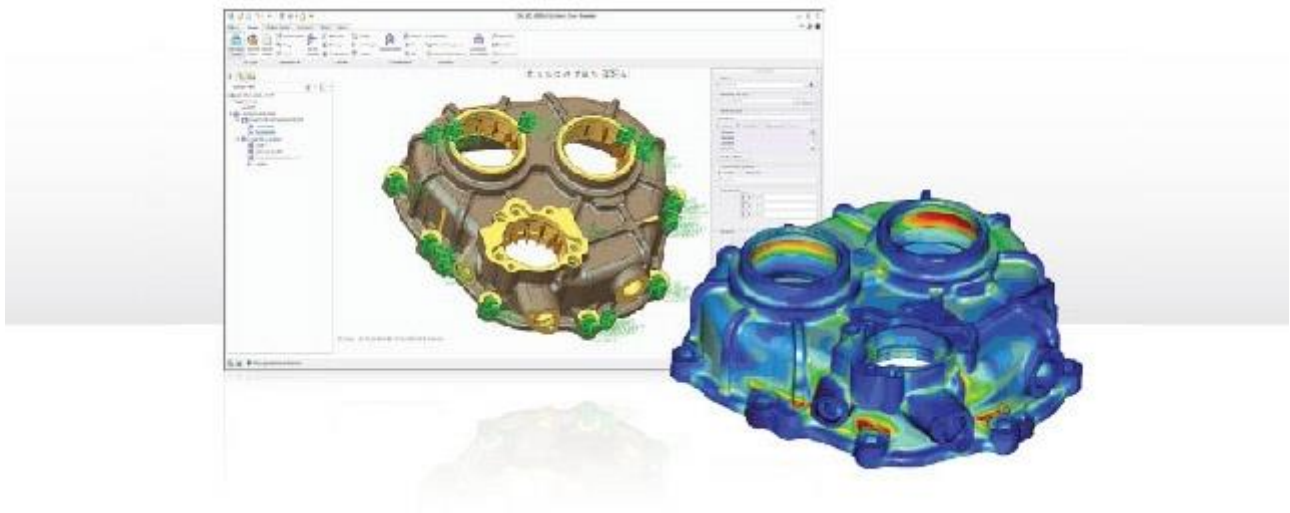


## Hands-on Training No.2

### Structural Simulation With Simulate



- **Part 1 : Introduction To Simulate**
- **Part 2 : Bolted Joint Analysis**
- **Part 3 : Shell Modeling**

## **Contents**

### ➤ **Part 1 : Introduction To Simulate**

- I. Introduction
- II. Creating a Part
- III. Starting an Analysis
- IV. Defining Constraints
- V. Defining Loads
- VI. Defining Materials
- VII. Running an Analysis
- VIII. Seeing the Results
- IX. Additional Analysis – Bending
- X. Additional Analysis – Torsion
- XI. Review
- XII. Where Next?

### ➤ **Part 2 : Joint Analysis**

- I. Introduction
- II. Bonded Analysis
- III. Creating the Holes
- IV. Edge Constraint
- V. Surface Constraint
- VI. Accurate Analysis of Bolted Joints
- VII. Review

### ➤ **Part 3 : Shell Modeling**

- I. Introduction
- II. Shell Modelling
- III. Analysing Shell Models
- IV. Review

## Part 1 Introduction To Simulate

### I. Introduction

**Simulate** can undertake different types of analysis. These are:

- **Structure** – Static loading of parts to calculate stresses. Also calculates vibrations.
- **Thermal** – Applying thermal gradients to calculate heat distribution.

This tutorial covers the structural analysis only.

### II. Creating a Part

- Before starting to learn the analysis process, you need to model the very simple part shown Figure 1. Setup the template to **en\_mmns\_part\_solid** and the **~\TP2\Part1\Bar** as working directory.

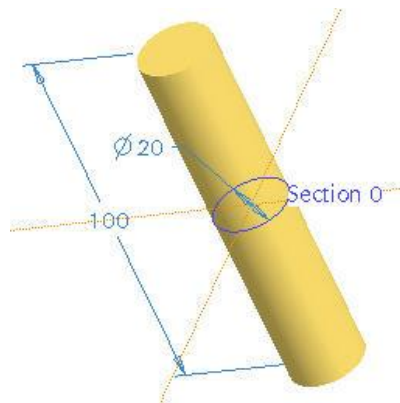


Figure 1 – The Bar part


### III. Starting an Analysis

You are now ready to start the analysis process. We can try several different load cases on this simple bar and you can compare these with manual calculations.

- Choose **Applications > Simulate** now to take your model into analysis.
- Check that the **Structure Mode** is activated.

### IV. Defining Constraints

The first step for this model is to define the constraints. Constraints determine where and how the model is held or fixed in position. We are going to apply a tensile (pulling) force to the bar so one end needs to be fixed.

- Choose **Constraints** > **Displacement** (or you could just pick the  icon). The Constraint dialog will appear.

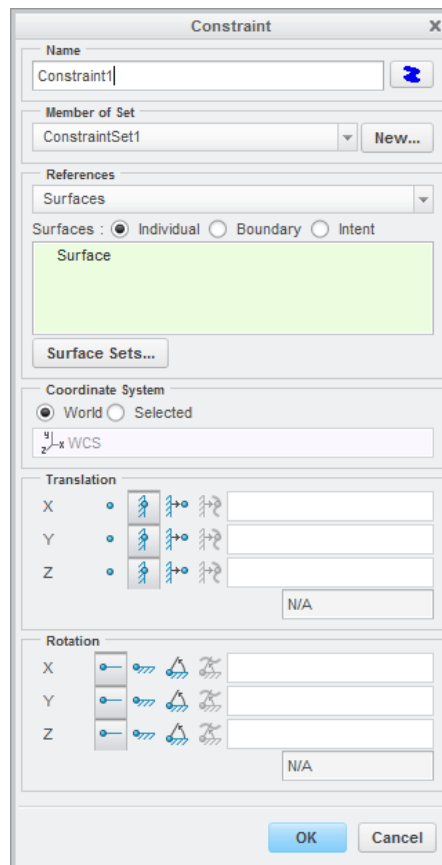



Figure 2 – The Constraint Dialog

- Make sure the References option is set to **Surface(s)**. Now pick one end of the bar. You have picked one surface to constrain and the  symbol show what translations are restricted – they all are, so this surface is fully constrained. Every model to be analyzed must be constrained at some point in all six degrees of freedom. Click **OK** in the constraint dialog to finish.

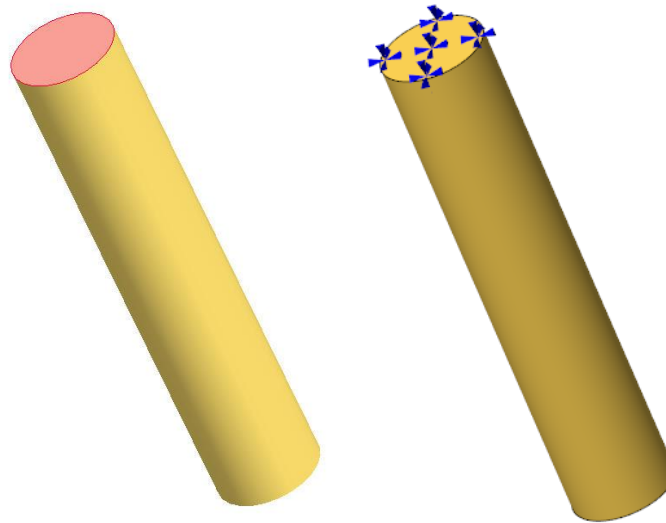



Figure 3 – Constraint Surface

## V. Defining Loads

- Definition of loads is similar to constraints. Choose **Loads > Force/Moment** or pick the  icon to apply a load over a surface. The Force/Moment Load dialog will appear.

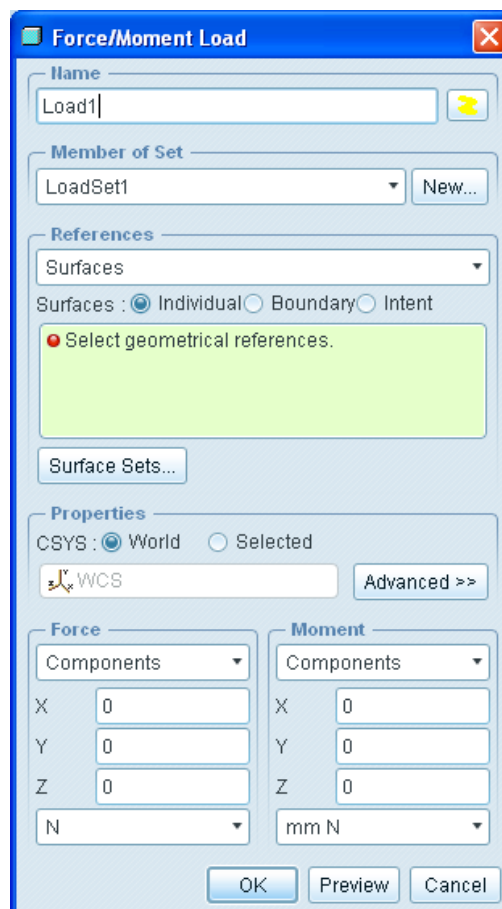


Figure 4 – Force/Moment Dialog

- Make sure the References option is set to **Surface(s)**. Pick the other end of the bar (spin the view with the middle mouse button if you need to).

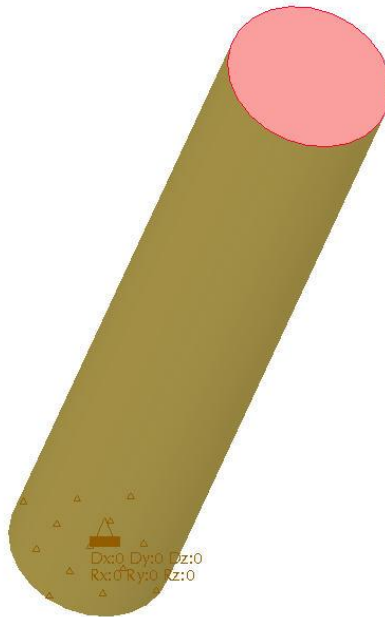


Figure 5 – Load Surface




- If you look carefully at the graphics window you should see a symbol displaying the X, Y and Z directions (if you can't see it make sure the  icon is clicked at the top of the window). This shows that for a tensile load to be applied along the bar we need to apply a load in the Y direction. So type a value of **100 000** in the **Y** field below **Force**. Press **Preview** – if the arrows point into the bar rather than out type -100000 instead. Click **OK** in the Force/Moment Load dialog to finish.



Figure 6 – Load Direction

## VI. Defining Materials

- The final definition for this analysis is to determine the material for the bar. Choose **Materials**  and the Materials dialog will appear. Scroll down the Material Directory to find **Steel** (en-steel-insa.mtl) and double click on it to transfer it to this model. If you choose **Edit** you will see the material parameters defined for steel – the most important ones are Young's Modulus and Poisson's ratio.
- You must now assign the chosen material to the part. Choose **Materials Assignment**  and the Materials Assignment dialog will appear. Click on the bar if needed and choose steel to assign the material. Close the material dialog.

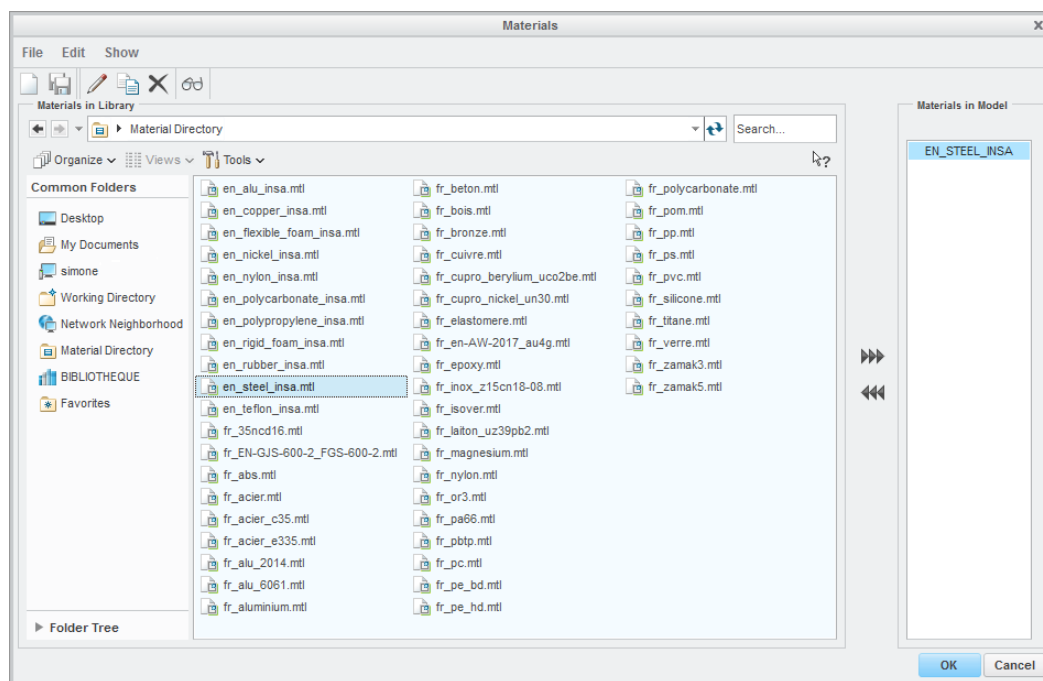





Figure 7 – Materials Dialog

## VII. Running an Analysis

That's it you are ready to run an analysis.

- Choose **Run > Analyses and Studies**  and the dialog in Figure 8 appears. From this dialog choose **File > New Static** and type the name **Bar**. Notice the method is Single-Pass Adaptive. This method is used for quick checks to ensure everything is defined correctly and to get rough results quickly. For more accurate results, you would change this to Multi-Pass Adaptive. Leave it as it is for now and **OK**. Choose the icon  to run this analysis choosing **Yes** for error detection. The Diagnostics dialog for the analysis named bar appears. After a few seconds (longer on a slower machine!) the report should state Run Completed. You can also press  to watch the report of the analysis as it runs. Close the Report dialog, Diagnostics dialog and the Analyses dialog.

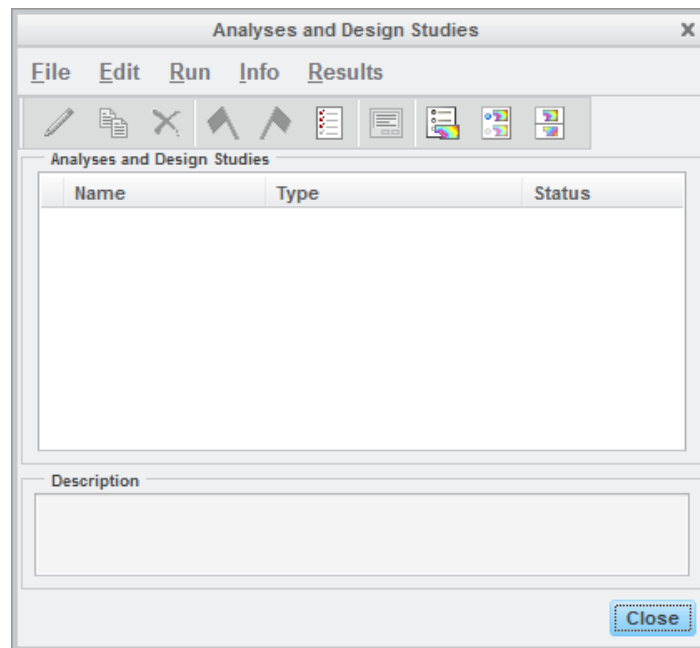



Figure 8 – Analysis and Design Studies Dialog

## VIII. Seeing the Results

Results are handled in a separate though integrated module of Creo.

- Choose **Run > Results** 

**Note** - This icon was available in the Analyses dialog as well. The main graphics window will go grey and the menus and icons will all change.

- Choose **Windows Definition > New** or the  icon. Make sure all the options are the same as in Figure 9 then click **OK and Show**.



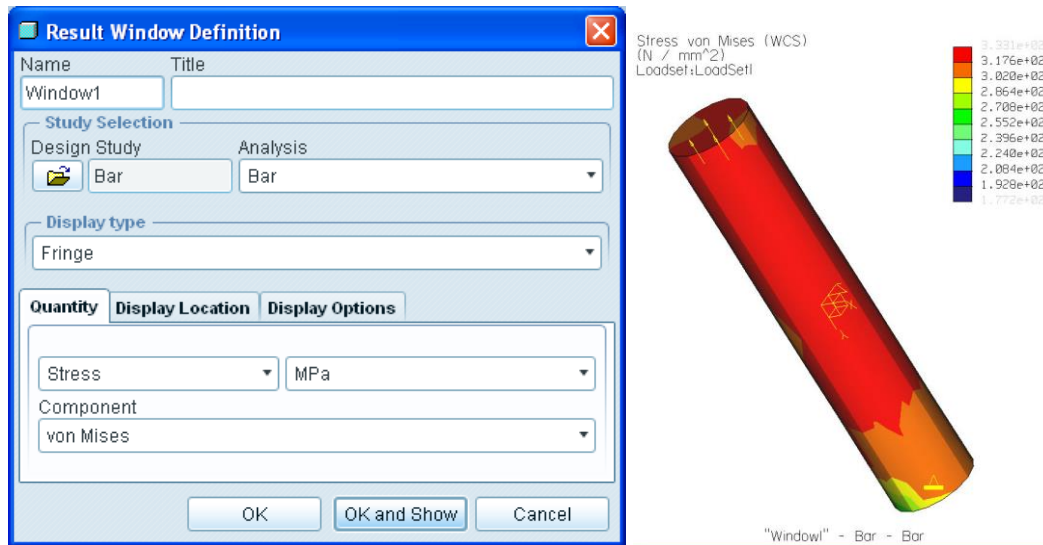



Figure 9 – Results Definition

The resultant plot shows the Von Mises stress distribution over the whole bar where the colors show the stress ranges and the values are shown on the scale to the right. The unexpected variation of stress at one end is due to being local to a constraint which can affect the result.

- Choose **Query > Dynamic Query** to get more feedback on actual values. Now as you move the cursor over the model you will get the actual value at the cursor reported in the dialog box. You will see that the majority of the model is at about 319 N/mm<sup>2</sup> (notice the units are reported at the left of main window).

**Question1** - Is this value correct?

**Answer1** - For tension stress is calculated by load/area. The cross sectional area of this bar is  $\pi \times 20^2 / 4$  or 314mm<sup>2</sup>. The load we applied was 100000 N so the normal stress should be 100000/314 or 318,5 N/mm<sup>2</sup>. Spot on - even though this was only a quick single pass adaptive check!

- What else can we show? Choose **Windows Definition > Edit**  to bring back the dialog in Figure 9. Below Quantity change Stress to **Displacement** then **OK and Show**. Again a colored plot appears with the colors relating to the amount of displacement. One end is blue with a displacement of 0 because it was constrained. The other end has stretched as the load is applied so this is shown in red with a displacement value of 0,15 mm.

**Question2** - Is this value correct?

- We can combine the display of displacement with stress in a very interesting and informative way. Choose **Windows Definition > Edit** to bring back the dialog in Figure 9. Below Quantity change Displacement back to **Stress** then on the Display Options tab tick **Deformed** and **Transparent Overlay**. **OK and Show** should now show the Von Mises stress plot on the stretched model. Close the results window with **File > Close** then **Don't Save**.

## IX. Additional Analysis – Bending

Let's try a different analysis. We will apply a bending moment to the bar rather than a tensile load.

- Pick the tensile load by clicking on the orange arrows in the main graphics window (they turn green) then choose **Edit Definition** (RMB). You should see the Force/Moment Load dialog (as in Figure 4). Change the load value to **0** in the **Y** direction and to **1000** in the **Z** direction. Check the load direction is as shown in Figure 10.

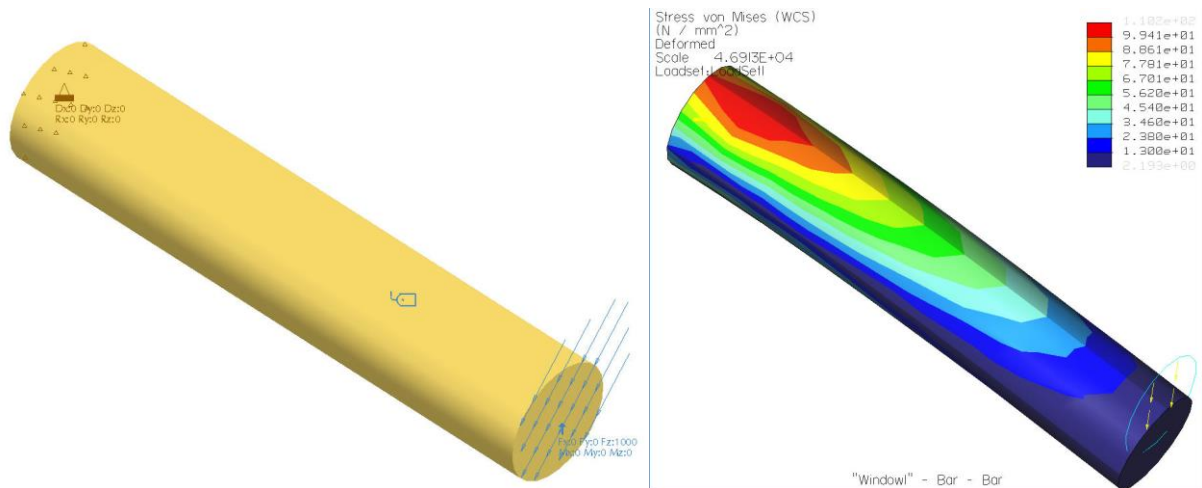


Figure 10 – Bending Moment

- Re run the analysis and view the results.

**Question3** - Note the value of the maximum stress and deflection. What should these values be?

	Analytic expression	Numerical maximum value	Simulate maximum value
Second Moment of Area for a Circular Beam			
Bending Stress			
Deflection			

- Do your Simulate results compare?

## X. Additional Analysis – Torsion

Now we will apply a torsional load to the bar rather than a bending moment load.

- Pick the bending load by clicking on the orange arrows in the main graphics window then choose **Edit Definition** (RMB). You should see the Force/Moment Load dialog (as in Figure 4). Next change the Distribution to **Total Load at Point** (located below **Advanced >>**) and then pick a point on the circular edge of the end of the bar (since we will apply a moment force the location of this

point is not critical). Enter a value of **100 000** for **Y** in the **Moment** column and **0** in all the other **Force** fields. Preview the load direction is as shown in Figure 11.

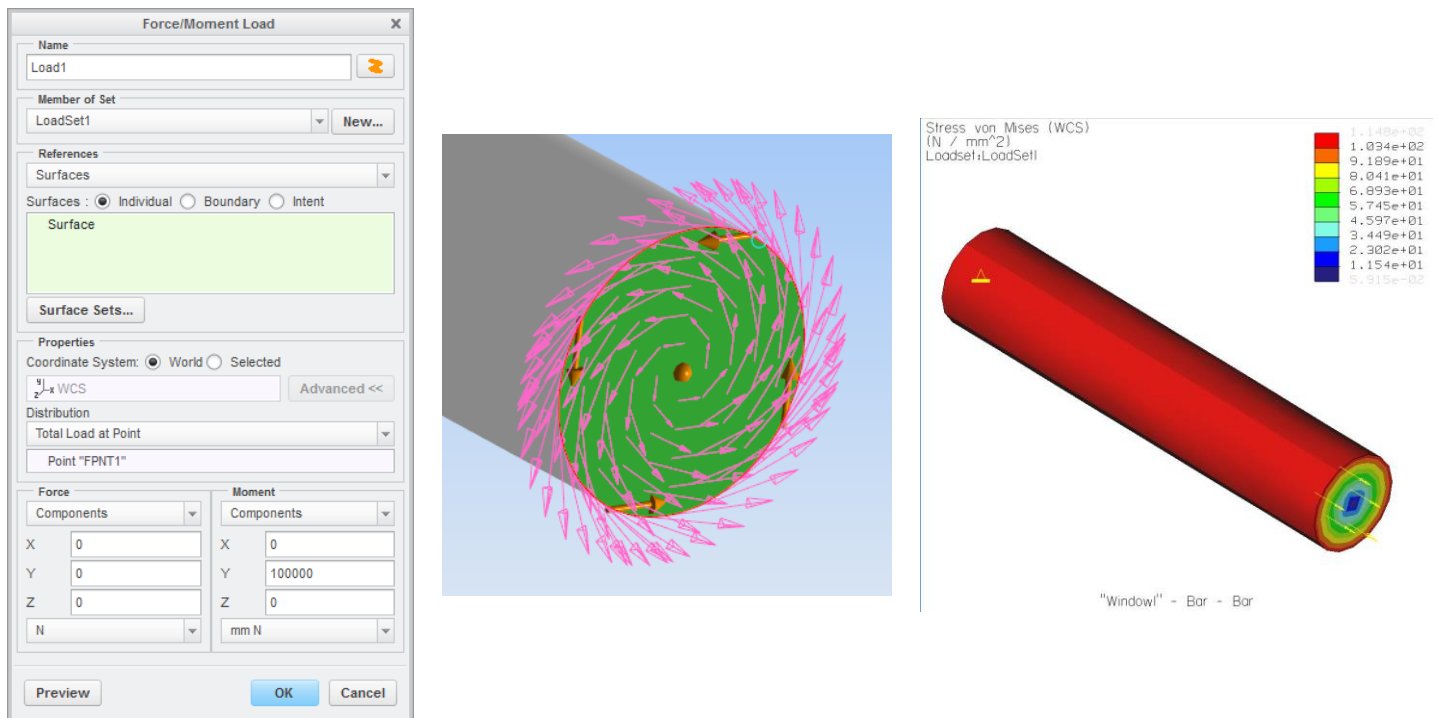


Figure 11 – Torsional Moment

Before re-run the analysis and to correctly explore a torsional load we will need to create a cylindrical coordinate system.

- Choose **Refine Model > Datum > Coordinate System**. In the Coordinate System dialog change the Type of the coordinate system to **Cylindrical** then pick the **RIGHT** and **FRONT** datums and then the end of the bar **IN THAT ORDER** – you should get the icon displayed as shown in Figure 12. **Close** the Coordinate System dialog box.

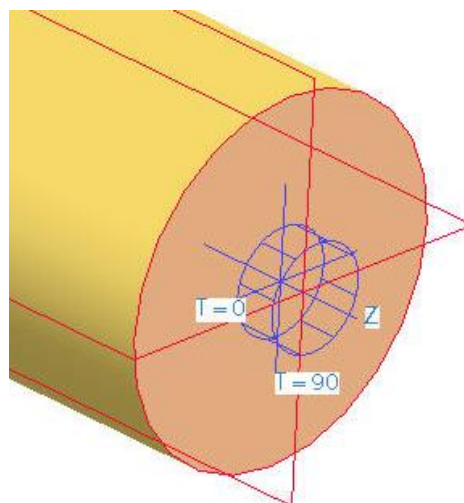



Figure 12 – Cylindrical Coordinate System

- Now re-run the analysis and view the results for maximum shear stress.

**Question4** - Note the value of the maximum stress. What should these values be?

	Analytic expression	Numerical maximum value	Simulate maximum value
Centroidal Moment of Area for a circular beam			
Torsional Stress			
Deformation			
Displacement			

- Bring back the **Result Window Definition** dialog box. **Below Quantity** change the coordinate system selected by tick on  and then pick the coordinate system you just created and **OK**. The load directions will change from X, Y and Z to R, Theta and Z. You can now display the results in this coordinate system.

## XI. Review

So what should you have learnt?

- How to start analysis.
- How to define loads, constraints and materials.
- How to run an analysis.
- How to show 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.

## XII. Where Next?

- Here is a more complicated model of a steel bracket which you can open on the [~\TP2\Part1-2\Bracket](#) directory. Try applying a **10 000 N** vertical load to the circular hole and constraining the back surface as though it were glued to a wall.

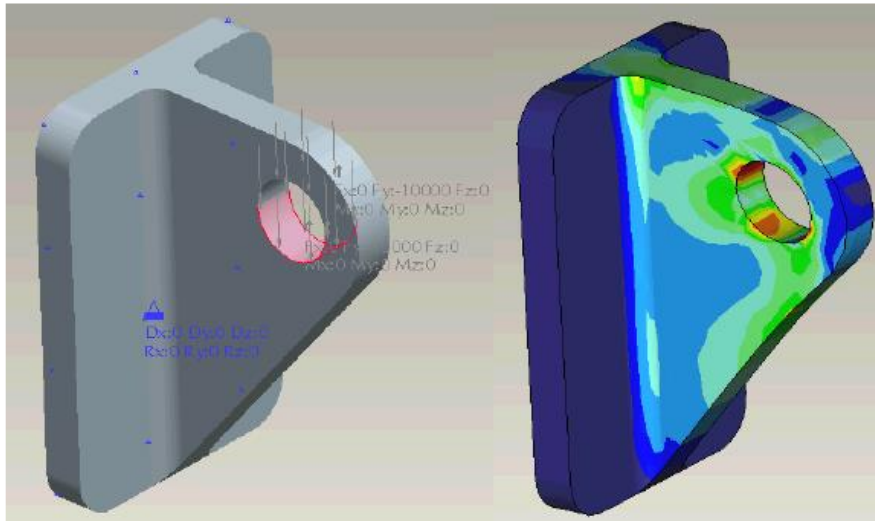


Figure 14 – F.E.A. Analysis Exercise

- Choose **Application > Simulate** now to take your model into analysis. From the Model Type dialog choose **Structure Mode**.

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.

- Fully constrain the back face of the bracket as shown in the following figure.

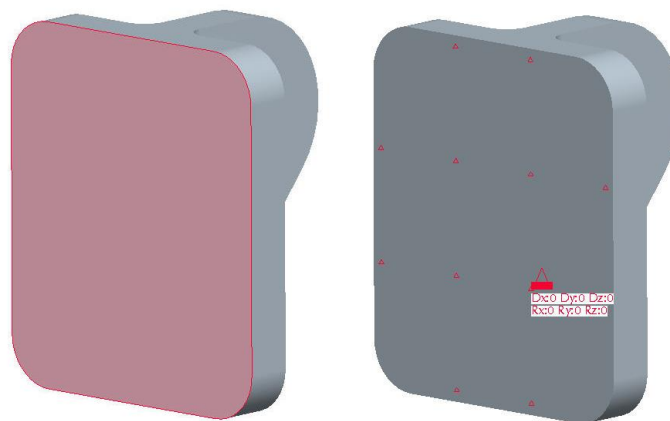



Figure 15 – Constraint Surface

- Next define the load on the bracket. **Loads > Bearing** or pick the  icon to apply a load over a surface. In the Bearing Load dialog pick the surface highlighted in Figure 16. Type a value of **-10 000** in the **Y** field below Force. Press **Preview** – the arrows should point the same way as in Figure 16. Click **OK** in the Bearing Load dialog to finish.

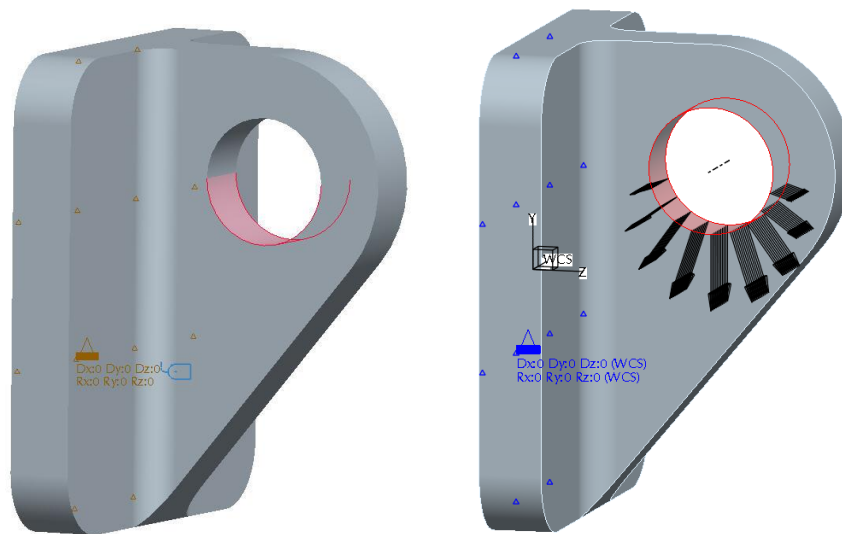


Figure 16 – Load Surface

- The final definition for this analysis is the material. So select and assign the **Steel** material to the bracket.
- That's it you are ready to run an analysis. So define a new static analysis named **Bonded** and run it.

## Part2

### **Bolded Joint Analysis**

## I. Introduction

This tutorial looks at different ways of analyzing bolted joints within Simulate. This is an important technique as correct analysis may have significant effects on the results for the whole part being analyzed. The tutorial uses the simple previous bracket.

## II. Bonded Analysis

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

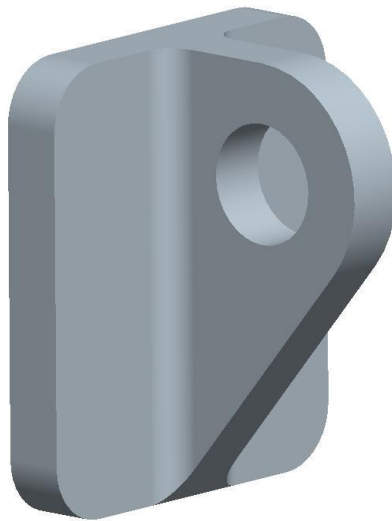



Figure 1 – The Bracket for Analysis

- Re run the analysis if necessary and view the results. After the analysis completes choose **Results > Define Result Window...**  menu. Make sure all the options are the same as in Figure 2 then click **OK And Show**.



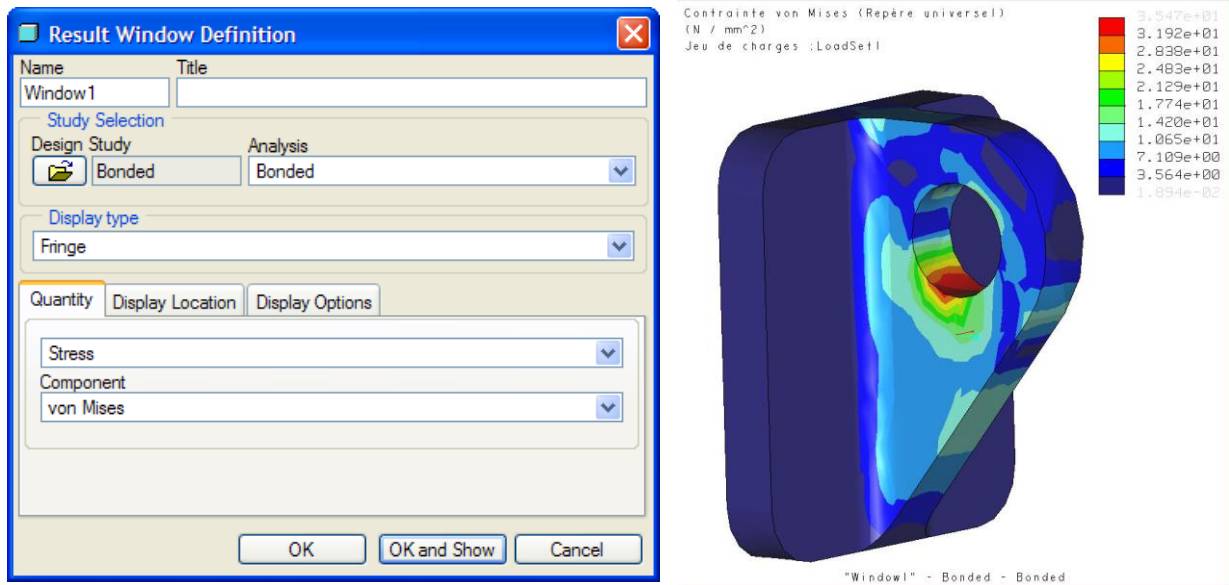


Figure 2 – Results Definition

- 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.

### III. Creating the Holes

- Now let's try to analyze a bolted joint – for which we need some holes! Modeling is done outside of Simulate so **Close** the Simulate application.
- Create four holes to lock on to the same size as shown in Figure 3. Note all of the holes are concentric with each corner of the bracket.

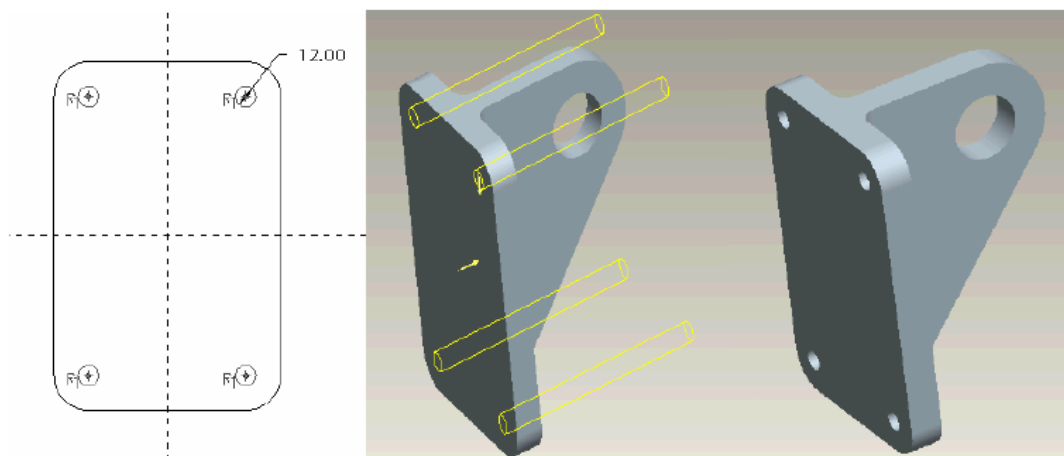


Figure 3 – Sketch for Four Holes

You should have four bolt holes.

### IV. 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 **Applications > Simulate** now to take your model back into analysis.

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 add a new one but we can learn another technique.







- Choose **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 **Edge(s)/Curve(s)** 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 finish defining this constraint.



Figure 4 – 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 **Run > Analyses and Studies**  and in the dialog choose **Edit > Duplicate**. 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 the report should state Run Completed. Close the Report dialog and the Analyses dialog. After the analysis completes choose  menu. In the Result Window definition dialog that appears if necessary press  and click (not double click) on the folder which is the same name as the analysis that is **Edges**. Make sure all the options are the same as in Figure 5 then click **OK And Show**.

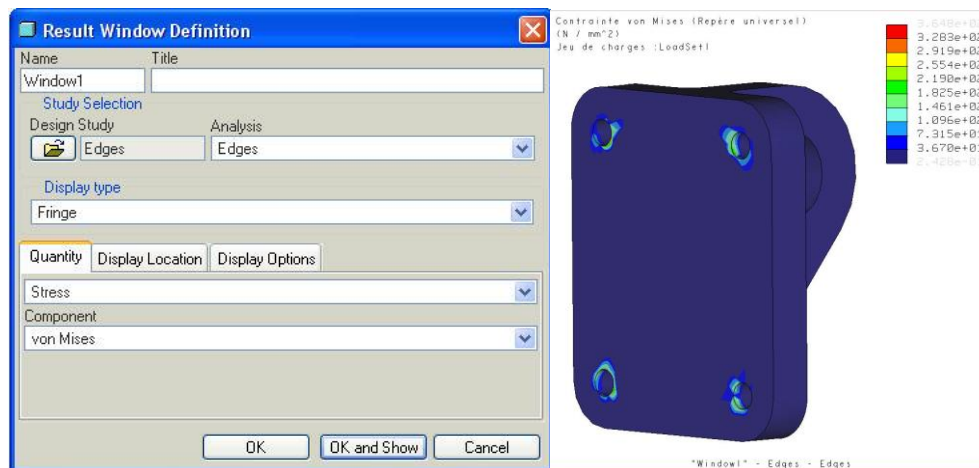



Figure 5 – Results Definition

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)  $2.10^{-1}$  MPa to  $2.10^2$  MPa whereas in the Bonded analysis it went from (about)  $4.4.10^{-2}$  MPa to  $3.6.10^1$  MPa. To make an accurate comparison we can show the two analyses side by side with the same legend values.

- Choose **Home** > **New** or the  icon. Open the **Bonded** analysis then click **OK And Show**. You will see the two analyses together. Click on each window in turn then choose **Format** > **Edit**. Type **2** as the minimum value and **20** as the maximum value so that you can get a fair comparison.

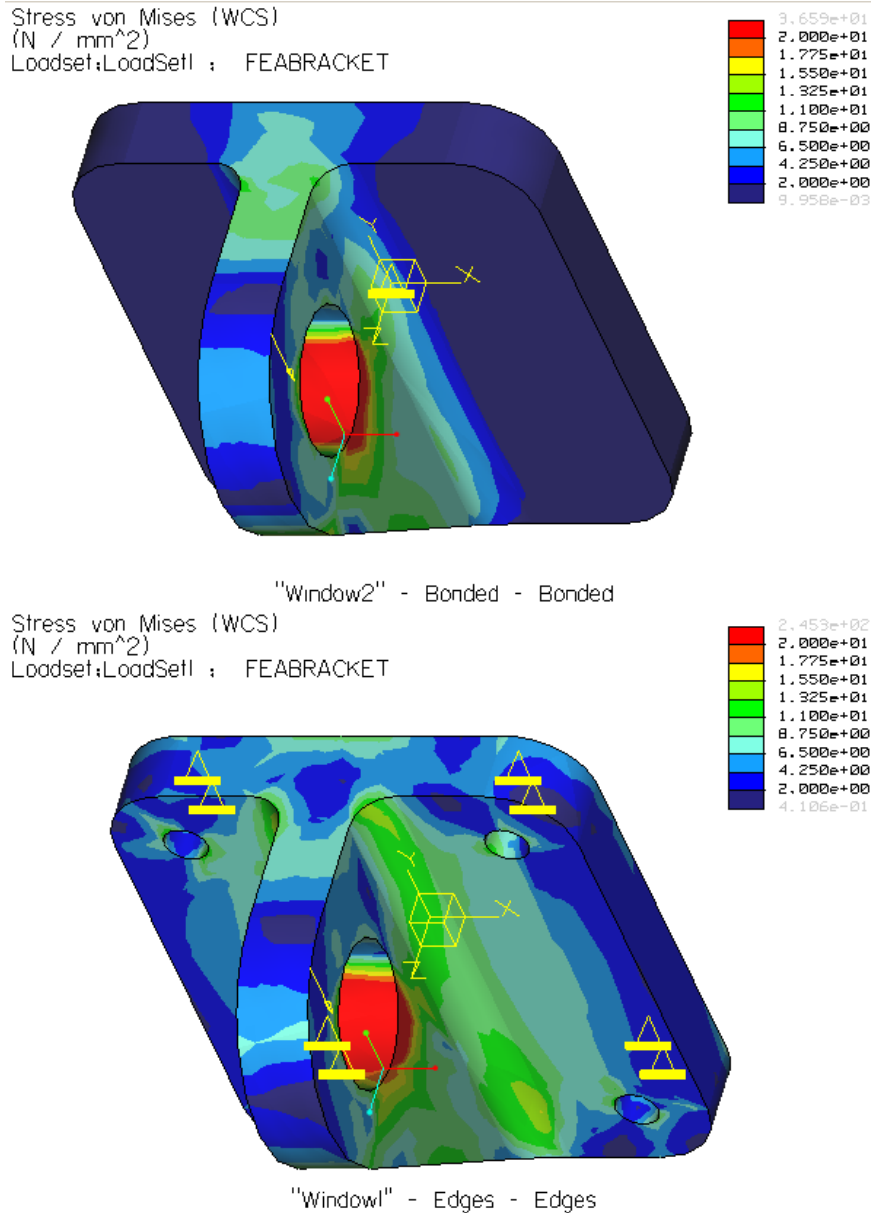


Figure 6 – Comparison 2

- Choose **File** > **Close** before continuing.

## V. 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 (**Displacement**) and create a new constraint set (this will be **ConstraintSet3**). Pick the internal surfaces of the holes.

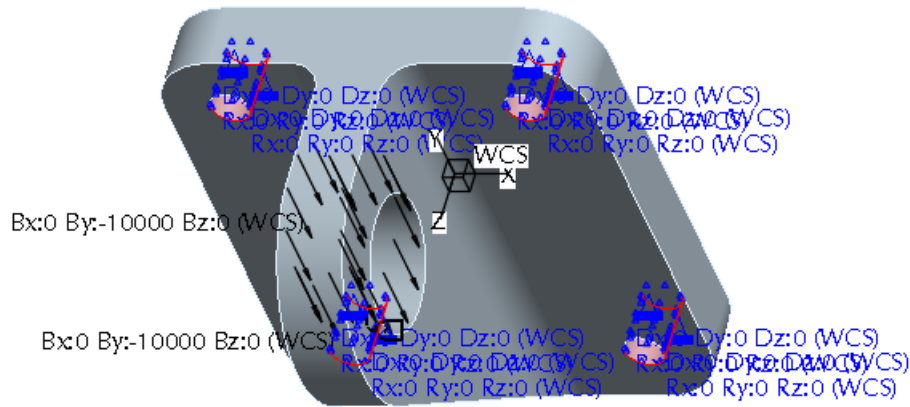


Figure 7 – Constraint Surface

- Choose **Run > Analyses and Studies** and **Edit > Duplicate** 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.

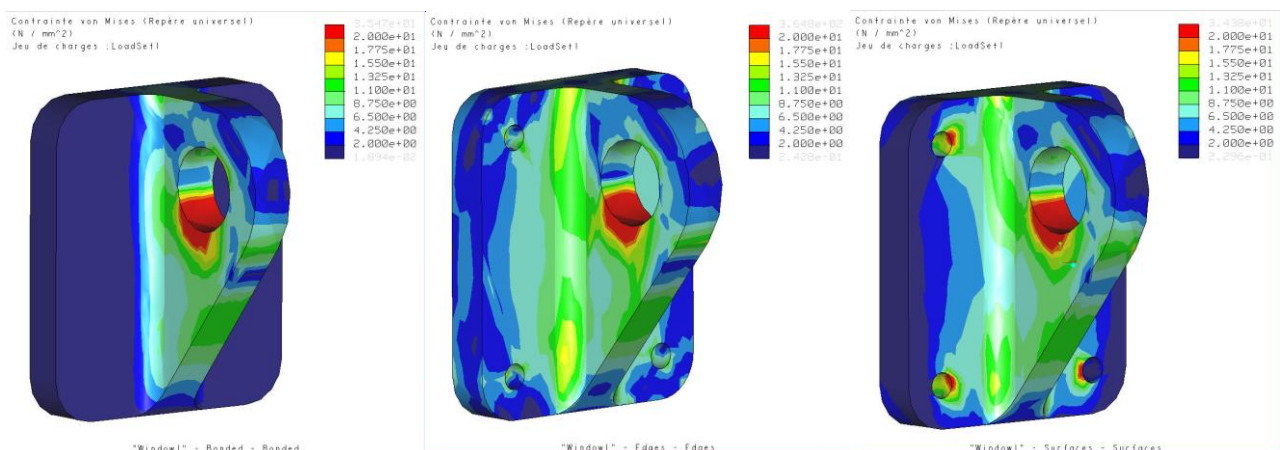


Figure 8 – Comparison 3

## VI. 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.

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 Simulate using **Refine Model > Regions > Volume Region > Extrude** (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 **OK > 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 hidden line display.

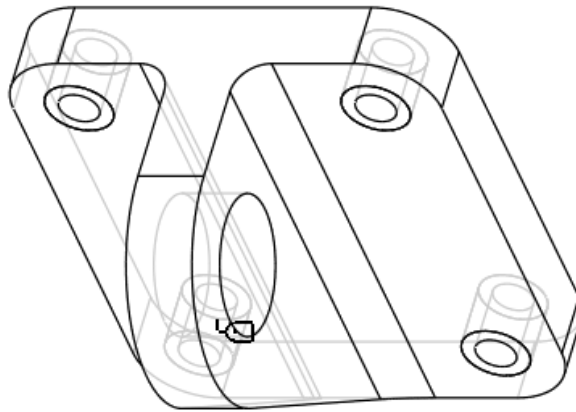


Figure 9 – Volume Regions

- We will also need to create eight datum points – one at each end of the four holes. Choose **Refine 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 center of the arc so in the Datum Point dialog click on the word **On** and change it to **Center** then click on **New Point**. Repeat this for the eight points. Don't forget to click on new point after each point is defined. Close the dialog with **OK**.

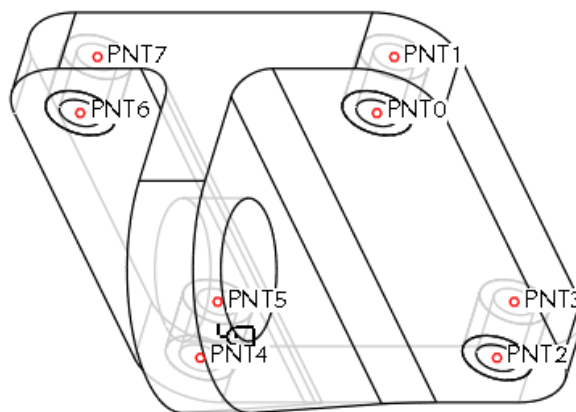


Figure 10 – Datum Points

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 **Idealizations > Beam**. In the Beam Definition dialog click on below Reference(s), choose the **Point-Point** method, 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 **Beam 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).

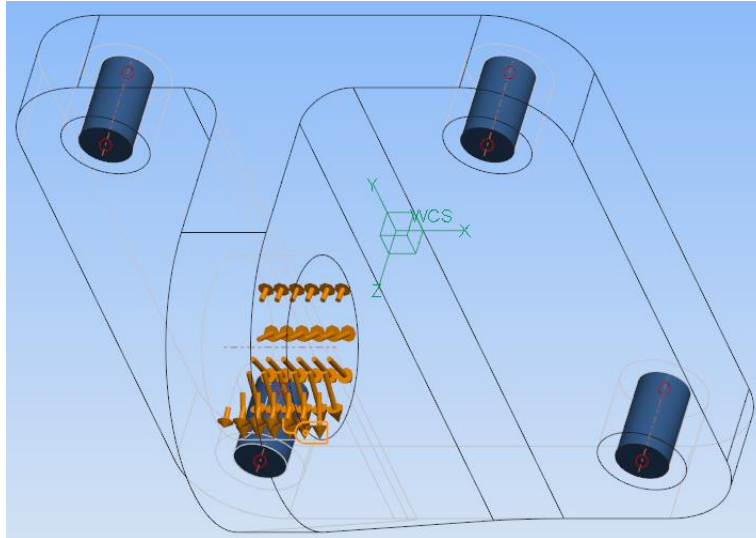


Figure 11 – Four Beams defined

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 **Connections > Rigid Link** and in the Rigid Link dialog click on icon then pick the surface 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).

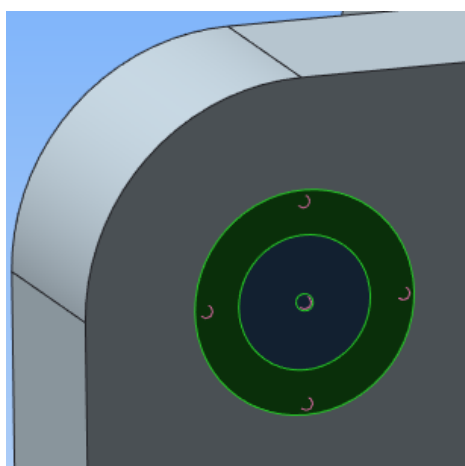


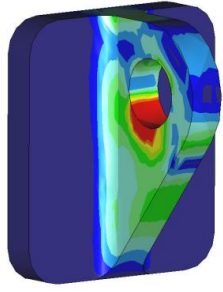
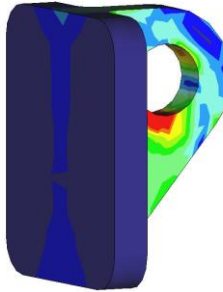
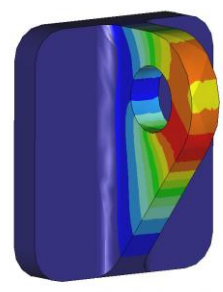
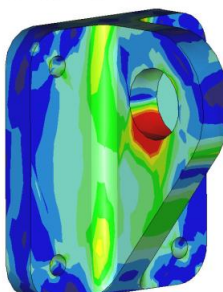
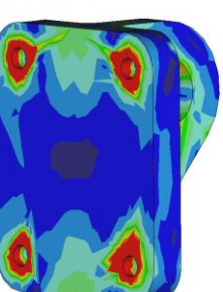
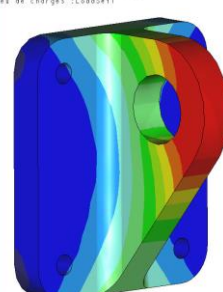
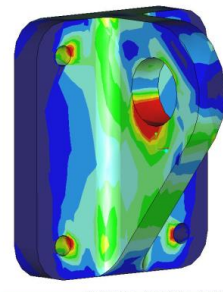
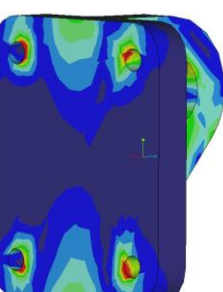
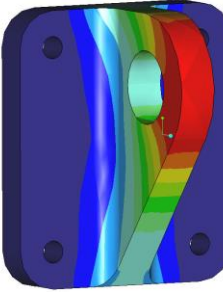
Figure 12 – Rigid Connection

Finally, all that is needed is to correctly constrain the model.



- Create a new **Displacement** constraint 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 analyze this model creating a new analysis called **Spider** which use the correct constraint set.

**Question1** – Compare the results of all four analyses. What should these values be?

	VM Constraint		Displacement
Bonded			
	Max = MPa		Max = mm
Edges			
	Max = MPa		Max = mm
Surfaces			
	Max = MPa		Max = mm

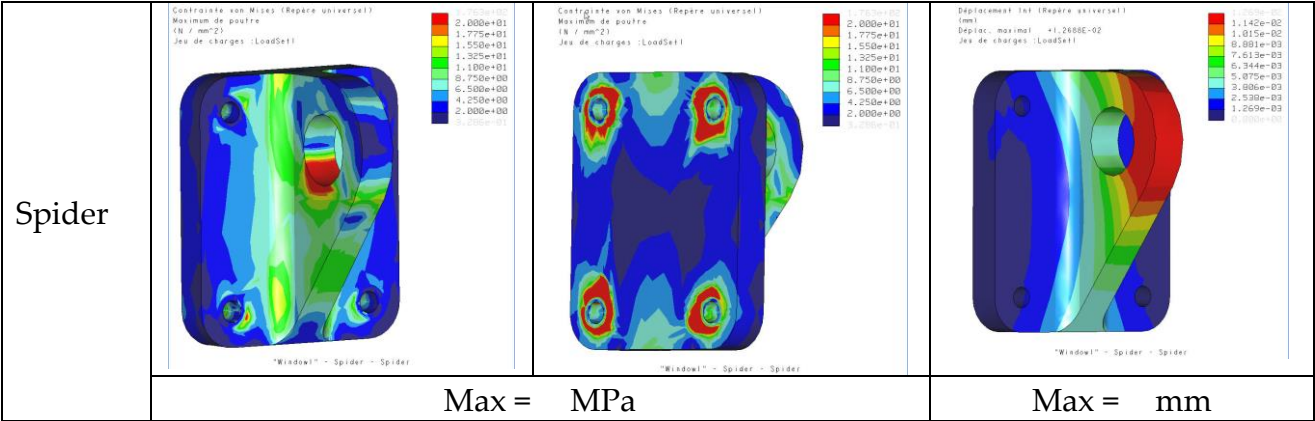


Figure 13 – Comparison 4



## Part 3

### Shell Modeling and Analysis

#### I. Introduction

You have probably already realised that the initial model is very important and can affect both the result accuracy and the time taken to perform the analysis. For example, analysis is often undertaken on models where the majority of radii and other small features which have no significance on the results have been removed or suppressed – this can reduce analysis time tremendously. Of course it is down to skill of the operator to decide which features can be suppressed without affecting the results.

A particular area where correct modelling can improve analysis speed is in parts which have lots of thin walls of constant thickness. Examples of these include sheet metal parts (simple brackets or complex car bodies) and even moulded parts (since good moulding practice requires constant wall thicknesses wherever possible). The modelling technique used for these parts is called shell modelling. Here the designer will model the centreline of a feature then assign a thickness to the feature. Creo combines the information to generate a solid model which looks identical to one made from normal modelling techniques. When analysing the model the shell information can be used to reduce the analysis time – experience has shown that this can be by as much as 100 times in extreme cases.

Here is an example of the techniques involved. The tutorial uses a realistic part so the process is quite complex. Even if you don't intend to use shell modelling the tutorial is worth completing as it introduces other techniques related to analysis.

#### II. Shell Modelling

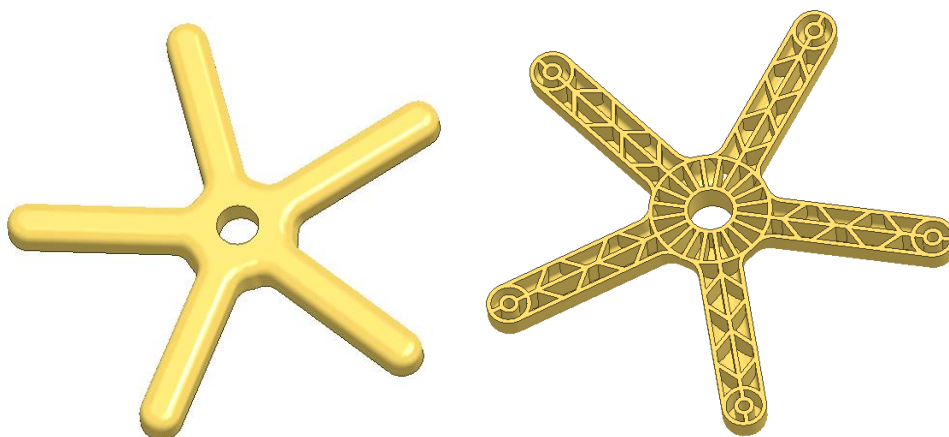


Figure 1 – The Chair Base

The part we are going to analyse is the injection moulded base to a swivel chair as shown in Figure 1. The first thing you should notice about such a part is that it has 5 identical legs. This should immediately show you that you can save both modelling and analysis time by only looking at one of the five legs. Even more time can be saved if you recognise that each leg has a plane of symmetry along its length (see Figure 2) so even more modelling and analysis time can be saved.

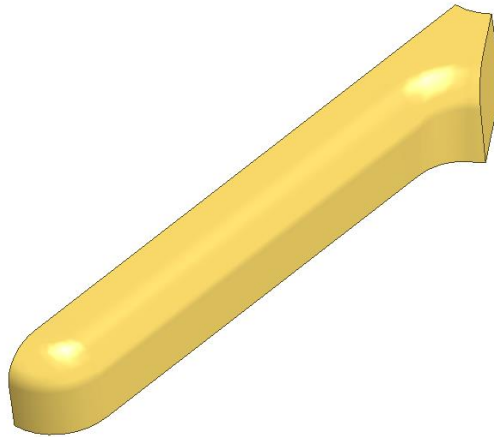


Figure 2 – The Leg Half Model

Here is how to model the leg.

- Create a new part using **File > New** with a name of **Chair\_Leg**. Setup the units to mm.N.s.
- Next create an extrusion. From the dashboard choose the Sketch icon then pick the datum plane **TOP** by clicking on it in the graphics window or in the browser. Draw the sketch in Figure 3. Exit sketcher ☒ and type in the extrusion distance of **30**. Finish the feature with ☒.

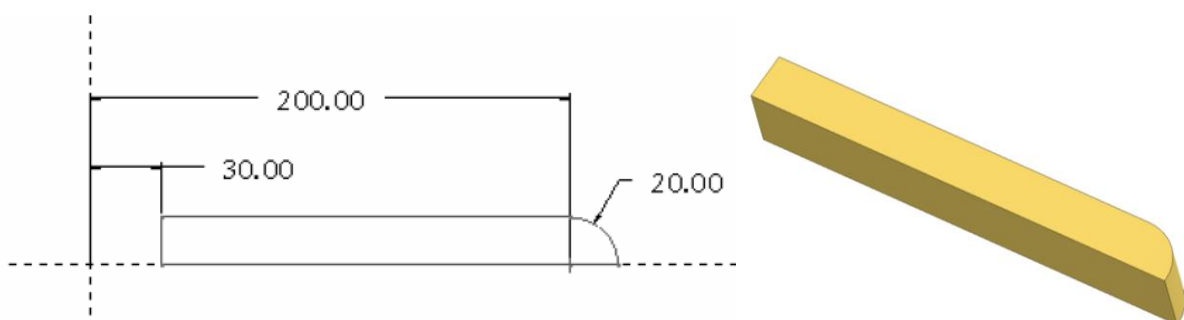


Figure 3 – First Feature Sketch

- Next create a revolution feature. From the dashboard choose the plane **FRONT** as sketch plane. Draw the sketch in Figure 4 – notice that the top line is inline with the top of the first feature. Draw a centreline on top of the **RIGHT** datum. Exit sketcher ☒ and type in the revolve angle of **36**. Finish the feature with ☒.

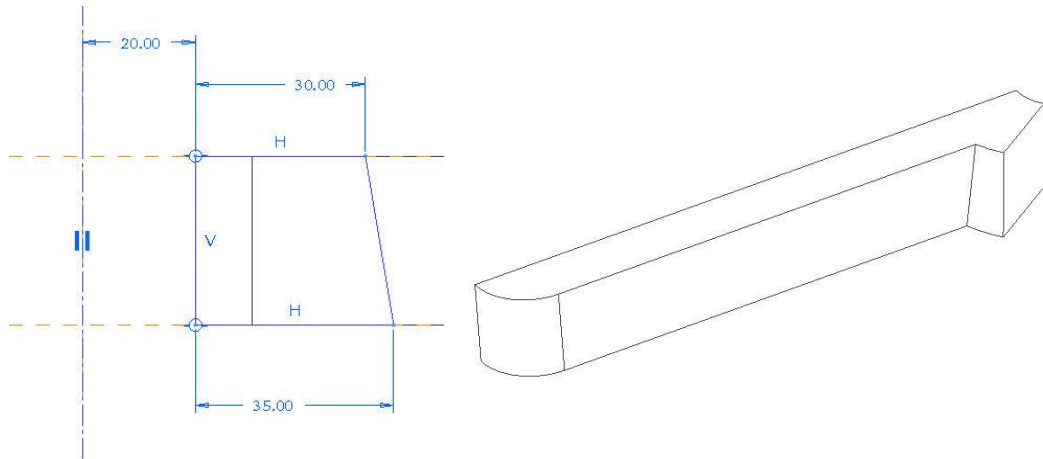


Figure 4 – Second Feature Sketch

- Add a **16 mm round** to the edge between the two features.

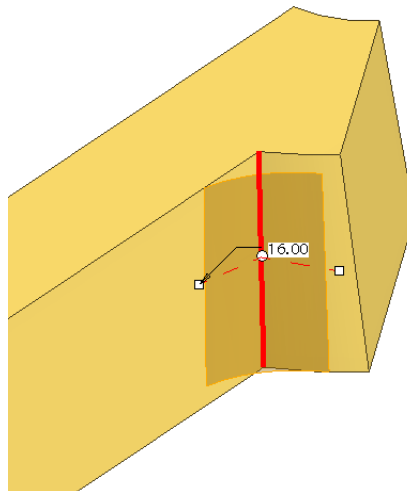


Figure 5 – A Round

- Add a **13 mm round** to the edge around the top of the leg – it should automatically propagate all around as the edges are all tangent.

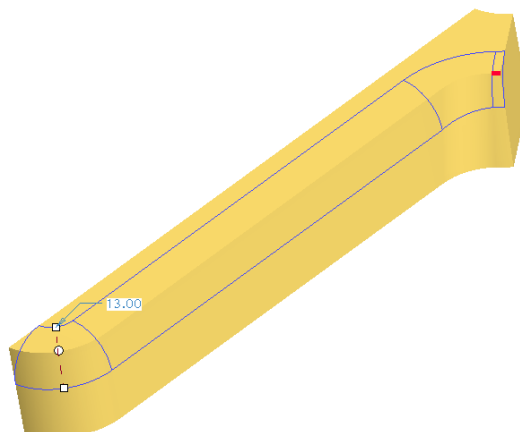


Figure 6 – A Second Round

The steps so far should be familiar to you – there is nothing new. The next step should also be known to you – shelling.

- Create a shell feature to hollow out the leg using **Engineering > Shell**. Pick the two surfaces shown in red in Figure 7a. Choose a shell thickness of 4. Before you finish this feature stop and think. The surface shown in Figure 7b is a web between two legs which should be 4 thick but only half of it is in this section of the model so it should be 2 thick here. This can be achieved in the shell command. Click on the **References** tab then click to activate the **Non-default thickness** pane you now can pick surfaces on the model which will have a different thickness to the rest of the model. Click the surface shown in Figure 7b and change the thickness for this surface to 2.

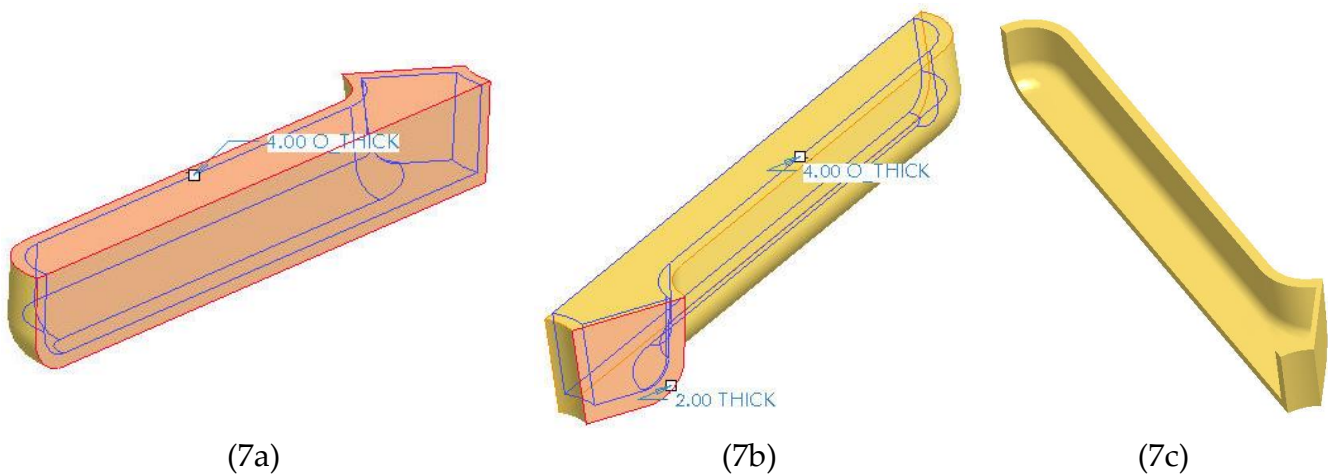


Figure 7 – Creating the Shell

- Now for something rare in extrusions. Create a new extrusion. From the dashboard choose the Sketch icon then pick the datum plane **TOP** by clicking on it in the graphics window or in the browser. Draw the sketch in Figure 9 (a simple semi circle) and exit sketcher . On the dashboard (Figure 8) choose the up to next surface option and use the button to make sure the extrusion is going the correct direction – towards the inside of the leg.

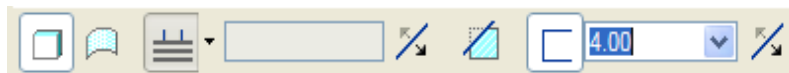


Figure 8 – The Dashboard

- Now click on the thicken sketch button . This button takes the single line sketch you have drawn and adds material to the thickness typed in the box next to the button – 4. Also the second button decides which side of the sketch to add material – click it till the material is outside of the arc. Finish the feature with .

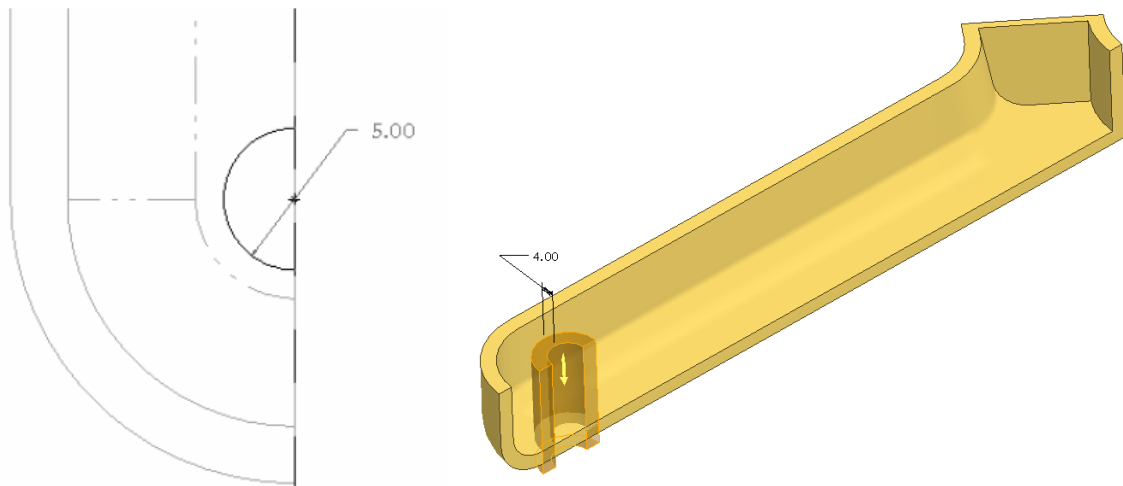



Figure 9 – Thin Protrusion

### III. Analyzing Shell Models

That's enough modelling for now – more later. We will perform an analysis.

- Choose **Applications > Simulate** now to take your model into analysis. Make sure the mode option is set to **Structure Mode**. The first step is to define some simplistic constraints. In this case the hole where the central pillar of the stanchion fits needs to be fixed. Choose **Constraints > Displacement** (or you could just pick the  icon). The constraint dialog will appear. Pick the surface in this model which is part of the hole (Figure 10) then **OK** to return to the constraint dialog and **OK** to leave this surface fully constrained.

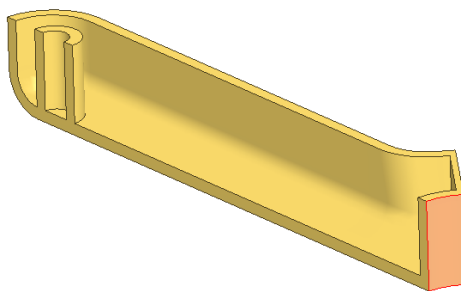



Figure 10 – Constraint Surface

- Definition of loads is similar to constraints. Choose **Loads > Force/Moment** or pick the  icon to apply a load over a surface. Pick the surface as shown in Figure 11. Type a value of **150** in the correct field for a vertical load on the leg (probably the Y direction). This will be half the total load applied to a single leg as we are only modelling half the leg. Press **Preview** to check the arrows point in the correct direction. Click **OK** in the Force/Moment dialog to finish.

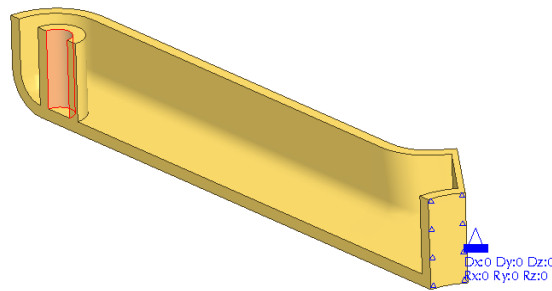







Figure 11 – Loaded Surface

- Choose **Materials > Materials**  and the Materials dialog will appear. Scroll down the Materials in Library to find **Nylon** and double click on it to transfer it to this model. Choose **Materials > Materials assignment**  and click on the leg and **OK** to assign the material. **Close** the material dialog.

That's it you are ready to run an analysis.

- Choose **Run > Analyses and Studies** . From this dialog choose **File > New Static** and type the name **Leg** 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.

**Question1** – Note how many elements are used in this analysis and the elapsed time to complete the analysis.

**Question2** - Note the value of the maximum stress and displacement.

- **Close** the Report dialog and the Analyses dialog.

The analysis should complete correctly and you could review these results. This has performed a normal analysis – it has not used any information about shells at all. So how do we use shell information?

- The easiest way to do this is to use the automated **Refine Model > Idealizations > Shell Pair** then choose **Detect Shell Pairs**. Deselect the detection method. This takes any shelled surfaces or thickened protrusions and automatically generates thin shells from them. Click on **Start**.
- After this command you can see the shells by selecting **Midsurface** option from the **AutoGEM** menu. Now click the **AutoGEM > Review Geometry...** to open the Simulation Geometry dialog box. Select **Shell Surfaces** and **Original Geometry** on the Simulation Geometry dialog box to review shell compression.

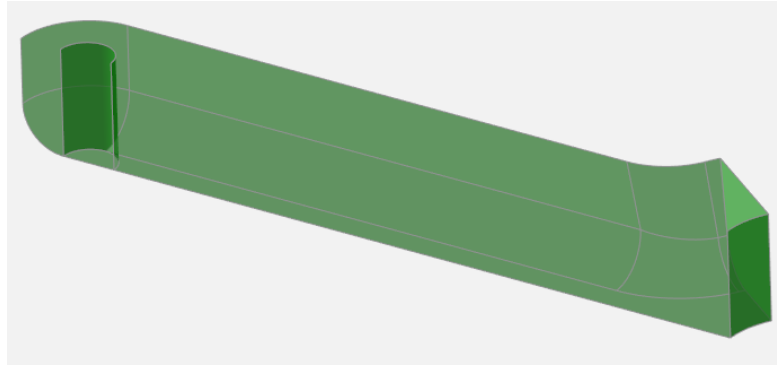


Figure 12 – Shell Display

- Before running the analysis you must modify the displacement constraints by fixing all six degrees of freedom. Indeed, the shell elements, unlike solid elements, have rotational freedoms which can be adjusted.
- So try running the analysis again.

**Question3** –In the Status dialog notice now how many shell elements are used and the time taken for the analysis will be much shorter.

**Question4** – Have a look at the results and show the max Von Mises stress and the max magnitude of displacement.

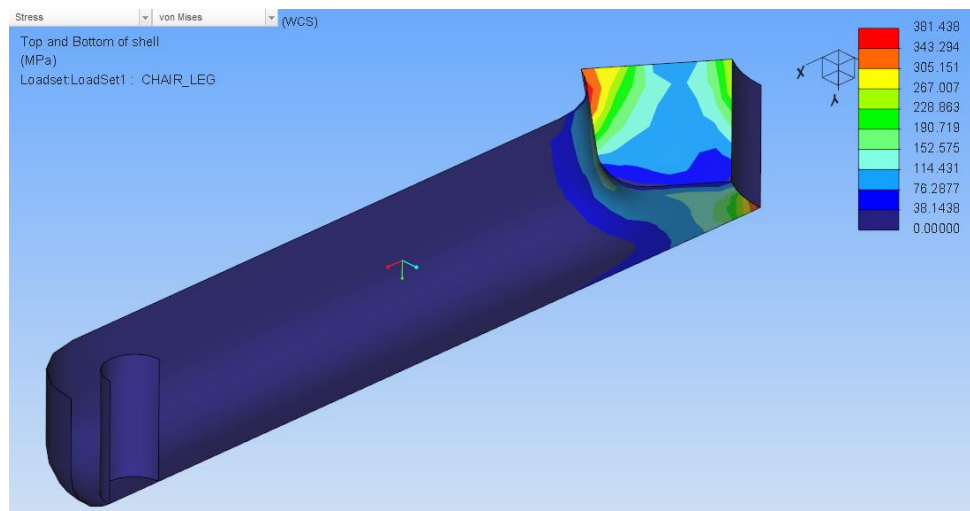



Figure 13 – Analysis Using Shells

There is a problem with the analysis! Look carefully at the leg and you will see that as it is loaded it twists. This wouldn't happen in real life because we would have a full leg not just half. We can correctly simulate the other half of the leg without having to model it by correct use of constraints.

- Choose **Home > Constraints > Symetry Constraint**  and the symmetry constraint dialog will appear. Pick the edges in Figure 14 then **OK** to finish.



**Note** - Edges are selected rather than the central surface because the surface ‘disappears’ when the model is collapsed into shells.

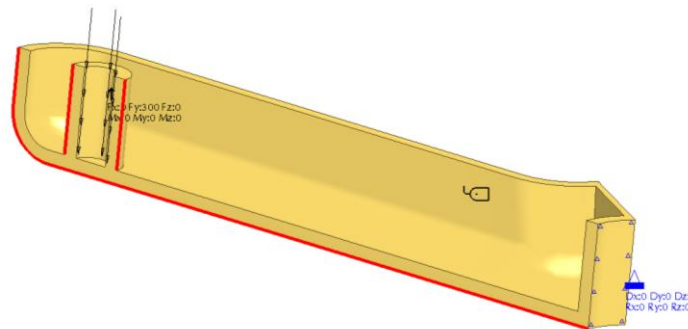


Figure 14 – Edge Constraints

There is another issue with the current problem setup since the cyclic symmetry of the legs has not yet been taken into account.

This time the constraint is not along the normal X, Y or Z axes. We need to make a new definition for the direction of X, Y and Z. This is done in Creo with a coordinate system. We need to create one now.

- Choose **Refine Model > Datum > Coordinate System**. The coordinate system dialog is displayed. This is an “intelligent” dialog – it will try and make sense of what you select. Click on the 3 surfaces/datums now in the order shown in Figure 15. Notice the new yellow coordinate system icon – the X direction is at right angles to the first surface you picked and this is the direction which we will constrain. In the properties tab type the name **Angled**. Click **OK** to close the dialog and Angled should appear in the model tree under Simulation Features.

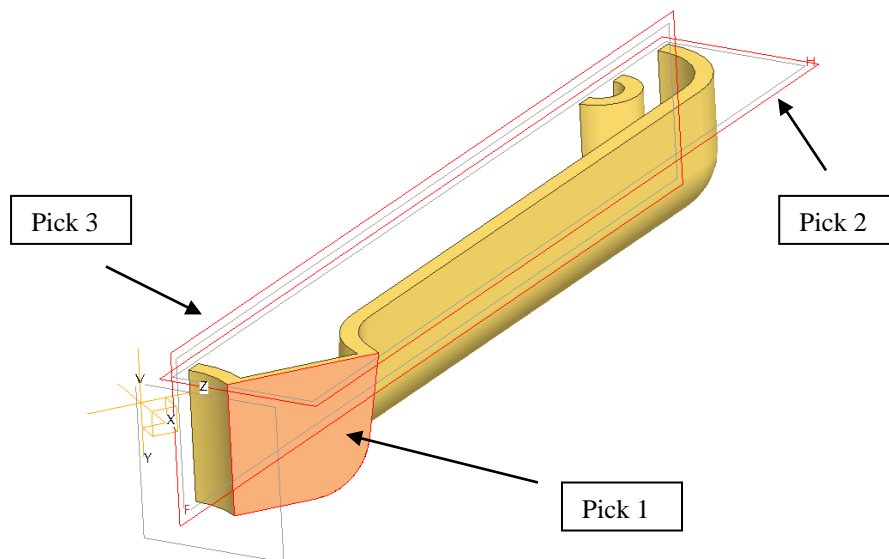



Figure 15 – Defining the Coordinate System

- We will now add another constraint using this coordinate system. Choose **Displacement** . The constraint dialog will appear. Pick the surface highlighted in red in Figure 16. To use another



coordinate system click on **Selected** below Coordinate System then pick the **Angled** coordinate system. This constraint needs to stop movement across the symmetry plane (X) whilst allowing free movement in the plane (Y&Z). Set the constraints as shown in Figure 16 for both Translation and Rotation.

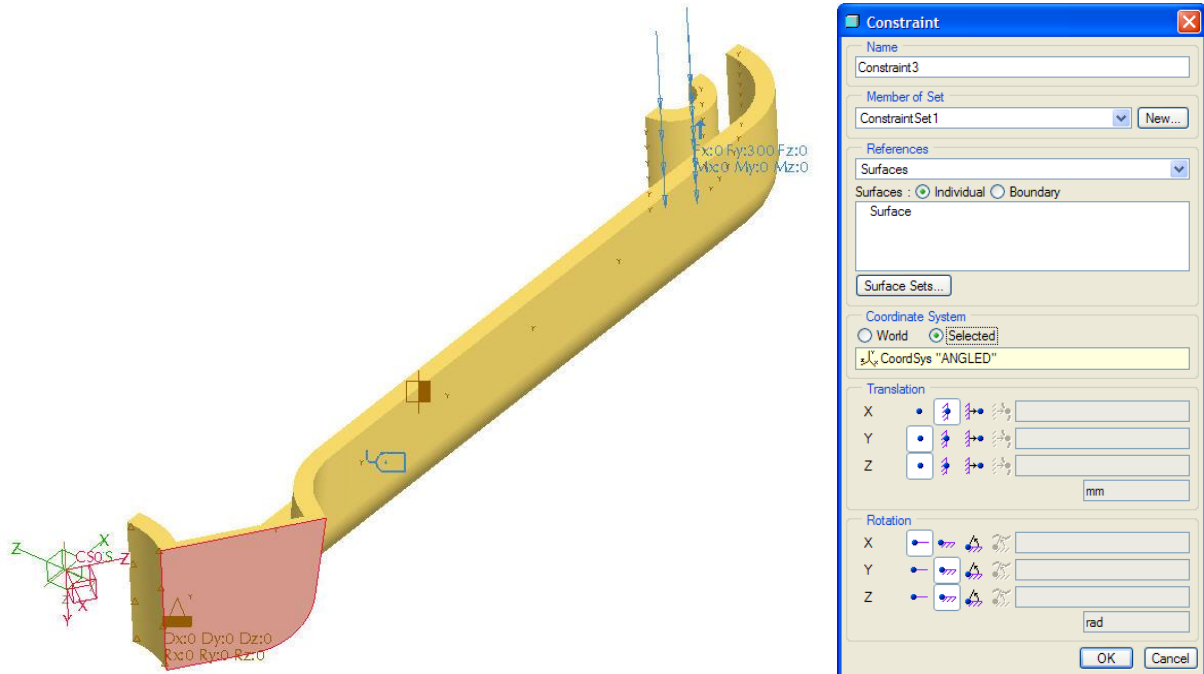


Figure 16 – Second Symmetry Constraints

- Reanalyse the part and the stress and deflection patterns should accurately mimic a real leg.

**Question5** – Have a look at the results and notice the max Von Mises stress and the max magnitude of displacement.

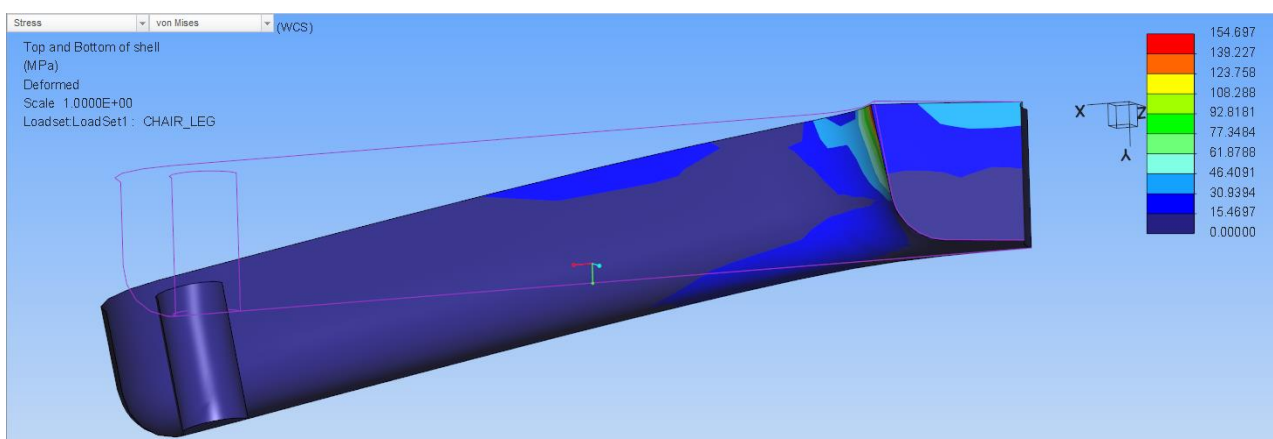





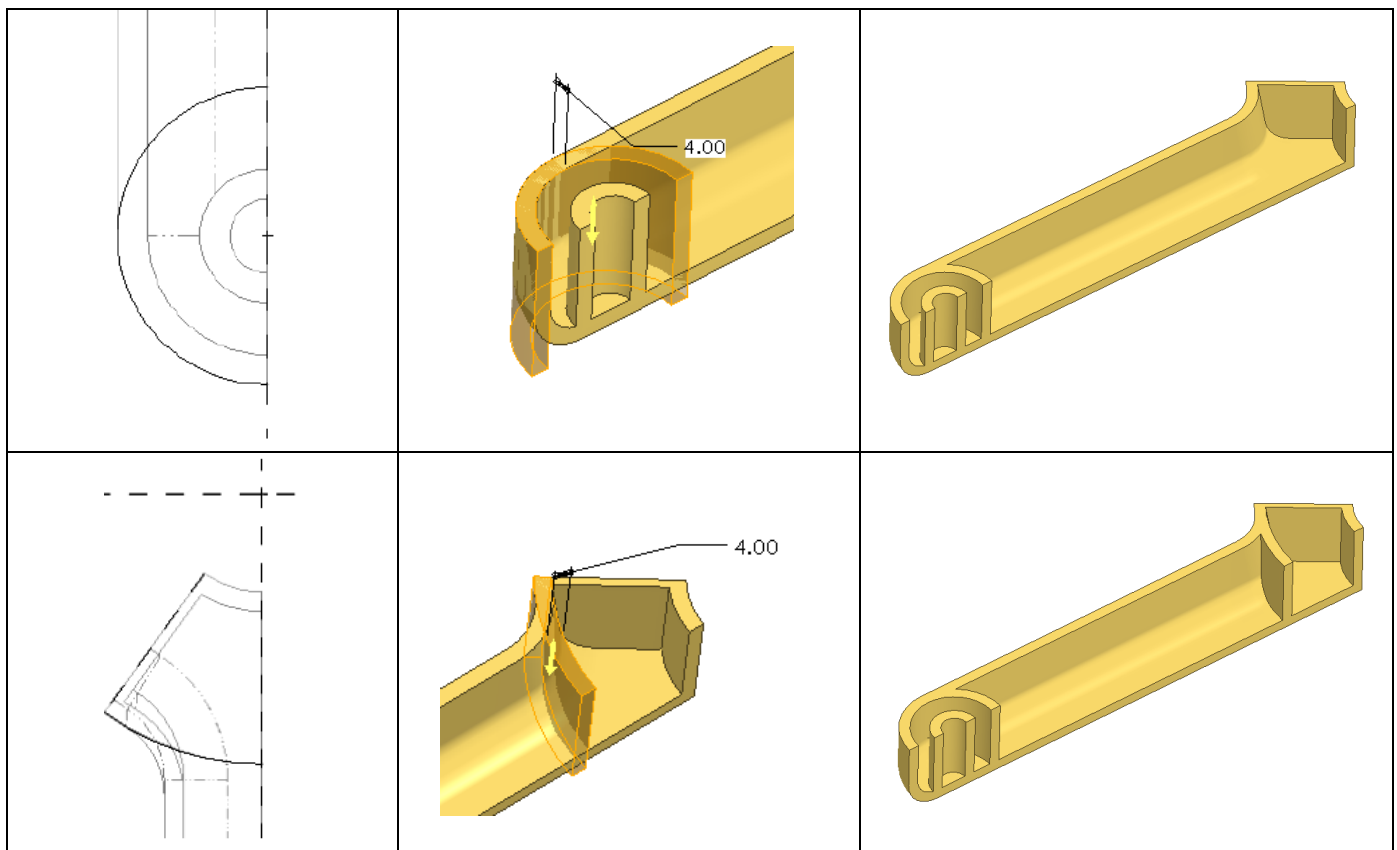
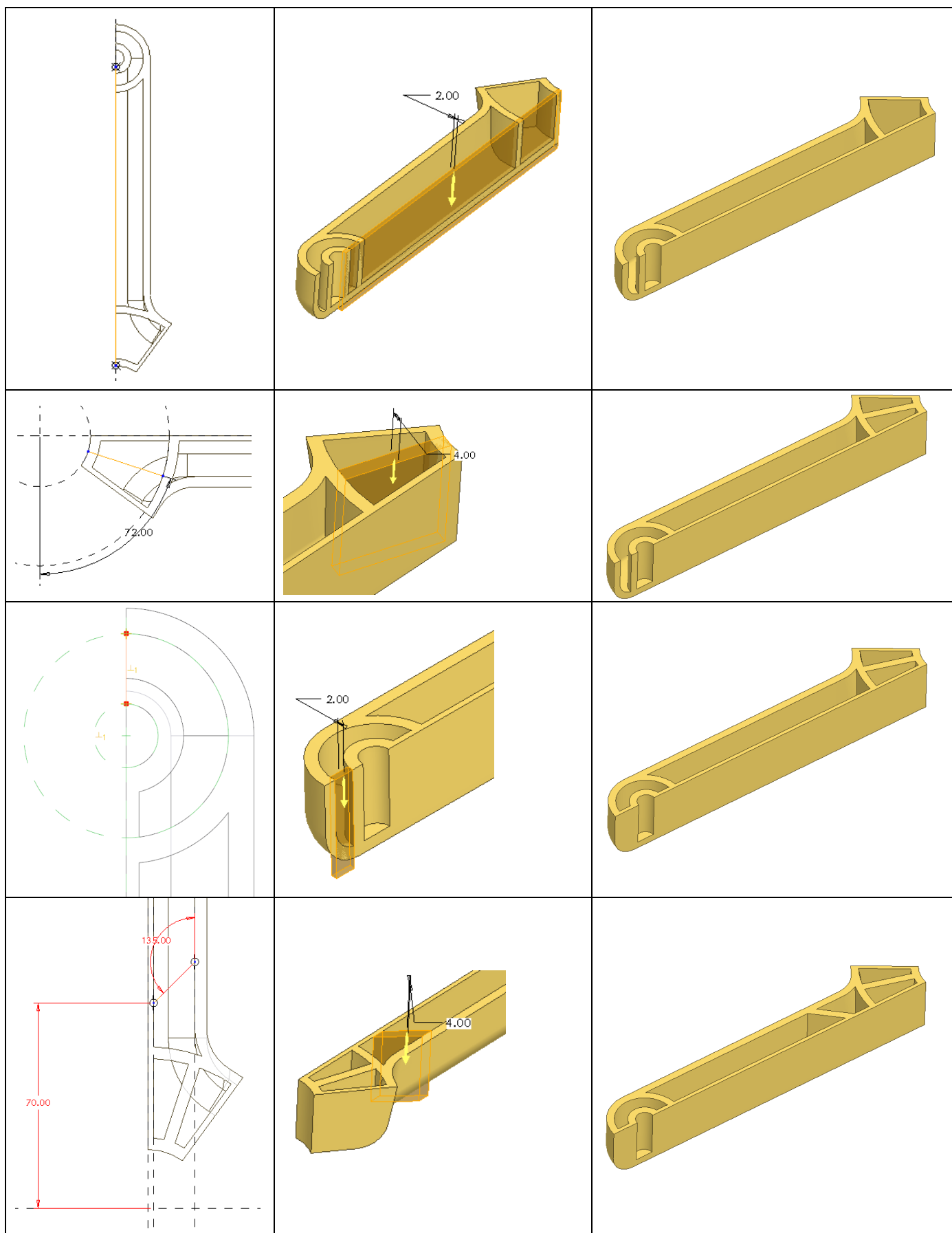


Figure 17 – The Analysis

If you look at the deflection at the end of the leg it is too high. Normally such a structure would have strengthening ribs to improve the strength. Here is how to add such ribs – they are all added as thickened protrusions.

- Create a new extrusion. From the dashboard choose the Sketch icon then pick the datum plane **TOP** by clicking on it in the graphics window or in the browser. Draw the appropriate sketch for this feature (see the comic strip in Figure 18) and exit sketcher . On the dashboard choose the up to next surface option  and use the button to make sure the extrusion is going the correct direction – towards the inside of the leg. Now click on the thicken sketch button . This button takes the single line sketch you have drawn and adds material to the thickness typed in the box next to the button – 4. Also the second  button decides which side of the sketch to add material – click it till the material is on the correct side. Finish the feature with .
- Perform all the ribs showed in Figure 18.





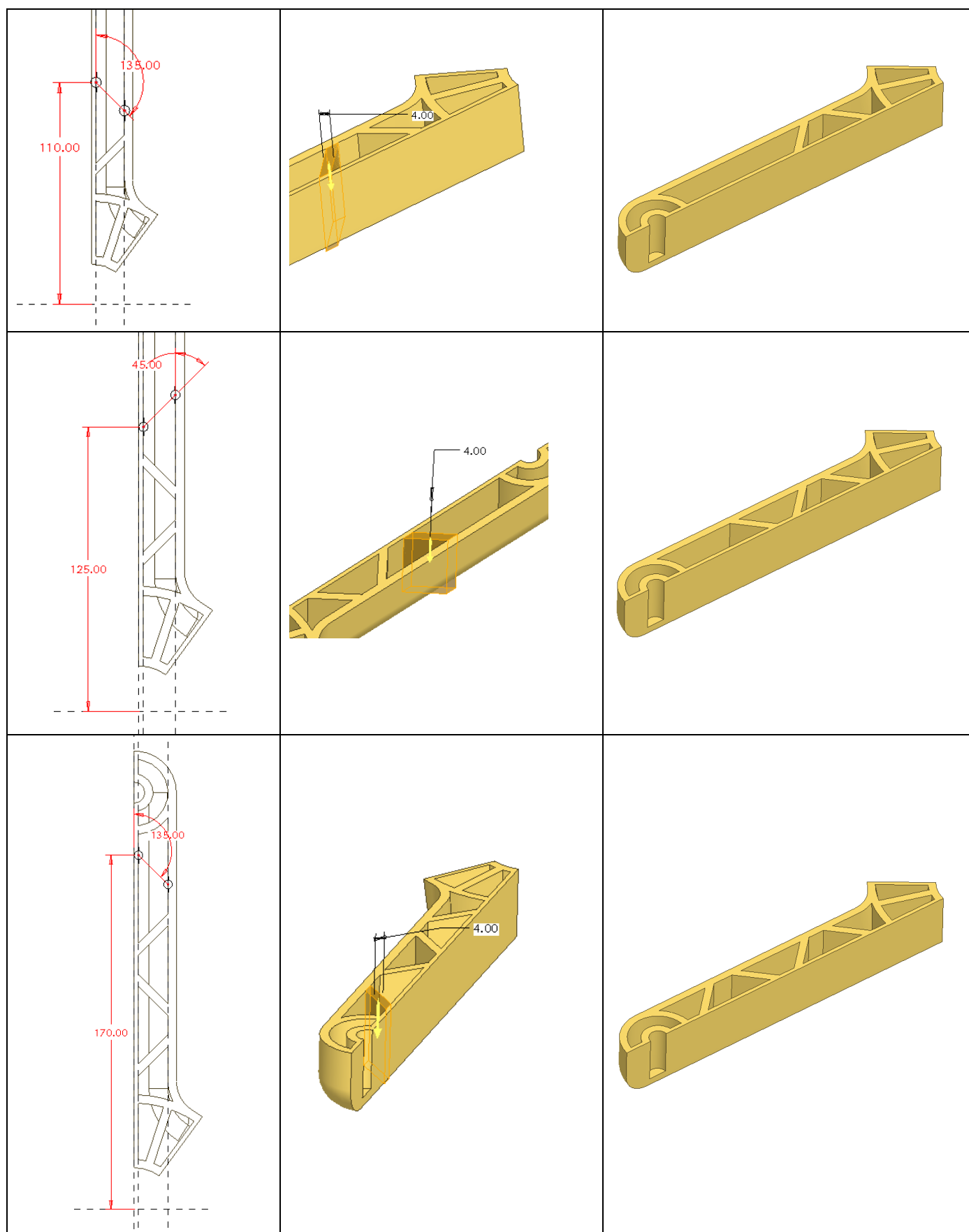


Figure 18 – Thin Protrusions

- Before running a new analysis on the ribbed model you have to create new shells and adjust the symmetry plane constraint.
- Re-run an analysis in order to assess the improvement in strength and stiffness of the leg.

**Q6** – Have a look at the results and notice the max Von Mises stress and the max magnitude of displacement.

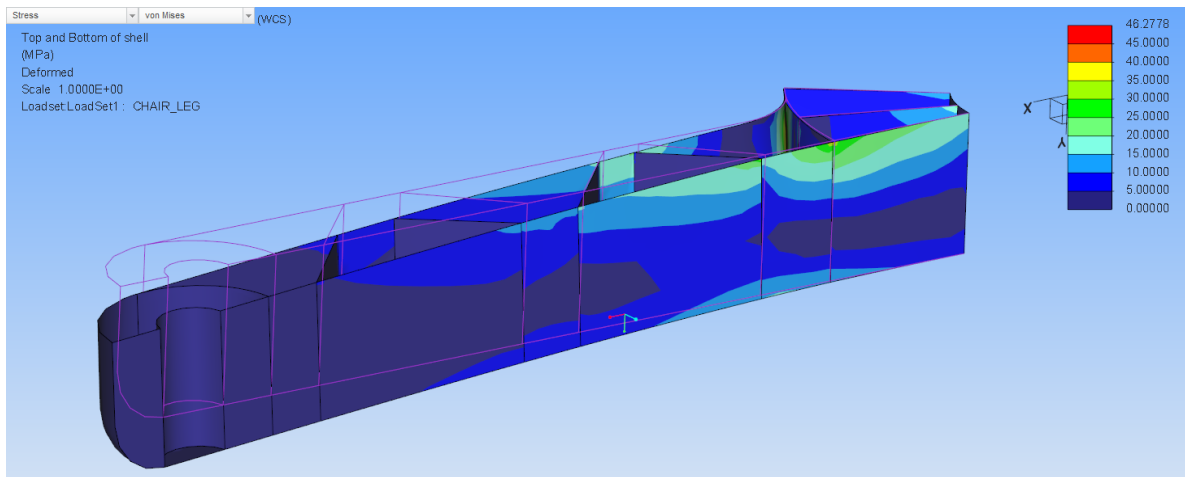


Figure 19 – The Analysis