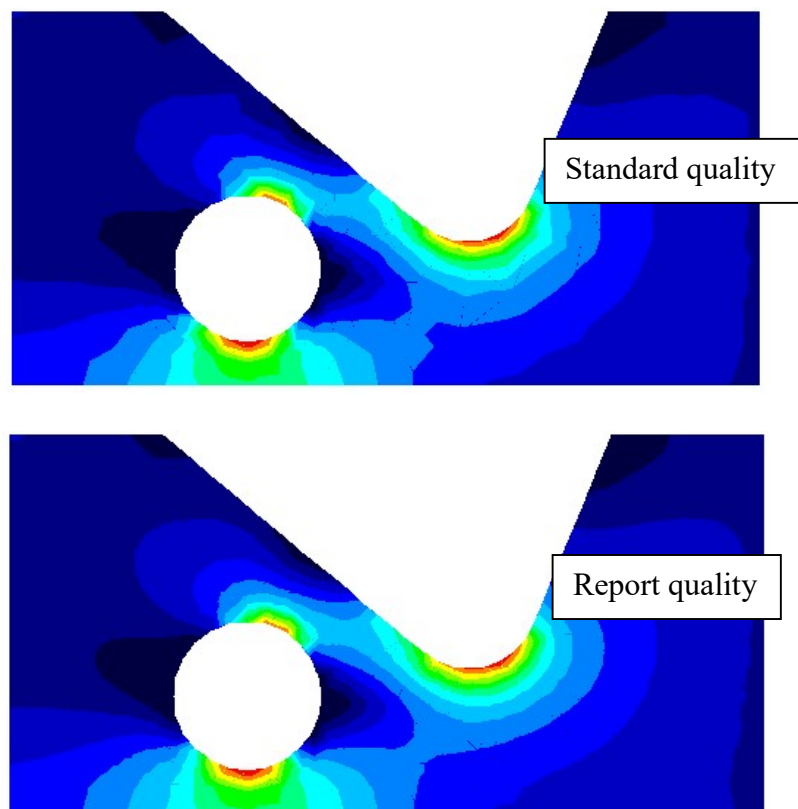

Figure 9-8. Comparison of standard and report quality contours

9.2.2 Continuous colour contours

The contours of results may be displayed using continuous colour instead of the normal colour bands having sharply defined edges. Here, 256 colours are used to display the results, giving rise to a contour plot without sharply defined edges, but in which the smooth distribution of results is more evident (figure 9-9).
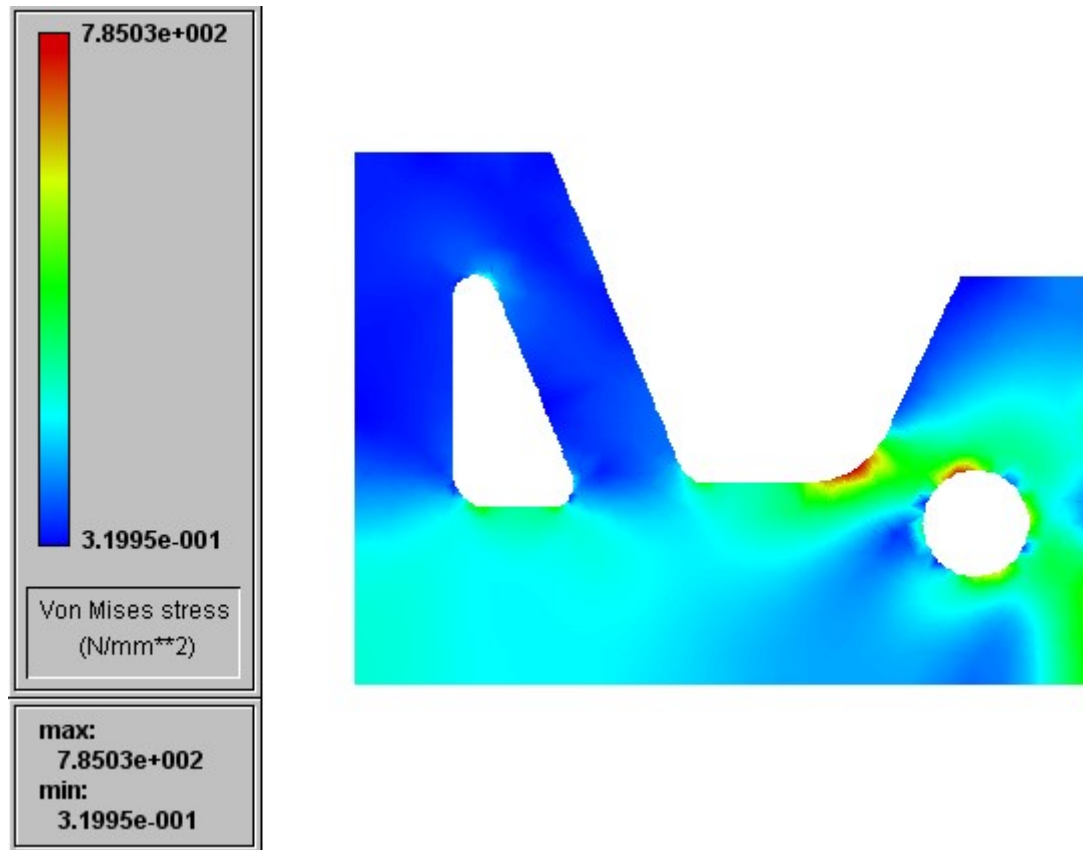

Figure 9-9 Example of continuous colour contour plot

Notice that the legend for continuous colour contours lists only the maximum and minimum values since it is not expected that users will be able to map accurately between the legend and the contour display.

Continuous colour contours are displayed following selection of the check box in the lower right hand corner of the contour dialog box (figure 9-2).

9.2.3 Greyscale contours

When printing *on a black and white printer* hard copies of documents containing contour plots of results, a conventional colour palette (blue for low values through the spectrum to red for high values) can generate some confusion. This arises because the darker blues and reds both appear as a dark grey shade, whereas cyan, yellow and lighter greens appear as light grey shades.

Concept Analyst includes a supplementary contour palette comprised entirely of grey shades. This is activated following selection of the check box in the lower right hand corner of the contour

dialog box (figure 9-2). Subsequent contour plots are drawn such that low values are displayed in a dark grey, changing uniformly to light grey for the higher valued contours (figure 9-10).
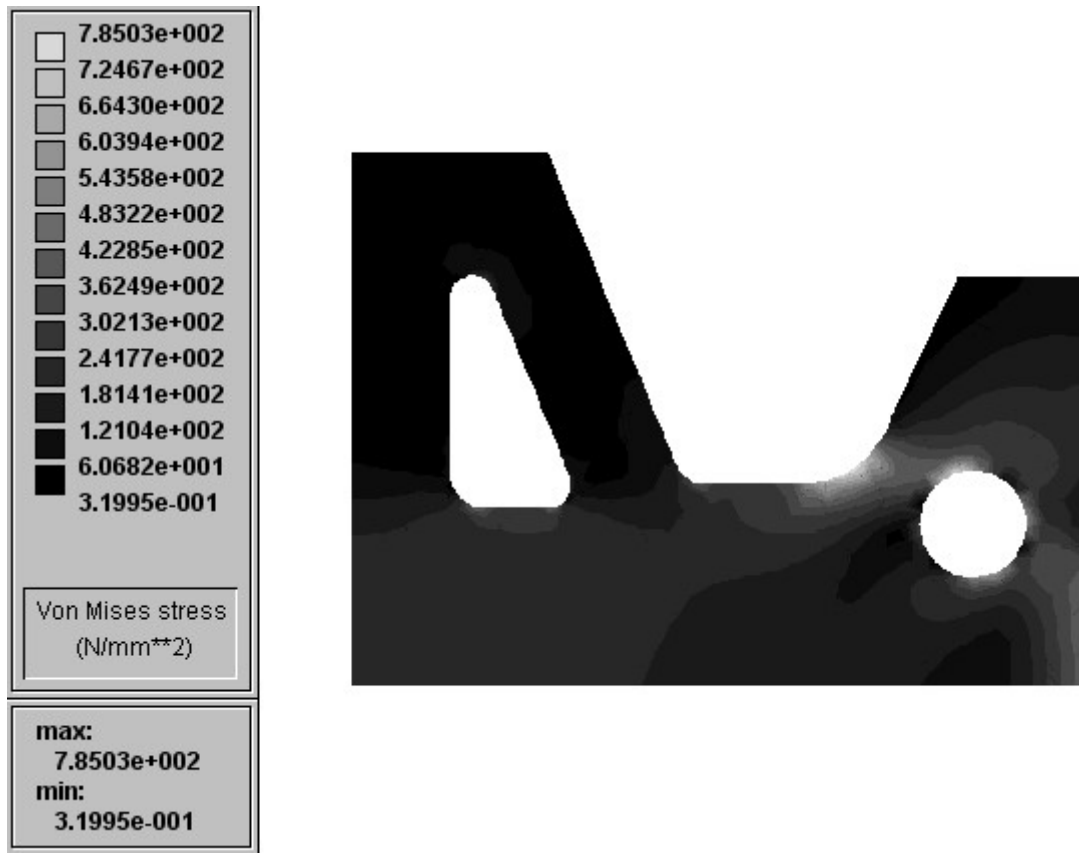

Figure 9-10. Greyscale contours avoid confusion when printed in black and white

9.3 Internal line plots

When a contour plot is displayed on the screen, it is possible to define any straight line which is to form the horizontal axis in an *xy* graph plot of results. This allows the distribution of stress or displacement to be seen over any arbitrary line. The line may be completely contained within the material, may extend to the boundary of the component, and may cross the boundary of the component. An internal line may leave and re-enter the component, for example when cutting through a hole.

The internal line plot is initiated by the command Results – Internal line, or icon . Following this, the internal line may be specified in one of two ways:

- Double-right-click anywhere in the screen area to type (*x,y*) coordinates of the start and end points into a dialog box

- Use a click-and-drag operation (left mouse button) to draw the line dynamically across the contour plot

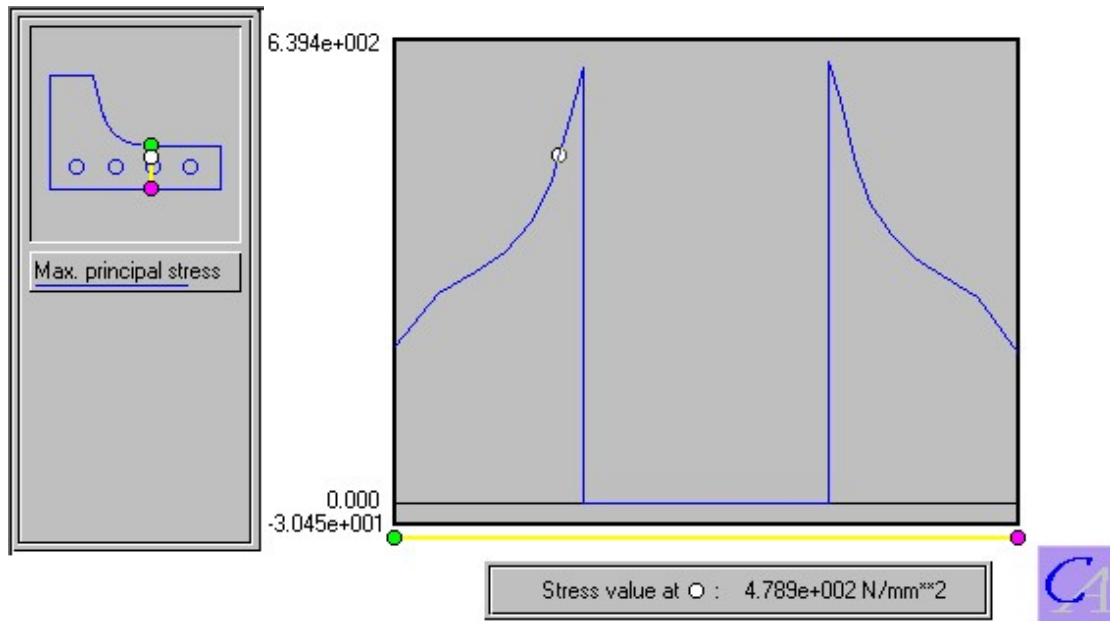The internal line plot is drawn (figure 9-9).

Figure 9-9. Internal line plot of results

The main screen area now shows a graph plot (in this example it is showing the variation in maximum principal stress across a section that crosses a hole). In the top left hand corner of the screen is shown a small display of the geometry of the component, on which is superimposed the line that represents the horizontal axis of the graph. The green and purple circles show the start and end of the line relating to the definition shown immediately below the graph.

The vertical axis of the graph does not display numbers in the usual way, but instead shows only the stress = zero axis (if applicable) and the values at the top and bottom of the axis. The top and bottom values are in excess of the maximum and minimum values being displayed, and this ensures that the graph stays within the screen area.

The value of the result (in this case, maximum principal stress) is shown for any location simply by moving the cursor around in the graph area. The horizontal location of the cursor relates to a location on the model, and a white circle displays this location in the small picture of the geometry in the top corner of the screen. Another white circle follows the graph line, and the caption at the bottom of the screen displays the value of stress or displacement at this location.

Users should be aware of two points:

- For internal lines ending at, or crossing, boundaries it is difficult to sketch a line (using the graphical method of definition) so that it reaches the peak stress location. If it is important that an internal line show the peak stress, then the double-right-click method should be used to define the line giving an accurate set of coordinates for, for example, the edge of a hole.

- For numerical reasons discussed in the Concept Analyst Theoretical Guide, internal point results can become inaccurate as the internal point becomes very close to the boundary.

Users should be aware that the best accuracy for internal lines is found when the line is more than a short distance away from the boundary.

9.4 Boundary graph plots

A graph plot similar to internal line plots may be displayed for any contiguous section of boundary. In this case, unlike the internal line plots (which are limited to the result quantity being displayed already in contour form), graph plots may show two or more result components at the same time.

The graph plot is initiated using the Results – XYplot… command, or icon [image]. Next, click-and-drag to define a section of boundary to form the horizontal axis of the graph. This may be done by pressing (and holding) the left mouse button with the cursor over a line, and then dragging the cursor to another line. The line segments in between will all become highlighted. If the mouse button is both pressed and released over the same line, this single line will be highlighted and will itself form the entire horizontal axis.

It should be remembered that circular shapes in Concept Analyst are defined as four 90° arc segments, so a graph plot of, for example, maximum principal stress around the hole would require all four arcs to be highlighted in the click-and-drag operation.

In the click-and-drag operation to highlight the line segments forming the horizontal axis, it is assumed that the direction will be following the normal sign convention:

- Counter-clockwise on the exterior boundary

- Clockwise on interior boundaries (holes)

Once the mouse button is released the graph plot dialog box appears, in which the user can specify the stress or displacement components to be plotted. Note that it is not permissible to display stress and displacement components at the same time, though multiple stress *OR* displacement components may be selected for display.
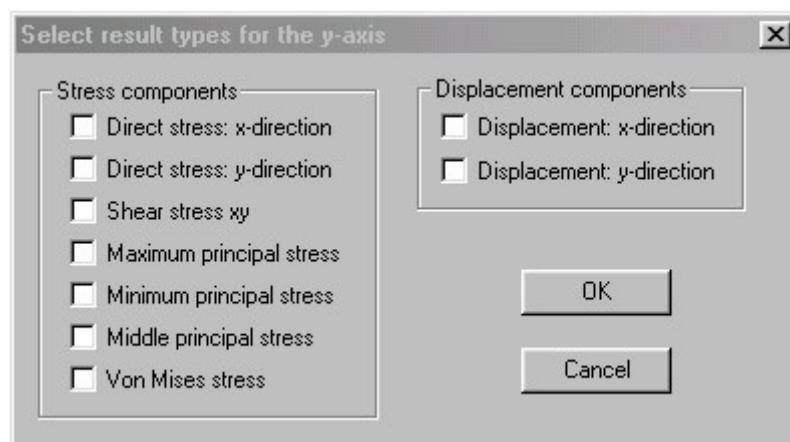

Figure 9-10. Graph plot dialog box

In the example shown in figure 9-11, the user has selected the direct stresses in the *x* and *y*-directions, and also the shear stress, to be displayed.
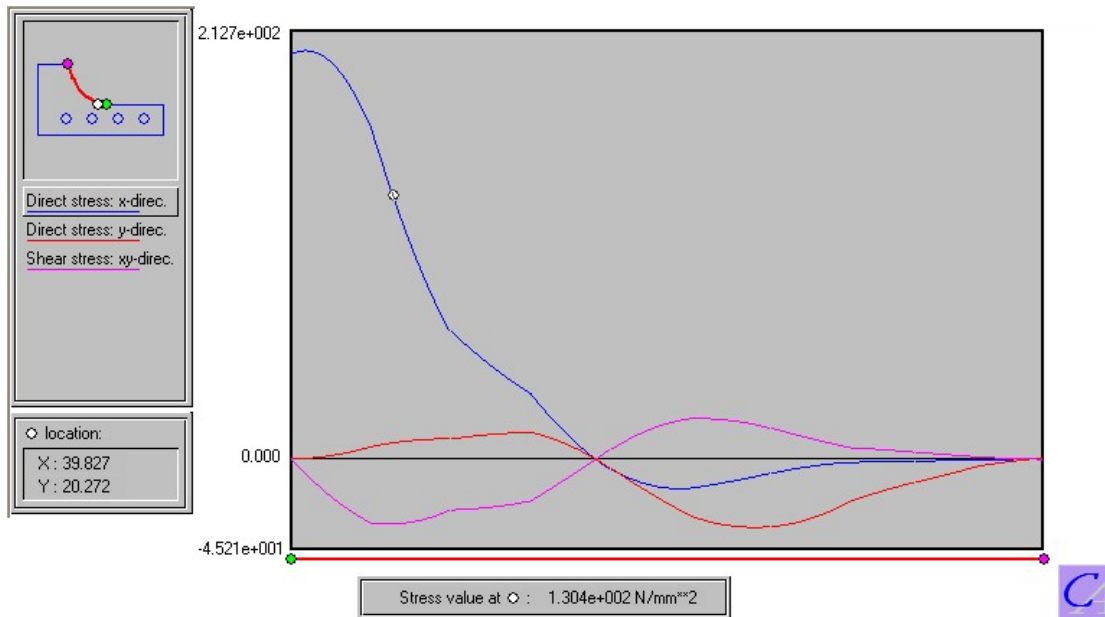

Figure 9-11. Graph plot showing multiple stress components

The plot, and the way the cursor works while the plot is on the screen, is much the same as for the internal line plots (section 9.3). In this case, though, the user may select from the left hand window the result component for which the white circle will follow the line, and for which the stress or displacement result will appear in the caption at the bottom of the screen. In the above example, since *x*-direction stress is shown highlighted in the left hand window, the white circle will follow this line on the graph according to the cursor location.

Horizontal motion of the cursor in the graph area will cause the white circle to move to the corresponding location in the small geometry display, and the new result component will be displayed in the caption. An approximate location of the white circle point, in model (X,Y) coordinates, is also displayed.

9.5 Stress intensity factors

For models containing cracks, the principal mode of displaying results is in the form of the mode I and mode II stress intensity factors, K_I and K_{II}. These may be superimposed on the geometry using the command Results – Stress intensity factors. A typical display is shown in Figure 9-12. A text window confirms the units in which the stress intensity factors are displayed; these units are MPa√m for *both* metric unit sets (N, mm, MPa) and (N, m, Pa), and ksi√in for the imperial unit set.

The display of these results is turned off either by giving the same command again or, more simply, by refreshing the display of the model by pressing the right mouse button once.
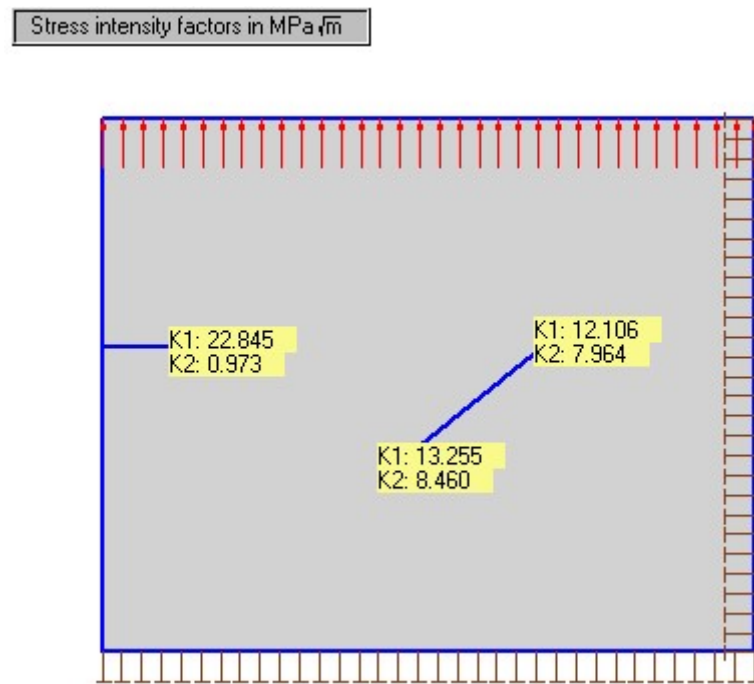
Figure 9-12: Screen display showing stress intensity factor results

Re-analysis

10.1 Justification for re-analysis

The sketching capabilities, and the ability to run a stress analysis on such a sketch, lead us to the conclusion that Concept Analyst is particularly suitable for a role in conceptual design of mechanical and structural components. In this phase of the design, it is expected that the engineer will wish to experiment with many similar design geometries in as short a space of time as possible. This is achieved in the program with the use of re-analysis.

Re-analysis may be defined as the analysis of a problem that is similar to one that has been run already. It may be that, for example, a hole has been moved, or its diameter might have changed. Perhaps a section of the outer boundary has moved, or a fillet radius increased. There are many such design changes that frequently happen in the early stages of a design.

If the design modification leaves many parts of the model unchanged, then it is possible to re-use much of the information calculated during the analysis of the first model. This saves time, and means that re-analysis may be performed in about one quarter to one half of the time required for the full analysis. For a continuously evolving design, this feature allows the stress and displacement results to evolve with the design geometry.

10.2 Initiation of a re-analysis

A re-analysis is started in the following way:

1. Display a contour plot of results

2. Make any one of the following design changes

   - Move shape
   - Move shape point
   - Resize fillet
   - Resize circle
   - Modify spline
   - Addition of a new circular hole

The re-analysis is then performed automatically and the updated contours displayed.

In order to make geometric changes without performing a re-analysis, users should simply revert to a plot of the geometry by clicking the right mouse button once.

The Concept Analyst Theoretical Guide provides detailed information about the operations that are involved in re-meshing and the re-solution. It should be noted, though, that the re-analysis is based upon the assumption that some of the boundary elements have changed their geometry, and that the contribution of those elements to the boundary element matrix system needs to be recomputed. The assumption is that the total number of elements is maintained, just that some of them are in new locations. The addition of a new circular hole is the exception to this rule. This command will cause the matrix to become larger, and this is accommodated by modified form of rapid re-solution.

Some modifications, requiring a more extensive change to the boundary element model, may cause a full analysis to be performed instead of a re-analysis. This is because the program has determined that it will not be possible to produce results of sufficient accuracy using the same number of elements as the previous run.

10.3 Accuracy of the re-analysis

An interesting test to perform goes like this: run a full analysis, and then several re-analysis runs for various design modifications. Note, for example, the peak stress value in the contour plot. Now, for this latest geometry, run a full analysis and compare the peak stress value with that noted at the end of the re-analysis runs. They are likely to be slightly different.

The reason for the difference is the way in which some of the terms in the boundary element matrix are calculated (the diagonal terms, in fact).

There is no intrinsic reason why one set of results should be any more accurate than the other. The mathematical models are slightly different. Intuitively, it would be reasonable to suggest the full analysis should be slightly more accurate though a strong theoretical case cannot be made to

support this. As a general principle, users are advised to run a full analysis from time to time as the design progresses.
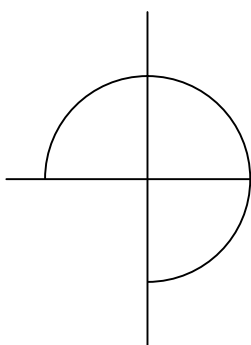
11

Limitations

11.1 Numerical approximation and the Concept Analyst philosophy

It should always be recognised that the results produced by Concept Analyst are an approximation to physical behaviour.

Like many forms of numerical approximation, the accuracy of the solution is directly related to the quality of the approximation, and this is in turn related to the time and effort invested in the approximation.

For programs like Concept Analyst, this essential trade-off is found in the definition of the 'mesh'. Finite element and boundary element software systems all use elements (of one sort or another) to describe the geometry and results. It is usually the case that using more elements will give rise to better results, but will require more time and greater usage of computational resources than the coarser mesh.

Most analysis systems leave to the user this decision about the number of elements to use. This has the advantage that the user is free to undertake what is called a 'convergence' analysis, in which models of increasing complexity are run, and when the solutions do not change markedly from one run to the next, it may be assumed that convergence has been achieved and the numerical model is proving to be adequate.

On the other hand, leaving the decision to the user requires that user to be reasonably expert in the use of this technology. Concept Analyst takes the approach that these important decisions should be made automatically by the program according to some rules contained within the software algorithms (although a limited convergence analysis is still available through the use of the coarse, standard and fine mesh density settings).

This has the advantage of ease and speed of use, particularly in the hands of non-expert stress analysts. However, it is possible that a particular geometry might cause the program difficulties in automatically defining a suitable mesh of boundary elements for that particular problem. This section presents information that should be considered by users as part of their analysis modelling work, in order that they should be aware of some of these special cases.

By turning on the display of elements (by the use of the Edit – Preferences… dialog box), users will be able to see the mesh refinement that is automatically carried out by the program to respond to particular geometric shapes.

11.2 Elements close together

The fundamental numerical difficulty that boundary elements have is with elements that face each other across a narrow gap, or across a small thickness of material. Figure 11-1 shows two geometries that potentially cause problems of this nature.
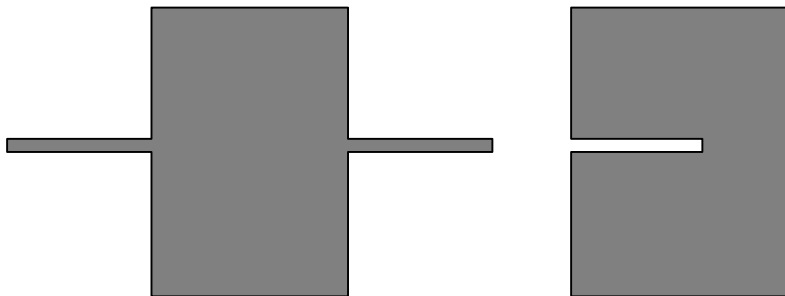

Figure 11-1. Elements close together can face each other across narrow sections or slots

The elements on the top and bottom surfaces of the narrow section or slot face each other across a gap as shown in close-up in figure 10-2.

Figure 11-2. Elements close together

Most boundary element systems normally recommend that boundary elements should be no longer than about two or three times their separation distance. This means that small elements should be used for areas of a model in which small separation distances are involved.

For many cases, the automatic meshing performed by Concept Analyst produces reasonable meshes and will produce reasonable results. Versions 1.5 and earlier of Concept Analyst did not refine meshes when elements facing each other were:

- on different shapes

or

- not closely connected on the same shape

These cases are illustrated in figure 11-3, which shows two cases in which the program has automatically refined the mesh in areas in which elements close together might otherwise cause numerical accuracy problems.
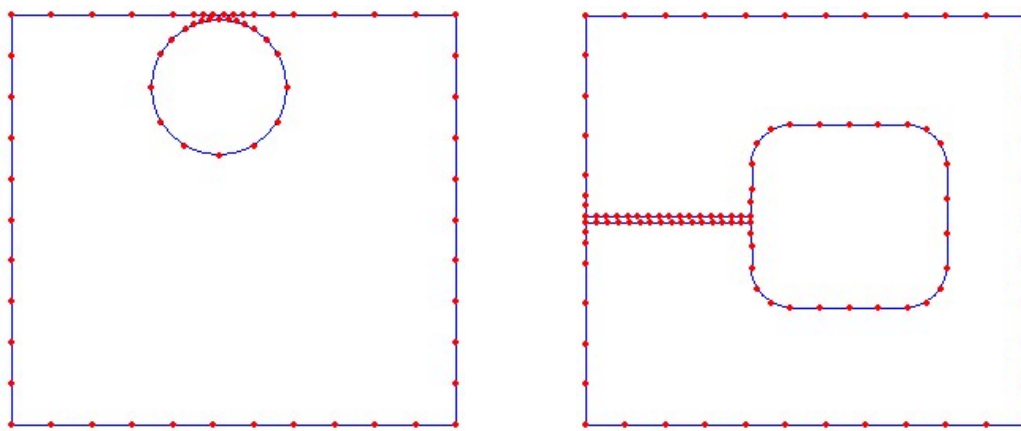

Figure 11-3. Elements close together causing localised mesh refinement

11.3 Thin sections

Boundary elements can have numerical problems when analysing very thin sections (fig 11-4).

Figure 11-4. Thin section analysis

Concept Analyst is capable of producing accurate solutions for slenderness ratios up to 100:1. Beyond this, inaccuracies start to develop in the solution. Users should exercise caution using the software to look at stresses in very thin sections.
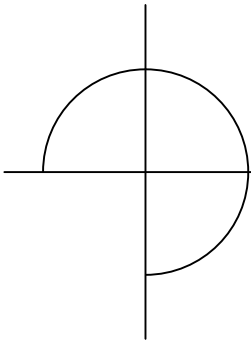
12

Files

12.1 Model files

At any point the current model may be saved using the icon ⊞ or the normal Windows commands File – Save or File – Save As…

The default file extension for Concept Analyst is "*.cna" so that a model file might be named "PlateWithHole.cna", etc.

A Concept Analyst model file may be opened using the icon 📂 or the Windows command File – Open.

Model files contain ***no results***, so the analysis must be run again if required on opening a file. This allows model files to be very small and easily archived. For the 2D models runs in Concept Analyst run times are almost insignificant.

12.2 Results file

In addition to the graphical displays of results available, as described in section 8, the program creates a text file containing a statement of the boundary element locations and the associated results. This file is created on any of the following operations:

- Full analysis

- Re-analysis

- Internal line operation

The results file is overwritten on any of the above operations so that, for example, on a second internal line operation, the results for the first internal line become unavailable.

The results file name is initially set to be "CAResults.out" and is placed in the directory indicated in the "Start in" field for the properties of the Concept Analyst shortcut icon.

The results file name may be modified at any stage by giving the command File – Results file name. This allows the full path name to be entered via the keyboard.

12.3 Report file

Concept Analyst contains a feature allowing users to produce reports on their work very quickly and easily. This is done using the File menu, under which a new set of commands relating to the report file have been implemented. Under the File – Report file sub-menu, there are commands:

> New report file…
> Add current results plot
> Close report file

The combined use of these commands will generate a proforma report file in html format, that can be viewed in a web browser, manually edited and printed.

Report files may be generated only after an analysis has been performed on the object. This is because the summary at the beginning of the report contains a brief description of the maximum stress, and this information is not available before analysis.

The stages of compiling a report file are as follows:

New report file…

This command prompts the user for the file location (default is found from the personal settings file) and name (default name is CAReport.htm). Prompts are made for the part name and author (default is the user's name taken from the personal settings file).

Add current results plot

Once the report file has been opened, as above, the 'Add current results plot' command may be given when any form of results plot is displayed on the screen. This applies to contour plots,

deformed shape plots, boundary or internal graph plots. For each plot, a new section is written to the file containing the plot and a brief paragraph of descriptive text referring to the figure and giving the maximum value of the results quantity in question. This command may be given several times, allowing multiple results plots to be compiled together in the report.

The 'Add current results plot' command is also given by the toolbar icon .

Close report file

This command prompts the user to enter some text to form the Conclusions and Recommendations section of the report.

The Concept Analyst report file contains automatically generated text that is not likely to be sufficiently complete for most reports. However, it does provide a framework for the report, and allows editing. Users are advised to edit the file to insert whatever extra text may be required for the particular problem under consideration. Editing may be performed in any text editor, such as the Microsoft 'Notepad' available on Windows operating systems.