Joint Analysis

This tutorial looks at different ways of analysing bolted joints within Pro Mechanica. This is an important technique as correct analysis may have significant effects on the results for the whole part being analysed. The tutorial uses a simple bracket which can be downloaded from http://www.staffs.ac.uk/~entdgc/WildfireDocs/tutorials.htm under the bracket link.

Bonded Analysis

As a base point for our comparison of different methods of simulating joints we will analyse the bracket as though it was bonded (glued) to the wall (if you did the Introduction to Mechanica tutorial you should already have done this as a further exercise). Even this is a simplification and if you wanted to correctly analyse a bonded joint you would approach it differently to account for bond flexibility and other factors.

You are now ready to start the analysis process so load the part into Pro Engineer.

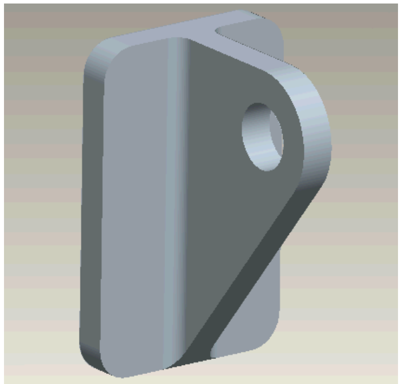


Figure 1 : The Bracket for Analysis

Choose APPLICATION > MECHANICA now to take your model into analysis. Click OK on the box notifying you of the units of your model. From the MODEL TYPE dialog choose STRUCTURE and OK.

As always the first step for this model is to define the constraints. We are going to roughly simulate the bracket being bonded to a wall so the back face needs to be fixed. INSERT > DISPLACEMENT (or you could just pick the icon). Click on below Surface(s) in the constraint dialog then pick the back surface of the bracket then OK to return to the constraint dialog. You have picked one surface to constrain and the & symbols show what movements are restricted – they all are so this surface is fully constrained. Click OK to finish.

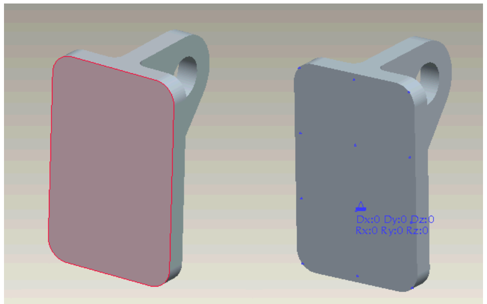


Figure 2 : Constraint Surface

Next define the load on the bracket. Choose INSERT > FORCE/MOMENT LOAD or pick the icon to apply a load over a surface. Click on below Surface(s) in the Force/Moment dialog then pick the surface highlighted in Figure 3 then OK to return to the Force/Moment dialog. Type a value of -10000 in the Y field below Force. Press PREVIEW – the arrows should point the same way as in Figure 3. Click OK in the Force/Moment dialog to finish.

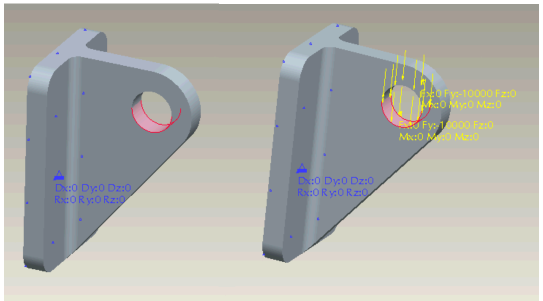


Figure 3 : Load Surface

The final definition for this analysis is the material. Choose PROPERTIES > MATERIALS and the MATERIALS dialog will appear. Scroll down the materials library in the Materials dialog to Find STEEL and double click on it to transfer it to this model. Press ASSIGN > PART and click on the bracket and OK to assign the material. CLOSE the material dialog.

That's it you are ready to run an analysis. Choose ANALYSIS > MECHANICA ANALYSES/STUDIES and in the dialog that appears choose FILE > NEW STATIC and type the name BONDED and press OK. Choose the icon to run this analysis choosing yes for error detection. Press to watch the report of the analysis as it runs. After a few seconds (longer on a slower machine!) the report should state RUN COMPLETED. Close the REPORT dialog and the ANALYSES dialog. After the analysis completes choose ANALYSIS > RESULTS menu. The main graphics window will go blank and the menus and icons will all change. Choose INSERT > RESULT WINDOW or the icon. In the RESULT WINDOW DEFINITION dialog that appears press and click (not double click) on the folder which is the same name as the analysis that is BONDED. Make sure all the options are the same as in Figure 4 then click OK AND SHOW.

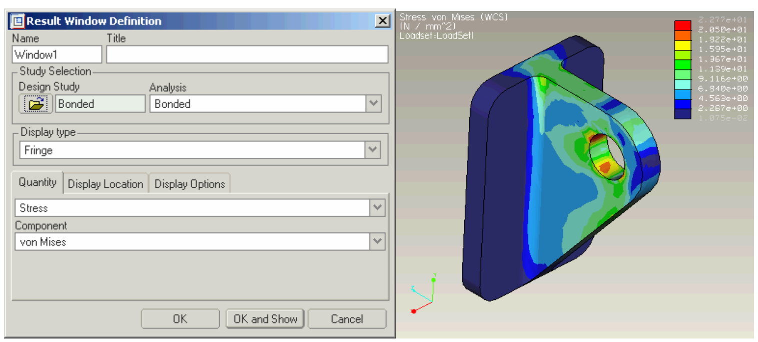


Figure 4 : Results Definition

That's the first analysis complete. Just note the stress distribution in the bracket for now and then try the next method. Choose FILE > EXIT RESULTS before continuing.

Creating the Holes

Now let's try to analyse a bolted joint – for which we need some holes! Modelling is done outside of Pro Mechanica so choose APPLICATION > STANDARD. Next choose INSERT > EXTRUDE from the menu. You should see a new toolbar appear like the one in Figure 5. This is called the dashboard and contains all of the options for the type of feature you are creating.

Figure 5 : The Extrude Dashboard

To start creating this feature choose PLACEMENT > DEFINE in the dashboard and the Section dialog appears. Notice that this dialog has many fields but the sketch plane option is highlighted in pale yellow awaiting your input. The sketch plane is a flat surface onto which you will draw your shape. Choose the back surface of the bracket (the one you constrained in Figure 2) by clicking on it in the graphics window. The other fields in the Shape dialog are filled in automatically so you don't need to worry about them at the moment – just click on the SKETCH button.

The graphics screen will change to a black background looking directly on to the sketch plane, and the drawing icons described will appear. You can CLOSE the References dialog.

You are now ready to use sketcher. Choose SKETCH > CIRCLE > CONCENTRIC and draw the circles with two clicks – the first click on the radius in the corner of the bracket, the second click to determine the size as shown in Figure 6 – press the middle mouse to finish drawing each circle. You should be able to get all four circles to lock on to the same size and showing an R1 symbol.

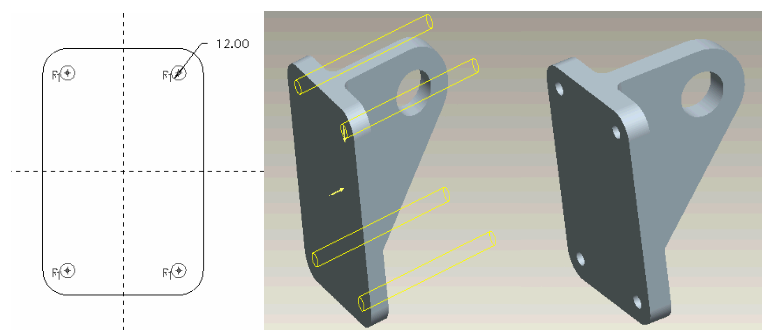


Figure 6 : Sketch for Four Holes

Your window should now look like **Figure 6** but the diameter of the circle will be different. To set the size of the circle to the correct value, choose the selection tool and double click on the dimension and type in the required value of 12. To end sketching choose and click OK in the Section dialog. To complete this first feature at the dashboard choose the thru all option and to remove (rather than add) material if the extrusion is going in the wrong direction press (see Figure 6b). Click the green tick to finish. You should have four bolt holes.

Edge Constraint

Now we have to modify the analysis to take account of these holes. The load and material will stay the same but the constraints will alter.

Choose APPLICATION > MECHANICA now to take your model back into analysis. From the MODEL TYPE dialog choose STRUCTURE and OK.

We are going to simulate a bolted joint by just holding the edges of the holes in position. We could delete the surface constraint we applied earlier and a new one but we can learn another technique. Choose INSERT > DISPLACEMENT (or you could just pick the icon). On the dialog there is a NEW button next to the MEMBER OF SET FIELD. Click on this button to create a new set of constraints by default called CONSTRAINTSET2. Click OK to return to the CONSTRAINT dialog. Below the word REFERENCES it will say SURFACE(S) – change this to EDGES(S)/CURVE(S) then click on below then pick the back edges of the holes in the bracket holding the CTRL key. When you have picked all 8 edges click OK to return to the constraint dialog. Click OK to finish defining this constraint.

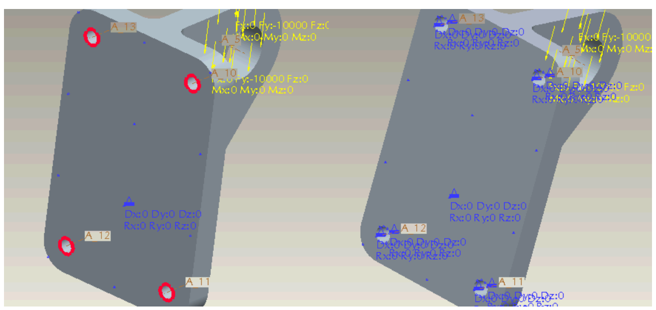


Figure 7 : Constraint Edges

The new constraints have been defined. They are stored in a separate set from the original constraint. You are ready to run an analysis. Choose ANALYSIS > MECHANICA ANALYSES/STUDIES and in the dialog choose EDIT > COPY and press OK. You now have two analyses which are currently identical except for their name. Choose EDIT > ANALYSIS/STUDY to edit the copy. Change the name of the analysis to EDGES. Below CONSTRAINTS choose CONSTRAINTSET2 so that the analysis will use only the edge constraints just defined then click OK. Choose the icon to run this new analysis choosing yes for error detection. Press to watch the report of the analysis as it runs. After a few seconds (longer on a slower machine!) the report should state RUN COMPLETED. Close the REPORT dialog and the ANALYSES dialog. After the analysis completes choose ANALYSIS > RESULTS menu. The main graphics window will go blank and the menus and icons will all change. Choose INSERT > RESULT WINDOW or the icon. In the RESULT WINDOW DEFINITION dialog that appears press and click (not double click) on the folder which is the same name as the analysis that is BONDED. Make sure all the options are the same as in Figure 8 then click OK AND SHOW.

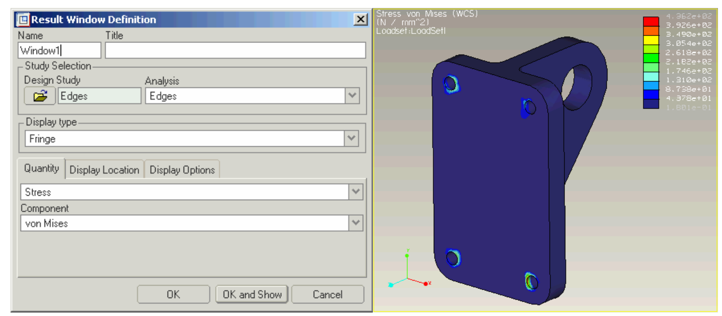
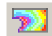


Figure 8 : Results Definition

The results are very different. There are some very high stress concentrations around the edges you constrained which are masking the stresses elsewhere in the bracket. Stress concentrations around constraints are common but can be ignored as they are not realistic. If you look at the stress legend it goes from (about) 1.8e-1 to 4.3e2 whereas in the BONDED analysis it went from (about) 1.07e-2 to 2.2e1. To make an accurate comparison we can show the two analyses side by side with the same legend values.

Choose INSERT > RESULT WINDOW or the  icon. In the RESULTS WINDOW DEFINITION dialog that appears press  and click (not double click) on the folder for the BONDED analysis then click OK AND SHOW. You will see the two analyses together. Click on each window in turn then choose FORMAT > LEGEND. Type 2 as the minimum value and 20 as the maximum value so that you can get a fair comparison.

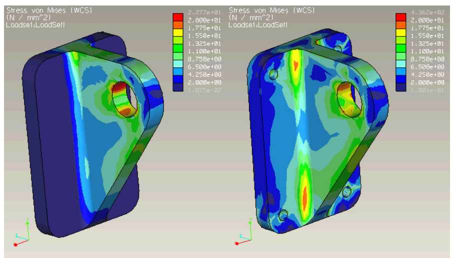


Figure 9 : Comparison 2

Choose FILE > EXIT RESULTS before continuing.

Surface Constraint

We are going to improve the simulation of a bolted joint by holding the surfaces of the holes in position. Follow the procedure for defining a new constraint as you did with the edge constraints. This time create a surface constraint (INSERT > DISPLACEMENT) and create a new constraint set (this will be CONSTRAINTSET3). Pick the internal surfaces of the holes.

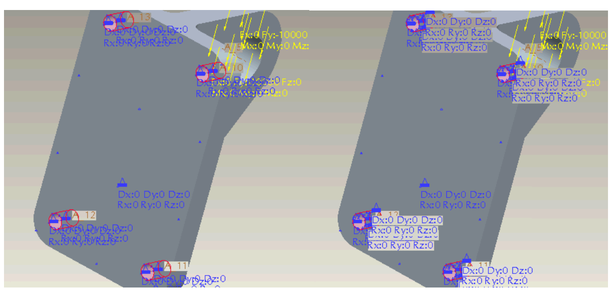


Figure 10 : Constraint Surfaces

Choose ANALYSIS > MECHANICA ANALYSES/STUDIES and EDIT >COPY a new analysis called SURFACES which uses CONSTRAINTSET3. Run the analysis and show the results of the three analyses side by side with the same legend values.

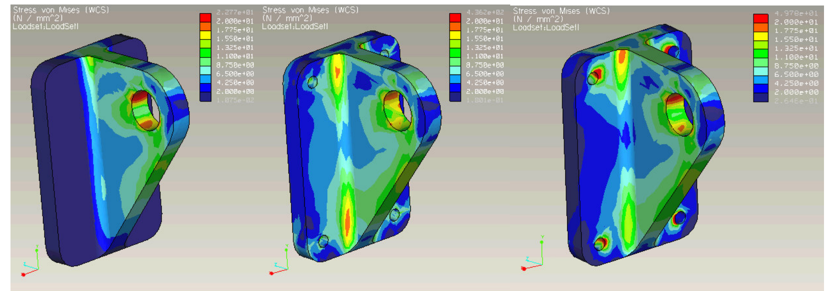


Figure 11 : Comparison 3

Accurate Analysis of Bolted Joints

In many situations where the area around the bolt is not critical the surface constraint technique would be acceptable (ignoring the stress concentrations around the holes). For greater accuracy a more complex technique is required.

To understand the technique it first important to understand how a bolted joint works. It may seem straight forward but it is not! A good source of research for this is http://www.boltscience.com/ where there is a tutorial on bolted joints.

We will need to define the area where the washer contacts the face of the bracket. Currently this is a single surface so we need to split it into what are called regions. This is done in Pro Mechanica using INSERT > VOLUME REGION > CREATE > EXTRUDE > DONE (we use volume region rather than surface region as we can split the front and back surfaces in one command). Pick the front face of the bracket as the sketch plane then OKAY > DEFAULT to enter sketcher. Draw 4 equal diameter circles concentric to the four holes. After exiting sketcher choose THRU ALL > DONE then OK. What this has done is created an imaginary extrusion. Where this extrusion passes through the bracket it has split the surfaces. This will only be visible if you are in a non shaded display.

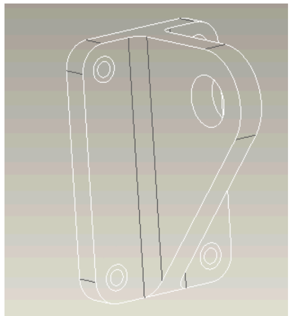


Figure 12 : Volume Regions

We will also need to create 8 datum points – 1 at each end of the four holes. Choose INSERT > MODEL DATUM > POINT > POINT. The DATUM POINT dialog appears. Click on the edge of one of the holes and a datum point is created where you pick. We want the datum point to be at the centre of the arc so in the DATUM POINT dialog click on the word ON and change it to CENTRE then click on NEW POINT. Repeat this for the 8 points. Don't forget to click on new point after each point is defined. Close the dialog with OK.

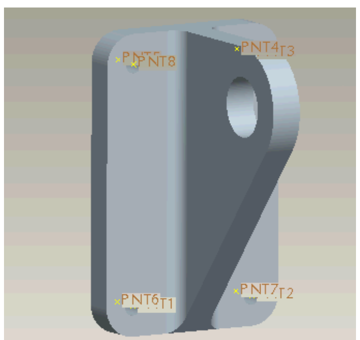


Figure 13 : Datum Points

That is all of the preliminary geometry defined. Next we will simulate the bolt with a special type of element known as a beam. This will appear like a simple line in the analysis but will have all the same properties as the bolt shank. Choose INSERT > BEAM. In the BEAM DEFINITION dialog click on below Reference(s) then pick the point at either end of one hole. Press the MORE button next to MATERIAL to set the material to STEEL. Press the MORE button next to SECTION. Choose NEW then set the type to SOLID CIRCLE with a radius of 6 (this represents the size of the bolt shank). Click all the OK buttons to finish defining this beam. Repeat it for all four holes (you won't need to define material or section as they are now the defaults).

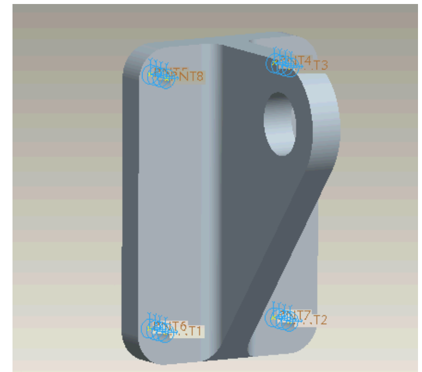


Figure 14 : Four Beams defined

Now the bolt needs to be connected to the bracket. If the bolt is designed and fitted correctly it should not move relative to the bracket so we will use rigid connections. These don't allow any movement. Choose INSERT > CONNECTION > RIGID CONNECTIONS > CREATE and in the RIGID CONNECTION dialog click on icon then pick the edges of the volume region and the adjacent datum point IN THAT ORDER. Click OK then close the dialog with OK. Repeat for each end of the four holes (8 times).

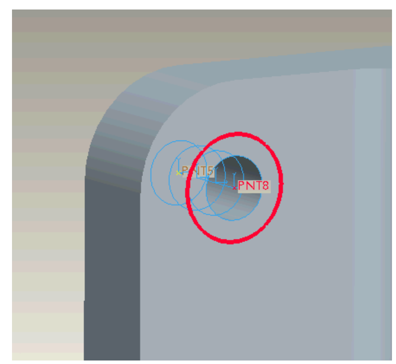


Figure 15 : Rigid Connection

Finally, all that is needed is to correctly constrain the model. Create a new DISPLACEMENT CONSTARINT in a new constraint set. Change the reference type to POINT and pick all four datum points on the back of the bracket. You should now be able to analyse this model creating a new analysis called SPIDER which use the correct constraint set. (It is called a spider connection for historical reasons).

Compare the results of all four analyses.

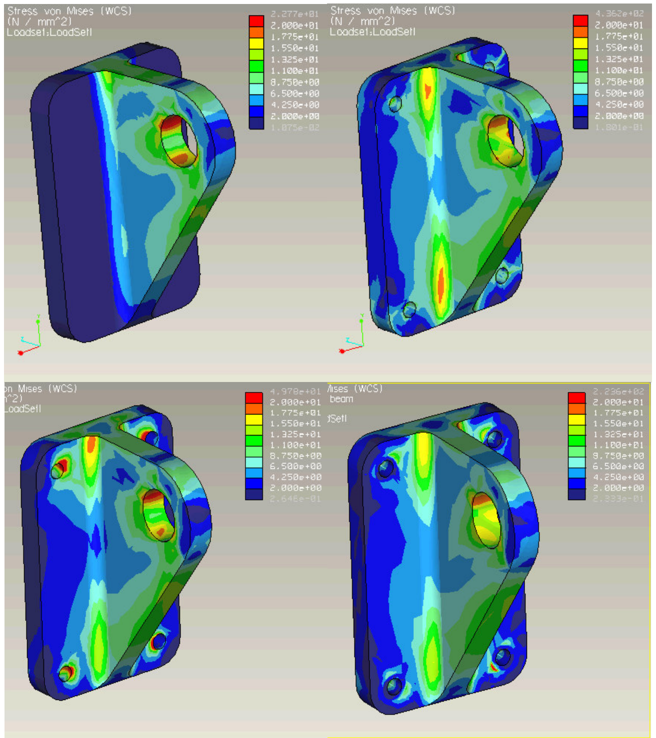


Figure 16 : Comparison 4

Review

So what should you have learnt?

- How to create various types of constraints.
- How to define volume regions.
- How to define rigid connections.
- How to compare results of an analysis.

Any problems with these? Then you should go back through the tutorial – perhaps several times – until you can complete it without any help.