Concept
Analyst

TUTORIAL

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 the University of Durham 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
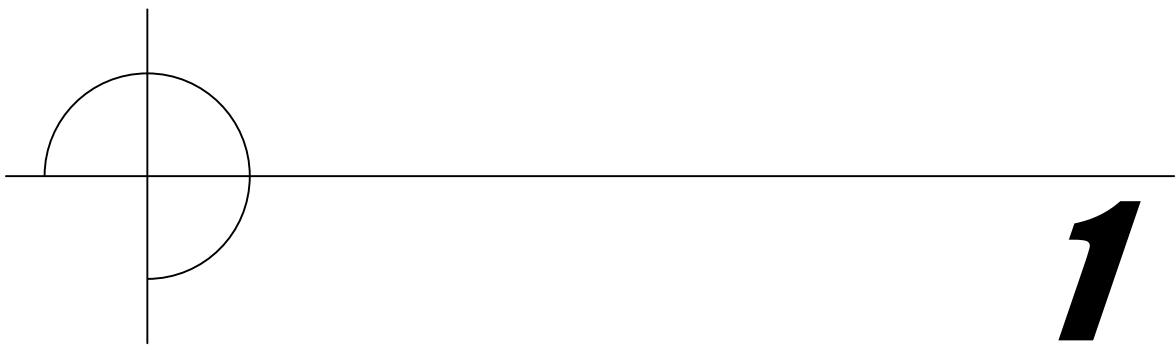
Introduction

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.

This document provides some sample interactive sessions that may be followed by engineers new to Concept Analyst in order to gain familiarity with the most commonly used features of the system.

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, 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

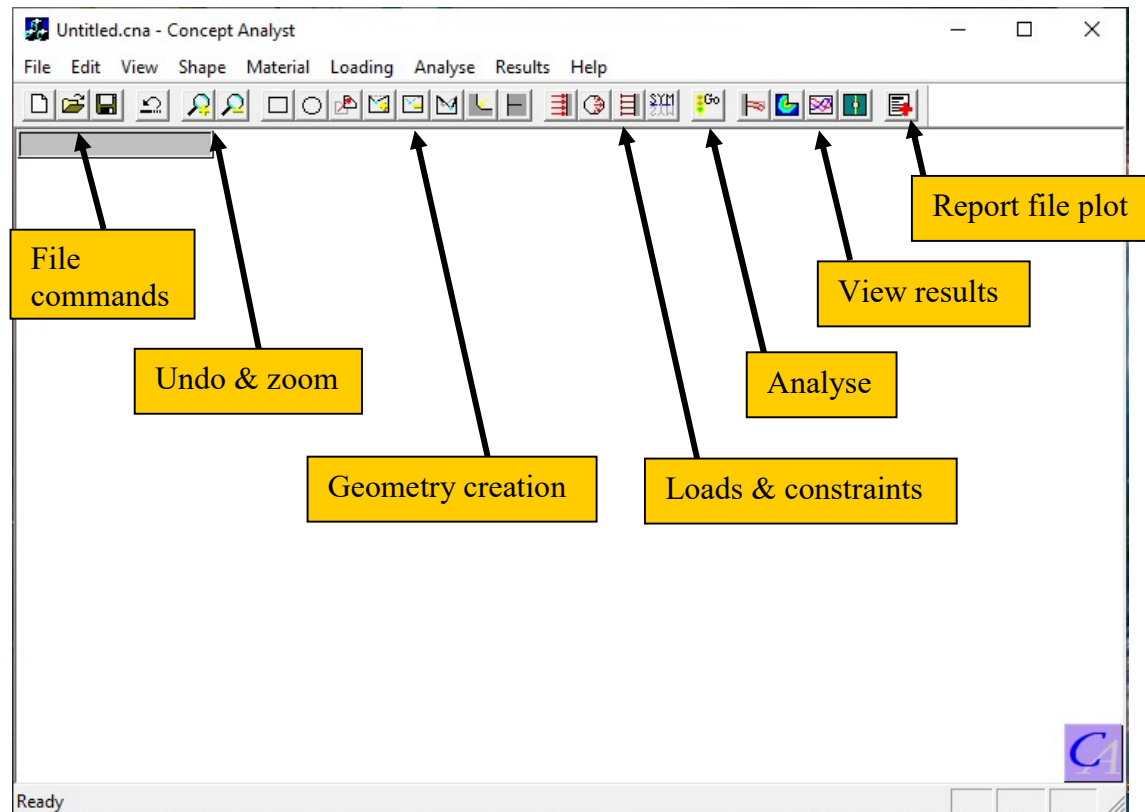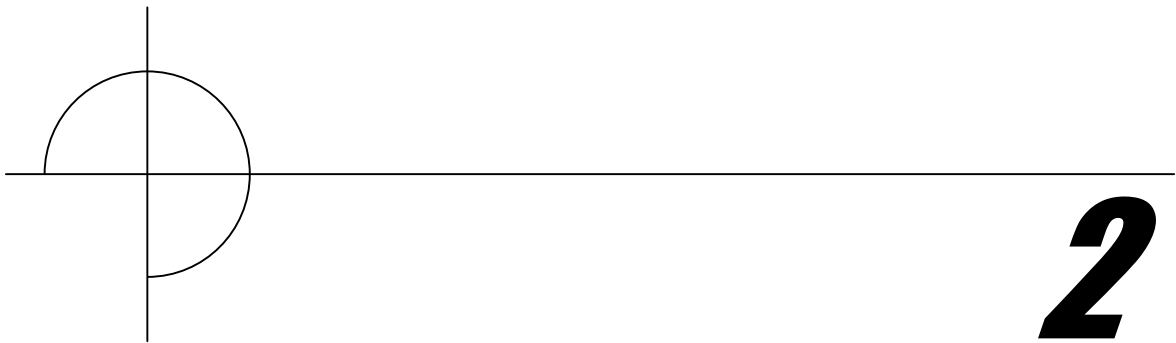The main screen layout appears as shown in Figure 1-1.

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. These are grouped by function, as shown in the figure.

2

Session 1

To start the Concept Analyst tutorial, generate a model of a rectangular plate with a circular hole under uniaxial tension.

Click on the rectangle icon ▣.

Now move the cursor around the screen, and notice how the X and Y coordinates at the top of the screen continue to show the cursor position. The default unit set is [Newtons, millimetres, N/mm^2]. Without being careful about the (X,Y) coordinates of the points concerned, position the cursor where you want one corner of your rectangle to be, press the left mouse button and, keeping the button pressed, drag the mouse to the opposite corner to draw a rectangle. Release the mouse button when you are happy with the size and shape of the rectangle. Try to achieve a rectangle of the approximate proportions shown below, but don't worry about the exact size of the rectangle. For now we are using Concept Analyst as a sketching tool.
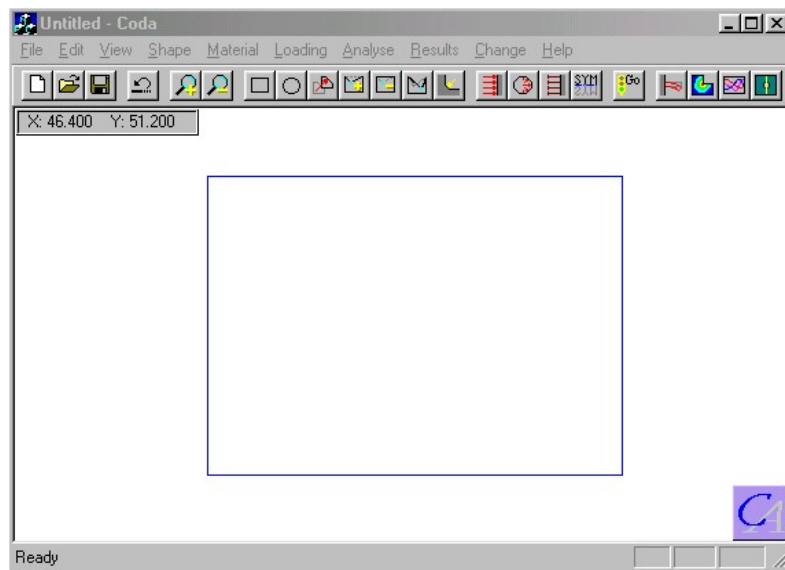
Notice now how the bottom left hand corner of your rectangle is at the origin, i.e. its coordinates are (0,0). The origin automatically snaps to this corner of a rectangle when it is the first shape to be defined.

Now click the circle icon .

Move the cursor to the screen location you choose for the centre of the circle. Press the left mouse button and, as for the rectangle, drag a circle shape with the button depressed, releasing the button when you are happy with the size of the circle. Notice how the circle radius is displayed during the click and drag. Try to create a circle that is roughly central in the plate.
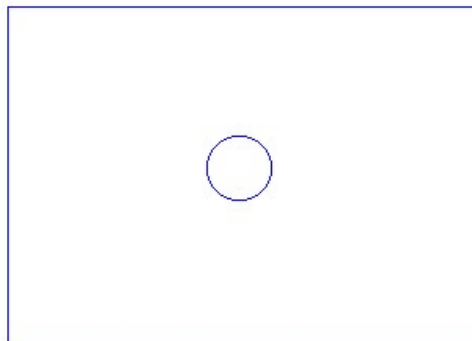

Now click the Move Shape icon . At this point, if you have not selected that points be drawn, they appear now in green. These are the base geometry points on which the shapes are formed.

Press and hold the left mouse button near one of the points highlighted in green on the circumference of the circle, and drag the circle to a new location. Release the mouse button when you are happy with the location of the circle. (Of course, this is optional – you don't have to change the position of a circle if you don't want to). The X and Y coordinates displayed represent the new coordinates of the point you are dragging.

The default material properties are those of mild steel, and these will be used. All that remains is to define the loads and boundary conditions.

Click on the pressure icon .

Move the cursor to the line on the right hand side of the rectangle, and click-and-release the left mouse button. The line segment becomes highlighted and a dialog box opens up. Apply a pressure (traction) of 100 units in the x-direction (this is the default for vertical lines) and press OK. The load is displayed on the screen. If your (default) units are Newtons, millimetres and MPa, you have applied a distributed load of 100 MPa (N/mm^2).
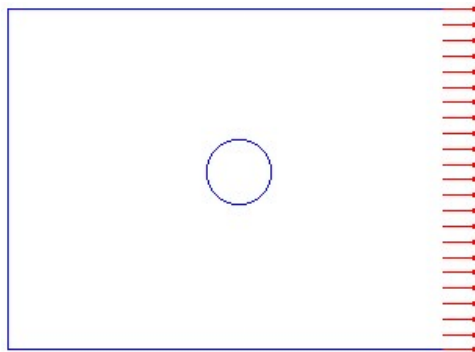
Click the constraint icon .

Move the cursor to the line on the left hand side of the rectangle, and click-and-release the left mouse button. The line segment becomes highlighted and a dialog box opens up. Apply a displacement of 0 units in the x-direction and press OK. Move the cursor to the line on the bottom edge of the rectangle, and click-and-release the left mouse button. Apply a displacement of 0 units in the y-direction and press OK. The constraints are displayed on the screen.
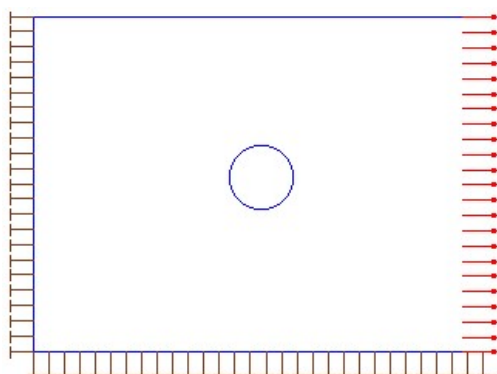
The model is complete. Initiate the analysis using the Analyse (Go) icon .

The analysis will take only a very short time. Click the contour plot icon .

A dialog box opens enabling the selection of the results quantity to plot. Select maximum principal stress and press OK. Contours of maximum principal stress are displayed, with a legend indicating the meaning of each contour colour.
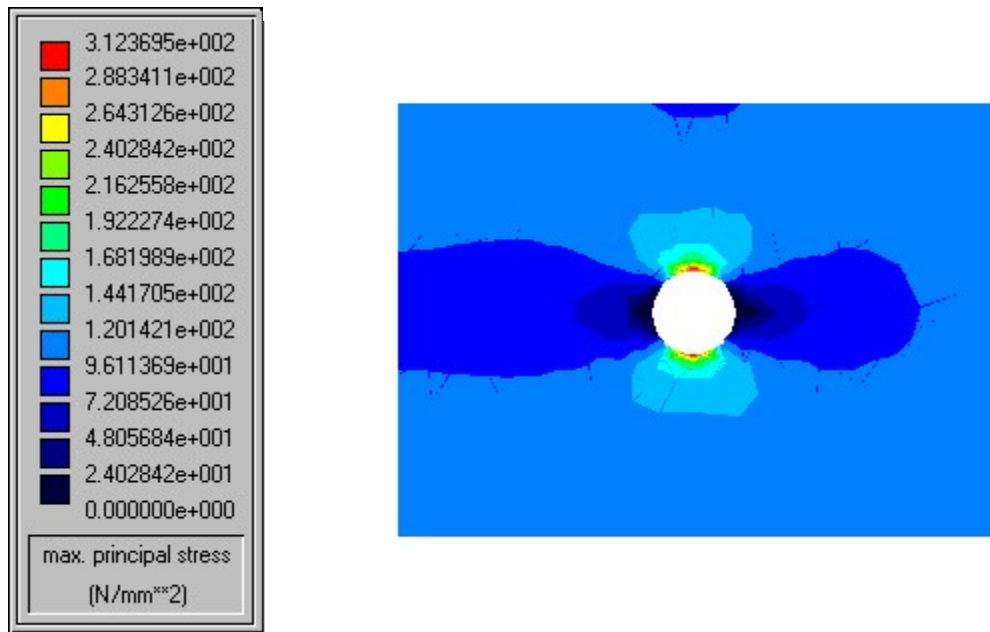
In this example the maximum stress is 312.4 N/mm^2, giving a stress concentration factor of 3.124 for the hole. Yours will be somewhat different because you have a different geometry, but the maximum value should be slightly over 300, depending on the size of the circular hole relative to the plate dimensions.

Click on the deformed shape icon.

The plate will be shown in undeformed shape and then a short animated display of the deformation will be shown, completing with the final deformed shape being superimposed on the original shape.
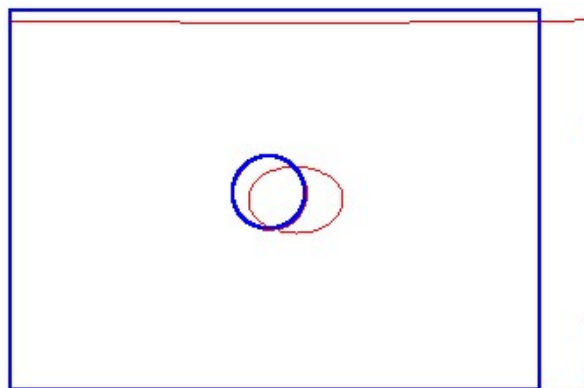

Notice the ovalling of the hole and the fact that the right edge of the plate deforms most in the centre, indicating the reduction in stiffness resulting from the hole.

To draw a graph plot, first restore the display to the normal setting of geometry and loading. Do this by clicking the right mouse button (the cursor can be anywhere in the window).

Now click the graph plot icon.

Click on one part of the circular hole and, keeping the mouse button depressed, move the cursor around the hole. Sections of the hole will become highlighted. When the entire hole is highlighted, release the mouse button. The highlighted section of boundary will become the horizontal axis of the graph plot. Select now the type of results to plot graphically on the vertical axis. Select both maximum principal stress and minimum principal stress, and then press OK. A graph plot is displayed (yours may be different, depending on the way you used click-and-drag to select the circular boundary to form the x-axis).
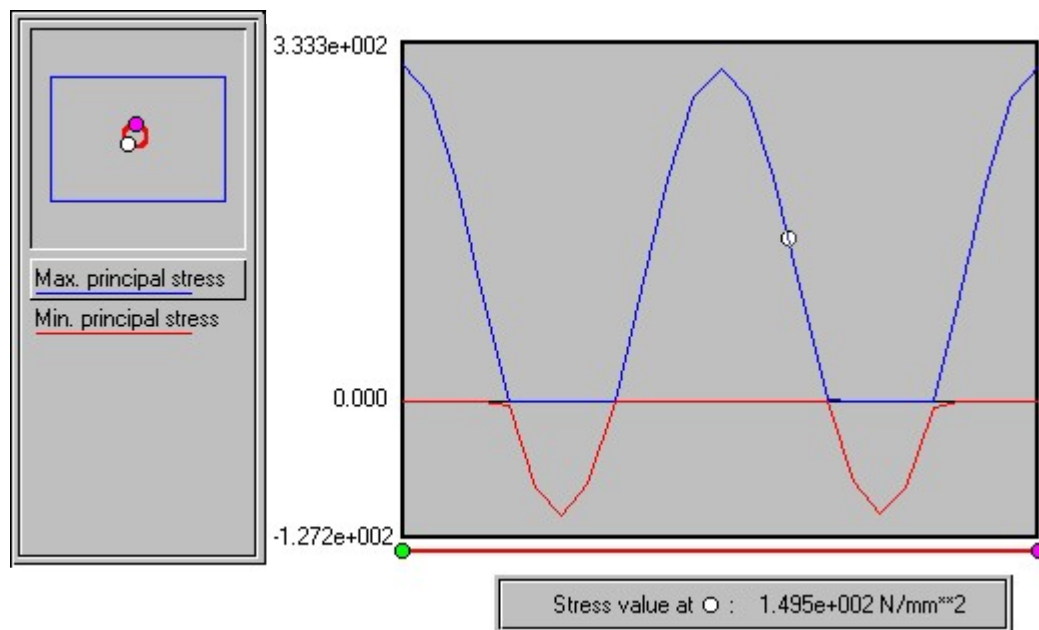
Now move the cursor over the graph itself. Notice that horizontal motion of the cursor causes the white circle to move in the top left hand corner of the window, indicating the position on the model that relates to the position of the cursor along the horizontal axis of the graph. Notice also that the white circle also follows the maximum principal stress line on the graph, and that the stress value at this location is shown in the box at the bottom of the screen. Placing the cursor over one of the peak stresses will immediately show both the location and the value of any desired stress.

Click on the "Min. principal stress" legend and you will find the white circle now follows that line on the graph, and the stress value shown at the bottom of the screen now shows the minimum principal stress.

This concludes session 1 of the Concept Analyst tutorial, in which you will have used the essential features of the program to determine a stress concentration around a hole in a plate.
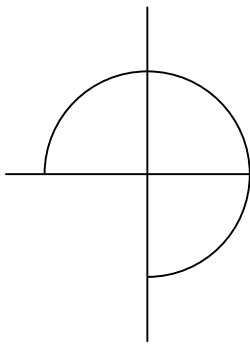
3

Session 2

In this session we introduce some of the geometry deformation and re-analysis capabilities of Concept Analyst.

If you have not left Concept Analyst since the end of the last session, simply click the New File icon ▢. Otherwise enter the program by double-clicking the shortcut icon.

Start by using the rectangle icon ▢ and using a click-and-drag operation to define a rectangle. Try to make your rectangle approximately square.

We will now make a change to the way the model is displayed on the screen. Click on the menu item "Edit" and then select "Preferences…" from the pull-down menu that appears. A 'dialog box' appears in which you can select various settings. Select the setting 'Draw points', and press OK. Now you will see the points at the corners of your rectangle displayed with small green circles. This can be useful when defining the model geometry. You might remember that this setting was applied automatically (though temporarily) in session 1; this is done when moving shapes.

Now click on the 'Add Shape Points' icon ▢ and use the left mouse button to click on the four locations marked ● on the figure below (the points will not be marked in this exaggerated way on the screen – the circles are used only in this document for

diagrammatic purposes). You should hold down the shift key while clicking, and this
will snap the points onto the nearest line location.
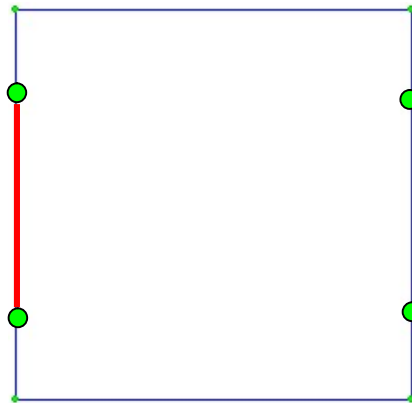
Now position the cursor over the line segment shown above in red, press and hold the
left mouse button and drag the section of boundary inward, releasing it when you are
happy with the dragged geometry. Repeat the operation on the right hand side.
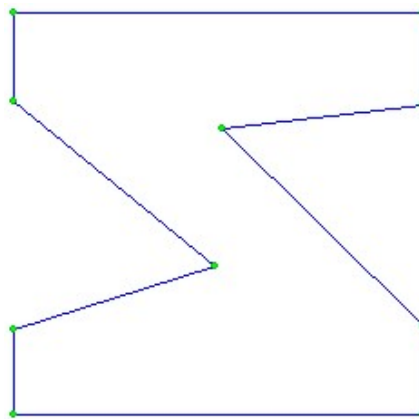
We will now create a fillet. Click on the 'Fillet' icon .

Now click and hold down the left mouse button with the cursor on one of the new
corners you dragged into the square. With the mouse button held down, drag the fillet
until the desired radius is achieved. Repeat the operation on the other interior corner
(you do not have to click the 'Fillet' icon again as this operation is still active). Try to
aim for a geometry similar to that shown below.
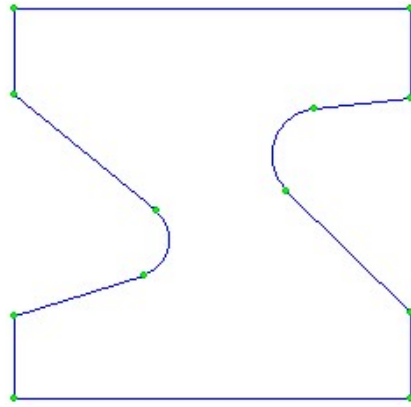
To demonstrate the manipulation of a shape, click the 'Move Shape Points' icon ⊡.

Press and hold down the left mouse button with the cursor over a tangent point where one of the fillet arcs meets a straight line. With the mouse button held down, drag the geometry around, releasing the mouse button when you are happy with the final position. Notice how the fillet feature is recognised and the geometry is distorted such that the radius remains constant throughout.

Using the menu bar, give the command 'Shape – Resize fillet'. Now place the cursor over one of the existing fillet arcs and drag to a new radius.

As for the first example we will use the default material properties, those for mild steel.

Click on the pressure icon ▤. Move the cursor to the line on the top of the geometry, and click-and-release the left mouse button. Apply a pressure (traction) of 100 units in the y-direction and press OK.

Click the constraint icon ▤. Move the cursor to the line on the bottom of the geometry, and click-and-release the left mouse button. Apply a displacement of 0 units in both the x- and y-directions and press OK. The loads and boundary conditions will be displayed on the screen.
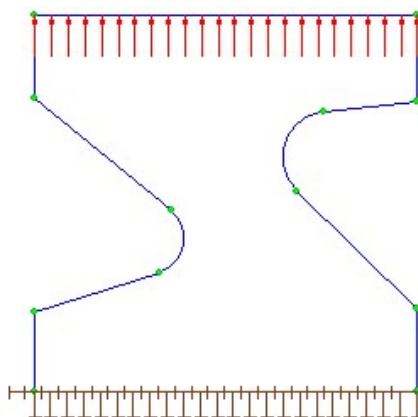
Initiate the analysis with the 'Analyse' icon ⊡.

Click on the deformed shape icon .

As in the first example, the geometry will be shown in undeformed shape and then a short animated display of the deformation will be shown, completing with the final deformed shape being superimposed on the original shape.
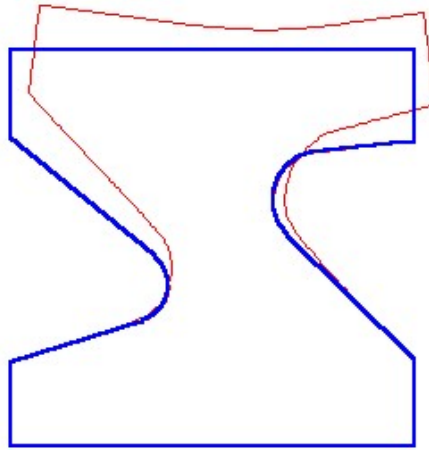
Click the right mouse button to redraw the geometry, and click the graph plot icon .

Click-and-drag to select a range of lines to form the horizontal axis of a graph plot. Include one (or both) of the fillets. Select whatever stress or displacement components you like and draw the graph, picking off the important values with the cursor positioning.

To complete the initial postprocessing phase, draw some contour plots. Click the contour plot icon .

Select a results quantity of your choice. Displayed below are the y-direction displacement, Von Mises stress and a failure criterion plot of Von Mises stress. This last plot displays in red the material that has failed by the Von Mises criterion, and the remainder of the material appears in blue. (Notice that Concept Analyst does not consider any material non-linearity, and so the material shown in red has been allowed to become stressed beyond yield. If the material non-linearity were to be included it would be found that the 'failed' region would extend over a greater area).
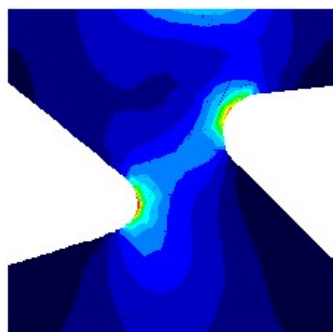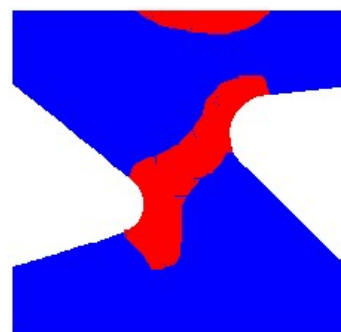
          y-displacement   von Mises stress   failure criterion

Notice that the material around both fillets and in the centre of the top surface has failed according to the Von Mises criterion.

Return to a view of contours of von Mises stress. Notice the location and value of the maximum stress value. In the example above, the maximum occurs on the left hand fillet. Yours will be different, of course. Now use Concept Analyst's re-analysis capability to investigate the effect of fillet radius on this maximum stress value.

Because there is a contour plot on the screen, Concept Analyst is now ready to re-analyse the problem given a geometric change. This is possible for the following operations:

- Move shape points
- Move shape
- Resize fillet
- Resize circle
- Modify spline
- Add a new circular hole

It is not currently available for other operations (e.g. Add shape points). This is because the re-analysis scheme relies upon the modified model having the same number of boundary elements with the same numbering as the original model. The re-analysis re-uses as much information as possible from the previous analysis in arriving more rapidly at the solution.

Any of the above geometric operations will now initiate a re-analysis. (Notice that if you want to change the geometry without doing a re-analysis, simply return to a plot of the geometry by a single click of the right mouse button; once contours are no longer displayed the re-analysis feature will be inactive). We will reanalyse the problem by resizing a fillet.

With the contour plot still displayed, use the menu bar to select
<p style="text-align:center">Shape
Resize fillet</p>

Now with the cursor on the fillet line on which the maximum stress occurs, press and hold the left mouse button while dragging the fillet to a larger radius. Release the mouse button when you are happy with a slightly larger fillet radius than the original model. Moments later a revised contour plot will be shown.
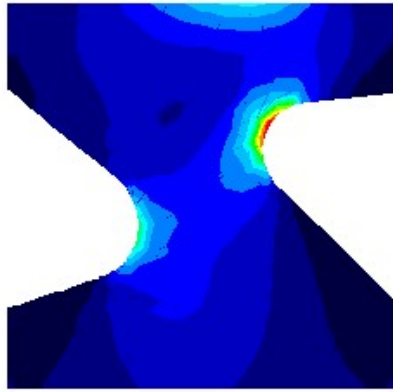
The maximum von Mises stress will have reduced with the increased fillet radius, and indeed the location of the maximum may well have moved to the other fillet. If your maximum stress is still on the same fillet, simply click-and-drag the radius again to a larger radius until the maximum moves to the opposite fillet.

Now click-and-drag to increase the other fillet radius, and see the stress level reduce further.

We will now do a re-analysis on deforming the boundary polygon. Select the move shape points icon ⊞. Move the cursor to the top right hand corner of the model and click-and-drag, releasing the mouse button when you are happy with the new location of this corner of the plate. A revised contour plot will be drawn.
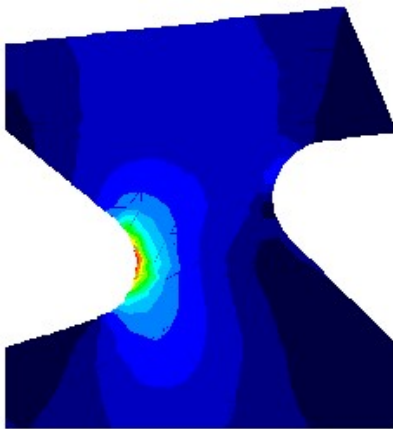
The original model has been changed in multiple ways. Both fillets have been increased in radius, and the top of the plate modified. At each stage, Concept Analyst has done a re-analysis that is very rapid in comparison with the first analysis. Only a very small approximation is made in the re-analysis, so the contours now displayed are very close to the stress contours which would occur if this were analysed in full in the usual way. Confirm this now by initiating a full boundary element analysis on the new geometry.

Remember the maximum stress value.

Press the Analyse icon ⊞.

A full analysis proceeds. On the example above, the revised maximum stress changes slightly, demonstrating that the accuracy of the re-analysis is of the same order as that of a full boundary element analysis.

Notice the generally low stress values in the top centre of the plate. Investigate now the effect of including a hole in the plate as a weight saving option.

Click the circle icon ☐.

Move the cursor to the centre of the circle to be added, and use a click-and-drag operation to create a circular hole similar in size and position to that shown in the contour plot below to generate a new set of Von Mises stress contours.
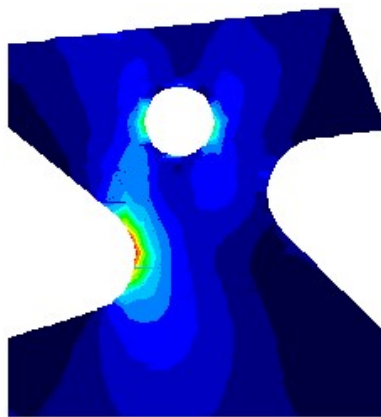
In this example (again, your contour plot will be different, of course), the maximum stress should increase slightly with the addition of the hole, and the hole is clearly attracting stress concentration effects. Zoom in around the stress concentrations to see more clearly the stress distribution.

Click on the zoom in icon ☐.

Use a click-and-drag operation to define a rectangular zoom window around the area of interest. A zoomed view will be displayed.
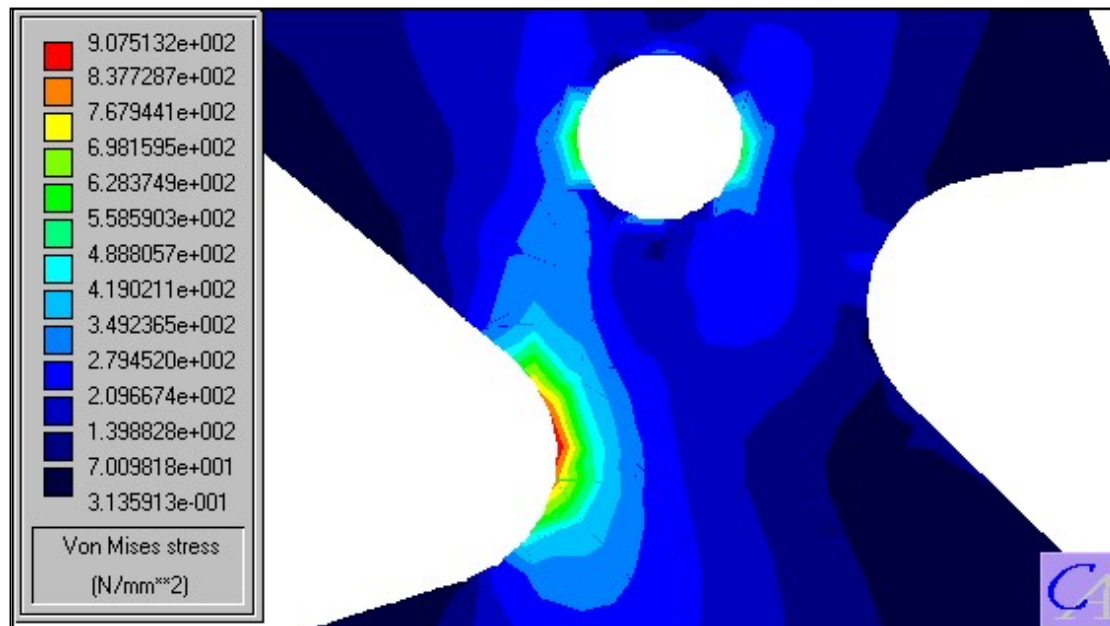
Finally, you may notice that the contours of results can be somewhat angular, and not quite as smooth as might be expected. This is all related to the issue of obtaining good quality results in a very short space of time. The smoothness of the contours can be improved by setting the 'Report Quality Contours' option. We will do this now. From the pull-down menu at the top of the window, give the command Edit – Preferences… and select the report quality contours option. On pressing 'OK' a short re-analysis is performed and a smoothed set of contours displayed.

The report quality contours will be set now allowing you to display, in report quality, more contour plots. However, the program reverts automatically to standard quality contours the next time you do a full analysis or re-analysis. It is not intended that the report quality contours be switched on as anything but a temporary measure.

This concludes session 2, which has demonstrated the use of the Concept Analyst system for interactive manipulation of planar geometries and for the rapid evaluation of the durability of design options.

4
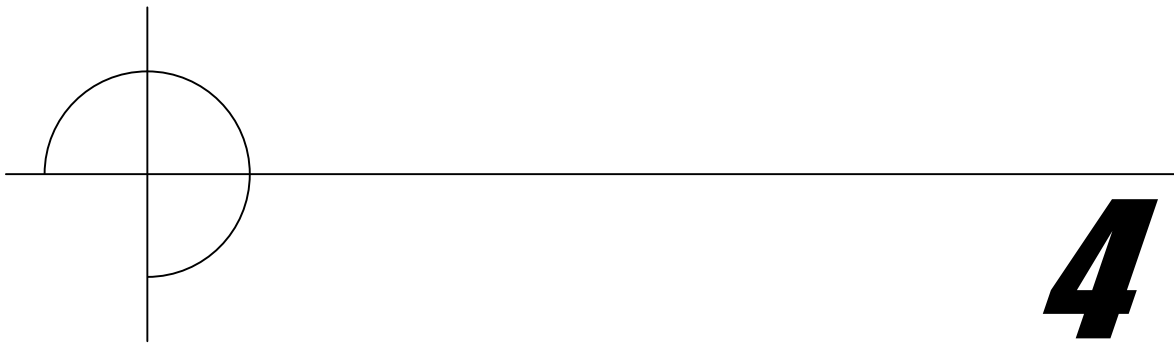
Session 3

In this session we introduce the use of bearing loads, context-sensitive manual editing of geometric details, the internal line display and file output. Open the Concept Analyst program in the usual way and sketch a rectangle. Now create two circles. At this stage, don't worry about trying to make the model look like the pictures below.

Place the cursor over one of the lines displaying the rectangle. Double-click the right mouse button (remember a single click of this button causes the model to redraw, e.g. after a contour plot). A dialog box appears showing the horizontal and vertical dimensions of the rectangle. Edit the numbers here to specify a horizontal dimension of 150 units and a vertical dimension of 100 units. The default length unit is millimetres.

Next click the 'Add Shape Points' icon and click and drag to deform the right hand edge of the rectangle outwards. Once again, do not worry about the exact coordinates of the new point. Now position the cursor over the new point you just created and double-click the right mouse button. This time, since the shape is a general polygon (no longer a rectangle) you are prompted for the coordinates of the point. Edit these now to give the point an X-coordinate of 200 and a Y-coordinate of 50 units. The point moves to its new location.

Place the cursor over one of the lines drawing a circle. Double-click the right mouse button, and this time, the program recognises the feature as a circle, and prompts you to confirm or edit its geometric definition. Give the circle a centre X-coordinate of 150, a centre Y-coordinate of 50 and a radius of 15 units. Perform a similar operation on the second circle, giving a centre X-coordinate of 80, a centre Y-coordinate of 50 and a radius of 25 units. You should have created a model like the one below.
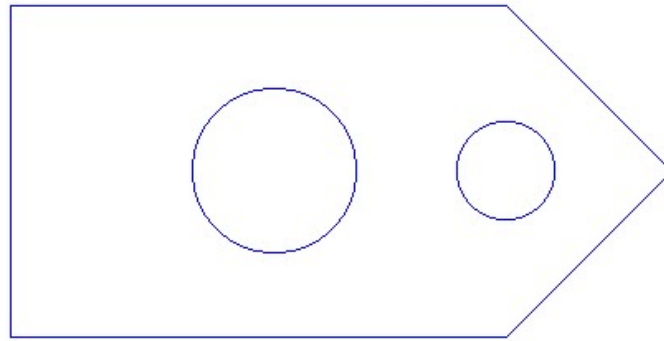
Now we define the loading on the component, which is to take the form of a bearing load through the right hand hole, this representing a normal load in a Cosine distribution over 180° of arc.

Click the 'Bearing force' icon and then the right hand circle (anywhere on the circle will do). Apply a total force of 2000 units in the direction to the right. The default force units are Newtons, and the program assumes a unit thickness. Concept Analyst will automatically compute the required Cosine load distribution and display it on the screen using variable length arrows around the arc. Note that this assumes that any shaft or pin through the hole makes a perfect fit with no clearance. This will not be a non-linear contact analysis – see Session 5 for a tutorial in the pin/hole contact analysis. Since this is a load that has been defined as a *force*, the load is displayed using blue arrows instead of the red that was used to display loads specified in terms of distributed, pressure, traction loads.

To complete the model, click the 'Constraint' icon then the left hand edge of the model, and fully constrain this edge by giving a zero displacement condition in both the x and y directions.
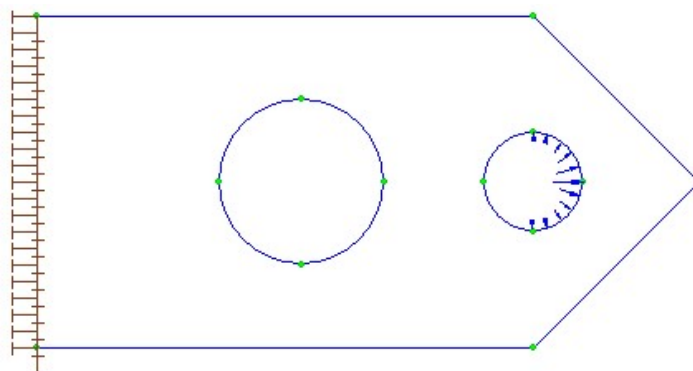

Run the analysis using the 'Analyse' icon .

Display a contour plot ![icon] of Von Mises stress. Depending on your window size, you may want to zoom out ![icon] to fill the screen with this plot.
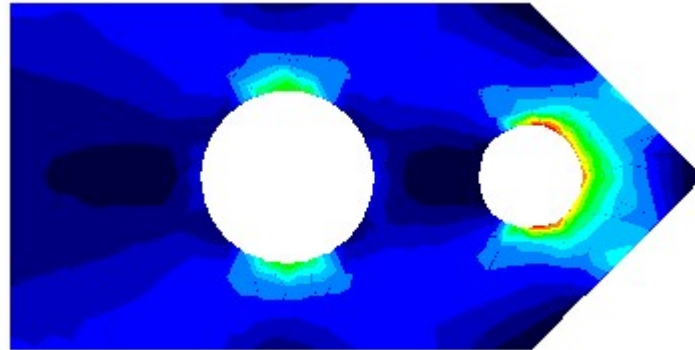

We can now find more information about how the stresses vary through the material. Concept Analyst includes a facility to display x-y graphs over internal lines. Whenever a contour plot appears on the screen, it is possible to define any line in the model and a graph will be drawn of results over that line, the graph showing the type of results being contoured. Thus, in order to obtain an internal line graph of y-direction displacement, for example, first display a contour plot of y-direction displacements and then define the internal line. In our example we will create an internal line plot for the Von Mises stress contours just displayed.

Click the 'Internal Line' icon ![icon] and use the mouse to click and drag a line on the model. Since the area around the loaded hole is the most active in this model, define a line that starts at the top of this hole and extends to the plate corner directly above it.
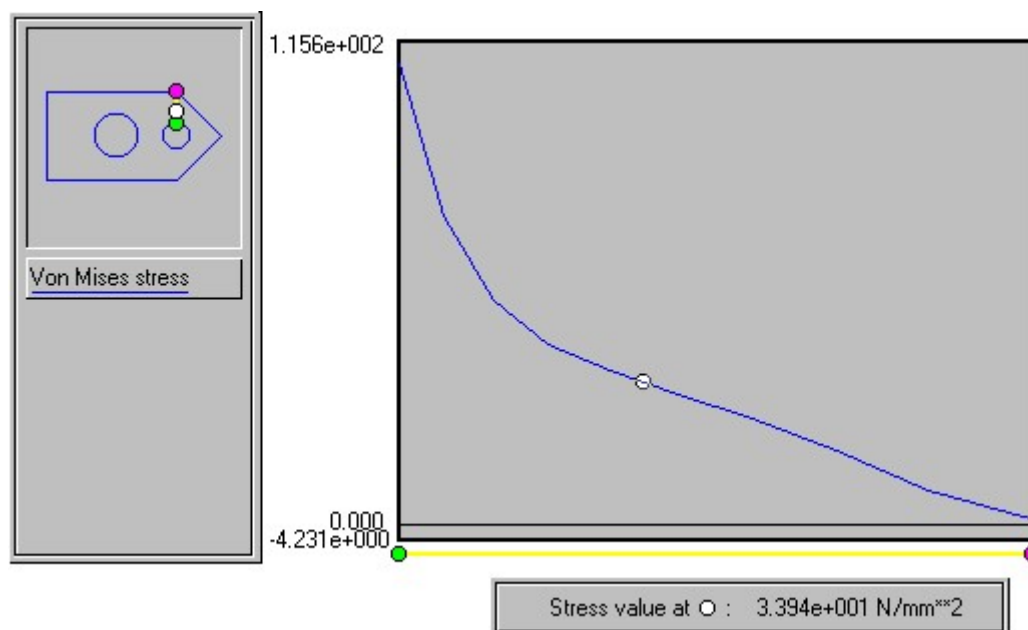
This display works in the same manner as the boundary graph plot seen earlier. The cursor location on the screen is reflected in the white circle on the small model display, to indicate the model location corresponding to the horizontal position in the graph, and the 'Stress value' box indicates the stress level corresponding to the vertical position of the cursor on the graph.

Internal lines can be wholly within the material or extend to the boundary. They can also cross holes and other features. Test this out now by redisplaying a contour plot and then defining an internal line crossing both holes.
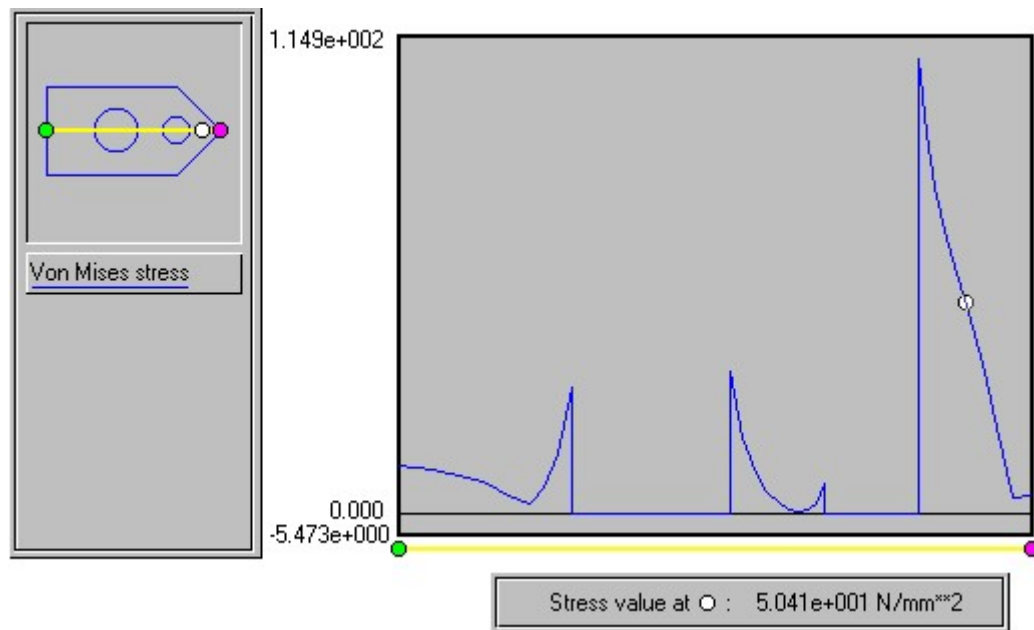
Finally, although Concept Analyst provides many graphical means of displaying results, it is possible that users may need the stress and displacement information in tabular form.

Give the command File – Results file name (from the pull-down menus at the top). A dialog box shows the name of the file that Concept Analyst uses to store a tabular listing of all mesh point displacements and results, and also of the results at a number of points along the latest internal line (previous internal line results are removed when a new internal line is created). The default file name is CAResults.out, and it is by default stored in the Concept Analyst models directory. To check the contents of this file, either:

- If you have permission to access this directory and file, simply open up the file in another program, e.g. Notepad or WordPad.

Or

- If you don't have permission to access the file, use the File – Results file name command to change the directory and/or file name. Then analyse the problem again and, if you like, display contours and an internal line plot. Now open up the file in another program, e.g. Notepad or Wordpad.

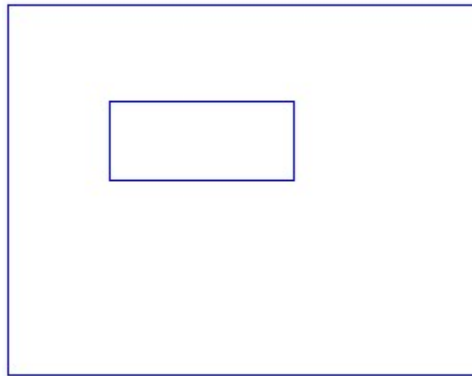This completes tutorial session 3.
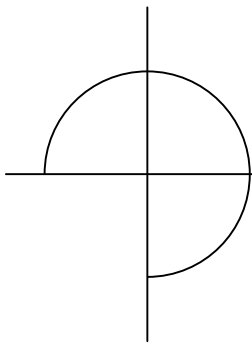
Session 4

In this session we introduce some new features, notably the use of splines, the moveable origin and load checking. We will be defining a geometry containing the following features and dimensions (in millimetres).
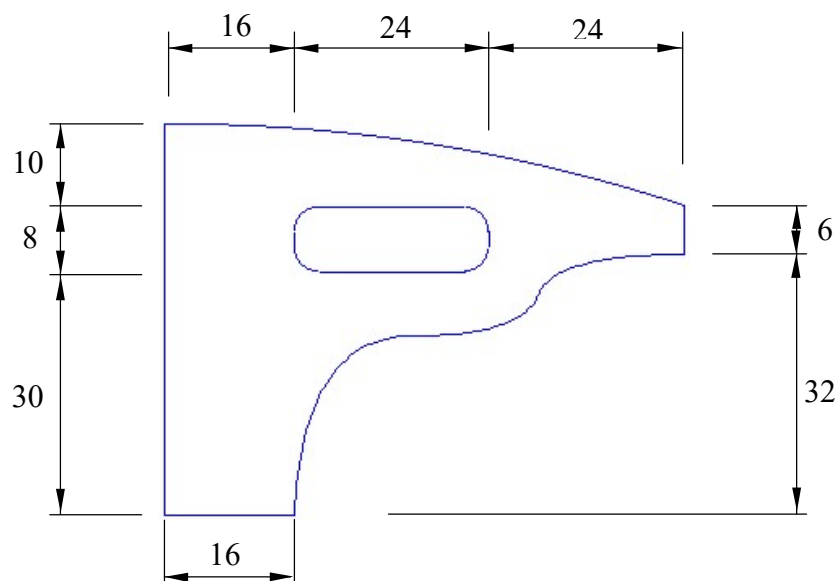
Enter Concept Analyst in the usual way and define first a large rectangle then a small rectangle.

Use the double right click to define the dimensions of the larger rectangle to be (64 x 48mm) and those of the smaller rectangle to be (24 x 8mm).

Click the Add points icon , then position the cursor at (16, 0) and click to create a polygon vertex at this point.

Place the cursor over the top right hand corner of the large shape and double right click. Move the point to a position (64, 38). Similarly, move the bottom right corner of the larger shape to a position (64, 32). Notice that, as soon as the shape is no longer a rectangle defined by four corner points, the double-right-click operation no longer allows the horizontal and vertical dimensions of the rectangle to be defined. Instead, the prompt asks, as we have seen, for the x and y coordinates of the nearest vertex.

Concept Analyst includes a feature that makes it relatively easy to define geometries by offsetting from other geometric details. This is achieved through a moveable origin. It was seen earlier that the origin initially snaps to the bottom left hand corner of a rectangle if it is the first shape to be defined. However, the location of the origin can be moved as many times as we like during construction of a model. The command Shape – New origin (there is no toolbar icon for this command – only this pull-down menu option) allows the user to pick a screen location for the origin. If this is close to (i.e. within a few pixels of) an existing point or shape vertex, that point will be used as the origin.

Give the command Shape – New origin and click the upper left hand corner of the model. Notice now how the (X,Y) coordinates change so that this corner is now the origin, and the rest of the model now has negative Y coordinates.

Now move the smaller shape into place. Click the Move shape icon . Position the cursor over the top left corner of the small rectangle and (with the left mouse button) drag the shape to a position in which the dragged corner lies at (16, -10). Use the coordinates displayed in the top corner of the screen to judge when to release the mouse button.

Give the Shape – New origin command again and select the bottom left hand corner of the model to be the origin once again.
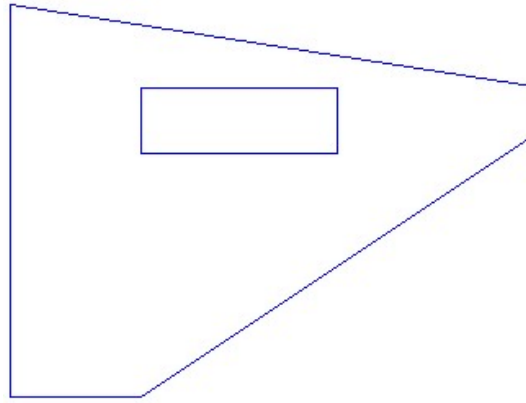
Click the fillet icon and use a click-and-drag operation to insert a radius of 3mm on all four corners of the small rectangle, releasing the mouse when the radius indicated in the radius box is correct. Now we will convert the inclined line at the top into a circular arc. Use the pull-down menus to give the command Shape – Line to arc. Click the inclined line at the top, and enter the radius of curvature to be 200. The arc is not yet drawn, since there are two possible arcs that have the required end points and radius. The program is waiting for you to click a point on the screen to indicate the side of the line on which the centre of curvature lies. Click with the mouse any point in the window to the lower left of the line. The required arc is drawn.
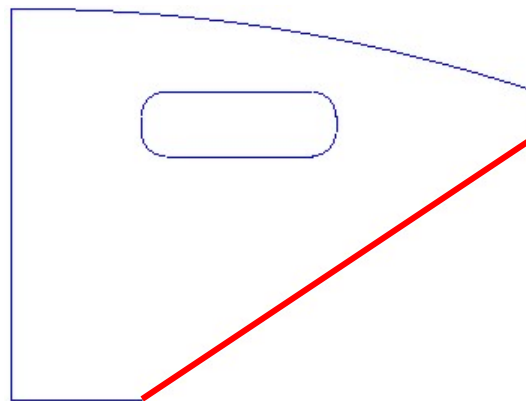
To complete the geometry of this profiled bracket, we will use a B-spline. Convert the line highlighted in red to a spline. You can do this by giving the command Shape – Line to spline. Then pick the line, which becomes highlighted. You are now ready to define the 'control points' defining the spline. For those unfamiliar with the operation of B-splines, it will become clear in a moment how these control points shape the line. Using the (X,Y) coordinates in the top corner of the screen as a guide, follow these steps:

1. Position the cursor at (17,22) and click the left mouse button
2. Position the cursor at (44,22) and click the left mouse button
3. Position the cursor at (48,32) and **double-click** the left mouse button

The double click indicates the last control point. The spline is drawn. Notice that it does not pass through the control points you defined, but is shaped by them. This will become clearer when we will modify the spline by moving its control points.
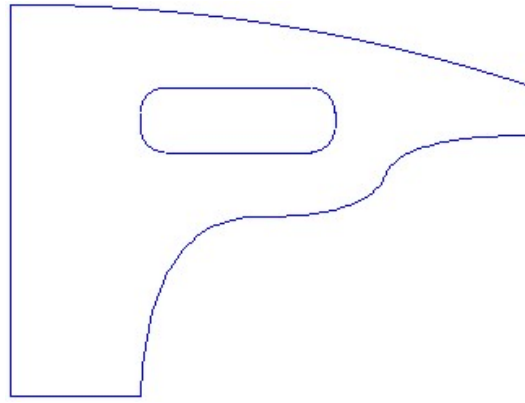
Define a pressure of 600 in the x-direction on the short edge at the right hand side of the bracket, and a pressure of 100 in the y-direction on the top of the hole. We will complete the model with displacement constraints, but this time apply them as symmetry/rollers conditions. This boundary condition constrains an edge against displacement in the direction perpendicular to the line, and this can model either a plane of symmetry or a 'rollers' constraint.

To define the symmetry/rollers constraints, click the symmetry/rollers icon ![SYM] and then select the vertical edge on the left. This applies the condition without the need to specify the value of displacement, as we needed to before, and the rollers are displayed. Select also the horizontal edge at the bottom and the rollers condition is applied there also.
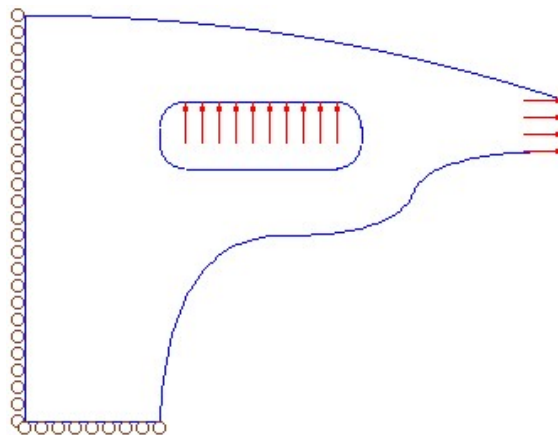
Before analysing the model, check the values of the pressure loads that have been applied to the lines. Use the pull-down menus to give the command Loading – Check. The values of the loads are written over the lines on which they are applied. (If you had defined some loads in terms of the force applied, then blue lettering would denote the force values). When you are happy that the loads have been applied correctly, click the right mouse button to redisplay the geometry in the usual fashion. Initiate the analysis now with the Analysis icon ![Go], and once the analysis is complete use the contour plot icon ![contour] to generate a set of Von Mises stress contours.
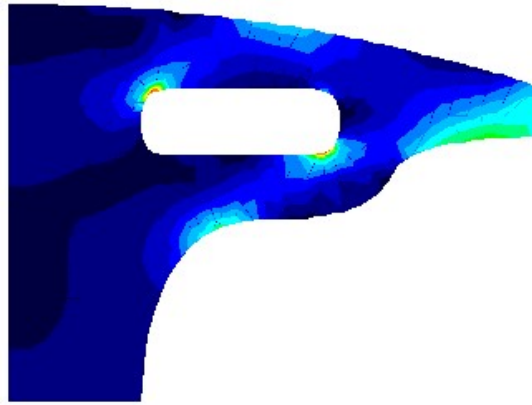
Now we will create a report describing the analysis. Give the command File – Report file – New report file, and you will see a dialog box appear as follows:

The report file directory and author's name will be taken from your personal settings file if you have one. Enter all fields, replacing the '*' in the Report file name by a name of your choice (leave the extension '.htm' to indicate that it will be an html file) and defining the 'Part name' as *bracket*.

Now select any results plot you choose. This may be a deformed shape, contour plot, xy-graph plot or internal line plot. When the display of the results plot is on the screen, give the command File – Report file – Add current results plot.

The icon has the same effect as the command File – Report file – Add current results plot.

Select another results plot you want to include in your report, and select this icon. Repeat for as many results plots as you choose.

Finally, give the command File – Report file – Close report file. You will be prompted in a dialog box to enter your text that will form the 'Conclusions and Recommendations' section at the end of your report. Type a few words in the box provided and press 'OK' to close the report file. We will view the file shortly.

To complete the tutorial, display a contour plot of some sort (setting up the re-analysis option) and then the command Shape – Modify spline. Click the spline and you will see the control points displayed. Click and drag one of the control points, and see how the spline geometry follows the control points. When you release the mouse button the re-analysis will be performed and a new set of stresses displayed.

Occasionally, if the program determines that a re-analysis will not be sufficiently accurate, it may perform a full analysis instead. If this is the case, you will see the normal full analysis progress bar appear in the top right hand corner of the screen as the analysis is performed. This is quite normal.

Now leave Concept Analyst and view the report file you have created. The easiest way to do this is to find the file you created in an Explorer window and double click on it to view the file.

Close the file, and in the Explorer window right click on the html file icon, selecting Open with… from the pull down menu. Select 'Notepad' as the program to use, and a Notepad window will open up showing the text format of the html file. Scroll down the file and you will notice the text of your report embedded amongst some html tags. You are able to edit the contents of your file like this to enter any more specific information that the automated report has not put in for your model.

This completes tutorial session 4.

6

Session 5

In this session we introduce the non-linear pin/hole contact analysis feature. We will analyse a plate containing a circular hole, within which a perfectly fitting pin exerts a force. A coefficient of friction of 0.2 will apply.

First click the rectangle icon and click and drag a rectangle shape on the screen. Double right click next to the rectangle boundary and define this to be a rectangle of size 100 in x and 80 in y.

Click the fillet icon to create fillets. Place the cursor over the top right corner of the rectangle and use a click-and-drag operation to create a fillet[1] of radius 20. Remember the dragging can be in any diagonal direction. If you have a sensitive mouse, remember you can always create the fillet to any arbitrary radius and then use the double right click operation with the cursor next to the fillet to type in the exact radius you want. Create another fillet of radius 20 at the bottom right corner and the geometry should look like the below.

---

[1] Strictly as this is an exterior corner we should not call this a fillet, but this command is helpful in creating an arc at any corner. In this case we are using it to generate an exterior corner arc.
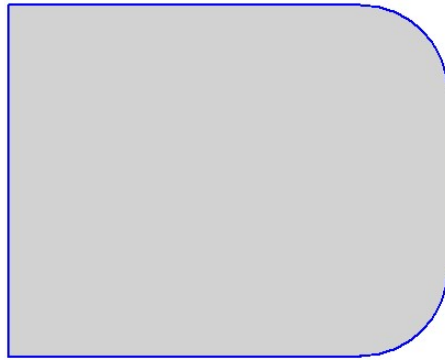
The next step is to create a circular hole of centre (60,40) and radius 15. It is easiest to do this by clicking the circle icon , using a click and drag operation to create a circle anywhere you like on the screen, and then using the double right click operation (with the cursor next to the circle boundary) to type in the exact geometry you want.

Now apply displacement constraints using the icon , clicking on the line on the left side to prescribe a displacement of zero in both the x- and y-directions. Now your model should look like the below.
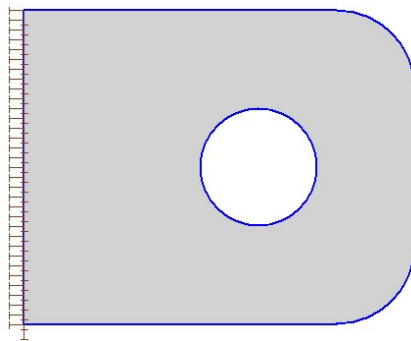

In order to do a pin/hole contact analysis we need to create a pin in the hole. Do this by giving the command Shape – Pin in hole, then clicking inside the hole. When prompted, give the coefficient of friction as 0.2. Notice the hole has been shaded now there is material contained inside. Both the pin and hole have the same material properties of the default material, so the colour is the same. Give the command Loading – Pin load, and apply loads of 1000 in the x-direction and 500 in the y-direction.
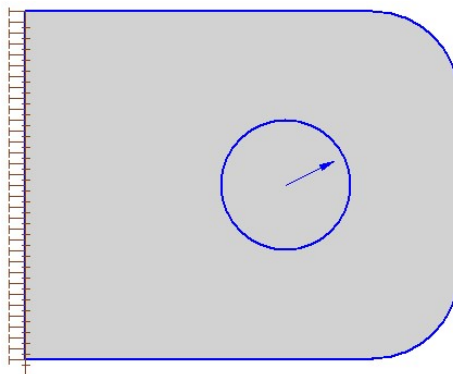
The model is now ready for analysis. Initiate the analysis with the icon [Go]. The progress bar in the top right corner of the screen will show that several analysis runs are being performed; this is because the non-linear analysis requires iteration to converge to the solution having the correct contact area and contact stress distribution.

When the analysis is complete plot a deformed shape using the icon [icon]. It should show the plate extending and bending upwards under the action of the pin load, and also some separation between the pin and hole developing on the left side of the pin.
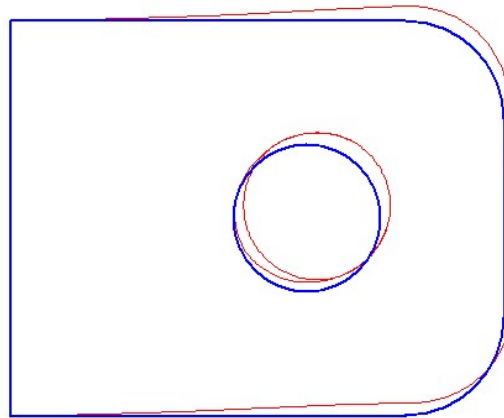
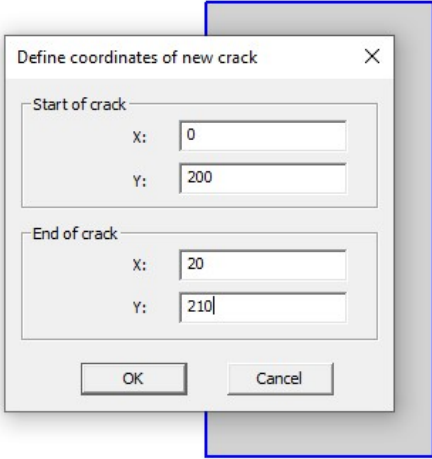The contour plot [icon] of von Mises stress should look like this.
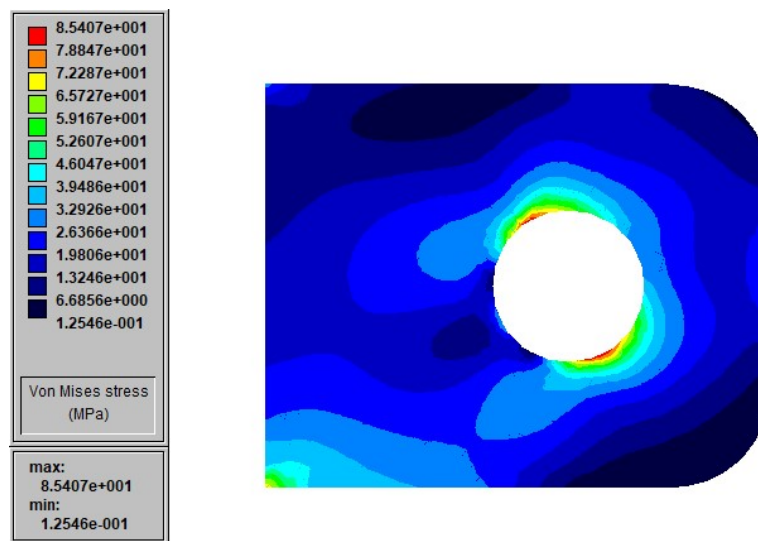
(Here we have used the report quality contours – you can turn this on in the Edit – Preferences… dialog box.) Notice that contours are not displayed inside the pin material. This is because the force is applied inside Concept Analyst as a point force at the centre of the pin. This would lead to unrealistic contours locally around the centre of the pin as they approach what would become an infinite stress at the centre.

7

Session 6

We will define first a rectangular plate with an inclined edge crack, calculate the stress intensity factors and then model some fatigue crack growth.

Start by using the rectangle icon ▢, and click-and-drag on the screen to create a rectangle. Now double-right-click with the cursor near the rectangle boundary and enter 200 and 400 as the horizontal and vertical dimensions of the rectangle. Click the crack icon ⊢, then place the cursor near the middle of the left edge of the plate and double-right-click. Enter the following crack geometry:

Define coordinates of new crack	✕
Start of crack	
X:	0
Y:	200
End of crack	
X:	20
Y:	210
OK	Cancel

A crack will be generated. Note we could have sketched the crack using a click-and-drag operation but chose on this occasion to enter the coordinates exactly using the double right click. If you are sketching, the program interprets a cursor click *close to* the boundary to be *on the boundary* so an edge crack is defined.
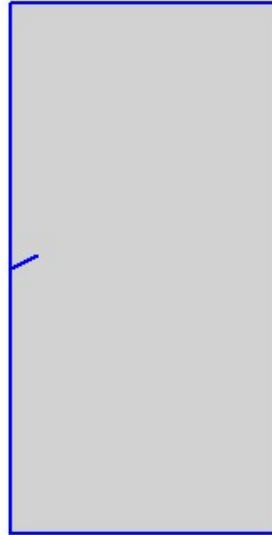
Now click the icon for distributed loads , then click the line at the top of the plate and define a y-direction distributed load of 100. Then click the constraints icon , click the line at the bottom of the plate and let the displacement in both x- and y-directions be zero.
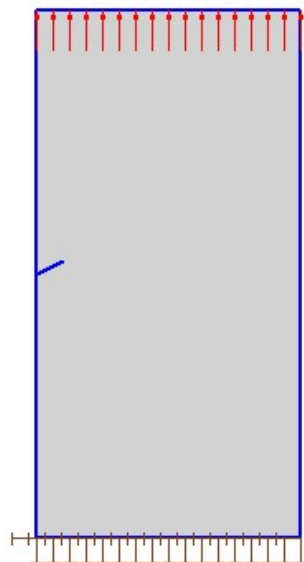

The model is ready to run, so initiate the analysis using the icon and specify the "No crack growth" option.

We can postprocess this first by displaying the stress intensity factors, K_I and K_{II} at the crack tip. These may be used to assess the stability of the crack, especially by making a comparison against the fracture toughness, a material property that can be

thought of as a limiting value of (typically) the mode I stress intensity factor K_I. Since we have not specified any material we will be using the defaulat material properties as read from the CAMaterials file – if you are using the file distributed with the software this will be a mild steel with a fracture toughness of 200 MPa $m^{1/2}$.

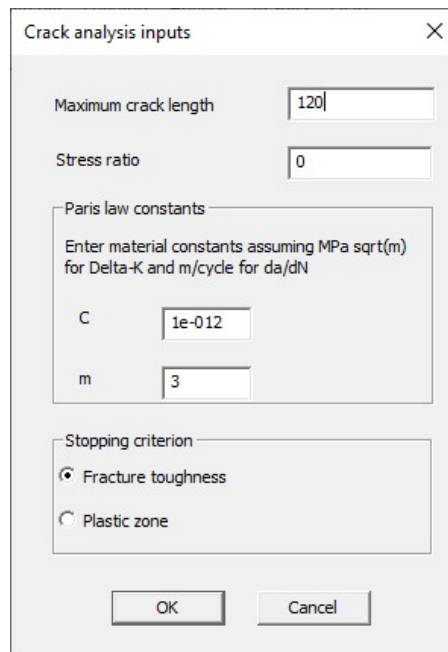So now display the stress intensity factors by giving the command Results – Stress intensity factors.

Stress intensity factors in MPa $\sqrt{m}$

K1: 26.744
K2: 2.468
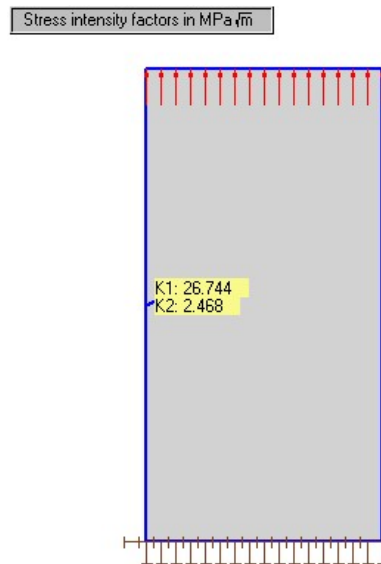
The units used for this display are always MPa $m^{1/2}$ or ksi $in^{1/2}$. Since we are using the a metric unit set of MPa, mm, N, on this occasion we see K_I and K_{II} in MPa $m^{1/2}$. The deformed plot can be displayed and shows the crack opening.
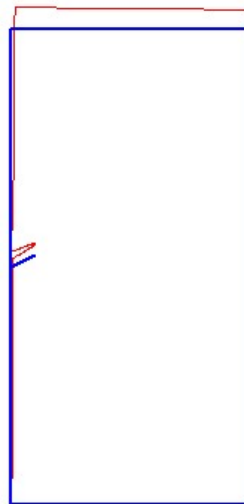
We will now model some fatigue crack growth. Initiate the analysis again with the icon , but this time selection the option for "Crack growth". You are prompted for information to be used in the crack growth simulation.

Enter 120 as the maximum crack length with a stress ratio of 0, and we will go with the default material properties (coefficients C and m) which will be used in the Paris

law that relates crack growth to load cycles. We will use "Fracture toughness" as the stopping criterion.

On pressing OK, the analysis proceeds. The progress box in the top right corner of the screen shows that several Concept Analyst runs are being performed as the crack growth is simulated. Cracks propagate in increments, and there are multiple runs at each increment to converge to the correct crack propagation path. For this analysis the run ends when the fracture toughness is reached and the crack path is displayed.
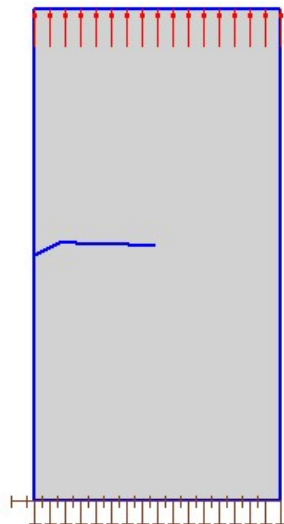
One of the key results of interest in a crack growth study is the growth of stress intensity factors with crack length. Also, engineers are interested in how the crack growth relatesd to the number of load cycles in order to make assessments of the remaining life of the component once the crack has grown to the initial length we specified. All this information is in the output file for which the default name is CAResults.out.

At the end of the file CAResults.out for this example we see the crack data:

```
Crack ID : 0            Tip : 0
===================================================================================================
  Crack increments   Crack length      TipX          TipY      KI (MPa root m)  KII (MPa root m)  Load cycles
         0            22.360674      20.000000     210.000000      26.983454         7.172400
         1            29.414028      27.053354     209.998795      35.808950        -0.731259        235355
         2            36.467382      34.100883     210.285355      46.871058         6.403032        338812
         3            43.520737      40.981857     208.735519      53.138853       -10.240149        395522
         4            50.574091      47.971027     209.684891      59.799875        25.722392        433074
         5            57.627445      55.012127     209.269272      72.565880        -2.083990        456666
         6            64.680799      62.059166     208.970886      84.331174        -1.375177        471441
         7            71.734154      69.106964     208.691025      98.364739        -1.700460        480805
         8            78.787508      76.155785     208.438217     115.511441         0.568689        486647
         9            85.840862      83.191467     207.939240     136.726587        -4.171229        490214
        10            92.894217      90.244476     207.869339     164.218777         3.578070        492319
        11            99.757365      97.097839     207.502991     200.078929        -0.468457        493477
```

This emphasises the fact that K_I has reached the fracture toughness as a stopping criterion. The maximum crack length we specified was not reached. We finally plot the deformed shape:
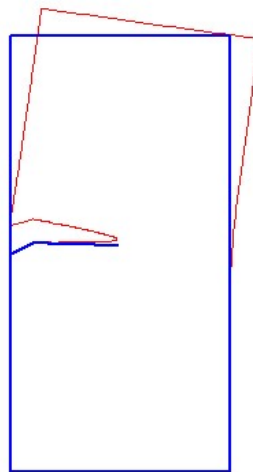
As a second example we will use the feature to generate a crack automatically as the location of largest maximum principal stress. We will assume that by now you are familiar with defining basic geometry, loads and displacement constraints. So open a new model using the icon and use the rectangle and circle commands with click and drag to sketch a rectangular plate containing a circular hole. Apply a load using to the right hand side and using constrain the left side against x-displacement and the bottom against y-displacement. You should see something like this.
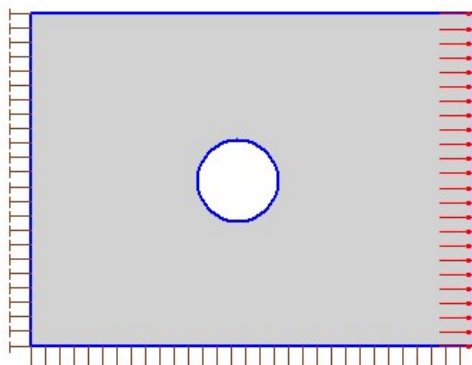
Run the analysis and display contours of the results. In the figure below we show contours of maximum principal stress.
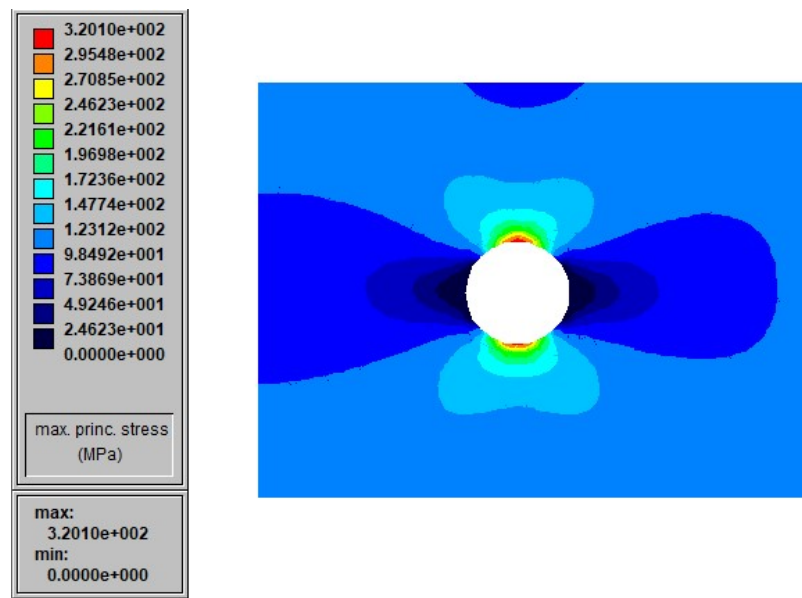
Now, with the contour plot still displayed on the screen, click the icon to create a crack at the location of largest maximum principal stress. You are prompted for an initial crack length (the default value is based on the local element size but you can enter your preferred crack length). The analysis is done automatically and the results displayed. In this case we have selected the "No crack growth" option and are displaying stress intensity factors.