Often when creating features in a model other features, created earlier, are referenced. These references are called parent child relationships. For example a hole cut into a block clearly references the block! The block is the parent the hole is the child. The child cannot exist without the parent. If the parent is removed the child must go too, or else another parent must adopt the child. It is important to understand the hierarchy that is being created in this way, as with a little thought it can be used to help your design and not hinder it. This is particularly true when the design requires modifying.

Relationships are created all the time within Pro Engineer. It is often possible to design a feature in several ways, with apparently the same result, but creating different parent child relationships. Some of these methods will capture the design intent better than others. As an example a hole may be created in another circular feature, a boss. The hole could be dimensioned independently of the first feature. If the boss is moved the hole will not move as well since there are no parent child relationships to the first feature. If however the hole is made concentric to the boss a relationship is built in that describes the design intent – the hole and boss are intended to always be concentric. Now the hole moves with the boss to maintain the concentric relationship.

Parent Child Relationships

This tutorial is designed to show that solid modelling in a parametric system need not be a rigorous fully structured procedure. Good technique can allow flexibility in the design process. It is assumed that the reader has already completed the previous modelling tutorials and is competent at creating models.

Start by creating a new part called COVER using the mmns_part_solid template. Next create an extruded protrusion as shown in Figure 1. The protrusion should use the TOP datum as the sketching plane and the FRONT datum as the BOTTOM reference and be created to a depth of 100.

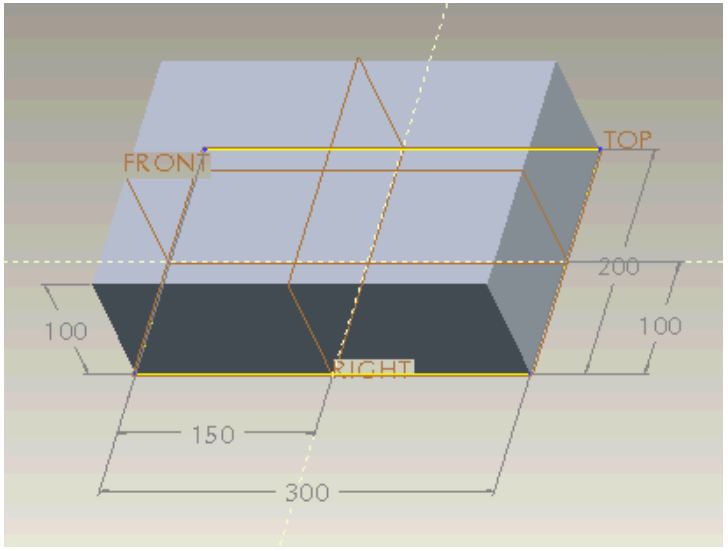

Figure 1 : The Base Protrusion

What parent child relationships have been created in this feature if any? As previously stated some dimensions (usually location dimensions) create relations. Which of the dimensions you entered have created a relationship? Which dimensions referenced other features for location? The 100 and 150 dimensions control the position of the box relative to the FRONT and RIGHT datums so a parent child relationship exists there. Is there any relationship to the TOP datum? You chose the TOP datum as the sketching plane (and the FRONT datum as a bottom plane) which also creates a relationship. So your block is related to all of the previous features! None of the datum planes could be deleted without deleting the block. You can prove this by choosing from the pull down menu INFO > PARENT/CHILD and picking on the block. A window appears as shown in Figure 2 which states that the block has no children but its parents are the TOP, RIGHT and FRONT datums.

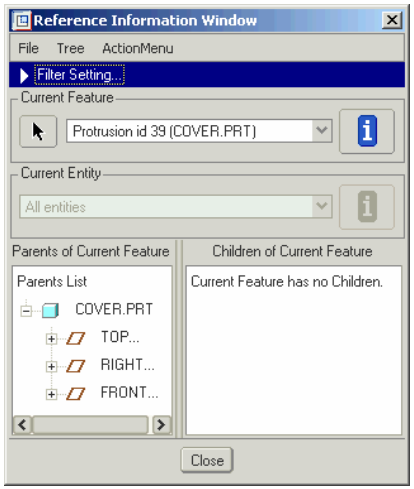

Figure 2 : Information On Parent Child Relationship

You may be already aware that the dimensions assigned to any feature are not fixed. Their value can be changed at any time by using the EDIT or command in the pop up menu. As a reminder – right click on the protrusion in the feature tree and choose EDIT. All of the dimensions used to define the block will be displayed. Double clicking on these dimensions will let you change the value. The modified value will be displayed in green indicating the change. The modification will not affect the 3D model until you choose EDIT > REGENERATE. Try this now by changing the 100 thickness of the block to 200. Regenerate to see the change and then change it back to 100.

Next create a second protrusion for a flange. The flange should use the TOP datum as the sketching plane and FRONT as the BOTTOM reference and should be created in the same direction as the first protrusion maintaining the overall height of 100. The sketch for the flange (Figure 3) should be created by using the offset edge icon  . The LOOP option and click on the top surface of the first block allows all four sides to be offset in one go – choose and offset of 10. Finish the sketch and choose the BLIND option with a length of 10.

This is an example of another type of parent child relationship. The use edge and offset edge both reference existing geometry and so a parent child relationship is formed.

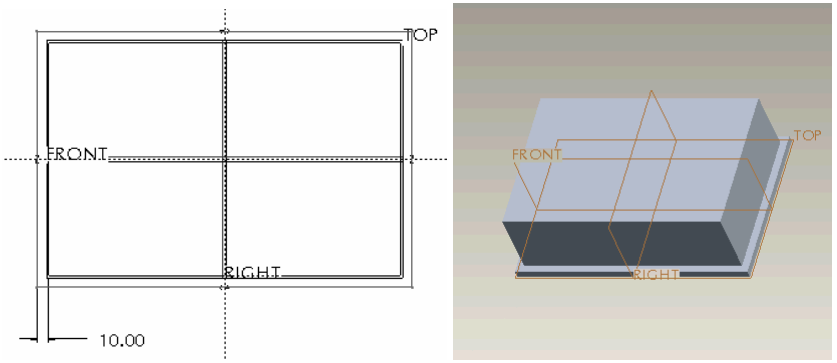

Figure 3 : The Flange

Reordering Features

Since this part is going to be a cover the centre needs hollowing out. A cut could be used for this but Pro Engineer has a special feature for this purpose. It is called a shell feature – you may have met it before. Use INSERT > SHELL and pick on the bottom most surface in Figure 3 enter a shell thickness of 10. The surface you picked will be removed and all of the remaining surfaces will be offset by 10 to make the shell.

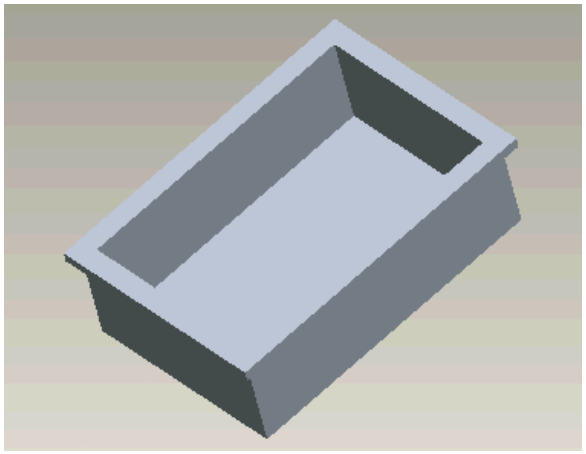

Figure 4 : The Shell

The cover is full of sharp corners so add rounds (INSERT > ROUND). There are eight edges to be rounded all around the outside of the cover. There are four around the top and the four vertical sides so you will need to hold the CTRL key whilst selecting them. Enter a radius of 25 for all rounds.

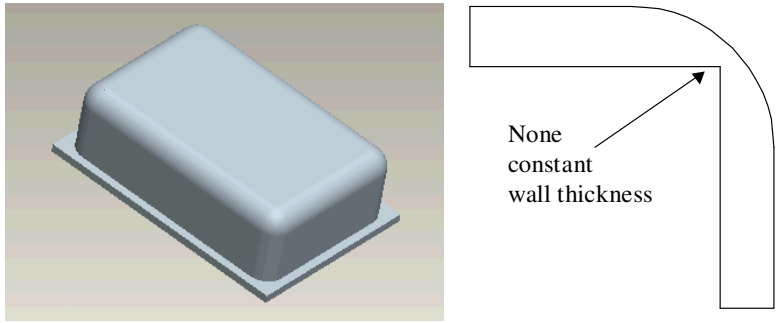

Figure 5 : Rounded Corners

The problem with this is that rounding the outside edges does not round the inside edges! The wall thickness is no longer constant. Ideally the rounds should have been added before the shell feature. Do we have to delete the shell and add it again after the round? No. The order of features can be changed – within the bounds of parent child relationships since you cant place a child before its parents. There are no parent/child relationships stopping this move. To reorder a feature click and drag the name in the feature tree. Drag the last round feature up the feature list – if you try and drag it before its parents the feature names will be highlighted in blue. The model will regenerate with internal and external rounds and the thickness of the whole model will be the same – just like you had added the rounds before the shell.

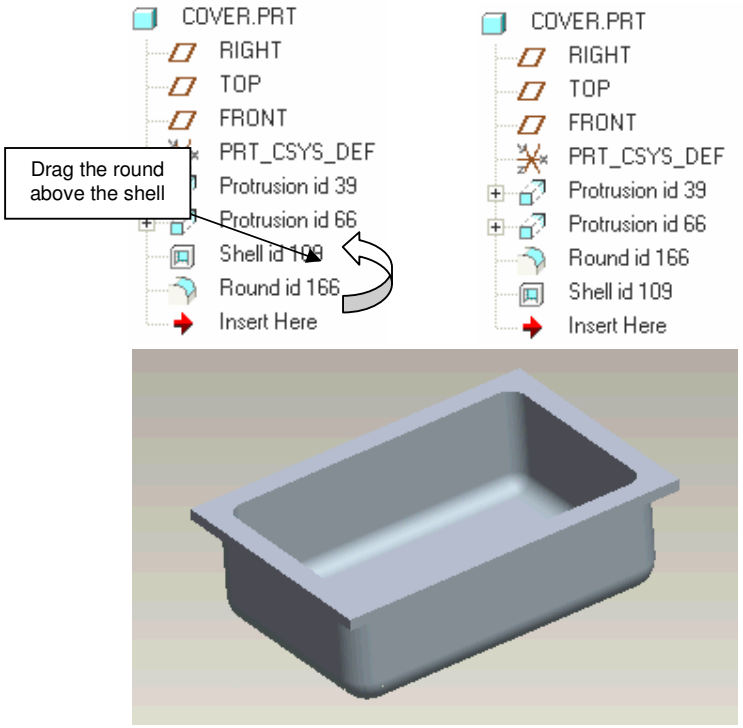

Figure 6 : Features Reordered

Inserting Features

The next step is to add a circular boss protruding from the top surface of the cover. If this is created as an extruded protrusion then once again there will be a material thickness discrepancy because the boss has been created after the shell. We could add it now and then re-order it to the correct position but since you noticed this problem early (you did didn't you!) there is an alternative method. The new feature can be inserted into the tree by dragging the Insert Here reference in the feature tree to below the second protrusion. The model will be taken back in time to the point before the rounds and shell were added. Now add the boss using the dimensioning scheme shown in Figure 7. The 80 and 30 dimensions reference the FRONT and RIGHT datums. The boss thickness is 15. After creating the boss bring back the rest of the model by dragging the Insert

Here reference in the model tree to the end of the list. The model will be regenerated with the boss before the shell maintaining a constant wall thickness.

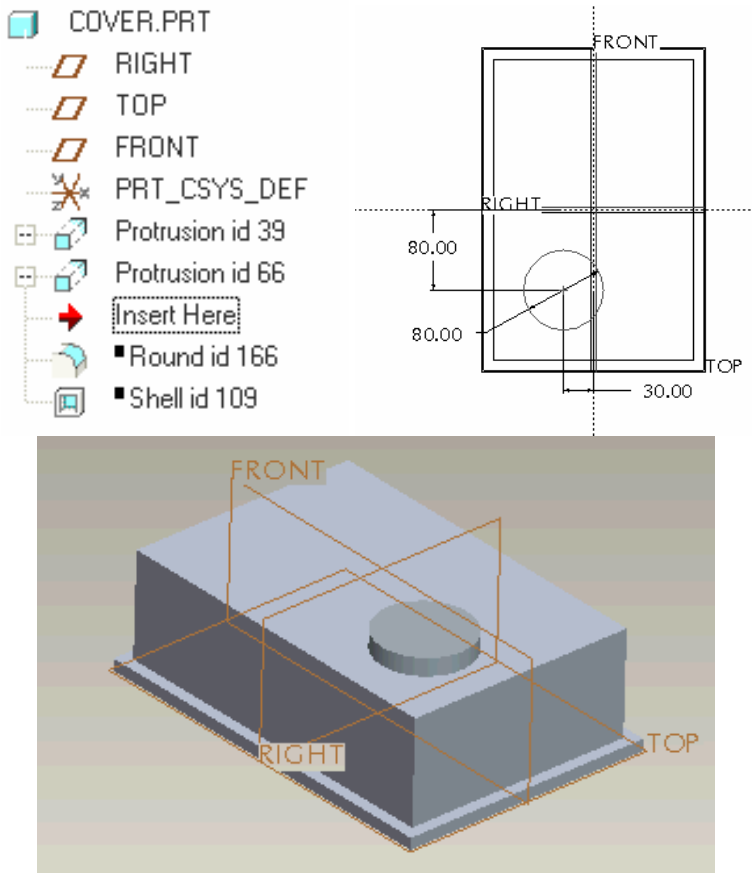

Figure 7 : Boss Dimensioning Scheme

Drag the Insert Here reference in the feature tree to the bottom of the list to continue modelling in the normal manner. Now complete the model with a hole through the boss. This should be an EXTRUSION using the THRU ALL option to remove material and should be created with the dimensioning scheme shown in Figure 8. This is an identical scheme to the boss and is not the obvious way to do it, it's not even the correct way of doing it – but it illustrates a point!

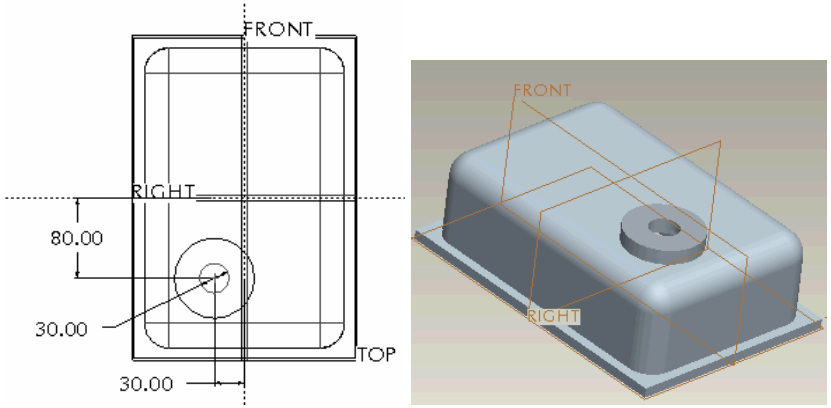

Figure 8 : Hole Dimensioning Scheme

Now further down the design cycle it is found that this boss (and it's hole) need to be moved. No problem! Right click on the boss in the feature tree (the third protrusion) and choose EDIT and change the 80 length to 60. Regenerate the model and all's well! Not quite – try it!

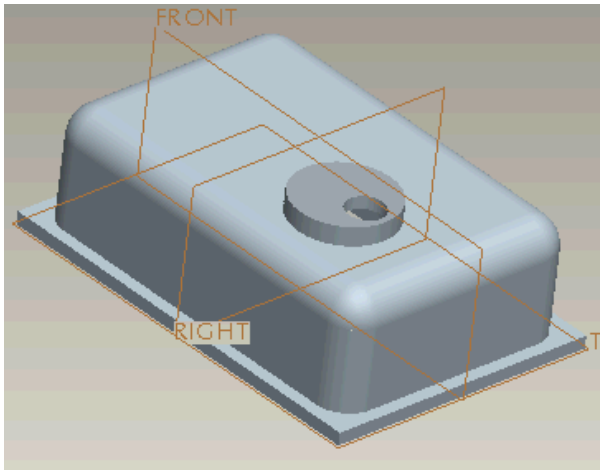

Figure 9 : The Boss Has Moved But Not The Hole!

The hole hasn't moved because it was dimensioned independently of the boss. No parent/child relationship was established even though this would be good practice in this case. You could just modify the dimensions of the hole as well but let's change the model to *capture the design intent*.

First modify the (now) 60 back to it's original 80 and EDIT > REGENERATE so the boss is back to its original position. The hole needs the dimensioning scheme changing. To do this, right click on the hole in the feature tree (the cut) and choose EDIT DEFINITION. Enter the sketch mode with PLACEMENT> DEFINE and delete the circle. Create a new circle this time using the concentric option  . Choose the icon click on the circle around the boss and click again to place the circle. You will notice that no linear (positional) dimensions are created because the circular hole is aligned to the boss thereby creating a parent/child relationship. End the sketch and the definition. Now try making the modification again and the hole should move with the boss.

Adding Draft Angles

Finally to show you the power of what you have learnt and as an excuse to introduce a new feature type lets assume that to allow the part to be removed from the injection moulding machine easily we need to angle the sides. These are called draft angles. Again to ensure we keep a constant thickness we need to add the draft before the shell. Drag the Insert Here icon to below the second protrusion in the feature tree. Now choose INSERT > DRAFT or the  icon.

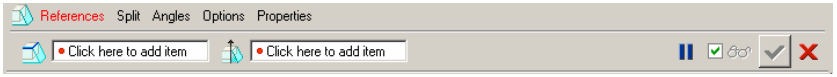

Figure 10 : The Draft Dashboard

In the dash board click on References menu and the options in Figure 11 will be shown. First click in the Draft surfaces pane then select with the CTRL held the four vertical walls of the cover (see Figure 12). Next click in the draft hinges pane and click on the large top surface (see Figure 12).

Type in a draft angle of 2 and click on  to change the draft direction if necessary.

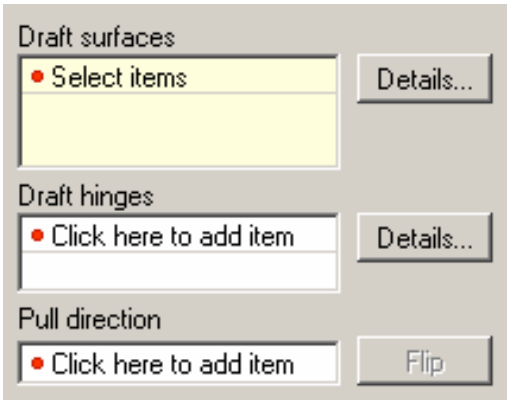

Figure 11 : Draft References

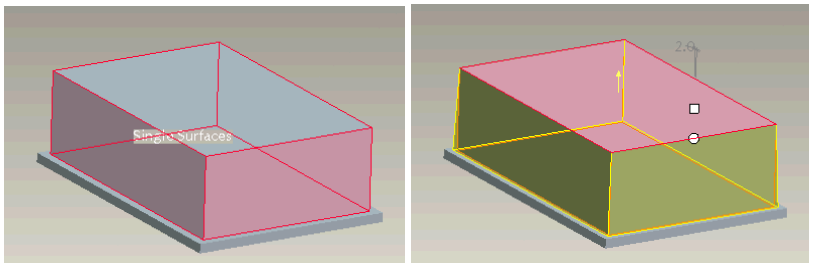

Figure 12 : The Draft and Hinge Surfaces

Drag the Insert Here reference in the feature tree to the bottom of the list to continue modelling in the normal manner.

Review

So what should you have learnt?

- Understand Parent Child Relationships.
- How to reorder feature.
- How to insert features.
- How to create Draft Features.

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

What Next?

You may wish to experiment yourself with these techniques whilst modelling the shape below. Scale or estimate all dimensions. Notice that most faces are angled to aid moulding so you will need to use Draft Features or other techniques.