

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. Pro Engineer 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. Pay careful attention as you read – especially if you have not completed all of the modelling exercises in this series.

Even if you don't intend to use shell modelling the tutorial is worth completing as it introduces other techniques related to analysis. If you find the modelling instructions difficult to follow then have you completed the modelling tutorials? If you haven't you might find it helpful to do so.

Shell Modelling

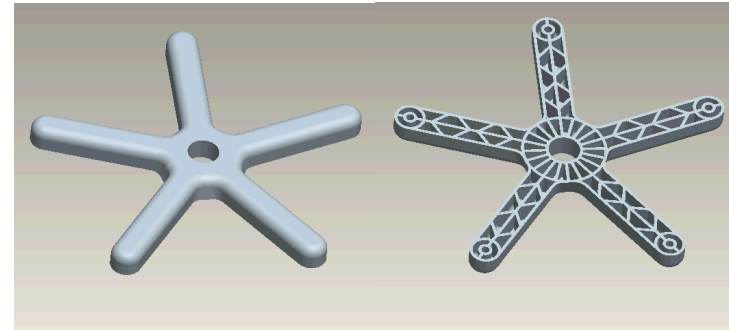


Figure 1 : The Chair Base

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.

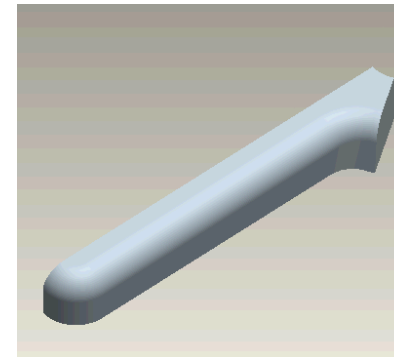


Figure 2 : The Leg Half Model

Here is how to model the leg. Create a new part using FILE > NEW with a name of chair_leg. Choose the mmns_part_solid template.

Next create an extrusion (INSERT > EXTRUDE). 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 then click on the SKETCH button.

Draw the sketch in Figure 3. Exit sketcher and type in the extrusion distance of 30. Finish the feature with .

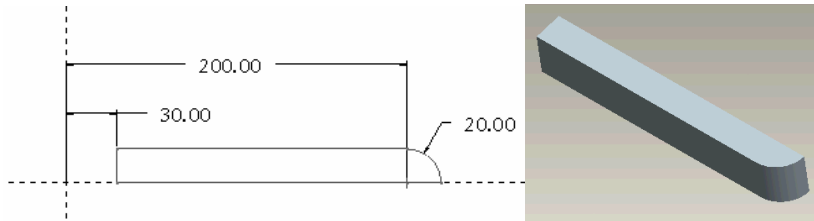


Figure 3 : First Feature Sketch

Next create an revolution (INSERT > REVOLVE). From the dashboard choose the SKETCH icon then pick the datum plane FRONT by clicking on it in the graphics window or in the browser then click on the SKETCH button.

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 datumFigure 3.

Exit sketcher and type in the revolve angle of 36. Finish the feature with .

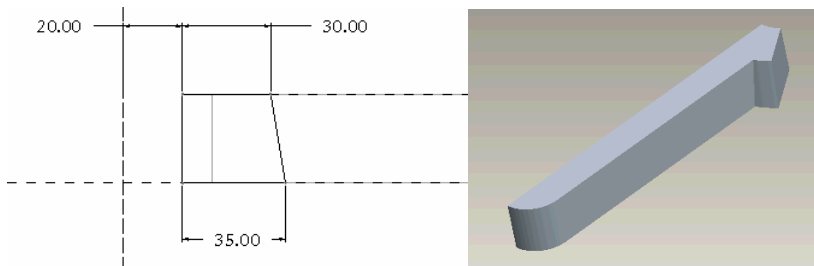


Figure 4 : Second Feature Sketch

Add a 16mm round (INSERT > ROUND) to the edge between the two features.

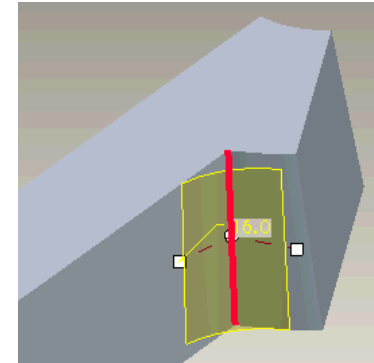


Figure 5 : A Round

Add a 13mm round (INSERT > ROUND) to the edge around the top of the leg – it should automatically propagate all around as the edges are all tangent.

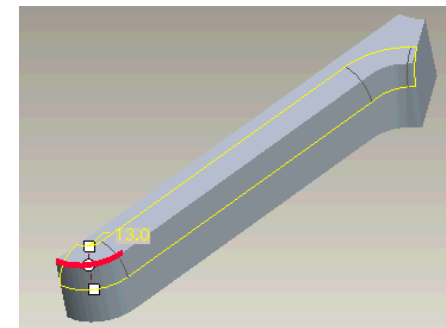


Figure 6 : A Second Round

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 INSERT > 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.

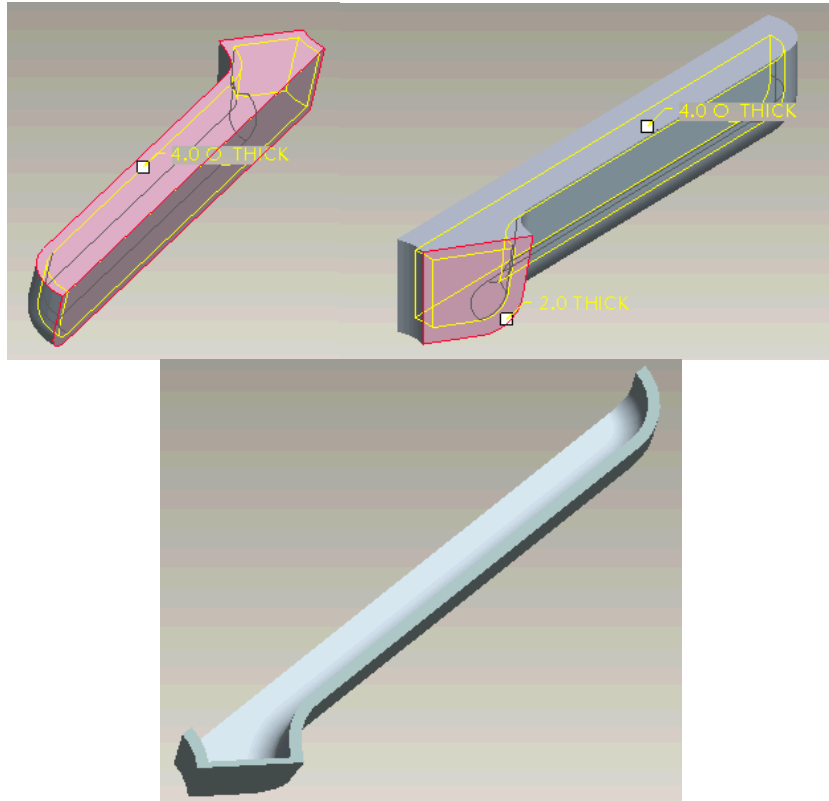


Figure 7 : Creating the Shell

Now for something new in extrusions. Create a new extrusion INSERT > 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 then click on the SKETCH button. Draw the sketch in Figure 9 (a simple semicircle) 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.

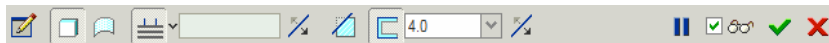


Figure 8 : The Dashboard

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 .

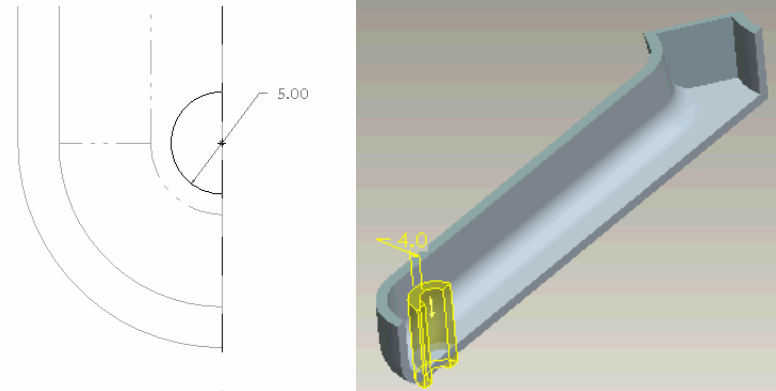


Figure 9 : Thin Protrusion

Analysing Shell Models

That's enough modelling for now – more later. We will perform an analysis. Choose APPLICATION > MECHANICA now to take your model into analysis. Click OK on the box notifying you of the units of your model. Make sure the MODE option is set to STRUCTURE and click OK.

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 INSERT > DISPLACEMENT CONSTRAINT (or you could just pick the icon). The constraint dialog will appear. Click on below Surface(s) then 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.

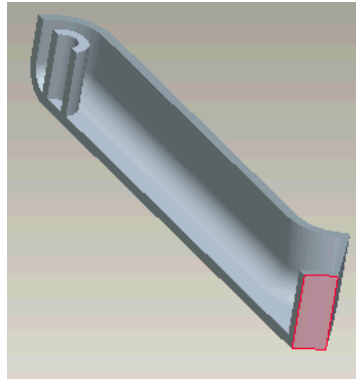


Figure 10 : Constraint Surface

Definition of loads is similar to constraints. Choose INSERT > FORCE/MOMENT LOAD or pick the icon to apply a load over a surface. Click on below Surface(s) then pick the surface in Figure 11 then OK to return to the Force/Moment dialog. Type a value of 300 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.

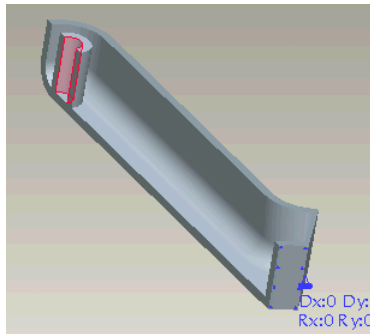


Figure 11 : Loaded Surface

Choose PROPERTIES > MATERIALS and the MATERIALS dialog will appear. Scroll down the materials library to Find NYLON and double

click on it to transfer it to this model. Press ASSIGN > PART and click on the chair leg 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 . From this dialog choose FILE > NEW STATIC and type the name leg and OK. Choose the icon to run this analysis choosing yes for error detection. Press to watch the report of the analysis as it runs. Note how many elements are used in this analysis and the elapsed time to complete the analysis. Close the REPORT dialog and the ANALYSES dialog.

The analysis should complete correctly and you could review these results if you wanted. 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 INSERT > MIDSURFACES then choose AUTO DETECT. This takes any shelled surfaces or thickened protrusions and automatically generates thin shells from them. After this command you can see the shells by choosing COMPRESS > SHELLS ONLY > SHOWCOMPRESS.

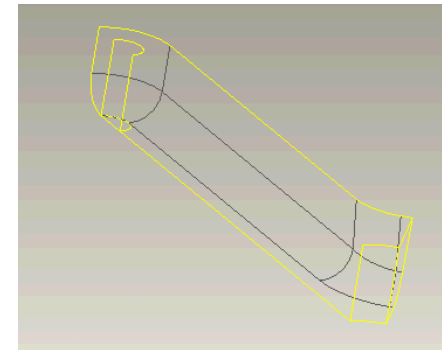


Figure 12 : Shell Display

Try running the analysis again – in the Status dialog you should notice it now uses Shell elements (Figure 13) and the time taken for the analysis will be much shorter. A look at the results will show you the display as shells too.

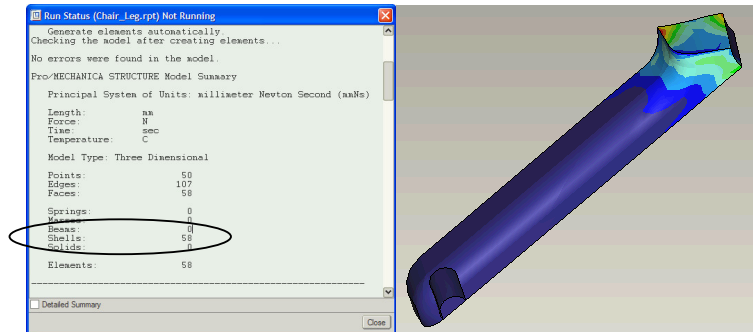


Figure 13 : Analysis Using Shells

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 INSERT > SYMETRY CONSTRAINT and the symmetry constraint dialog will appear. Pick the edges in Figure 14 then OK to return to the constraint dialog and OK again to finish.

NOTE: edges are selected rather than the central surface because the surface 'disappears' when the model is collapsed into shells.

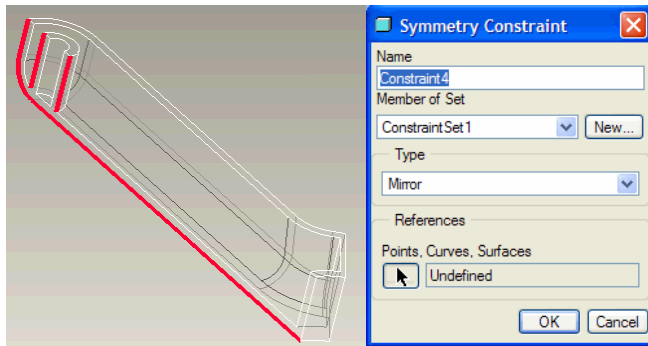


Figure 14 : Edge Constraints

There is another problem with the analysis! There is another symmetry on the surface in Figure 16.

NOTE:- Cyclic Symmetry constraints (as these are called) can be defined using the symmetry constraint type. If we were analysing a whole leg this

type of constraint could be used Since we are only analysing half a leg we cannot use this automated method – we will have to replicate this using displacement constraints.

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 Pro Engineer with a coordinate system. We need to create one now. Choose INSERT > 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.

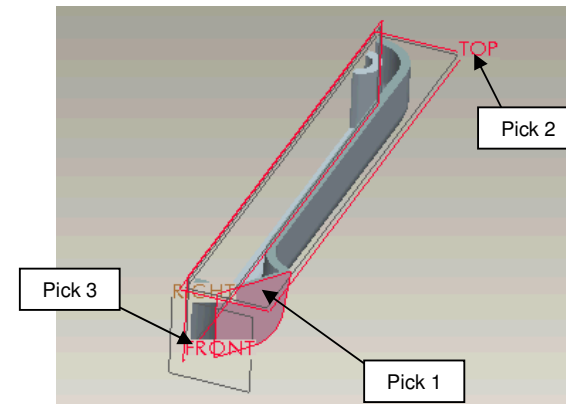


Figure 15 : Defining the Coordinate System

We will now add another constraint using this coordinate system. Choose INSERT > DISPLACEMENT CONSTRAINT . The constraint dialog will appear. Click on below Surface(s) then pick the surface highlighted in red in Figure 16 then OK to return to the constraint dialog. To use another coordinate system click on 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 16b for both Translation and Rotation. (Hint – For symmetry rotation and translation constraints are opposite).

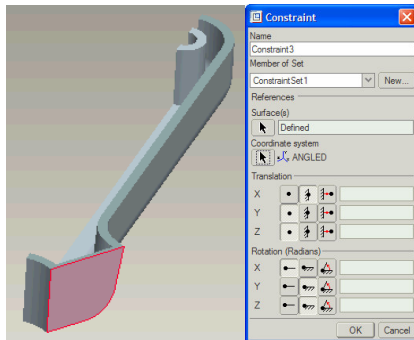


Figure 16 : Second Symmetry Constraints

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

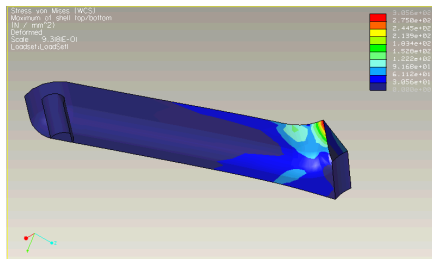
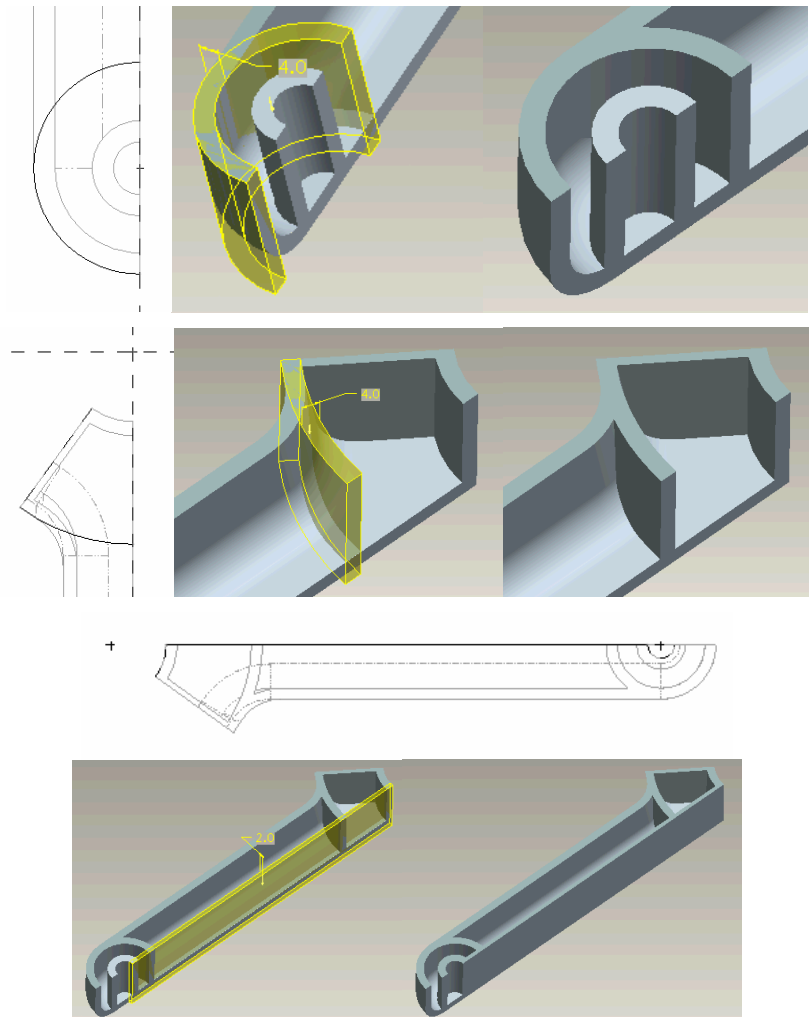


Figure 17 : The Analysis

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 INSERT > 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 then click on the SKETCH button. 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 .



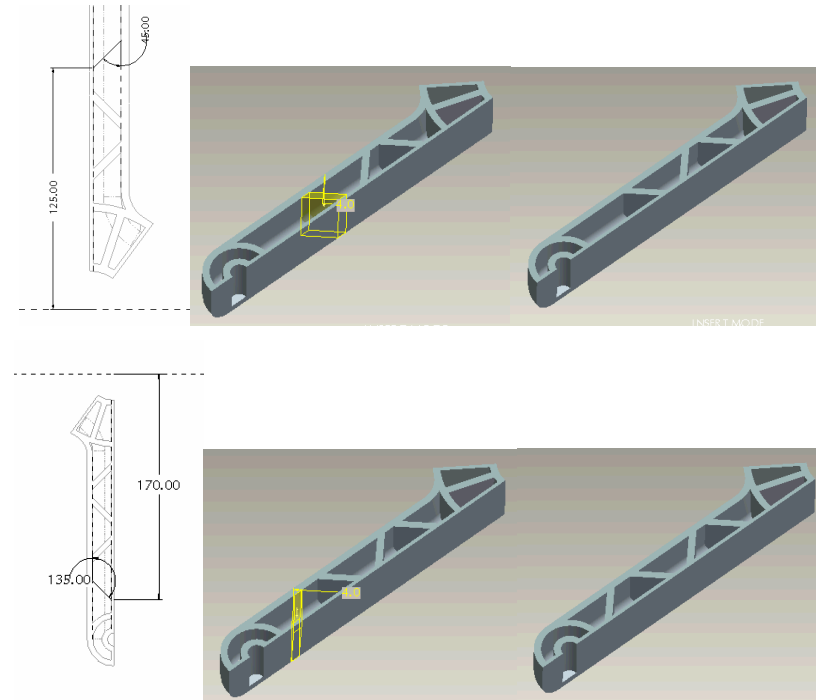
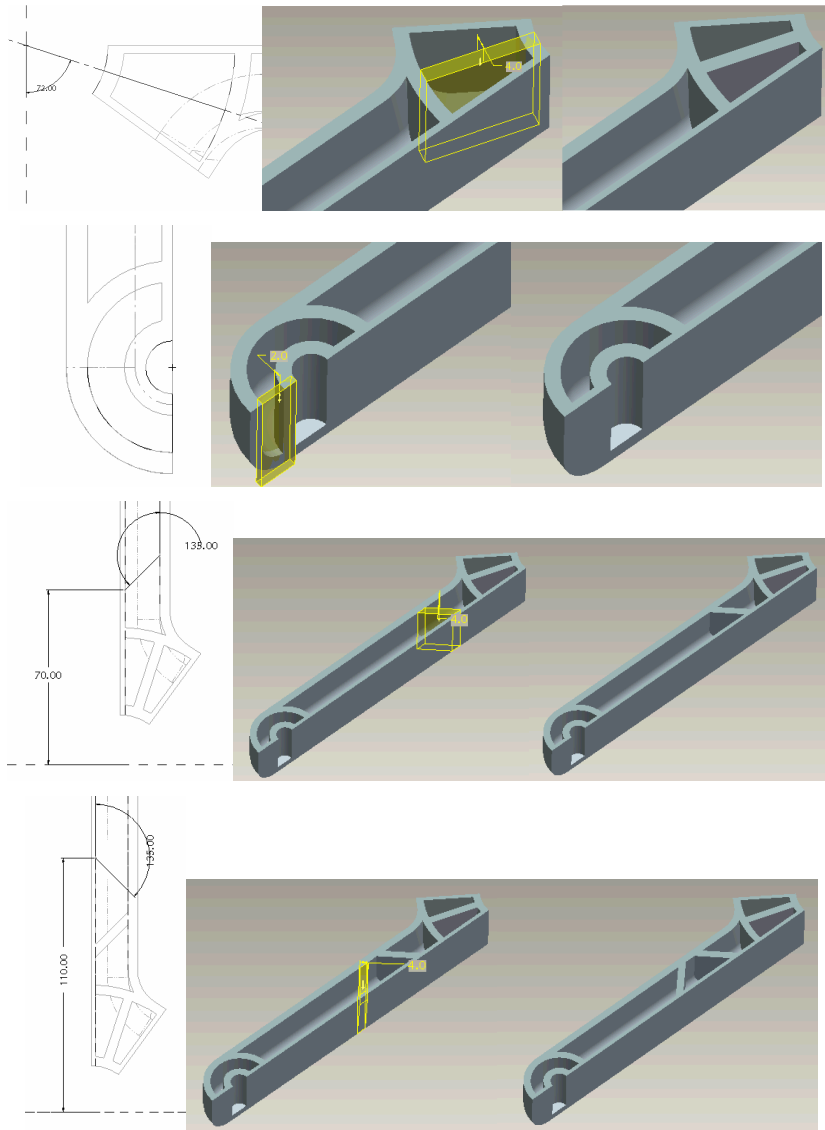


Figure 18 : Thin Protrusions

Perform an analysis on the strengthened model to determine the improvement in strength and stiffness of the leg.

Review

So what should you have learnt?

- How to create variable thickness shells.
- How to create thin extrusions.
- How to create symmetry constraints.
- How to use coordinate systems in analysis.

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