


Here are some answers to questions that are often asked. The explanations are brief as you are expected to be a competent user of ProEngineer.

Underlay

It is a common technique to use a photograph as an underlay or guide when sketching in a CAD model. Though not an explicit function of Pro Engineer this can be achieved with a little work as follows.

Before you start you will need one or more orthographic digital pictures of a product (photographs of a real object or scanned sketches of your own design).

Create a new part using the mmns_part_solid template. Insert a new sketch (INSERT > MODEL DATUM > SKETCH ) and draw a simple rectangle on the FRONT datum. Adjust the size of this rectangle to be the overall size of part to be modelled.

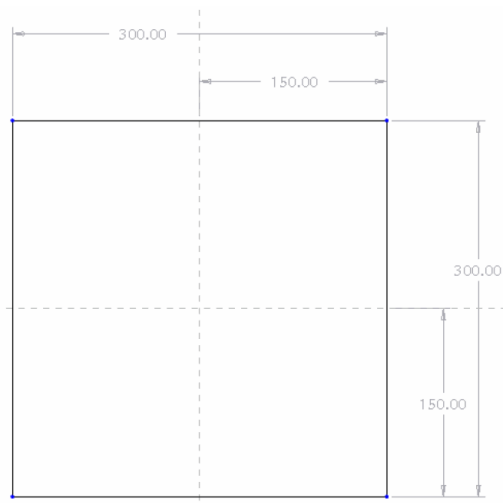


Figure 1 : Outline Sketch

The next step is to fill this rectangle with a surface. Make sure the sketch is selected in the model tree then choose EDIT > FILL. A surface will be created to fill in the rectangle – it will be more visible if you shade the model.

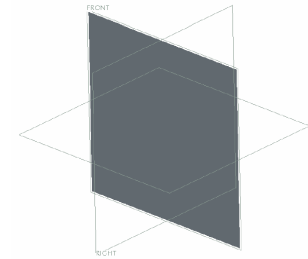


Figure 2 : Filled Surface











Now we need to associate a picture with this surface. Choose View > Color and Appearance to show the materials palette. In this dialog choose MATERIAL > NEW  and change the name of the material to FrontPicture . Click on the MAP tab  and click on the square button next to DECAL . At the top of the dialog choose TEXTURE > ADD  and using the dialog that appears to locate the texture. This can be any of the common picture file formats (bmp, jpg etc) but you are strongly advised to copy the picture file to the same directory as the model before you apply it. After loading the picture click on its name  to apply it to the surface and CLOSE  the Appearance Placement dialog. In the Advanced tab  of the Appearance dialog you may wish to adjust the Transparency  to say 80% so the texture is see through. Finally below Assignment choose Surfaces  and pick the fill surface choose OK and BOTH. You can now use this picture as an underlay when drawing. You can visually sketch on it but you cannot lock the points onto the picture.



Figure 2 : Completed Underlay

You may want to repeat this procedure for the other datum planes adding pictures of the model from other directions.

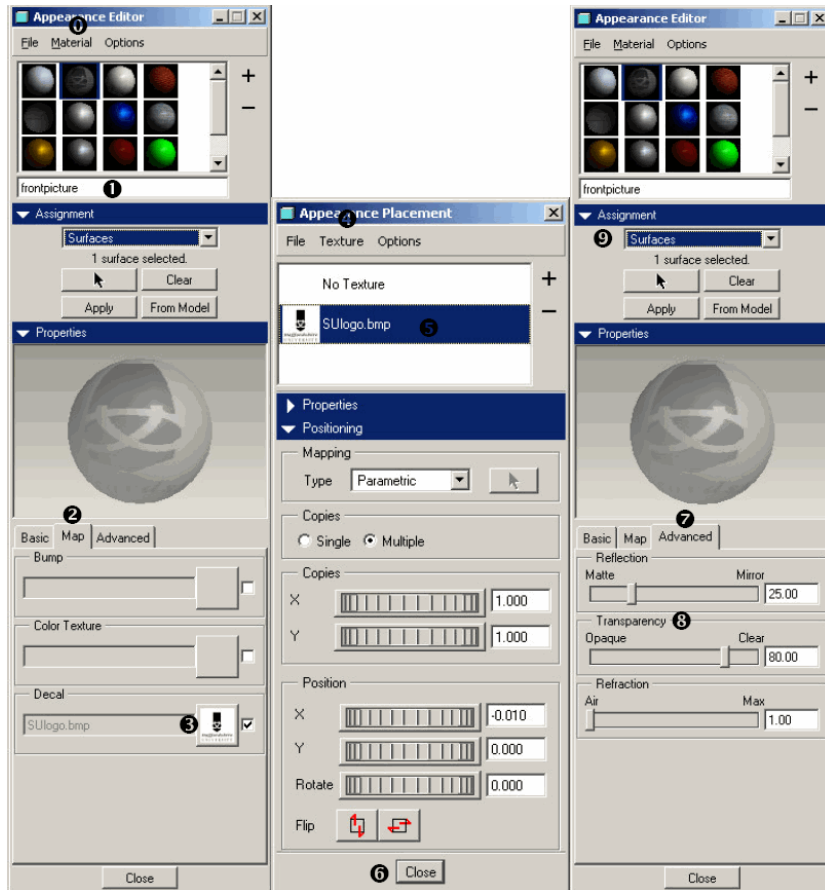


Figure 3 : Dialog Boxes

Tubes

Creation of planar (2D) tubes in Pro/ENGINEER is easy using the Variable Section sweep tool. What is not so obvious is how to create tubes that bend in all 3 dimensions. Here are two methods...

Method 1 – Multiple 2D tubes

The first method recognises that many 3D tubes can be made up of several planar tubes connected at their ends. Think of a traditional racing bicycle handle bars. These can be made up of two planar curves as shown in Figure 4. This shows two curves being drawn on separate datum planes. The ends of these curves meet and they are tangent. The tube was made using the Variable Section sweep tool and the References > Details button was used to add both curves to the Origin chain.

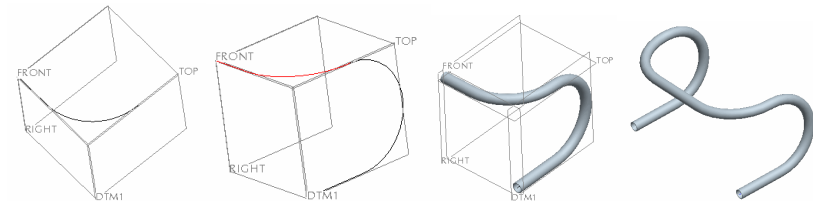


Figure 4 : Multiple 2D Tubes

Method 2 – Tubes Thru Points

If the shape you want is a genuine 3D shape, use this procedure...

First create a series of datum points to control the shape. Use INSERT > MODEL DATUM > POINT > OFFSET COORDINATE SYSTEM and pick on a coordinate system as a reference. Now enter the coordinates shown in Figure 5.

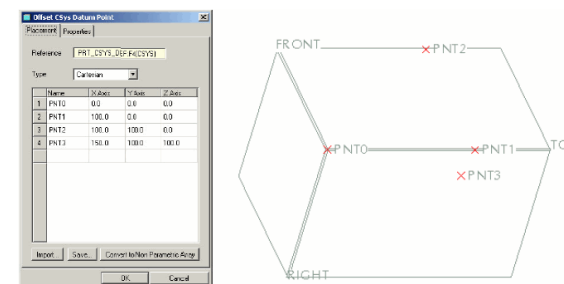


Figure 5 : Adding Datum Points

You can now use these points to create a datum curve using INSERT > MODEL DATUM > CURVE (not SKETCH). Choose THRU POINTS | DONE then pick one of the points (since they are a group they will all be selected). Choose DONE and OK in the Dialog box to create the curve.

Figure 6 shows the difference between the curve type options. Now use the Variable Section sweep tool to create a tube from this curve.

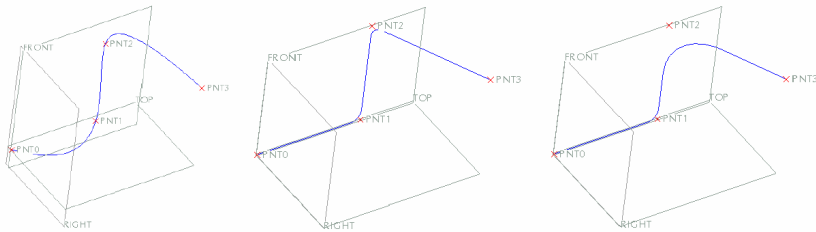


Figure 6 : Curve Types (Spline, Single Rad, Multiple Rad)

Springs

There is a special function for creating springs in ProEngineer. Choose INSERT > HELICAL SWEEP > PROTRUSION. From the side menu choose the parameters you want for the spring CONSTANT | THRU AXIS | RIGHT HANDED | DONE. Next choose a sketching plane such as the FRONT datum then OK and DEFAULT. You should now be in sketcher. Draw a line representing the outside shape of the spring. In the case of the 80 dimension represents the length of the spring and the 20 is half the diameter of the spring. You will also need to draw a centre line for the spring. After leaving sketcher you will be asked for the pitch of the spring – type in 20. You will now be back in sketcher this time to draw the cross-section of the wire from which the spring is made – draw a 5 circle on the end of the line. Exit sketcher and choose DONE to see the spring.

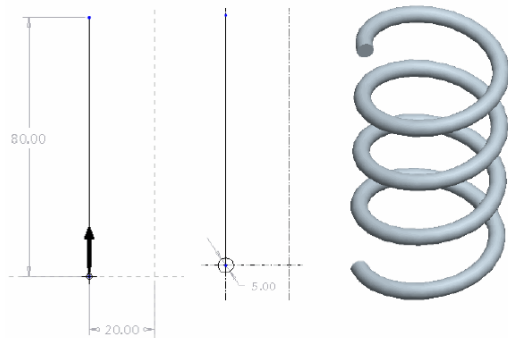


Figure 7 : Springs

If you change the length of the spring now by editing the 80 dimension to say 100 you will see that an extra coil is added. The spring hasn't stretched as you might expect. To make the spring act more realistically you can build in some relationships. Choose TOOLS > RELATIONS to see the dialog in Figure 8. Click on the spring in the main window and choose to show ALL dimensions. The dimension names – not their values will be shown. Notice that the pitch of the spring is d2 and the length is d0 (if yours are different use your values. Enter the formula shown in into the dialog. Click OK and now when you change the length of the spring the coils should stretch just like the real thing. Can you work out what this formula is doing? (Hint there are 4 coils in this spring)

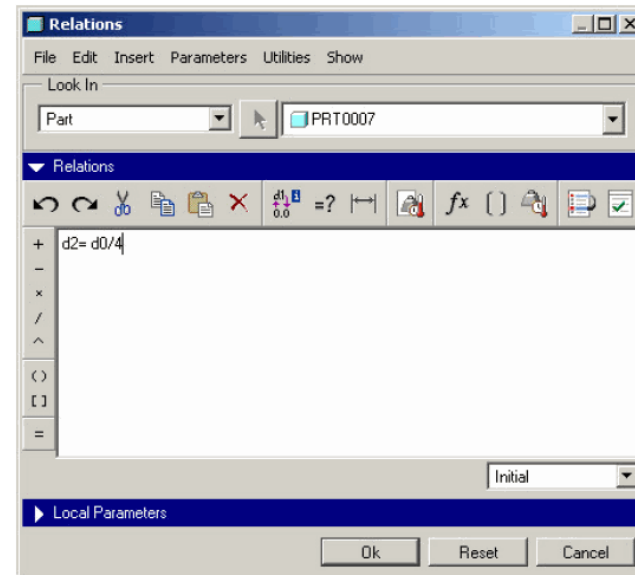


Figure 8 : Spring Relationships