

An assembly is a collection of parts oriented and positioned together. As such it is the highest level of data that can be manipulated within ProEngineer as shown in Figure 1.

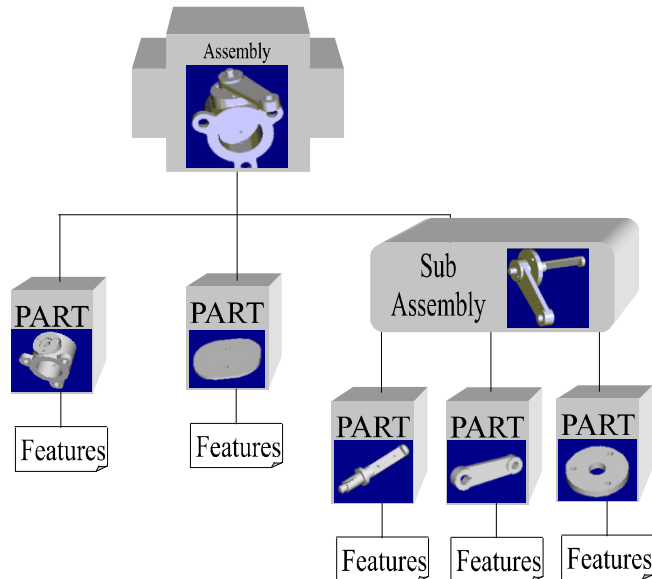


Figure 1 : Structure of ProEngineer Models

The first part placed in an assembly is known as the base component. All other parts are assembled to this part and are located by applying constraints. A constraint is a restriction of movement. Any part located in free space has six degrees of freedom, three translational (movements along x, y and z axes) and three rotational (rotations around x, y and z axes). For a part to be completely fixed in space all six degrees of freedom need to be constrained. ProEngineer allows components to be assembled together without constraining all six degrees of freedom – so a shaft can be left free to rotate in a hole.

Constraints are applied by using typical engineering metaphors. For example, two planar surfaces can be mated together so that they touch. This effectively restricts movement in 3 degrees of freedom (2 rotational and 1 translational). The six most common constraints are described

diagrammatically in Figure 2. It may be interesting to work out what degrees of freedom are constrained by each.

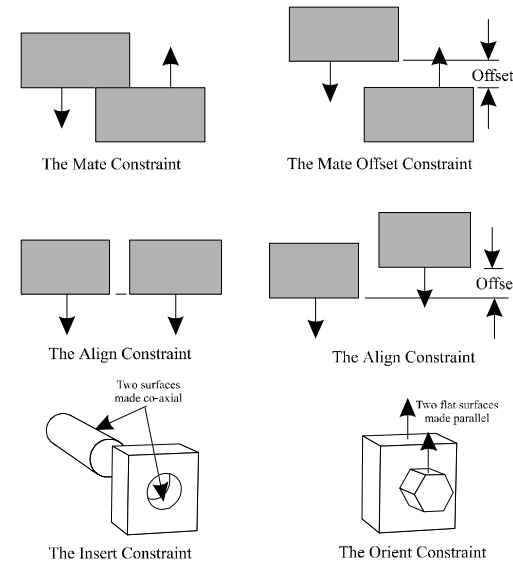


Figure 2 : Constraint Options

The first stage in assembling two components is to determine how you would assemble the parts in real life – “I would push that shaft in that hole until this face butts up against this face”. Then you have to translate these words into the types of constraints that Pro Engineer offers – “INSERT the shaft in the hole then MATE the two faces”. Then the constraints are applied to the new part. In each case, the constraint will require a location on the assembly and a location on the component to be chosen. For example, if a mate constraint is chosen two planar faces (or datum planes) need to be chosen - one on the assembly and one on the component. The assembly constraints are maintained even if modifications occur to the original parts.

The assembly file itself is saved with a .ASM file extension. This file does not contain any geometry defining the components. It contains references to the original part files. If the original part files are deleted, moved or renamed the assembly model will report an error and will not open correctly.


Creating a Sub Assembly

If you have not already been given them, the parts for the valve assembly used in this tutorial can be found in the same directory on the Web as this document (<http://www.staffs.ac.uk/~entdgc/tutorials.html>). All 5 parts should be copied to your local system before you start.

A sub assembly is a small collection of parts which are assembled first then later they are added to the main assembly. As an example a car engine would be completely assembled first before it is added as a single unit to the car on the main assembly line. Pro Engineer can treat any assembly as a sub assembly.

Create a new assembly component using NEW from the FILE drop down menu choosing the Assembly option and giving a suitable name - in this case *valve_sub*. Choose the empty template and click OK. A new window contain default assembly datums will be displayed into which parts will be assembled.

The first part in the assembly is known as the base component. It is usually obvious which part in your assembly should be considered as the base part since other parts are attached to it. Choose the command

INSERT > COMPONENT > ASSEMBLE or  icon and pick the base part called *valve_shaft* from the file list. The base part will now be shown in the assembly window. Since this is the first component it is automatically located at the default position. (If you had used a template which contained some datums then this would not have been the first component and you would have had to locate the part using the techniques you are about to learn).

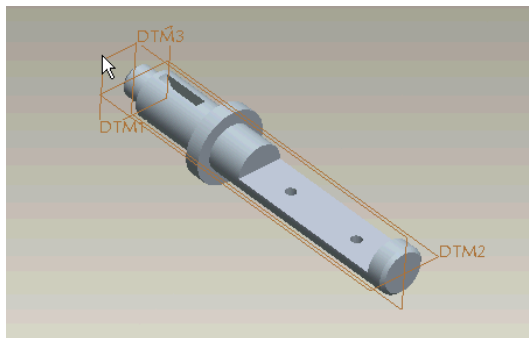


Figure 3 : The shaft in default position

The second part can now be placed using INSERT > COMPONENT > ASSEMBLE as before. Choose the name of the second component *valve_cover*. The part will appear at some random position in the assembly and the Component Placement Dialog box is shown ready to add the first constraint.

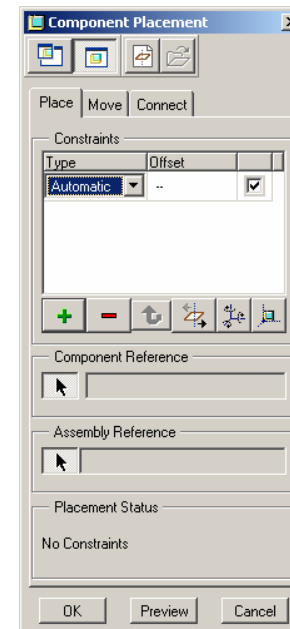


Figure 4 : The Component Placement Dialog

It is now time to apply constraints to the *valve_cover*. As each constraint is applied the window updates to show the constraint and states whether the component is sufficiently constrained to be placed. The *valve_cover* needs to have (at least) two constraints. You will see that under Constraints Type in the dialog there is a drop down list. This list contains the constraint types. It also has the option of Automatic. If Automatic is selected (the default) Pro/ENGINEER will decide on the type of constraint based on the type of geometry you choose. Leave the choice as Automatic and pick on the central hole in the cover. Now pick on the shaft on the 14mm dia. near where the keyway is located (see Figure 5). This adds the first constraint, which is reported in the component placement dialog box. Since you have chosen 2 cylindrical surfaces Pro/ENGINEER assumes you want an Insert constraint.

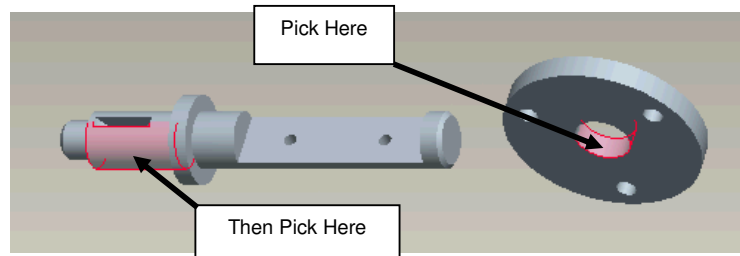


Figure 5 : The First Constraint

This is not enough to place the cover. A second constraint needs to be added. Pick on one of the flat faces of the cover then pick on the flat face of the largest shoulder of the shaft. Type in a distance of 0.

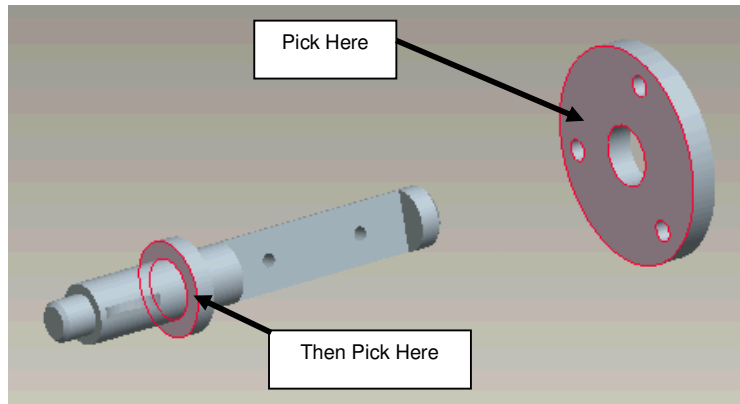



Figure 6 : The Second Constraint

This adds the second constraint, which may be reported in the component placement dialog box as an Align type. This is not correct it should be a Mate type. You can see the difference from Figure 2. This error is easily corrected by pressing the change orientation icon . This is now sufficient constraints to place the cover as you can tell since Fully Constrained is reported at the bottom of the dialog box. Choose **OK** to place this second component as shown in Figure 7.

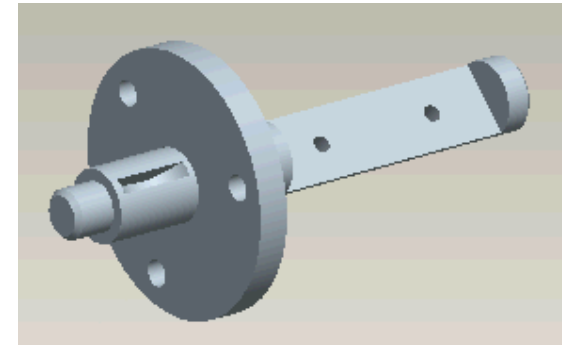


Figure 7 : Assembly With 2 Parts

The remaining component can now be placed by following the same procedure for starting with **INSERT > COMPONENT > ASSEMBLE**. Place the *valve_arm* with three constraints. For the first constraint pick the shaft on the 14mm dia. then pick the hole in the arm. This will be reported as an Insert.

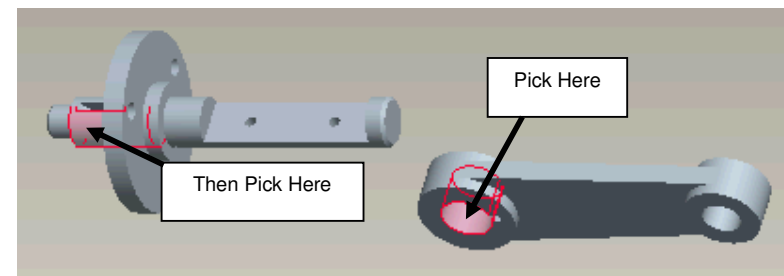


Figure 8 : Constraint 1

Next pick one of the flat faces of the arm with the front face of the first step in the shaft typing a distance of 0. This will be reported as an Align constraint.

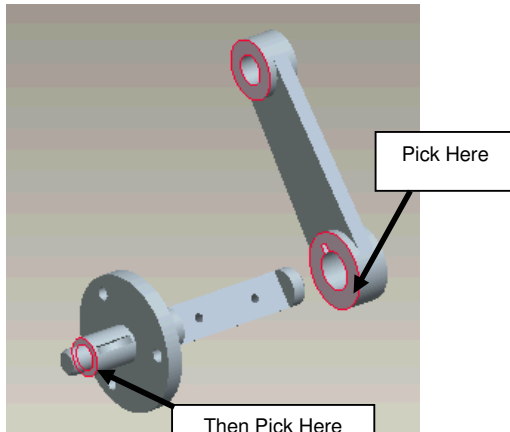


Figure 9 : Constraint 2

The component will be reported as fully constrained but this assumes that the rotation of the handle around the shaft is not important. In this case it is important because the keyways must align (unfortunately the default positions of the two parts happens to make the slot and keyway align but this is purely coincidental!). This can be achieved with a further constraint by pressing the add constraint icon . We now need to pick the side of the keyway and then the side of the slot. But these faces are 'inside' the model and so cannot be picked in the normal way. Pick the side of the keyway using the RIGHT mouse button and one of the outside surfaces will be selected. Keep clicking with the RIGHT button until the correct face is selected then press the LEFT button. Repeat this for the side of the slot

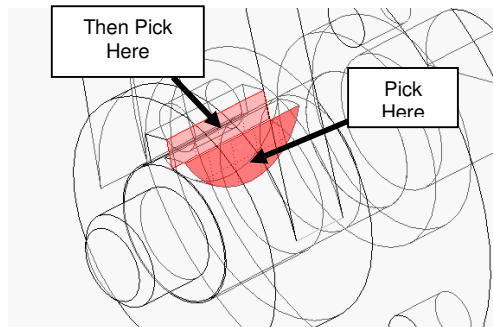


Figure 10 : Arm Orientation Constraint.

The first stage of this assembly is now complete as shown in Figure 11. Don't forget to save the assembly using FILE > SAVE.

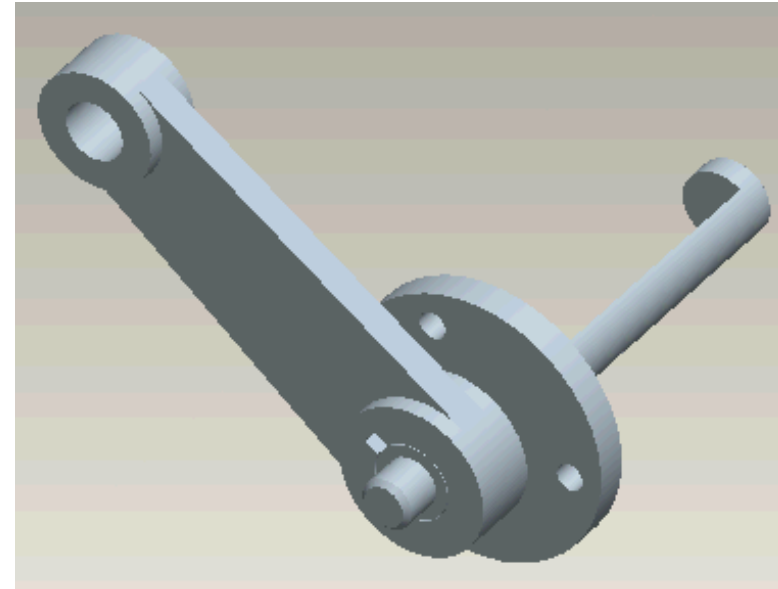


Figure 11 : Assembly after 3 parts

Creating Assemblies

Any assembly created within ProEngineer can be used as a subassembly within a larger assembly. In fact the assembly that was created in the tutorial earlier is going to be used as a sub-assembly. So lets now create the main assembly for this valve.

Create a new assembly called *valve* using FILE > NEW. Choose the *Empty* template. Add the base component, called *valve_housing*, using

INSERT > COMPONENT > ASSEMBLE or the icon. Having successfully placed the base component the next stage is to add the other parts and apply constraints to each in turn. Using INSERT > COMPONENT > ASSEMBLE, assemble the following parts with the constraints specified.

1. *VALVE_SUB* (the sub assembly you made earlier) with three constraints. Pick the shaft on the 14mm dia. and the hole in the

top of the housing. This will be reported as an Insert type. Now pick bottom face of the cover in the sub-assembly and the top of boss in the housing. If necessary change orientation using to make a Mate type. Add another constraint using and pick one of the 3 holes in the cover with the matching hole in the housing. This will be an Insert type.

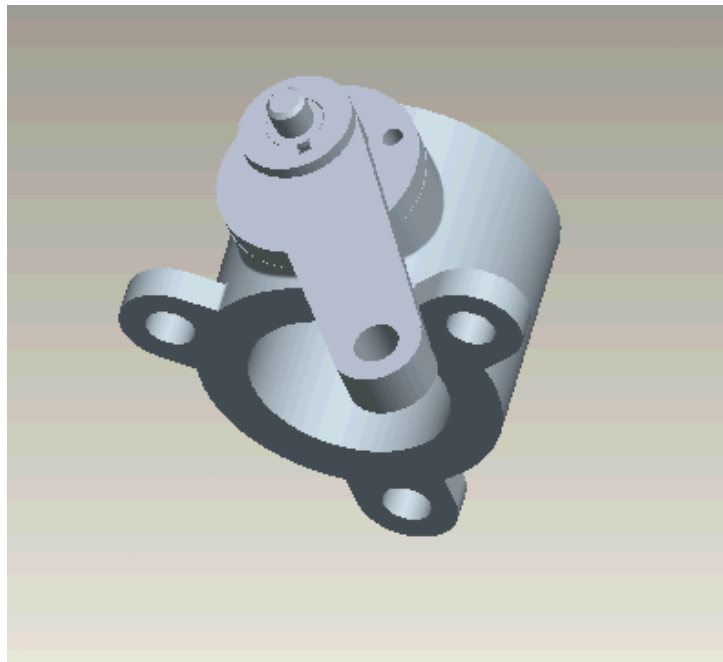
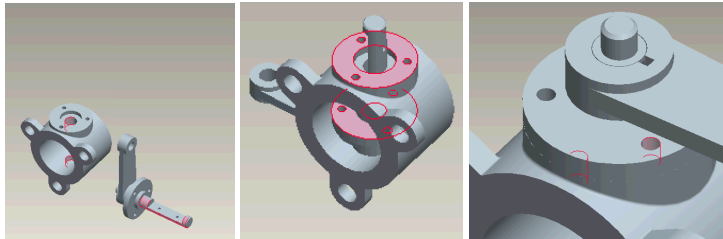


Figure 12 : Assembly after 2 parts.

2. *VALVE_BUTTERFLY* with three constraints. Pick one small hole in the butterfly and one hole in the shaft to make an Insert type. Pick the other small hole and the other hole in the shaft to make and Insert type. Pick one of the flat faces of the butterfly with the flat on the shaft. If necessary change orientation using to make a Mate type.

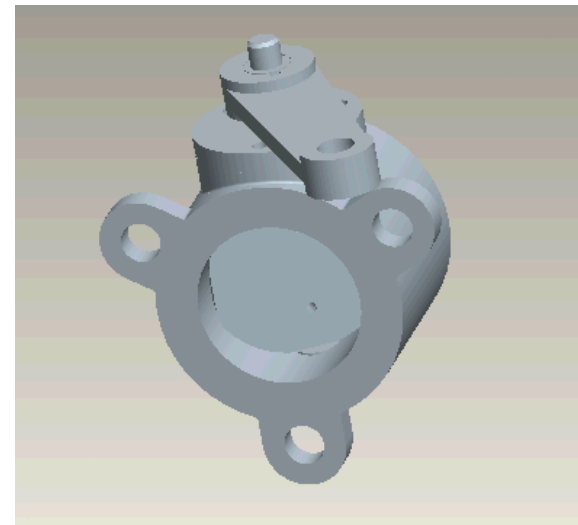
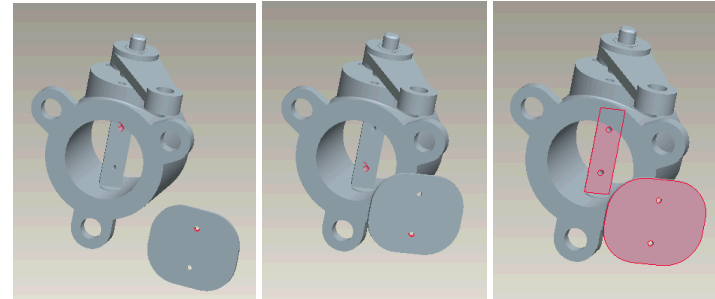


Figure 13 : Assembly after 3 parts

The assembly is now complete. Don't forget to save the assembly using FILE > SAVE.

Modifying Assemblies

Choosing EDIT > DEFINITION and picking one of the components allows modification of the constraints you have applied (Alternatively you can right click on the component name in the feature tree on the left then choose EDIT DEFINITION). The familiar Component Placement dialog box will appear showing the placements already applied. Clicking on one of the constraints makes it current. As each constraint is made current the references (the surfaces you selected to define the constraint) on the assembly and the component are highlighted. The current constraint can be deleted and/or an additional constraint can be added.

It is also possible to modify component dimensions of parts from within the assembly. First you need to select one of the features in one of the parts. At the very bottom of the Pro Engineer window you will see the selection list. This controls what will be selected when you click on an object in the graphics window. This will probably be set to SMART so that ProEngineer tries to 'guess' what to select for you. Change this option to FEATURES. Now when you move the cursor over a part in the graphics window the individual features are highlighted and selected if you left click on them. Select a feature then press and hold the right mouse button. A popup menu will appear and you can choose Edit – the dimensions of the selected feature will appear and you can change them by double clicking on them. To see the changes you will need to choose EDIT > REGENERATE. Don't forget to reset the selection list to smart.

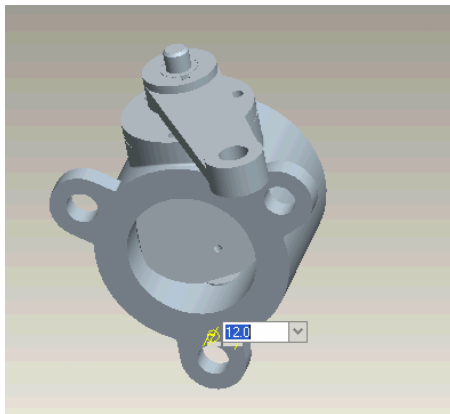


Figure 14 : Editing a Feature in an Assembly

Exploding Assemblies

Sounds exciting! Bang! Actually an exploded assembly is nothing more than the parts shown separated from each other. This can be achieved in ProEngineer using a simple command VIEW > EXPLODE > EXPLODE VIEW – try it in the *valve* assembly. ProEngineer will make certain assumptions based on the type of constraints applied and 'guess' at a suitable exploded state. As you can see from Figure 15 this is rarely perfect but it is a good starting point.

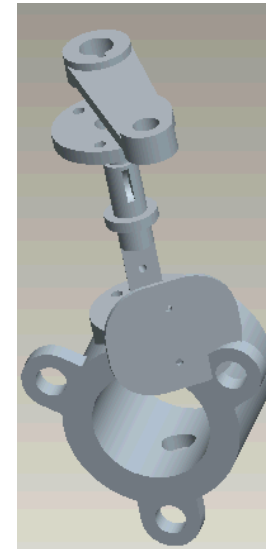


Figure 15 : A Poorly Exploded Assembly

Having exploded the assembly and found it is not quite correct the exploded positions can be modified and the parts moved to a more appropriate position. You are moving the parts in the exploded state. This means that you are not actually altering the assembled model, only how the parts will be shown when exploded. The modified positions of the model when exploded will be saved so that next time you explode the model it will be shown correctly.

In this case the exploded directions chosen by ProEngineer are fine, but the distances it has been exploded are insufficient and some parts overlap. Looking at the assembly you will need to move the *valve_butterfly* and *valve_shaft* up by the same amount. To make room for this movement

the *valve_cover* and *valve_arm* will need to be moved up as well. It may also look better if the *valve_butterfly* is moved forward.

To make sure these modifications are permanently saved choose VIEW > VIEW MANAGER. To show the view manager dialog in Figure 16. In the Explode tab choose EDIT > REDEFINE > POSITION to reposition the parts. The explode position dialog will be shown as in Figure 16.

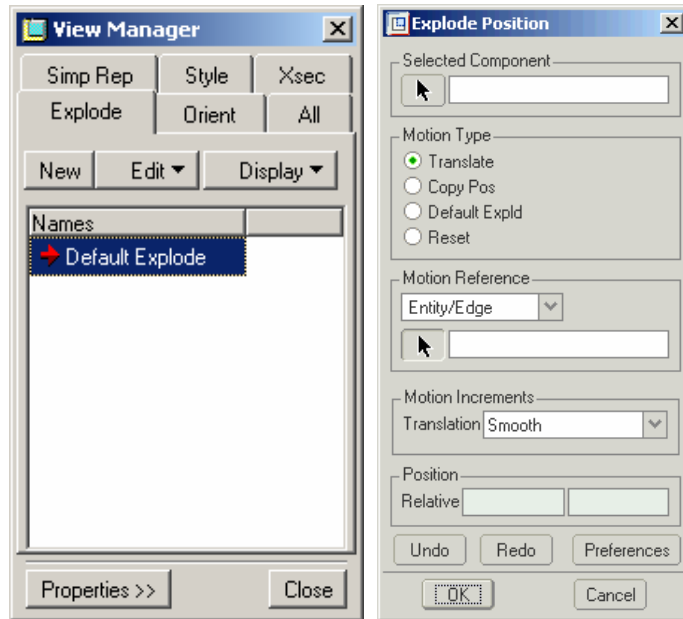


Figure 16 : View Manager & Explode Position Dialogs

There are two important fields in this box. First the motion reference must be set which defines the direction in which subsequent components will be moved. Choose PLANE NORMAL and pick the large flat surface on top of the valve cover. This defines the direction of movement i.e. along a line at right angles to this surface. Now click on one of the parts to be moved for example the shaft. As you move the mouse the part will move and the amount of movement will be reported in the dialog. Move the part to the desired position and click again. Return this part to the original position with the UNDO button. We really want to move the shaft cover, arm and butterfly by the same amount. Click on the preferences button and choose Move Many. You can now pick several parts to move – hold down the

CTRL key whilst picking (pick the *valve_butterfly*, *valve_shaft*, *valve_cover* and *valve_arm*) and click OK. You can now click and drag to move the parts. You may also decide to move the butterfly forward a little. To maintain a true explosion this part should only be moved in a direction at right angles to the flat on the shaft. Click on the arrow button in the motion reference field and click on the flat face of the butterfly. This now sets the movement direction correctly and the butterfly can be moved as before. Choose **OK** to save these modifications.



Figure 17 : A Correctly Exploded Assembly

You can now use VIEW > EXPLODE > UNEXPLODE VIEW to show the model in its assembled position.

Creating Bills of Materials

A bill of materials is a list of each component needed to make up an assembly. Often it contains more than just a simple list of parts. It may be important to list part number, material type, cost and weight and other data

for each part. Also subassemblies may be recorded as either a single part or they may be broken down as well to show their make-up. These would normally be indented in the list to show they are part of a subassembly. The bill of materials list is a good example of the way in which a CAD system can form the core data for more general data retrieval system. The bill of materials information is needed for stock control and order processing systems.

To create a simple Bill of Materials report from an assembly choose INFO > BILL OF MATERIALS. A dialog box appears as shown in Figure 18.

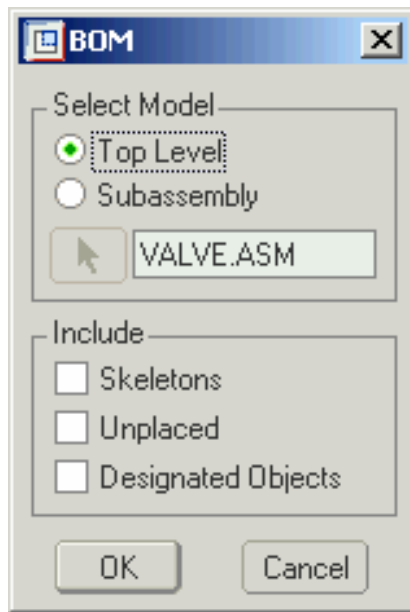


Figure 18 : BOM Dialog Box

The default options are probably fine so just click on **OK** and a window will appear showing the details of the assembly. You can see from this that the *VALVE* Assembly contains 2 parts and one sub-assembly. The sub-assembly itself contains 3 parts. The valve assembly contains a total of 5 parts. This listing appears on the screen and is also written to the disk as a text file under the name *valve.bom.1*. You could add this text to a drawing (not an assembly) by choosing INSERT > NOTE then FILE ⇒ MAKE NOTE locate the notes position and type in the filename *valve.bom.1*.

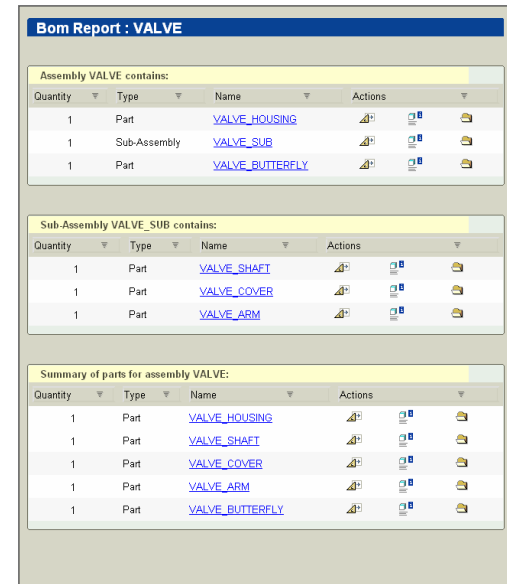


Figure 19 - A Bill of Materials Report for Valve

Review

So what should you have learnt?

- How to create assemblies.
- How to assemble components using insert, align and mate.
- How to edit features in an assembly.
- How to explode an assembly.
- How to create simple parts lists.

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 need to experiment with other assemblies of your own as there are more constraints which we have not covered here.

Also assemblies are the starting point for mechanism analysis so take a look at the mechanism tutorials.