Creating a Part

In this part of the tutorial we will introduce you to some basic modelling concepts. If you are already familiar with modelling in Pro Engineer you will find this section very easy. Before starting to work through this tutorial you need to be sitting in front of a computer terminal which has access to Pro Engineer and be logged on. You tutor should have advised you of how to log in already.

Start Pro Engineer by double clicking on the [icon] icon on your desktop or from the START menu. The main application window should appear shortly.

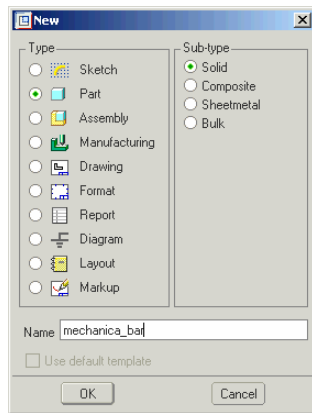The next step is to create your first part. To do this use the menu FILE > NEW.

Figure 1 : The New Part Dialog Box

After choosing the new command a dialog box will appear as shown in Figure 1. Notice that the Part option is already checked and type in *mechanica_bar* as the name of this part (Note : Pro Engineer does not allow spaces and other special characters in names).

A second dialog will appear offering different options for parts – in particular different units of measurement. Choose mmns_part_solid which means the units of length will be millimetres and units of mass will be Newtons and click on the OK button.
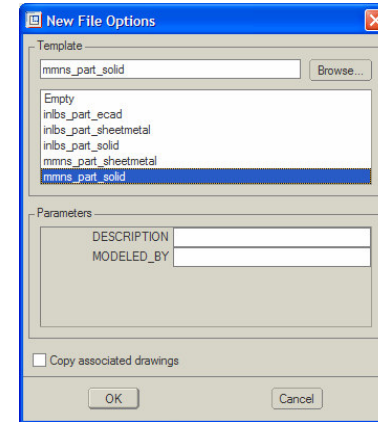

Figure 2 : Part Options

Well done – you have made your first part! The part contains some features already. The browser on the left shows 3 datum planes and a coordinate system. So what are datum planes? As the word plane implies these are flat areas that can be used as references for defining parts of your model. In some case you can define models with out any datum planes, in other cases they are essential. Many people choose to always have a basic set of default datum planes (like the ones in your model) defined as a starting point for their model. Datum planes are displayed as rectangles that are just big enough to enclose the model. They are given names by the system such as RIGHT, TOP and FRONT.
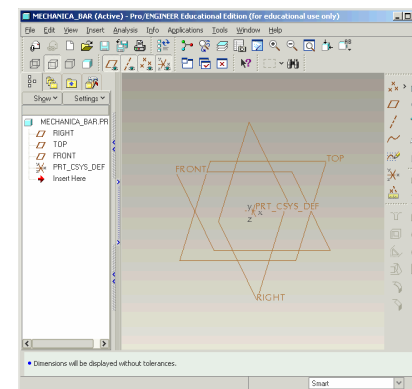

Figure 3 : Start of the Part

Now let's start modelling. Models are made from a series of building blocks called features. This model is so simple it will have only one feature called an extrusion.

Choose INSERT > EXTRUDE from the menu. You should see a new toolbar appear like the one in Figure 4. This is called the dashboard and contains all of the options for the type of feature you are creating.

Figure 4 : The Extrude Dashboard

To start creating this feature click on the first button 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 datum plane TOP by clicking on it in the graphics window or in the browser. 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 should also see a References dialog. References are used by Pro Engineer to locate dimensions. Pro Engineer guesses at suitable references and in this case will have chosen the Right and Front datum's as shown in the main graphics window by the dotted lines. This is a good choice in this case so you can CLOSE this dialog.

You are now ready to use sketcher. Choose the circle tool [ ] or SKETCH > CIRCLE > CENTER AND POINT and draw the circle with two clicks as shown in Figure 5.

Your window should now look like Figure 5 but the diameter of the circle will be different. If the dimension isn't positioned exactly as in Figure 5 don't worry, just choose the select tool [ ] and click and drag the dimension text to a new position. Now 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 20. To end sketching choose

[ ] and click OK in the Section dialog. To complete this first feature type

100 into the depth field of the dashboard (See Figure 4) and click the green tick to finish.
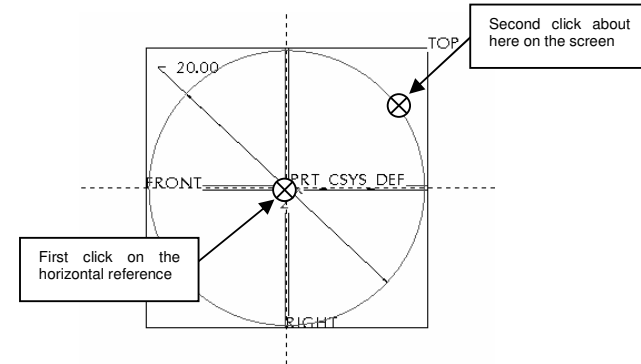

Figure 5 : Outline Sketch

To see this block in all its glory choose the command VIEW > ORIENTATION > STANDARD ORIENTATION and try the different display option icons [ ][ ][ ][ ]. You can also look around your design – press the middle mouse button and move the mouse to spin the model around. Middle mouse button and SHIFT key moves the model around the screen. Middle mouse button and CTRL key zooms into the model – you can use the mouse wheel for this too.
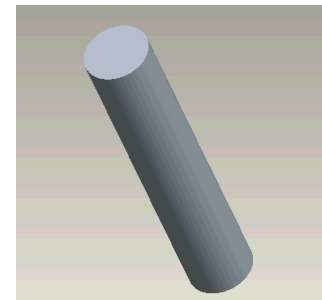

Figure 6 : First Feature

This is all the modelling you will need to do for this very simple part. You are referred to the other modelling tutorials at http://www.staffs.ac.uk/~entdgc/WildfireDocs if you want to learn more.

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.

Pro Engineer has two ways of running analyses. You can run the Pro Mechanica software completely independently or you can run it in integrated mode from within Pro Engineer – this has some small limitations but is much more streamlined. We will use integrated mode. Choose APPLICATION > MECHANICA now to take your model into analysis. Click OK on the box notifying you of the units of your model.

The top level Mechanica menu should appear to the right. If the bottom option in the menu FEM Mode is ticked you are running on a machine that does not have a licence for Pro Mechanica. You will not be able to continue with this tutorial.

Pro Mechanica can undertake different types of analysis. These are

- Motion – Analysis of mechanisms and assemblies.

- 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. From the side menu choose STRUCTURE

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 from the STRC MODEL menu then NEW > SURFACE (or you could just pick the ⬚ icon). The constraint dialog will appear.
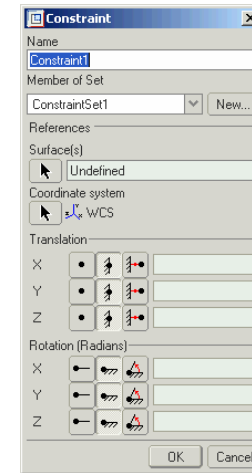

Figure 7 : The Constraint Dialog

Click on ⬚ below Surface(s) then pick one end of the bar 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. Every model to be analysed must be constrained at some point in all six degrees of freedom. Click OK in the constraint dialog to finish.
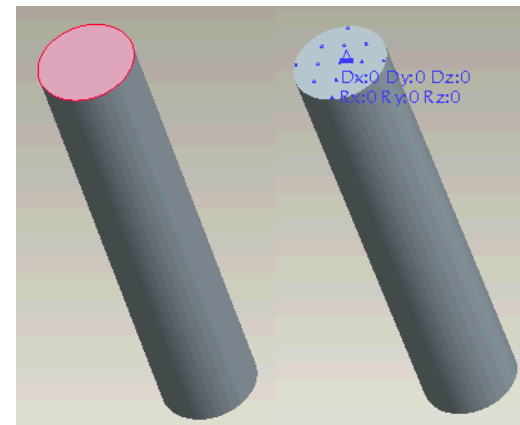

Figure 8 : Constraint Surface

Defining Loads

Definition of loads is similar to constraints. Choose LOADS from the STRC MODEL menu then NEW > SURFACE or pick the [icon] icon to apply a load over a surface. The Force/Moment dialog will appear.
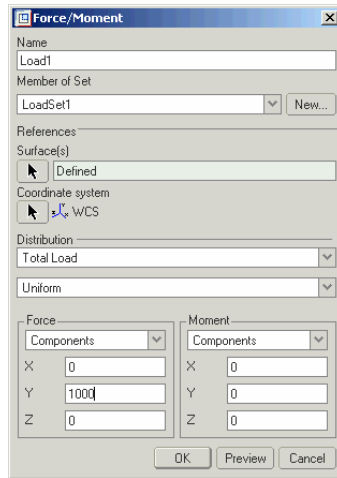

Figure 9 : Force/Moment dialog

Click on [cursor] below Surface(s) then pick the OTHER end of the bar (spin the view with the middle mouse button if you need to) then OK to return to the Force/Moment dialog.
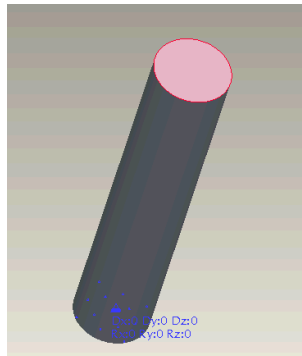

Figure 10 : Load Surface

If you look carefully at the graphics window you should see a symbol displaying the X, Y and Z directions (if you cant see it make sure the [icon] 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 100000 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 dialog to finish.
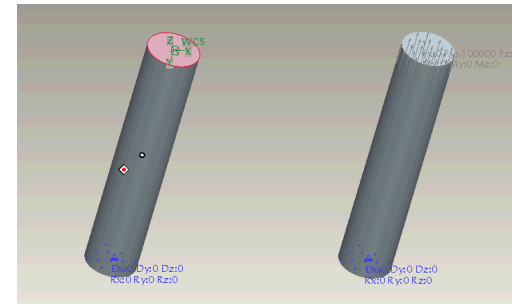

Figure 11 : Load Direction

Defining Materials

The final definition for this analysis is to determine the material for the bar. Choose MATERIALS from the STRC MODEL menu. The Materials dialog will appear. Scroll down the materials library to Find STEEL 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. Press ASSIGN > PART and click on the bar and OK to assign the material. CLOSE the material dialog.
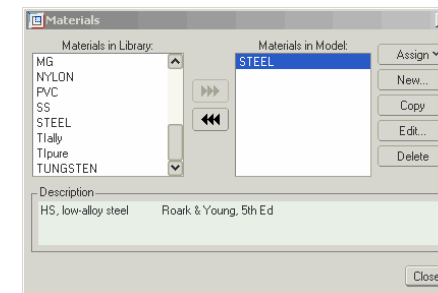

Figure 12 : Materials Dialog

Running an Analysis

That's it you are ready to run an analysis. Choose ANALYSES/STUDIES from the MEC STRUCT menu. The dialog in Figure 13 appears. From this dialog choose FILE > NEW STATIC and type the name TENSION. 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 ![run icon] icon to run this analysis choosing yes for error detection. Press ![report icon] 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.
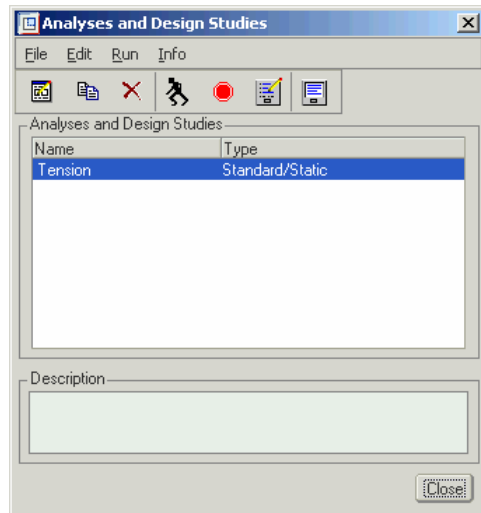

Figure 13 : Analysis Dialog

Seeing the Results

Results are handled in a separate though integrated module of Pro Engineer. Choose RESULTS from the MEC STRUCT menu. The main graphics window will go blank and the menus and icons will all change. Choose INSERT > RESULT WINDOW or the ![icon] icon. In the RESULTS WINDOW DEFINITION dialog that appears press ![folder icon] and click (not

double click) on the folder which is the same name as the analysis that is TENSION. Make sure all the options are the same as in Figure 14 then click OK AND SHOW.
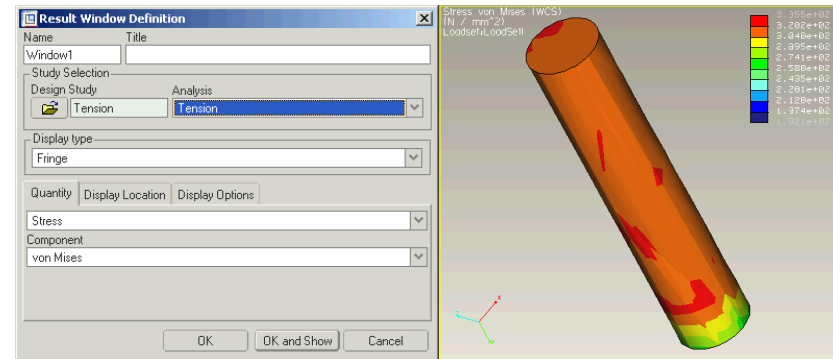

Figure 14 : Results Definition

The resultant plot shows the stress distribution over the whole bar where the colours 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 INFO > 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 3.18 N/mm^2 (notice the units are reported at the top left of main window). Is this value correct? For tension stress is given by load/area. The cross-sectional area of this bar is $\pi \times 20^2/4$ or 314mm^2. The load we applied was 100000N so the stress should be 100000/314 or 318 N/mm^2. Spot on - even though this was only a quick single pass adaptive check!

What else can we show? Choose EDIT > RESULT WINDOW to bring back the dialog in Figure 14. Below QUANTITY change STRESS to DISPLACEMENT then OK AND SHOW. Again a coloured plot appears with the colours 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 0.00158mm.

We can combine the display of displacement with stress in a very interesting and informative way. Choose EDIT > RESULT WINDOW to bring back the dialog in Figure 14. Below QUANTITY change DISPLACEMENT back to STRESS then on the DISPLAY OPTIONS tab

tick DEFORMED and ANIMATE. OK AND SHOW should now show the original stress plot on a model that is stretching as the load is applied. You can control the animation with the icons at the top

▶ ■ |◀ ▶| Slow [____|___] Fast .

Additional Analysis – Bending

Let's try a different analysis. We will apply a bending moment to the bar rather than a tensile load. Edit the load using LOADS > EDIT from the STRC MODEL menu and pick the tensile load. You should see the Force/Moment dialog (as in Figure 9). Change the load value to 0 in the Y direction to 1000 in the Z direction. Check the load direction is as shown in Figure 15.
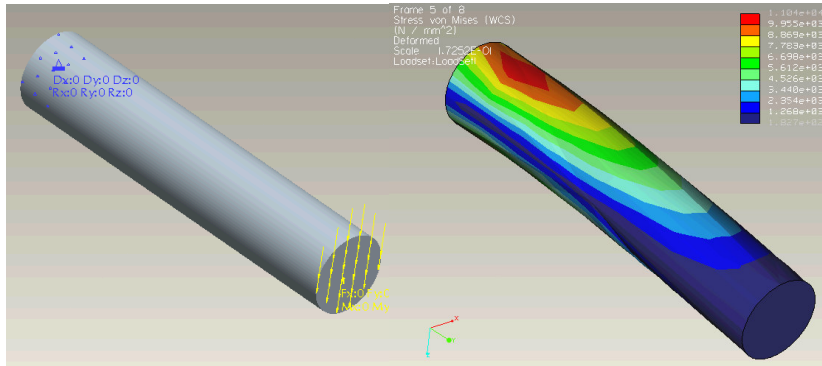

Figure 15 : Bending Moment

Re run the analysis and view the results. Note the value of the maximum stress. What should this value be?

$$\text{Bending Stress} = My/I$$

$$\text{Second Moment of Area (I) for a circular beam} = \pi d^4/64$$

$$= \pi 20^4/64$$

$$= 7854mm^4$$

$$\text{Bending Stress} = 1000*100*10/7854$$

$$= 127N/mm^2$$

Do your results compare? You may find they are close but not exact. The analysis you performed was a quick single pass adaptive analysis. Just by

looking at the results you will see that the maximum value does not occur at the end of the bar where it should. If you were to change to a multi pass adaptive analysis (by choosing EDIT > ANALYSIS/STUDY in the dialog in Figure 13) you should see improved results – especially if you decrease the percent convergence parameter and increase the maximum polynomial order. Of course you would have to re run the analysis if you change these parameters.

Additional Analysis – Torsion

To apply a torsional load we will need to create a cylindrical coordinate system. Choose FEATURES > COORD SYSTEM > CREATE from the STRC MODEL menu. 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 16.
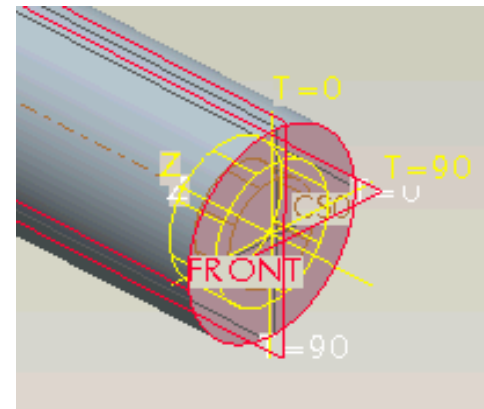

Figure 16 : Cylindrical Coordinate System

Edit the load using LOADS > EDIT from the STRC MODEL menu and pick the bending load. Click on ↖ below coordinate system 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. Next change the distribution to TOTAL LOAD AT POINT and click on ↖ below DISTRIBUTION. You will be given the chance to SELECT or CREATE a point. Choose CREATE and pick the cylindrical coordinate system to create a point at the origin. Enter a value of 100000 for Z in the MOMENT column and 0 n all the other FORCE fields. Preview the load direction is as shown in Figure 17.
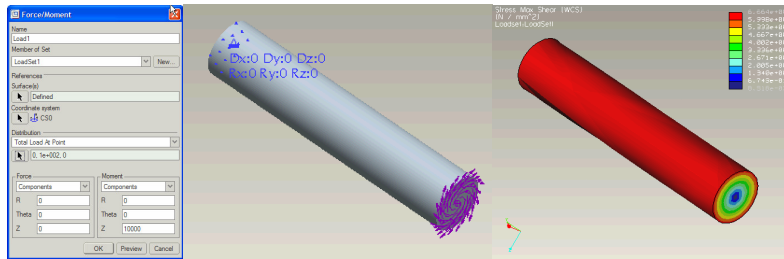
Figure 17 : Torsional Moment

Re run the analysis and view the results for maximum shear stress. Note the value of the maximum stress. What should this value be (Torsional Stress = Tr/J)?

Additional Analysis – Natural Frequency

Another analysis type that can be undertaken by finite element analysis is to determine the natural frequency of vibration. This type of analysis does not require any loads to be applied (if any are applied they will be ignored). You doe have to create a new analysis to perform this type on analysis. Choose ANALYSES/STUDIES from the MEC STRUCT menu. The dialog in Figure 13 appears. In this dialog choose FILE > NEW MODAL and type a name. Having created the analysis choose the 🏃 icon like before. The results for this analysis show the vibrations which will occur at different frequencies. Create four different results windows, one for each of the four modes of vibration calculated. The first two are very similar vibrating in the vertical and horizontal planes. The third is a torsional vibration and the fourth is a second order vibration.
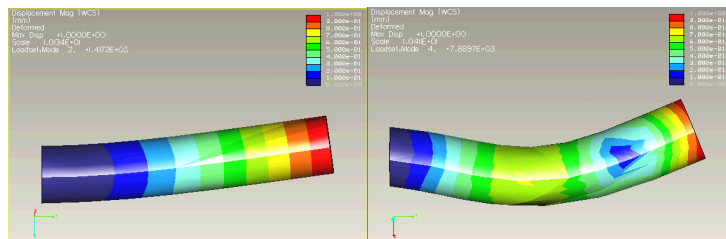

Figure 18 : Modal Frequency Analysis

From Blevins' "Formulas for Natural Frequency and Mode Shape", the natural frequency of a clamped-free beam is:

$$F = (\lambda^2 \sqrt{(EI/M)}) / (2\pi L^2)$$

where: λ = 1.875 for first mode
L= length of cantilever (m)
E= modulus of elasticity (N/m^2)
I= area moment of inertia (m^4)
m= mass per unit length of beam (kg/m)

Which calculates the first natural frequency as 1420 Hz

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.

Where Next?

Here is a more complicated model of a steel bracket which you can download from http://www.staffs.ac.uk/~entdgc/WildfireDocs under the bracket link. Try applying a 10000N vertical load to the circular hole and constraining the back surface as though it were glued to a wall.
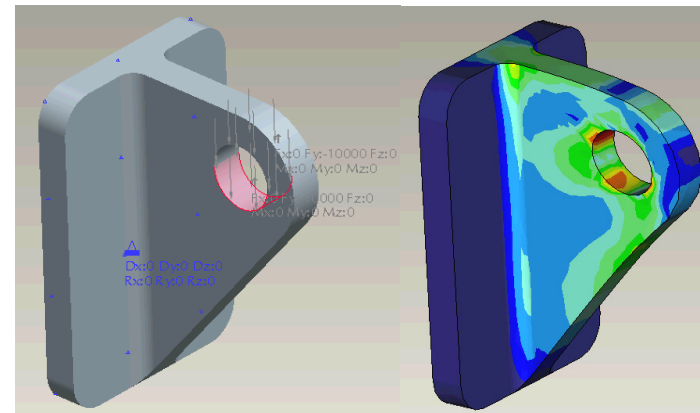

Figure 19 : FEA Analysis Exercise