CATIA V5 TRAINING

Foundation Course
Part Design
## Amendment History

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Session 4 – The Part Design Workbench

On completion of this session the trainee will:

♦ Be able to access the Part Design Workbench.
♦ Understand the Part Design Toolbars and Icons.
♦ Be able to create Sketch and Surface Based Features.
♦ Be able to Perform Boolean Operations.
♦ Be able to create Dress-Up Transformation Features.
♦ Be able to use the Compass Manipulation.
♦ Be able to create a User Defined Axis System.
An Introduction to Part Design

The Part Design Workbench is used to create Solid geometry using a Feature based approach. In general the features are produced from sketches created in the Sketcher workbench.

The specification tree contains all the features created along with the sketch used to define them. All the Solid features are contained within a node called a **PartBody**. They also contain wireframe sketches that are used to create the features. As you create features they are added to the tree in order of creation. There may be multiple **Partbodies** within a CATPart which can be Booleaned together in order to form complex solid models. **Partbodies** can be added to the Specification Tree by selecting **Body** from the **Insert** drop down menu when in the Part Design Workbench. The **Partbody** can then be renamed by editing its properties.

If an existing feature is used in the creation of a new feature i.e. the sketch plane for the new feature is created on an existing Face of another feature then the new feature is dependent on it. Therefore the existing feature becomes the **Parent** of the new feature which is known as the **Child**. If the Parent feature is deleted the Child feature will need to be edited and attached to a new parent or in some cases it may be isolated otherwise it will be deleted. Prior to deleting a feature you can ascertain its dependencies by selecting the node on the tree and by using **MB3** select **Parent/Child** on the Contextual Menu. A pop up **Parent and Children** window will appear displaying the features dependencies.

In the example opposite **Pocket.3** is dependent on **Sketch.2**, also **EdgeFillet.4** is depend on **Pocket.3**. Therefore if the sketch is deleted then both the pocket and fillet features will be deleted. By using MB3 on a selected node in the window you can edit the feature to change its dependencies. Once you have finished click **OK** or **Cancel** to closed the window.
When you are modifying or creating a node in the Specification Tree, the node is underlined.

If you select another node on the tree and by using MB3 and selecting **Define in Work Object** on the Contextual menu, you can step back in history and display the solid model at earlier stages of its evolution and then proceed to edit the command.

In this case, before the Holes and Fillets were created.
If you select a **Partbody** and Define it as the Work Object then it is possible to Scan through the entire history of the Partbody by selecting **Scan or Define in Work Object** option on the **Edit** drop down menu. A Scan panel will appear and by using the arrow button you can step through the history.

It is best to select the the **First** button to begin at the start of history. The **Next** button will move forward one command and the **Previous** button will step back one command. The **Last** button jumps to the end of history. Finally the square button exits Scan mode.
Accessing the Part Design Workbench

To access the Part Design workbench you can either Select Start > Mechanical Design > Part Design from the Start drop down menu

Again if a CATPart is not active you will be prompted to create a new part by the appearance of the Part name panel.
Part Design Toolbars and Icons

There are eight main toolbars within the Part Design workbench:

1. **Sketch based Features** – creates geometric features from sketches.
2. **Dress-up Features** – creates dress-up features on existing geometry.
3. **Boolean Operations** – used to Boolean Partbodies.
5. **Annotations** – attaches text annotation to features.
6. **Constraints** – creates constraints on features.
7. **Transformations** – applies transformation Operation to features.
8. **Surface based Features** – creates surface based Features.

The Part Design Toolbars are also accessible via the **Insert** Drop down
The Sketch Based Features Toolbar

This toolbar is used to create features from Sketches.

- Creates a Pad
- Creates a Pocket
- Creates a Shaft
- Creates a Groove
- Creates a Hole
- Creates a Rib
- Creates a Slot
- Creates a Stiffener
- Creates a Loft
- Creates a Removed Loft
- Creates a Pad
- Creates multiple depth Pads
- Creates a Filleted Pad

Creating Pads

Creates different type **Pad** features by extruding a sketch profile linearly to produce a solid.

Creates a pad from a Sketch to form a Solid feature by extruding a selected profile in a linear direction define by the user. If an solid geometry already exists in the PartBody then the new pad feature will automatically added to it even if it is not connected geometrically.
Select the **Pad** icon followed by the required sketch either graphically or from the specification tree. If you have just exited a sketch then it will already be selected. A **Pad Definition** panel will now appear and a wireframe preview of the Pad be displayed. Enter a value in the **Length** field (default 20mm) or you can **Grab** the LIM1 and LIM2 text using MB1 to control the length of the Extrusion in two directions and then dynamically drag the extrusion to its desired size. Now press **OK** to create the pad. The default is to create a pad extruded distance normal to the sketch plane. The **Reverse Direction** and the Orange Arrow button swaps the direction of the Limits.

The Pad feature is added to the specification tree and the sketch that was used to define it is now linked to the node.
There are various options available on the Pad Definition panel. By clicking on the More tab the panel expands to reveal all the options.

The following options are available:

**First** and **Second Limits** control the length of the extrusion about the selected sketch plane. The **Type** option allows you specify whether the extrusion is a length or up to a surface or plane. The **Length** field is only available when **Dimension** is selected in the **Type** field. The **Limit** field is available when up to a surface or plane is selected in the **Type** field.

**Profile/Surface** allows you to specify the sketch or surface that is used to create the pad. The **Selection** field indicate the sketch or surface that has been selected. The **Thick** check box allows you the create a shelled extrusion by entering the thickness value in the **Thin Pad** fields.

**Mirrored extent** if checked will deselect the Second Limit and create a pad extruded equally about the sketch plane.

As mentioned previously **Reverse Direction** flips the direction of the extrusion. You can also click on the orange Flip Arrow.

Direction controls whether the extrusion is **Normal to the profile** or along a **Reference** element.

**Thin Pad** is used in conjunction with the **Thick** check box to create a shelled pad. **Thickness 1** and **2** controls the thickness of the shell about the profile. If **Neutral Fiber** is selected the shell is created equal about the profile and **Thickness 2** is deselected.

The **Preview** button allows you to preview the pad.

Clicking **OK** finishes the Command and creates the Pad Feature.
The following are examples of the different ways of creating a pad:

If you select **First Limit > Type > Up to surface** the pad is extruded up to the selected surface.

You can create a pad up to but offset from the selected surface by entering a value in the **Offset** field.

If **First Limit > Type > Up to next** is used the resulting pad will extrude up to the next face of an existing feature. **Up to Last** will extrude up to the last face on an existing feature.
The **Thick** option creates a pad with a thickness around the sketch profile rather than just a solid. The **Neutral Fiber** applies the **Thickness 1** value equally about the profile.
If Reference is select rather Normal to profile then the resulting pad is extruded along the select Line and not normal to the sketch plane.
**Hints and when creating Pads**

It is possible to create a pad from a sketch that contains multiple profiles. Catia will automatically select all profiles in the sketch and extrude them into a pad including holes.

Avoid creating multi profile sketches with open profiles as this will result in Catia trying to create open pads and failing.

When using **Up to Next**, **Last** and **Surface** the face or surface that the pad is being extruded to must be large enough to fully intersect with the profile. This limitation does not apply to a **Plane**.

When you use an entire Sketch to produce a Pad any elements contained within the sketch but not used in the profile must switched to **Construction** i.e. Points.
When using a sketch with multiple profiles you can choose which of the profiles to include in the pad. After selecting the sketch to extrude the Pad Definition panel will appear. Use MB3 on the Selection field to reveal a Go to profile definition pop up. By selecting this option you can deselect profiles from within the sketch.

A Profile Definition panel now appears which list the current selection i.e. Sketch.1. Select Sketch.1 and click on the Remove button. The sketch is now removed from the list.

Now click on the Add button and select the required closed profiles or Sub-elements from the sketch followed by the OK button to create the desired pad.
Creates a **Drafted Filleted Pad** on an existing feature using a selected profile.

Select the icon followed by the profile to be used to create the pad. The following options are available:

- **The Length** is the length of the extrusion.

- **The Second Limit** must be selected and is usually the face of an existing feature on which the profile for the pad exists.

- **The Draft** applies a draft angle to the side faces of the pad. The **Neutral element** either **First** or **Second limit** controls which limit the draft angle is applied from i.e. the profile is at its original size.

- **The Fillets** options control the **Lateral radius**, **First limit radius** (the fillet nearest the **First limit**) and the **Second limit radius** (the fillet nearest the **Second limit**).

**Reverse Direction** reverses the direction of the pad.

Clicking **OK** creates the feature.
The **Specification Tree** will now have a Pad created in it together with a **Draft** and three **EdgeFillet** features rather than one all encompassing feature.

Create multiple pads with different extrusion lengths.

Select the sketch to use in the multi-pad command follow by the icon. **A Multi-Pad Definition** panel will now appear, use the **More** button to expand the panel to show all options.

Use the **First** and **Second Limit** to define the length of the extrusion. **Note: The only option is a Dimension.**

The **Direction** option allows you extrude normal to the sketch plane or along a Reference Line.

The **Domains** portion of the panel lists all the profiles contained within the selected sketch. By selecting one of the **Extrusion Domains**, the values that are entered in the **Dimension** fields control its length. Remember you can also **Grab** the **LIM1** and **LIM2** graphically to control the extrusion length.

Click the **OK** button to create the **Pad**.
Creating a Pocket

Creates different **Pocket** features by extruding a sketch profile linearly to create a subtraction feature.

Select the sketch containing the pocket profile and then select the icon. A **Pocket Definition** panel will now appear which you can select the **More** button to display all options. The options on this panel are identical to the **Pad Definition** panel except the resulting feature is subtracted from the previous feature in the **Specification Tree**.

The **First** and **Second Limits** control the **Depth** of the pocket about the selected sketch plane. The **Type** option allows you to specify whether the extrusion is a depth or up to a surface or plane. The **Depth** field is only available when **Dimension** is selected in the **Type** field. The **Limit** field is available when up to a surface or plane is selected in the **Type** field.

**Profile/Surface** allows you to specify the sketch or surface that is used to create the pocket. The **Selection** field indicates the sketch or surface that has been selected. The **Thick** check box allows you to create a thin pocket about the profile by entering the thickness value in the **Thin Pocket** fields.
Mirrored extent if checked will deselect the Second Limit and create a pocket extruded equally about the sketch plane.

Reverse Direction flips the direction of the pocket or use the orange Flip Arrow.

Direction controls whether the pocket is created Normal to the profile or along a Reference element.

Thin Pocket is used in conjunction with the Thick check box to create a narrow or thin pocket. Thickness 1 and 2 controls the width of the pocket about the profile. If Neutral Fiber is selected the pocket is created equal about the profile and Thickness 2 is deselected.

There is an option to Invert the pocket by selected the second arrow that appears when the Pocket Definition panel is displayed. Note: This option does not appear on the panel. Click the OK button the finish.

A Pocket node is added to the specification tree and the sketch that is used in its creation is added to it.

Note: If you extrude the pocket in the wrong direction i.e. into open space and not through an existing feature you will see a warning alert informing you that this operation is unnecessary unless the Pocket is in its own Partbody.
Creates a **Drafted Filleted Pocket**.

Select the sketch that is to be used to create the pocket followed by the icon. A **Drafted Filleted Pocket Definition** panel will now appear. The options are the same as the **Drafted Filleted Pad Definition** panel.

The **First Limit** is the **Depth** of the pocket.

The **Second Limit** must be selected and is usually the face of an existing feature on which the profile for the pocket exists.

The **Draft** applies a draft angle to the side faces of the pocket. The **Neutral element** either **First** or **Second limit** controls which limit the draft angle is applied from i.e. the profile is at is original size.

The **Fillets** options control the **Lateral radius**, **First limit radius** (the fillet nearest the **First limit**) and the **Second limit radius** (the fillet nearest the **Second limit**).

**Reverse Direction** reverses the direction of the pad.

Clicking **OK** finishes the command.

Again you can use the **Invert Arrow** to invert the pocket.
Creates multiple Pocket features with different depth.

After selecting the sketch containing the pocket profiles and the icon a Multi-Pocket Definition panel will appear which is identical to the Multi-Pad Definition panel.

Use the First and Second Limit to define the depth of the pocket. Note: The only option is a Dimension.

The Direction option allows you create the pocket normal to the sketch plane or along a Reference Line.

The Domains portion of the panel lists all the profile contained within the selected sketch. By selecting one of the Extrusion Domains you can enter the value for the depth of the pocket in the Dimension fields control its depth. Remember you can also Grab the LIM1 and LIM2 graphically to control the extrusion length.

Click the OK button to finish.

Note: There is no invert pocket option for this command.

Creating a Shaft

Creates a revolved solid feature from a selected sketch profile.

Select the sketch containing the profile followed by the Shaft icon. A Shaft Definition panel will now appear containing the follow options: -

Limits defines the First (start) and Second (end) angles for the revolution of the profile. Default is a 360° revolution.

Profile>Selection field indicates which sketch you have select by using MB1. Note: As with the Pad command by using MB3 on the Selection field you can access the Go to Profile option and select different profiles contained within the sketch.

Axis allows you to define the axis of rotation. Note: You can create an Axis Line within the defining sketch and Catia will automatically use it to produce the shaft.

Click OK to create the Shaft.
Below is an example of a **Shaft** feature using a Profile sketch containing an Axis Line.
Note: The Axis Line closes the profile and that the total Angle in the Limits can not exceed 360°.

A warning symbol will appear on the **Shaft Definition** panel if there is no Axis selected or if there isn’t one present in the sketch. A **3D Reference Line** may be used as an **Axis Line**.

The Specification Tree will have a **Shaft** node added to it and the defining sketch will be linked to it.

![No Axis Selected warning symbol](image)

**Creating a rotational Groove**

**Creates a rotational Groove feature about an Axis Line.**

Select the sketch containing the groove profile followed by the **Groove** icon. A **Groove Definition** panel with the same options as the **Shaft Definition** panel:

- **Limits** defines the First (start) and Second (end) angles for the revolution of the profile.

- **Profile>Selection** field indicates which sketch you have selected. Again you can the **Go to profile** option to select sub-profiles within the sketch.

- **Axis** defines the axis of rotation. Again you will see the no axis warning symbol if no axis is selected. **Note:** By selecting an existing shaft feature Catia will use its Axis as the Axis of rotation thus ensuring concentricity between the Groove and the Shaft features.

Finally click the **OK** button to create the feature.
Below is an example of a **Groove** with an Axis Line defined in the sketch profile.

A Groove node is added to the Specification Tree with the defining sketch attached.
Creating a Hole

Creating a hole in an existing feature without requiring a sketch profile.

Select an entry face for the hole on an existing feature and then click on the Hole icon. A Hole Definition panel will now appear with three tabs along the top Extension, Type and Thread Definition.

The Extension tab has the following options:

The hole limit tab allows to define whether it is a Blind hole or that it is controlled Up to Last, Up to Next, Up to Plane or Up to Surface. The default is a Blind hole, which allows access to the Depth field as well as the Diameter. If one of the other options is select then you have to select a Face, Plane or Surface to control the hole depth. There is also the option to Offset the depth from the selected element.

Direction controls the direction of the hole into the feature. The Reverse tab reverses the depth (Note: If the direction is away from the feature i.e. into open space, then a Hole node is created in the Specification Tree but there will be no visibility of a hole in the feature). The Normal to surface check box ensures the hole axis is normal to the entry surface, if this is unchecked you can select a Reference Line as the direction of the Axis.

The Positioning Sketch button allows you to enter a sketch to define the position of the hole on the entry face.

The final option is the Bottom style for the hole, which is either Flat or V-Bottom, which allows you to enter an inclusive Angle.
The Type tab allows you to specify the type of hole that you require. The options are Simple, Tapered, Counterbored, Countersunk and Counterdrilled.

The Parameter option allows you to control the size of the hole features i.e. Counterbore size, depth of the Countersink, etc.

The Anchor Point is only available when you select either Counterbored or Counterdrilled and is used to control the datum point on the hole in relationship to the selected Entry Face.

The Thread Definition tab can be used to assign thread attributes to the hole feature which can be extracted by other processes within Catia.

After selecting the required options click OK to complete the feature creation.
The following are examples of creating Hole features.

A 10mm Diameter **Blind** hole.

Select an Entry Face and enter the Diameter value.

Select the Positioning sketch button and apply a **Concentricity Constraint** between the **Hole** centre point and the base plate radius using **MB3**

Click **OK** to create the hole.
A 10mm Diameter Hole with a 15mm Diameter Counterbore by 5mm deep.

Select the **Entry Face**. Enter the **Diameter** value and then select the **Type** tab. Select the Counterbored option and enter the **Counterbore** values.

Select the **Extreme Anchor Point** to ensure the top of the Counterbore is positioned on the **Entry Face**.

Select the Positioning sketch button on the **Extension** tab and apply a **Concentricity Constraint** between the **Hole** centre point and the base plate radius using **MB3**.
A quick way of constraining the hole centre to the base plate radius is to select the edge of the radius followed by the **Insert Hole Icon** and finally select the **Entry Face**. The hole centre will then automatically be constrained to the radius edge.

You can also multi-select two edges to constrain to by holding down the **Ctrl** key and then selecting the two edges before clicking on the icon. This will result in two constraints being created between the selected edges and the radius centre.
Creating a Rib feature

Allows you to extrude a profile along a reference element or pulling direction.

Select the create rib icon. A Rib Definition panel will now appear. You have to select a Profile and a Center curve. As with Pad creation you can use MB3 over the Profile field and select Go to profile definition to select sub element profiles to create the Rib.

Profile control defines how the Profile is extruded along the Center curve:

- **Keep angle** is the default option and this ensures the Profile stays at the same angle to the Center curve as it is extruded. **Pulling direction** keeps the profile parallel to a selected Line or Plane. **Reference surface** maintains a constant angle between the H Axis of the sketch profile and a selected surface.

**Merge ends** will extrude the profile ends to join with an existing feature if it encloses the swept Rib.

To finish the operation click OK.
Below are examples of **Rib** features.

A Rib created with **Keep angle**

A Rib created with a vertical **Pulling direction**

A Rib created using **Merge**

A Rib created with **Keep angle**
with a sketch profile containing two circles

The Specification tree displays the **Rib** feature and its associated sketches.
Creating a Slot Feature

Creates a slot feature on an existing solid based on a sketch profile.

Select the Slot icon this will display the Slot Definition panel, which has the same options as the Rib Definition panel. Now select the sketch containing the slot Profile followed by the Centre curve. You can use MB3 on the Profile field to select sub element profiles for the slot creation.

Select the method of Profile control to control how the profile is extruded along the Center curve. Again the options are the same as the Rib creation, Keep angle, Pulling direction and Reference surface.

Merge ends will extrude the profile ends to join with an existing feature if it encloses the Slot.

Finally click OK to complete the command.

The Slot feature is created in the Specification Tree below the feature that it was removed from. The sketches containing the Slot Profile and Centre curve are attached below it.
Creating a Stiffener

Creates a stiffener on an existing feature using Line(s).

Select the sketch profile followed by the Stiffener icon. A Stiffener Definition panel will now appear with the following options:

- **Mode** controls the direction in which the stiffener profile is extruded. The options are **From Side** or **From Top** relative to the sketch plane containing the stiffener profile.

- The **Thickness** controls the width of the Stiffener via **Thickness 1** and **Thickness 2** fields. The **Neutral Fiber** option allows you to apply the **Thickness 1** value equally about the profile. **Thickness 2** is not selectable. **Reverse direction** swaps the direction of the thickness values. You can also use the Flip arrow to change the direction.

- **Depth** controls the direction of the stiffener extrusion.

- **Profile** allows you to select the sketch profile to be used in the creation of the Stiffener. You can use MB3 over the **Selection** field to select sub element profiles from the sketch.

After selecting your options click **OK** to complete the command.
Below are examples of the two different Extrusion Modes.

The sketch profile may consist of several elements that are connected together to form the stiffener.

Note: The Sketch Profile must lie fully within the existing feature for the stiffener to be created.

The Stiffener node is added to the Specification Tree together with the defining Sketch Profile.
Creating a Loft Feature

Creates a Loft through a series of Sketch Profiles.

Select the Loft icon to display the Loft Definition panel.

The top portion of the panel displays which profiles have been selected and the order that they have been selected. **Note: At least two non-intersecting profiles must be selected.**

The bottom portion as four tabs:

The **Guides** tab lists the selected guides. The purpose of guides is to control how the Loft cross section is controlled between the profiles.

The **Spine** tab lists any Spine that is used. When a Spine is use, the transition shape between profile is kept normal to the Spine curve. By default if no spine is selected then Catia will compute one based on the profiles and their orientation to each.

Both the spine and guides are optional.

The **Coupling** tab allows you select how the transition is mapped between profile. The are four **Section coupling** options:

- **Ratio** maps the profiles together by a curvature ratio.
- **Tangency** maps the profiles together by their tangent discontinuity points. If there are not the same number of points in each curve then this option will cause the Loft to fail.
- **Tangency then curvature** maps the profiles together by their curvature discontinuity points. As with tangency if there are not the same number of points in each curve then this option will cause the Loft to fail.
- **Vertices** maps the vertices of the profiles together. Again there must be the same number vertices in each profile for the Loft command to succeed.

The **Relimitation** tab allows you to re-limit the start and end sections of the Loft. The two options are to limit the loft to the first profile and the last profile. If either of these options is not selected then the Spline curve or the Guide curves control the relimit.
To create a default Loft using the **Vertices** of the profiles as the **Coupling** mapping, select the profiles required either graphically or from the Specification Tree. As the profiles are selected a section identity appear on the profile together with the **Closing Point**.

Note: The sketch can only contain a single closed Profile.
The Closing points must be aligned with each other along the length of the loft otherwise the resulting loft will twist. In this example the Section 3 Closing Point is incorrect which resulted in a twisted Loft.

To change a Closing points location it is best to start the loft again. After selecting the loft profile you have to select the profile in the top portion of the Loft Definition panel, then using MB3 to access the contextual menu. Select Replace Closing Point and then pick a new closing point on the profile, in this case a Vertex. After replacing the closing select Vertex as the Coupling and then click OK to complete the command.
The following is an example of a Loft with a user-defined **Spine** and the **Relimitation** of the end section controlled by the spine.

The Loft and its associated sketches are added to the Specification Tree.
The Remove Loft Feature

The Remove Loft feature creates a loft that is removed from an existing feature. The method for producing the loft together with the options is identical to the Loft command. A Loft feature and its associated sketches are added the Specification Tree.

Creating Reference Elements Toolbar

Allows you to create Points, Lines and Planes in 3D space.

Create a Point in 3D Space

Creates a point using ordinates or by reference existing elements.

Select the point icon to display the Point Definition panel. The Point Type tab as seven options:

Coordinates (default option10) allows you enter X, Y and Z co-ordinate using the ‘X=’, ‘Y=’ and ‘Z=’ fields to position a single point in 3D space. The Reference Point field allows you to specify the origin for the co-ordinates, by default this is the origin of the CATPart although you can specify the origin to be relative to existing elements i.e. Points and Vertices.
On curve allows you to create a single point on an existing curve. The Curve field is the name of the element you that you have selected. You have the option to enter a value in the Length/Ratio field that corresponds to either a Distance on curve or a Ratio of curve length (this value must be between 0 and 1). The option Geodesic is used with the Distance on curve and applies the true distance along the curve. The Euclidean option is also used with the Distance on curve and this applies a distance that is measured in relation a select Reference Point. The Nearest extremity option snaps the point to the nearest end point on the curve to where you clicked to select the curve. Middle point creates a point midway along its length. The Reverse Direction toggle drives the distance or ratio from the other end of the curve and finally the Repeat object after OK toggle redisplay the Point Definition panel when you click OK the create the point.

The On Plane option allows to create a point on a select Plane. The Plane field indicates the plane you have chosen to place to the point on. The ‘H’ and ‘V’ field allows you to enter a co-ordinate relative to the Reference Point, which by default is the origin of the plane.
On surface allows to create a Point on a selected Surface. The Surface field displays the name of the selected Surface. The Direction allows you to define which direction the Distance value is applied to or can indicate on the surface the position of the point using MB1. By default Reference Point is the centre of the surface.

Circle Center creates a point on a selected circle.

Tangent on curve creates a point on a curve tangent to a direction.

The Between option creates a point between two existing points either by Distance or Ratio positioning.

As mentioned previously in this session the reference elements are generated in an Openbody.
Creating Lines in 3D Space

This command allows you to create Lines in 3D space.

After selecting the icon a **Line Definition** panel will appear with the following six options for creating a line via the **Line Type**: tab: -

**Point - Point** (default option) creates a line between two existing points that you select. There is a option to locate the line on a **Support** plane if required. You can offset the ends of the line from the points by entering values in the **Start** and **End** fields. Finally **Mirrored extent** mirrors the **End** offset value at both points.

**Point - Direction** allows you to define a line by selecting a point as a start location and direction vector by selecting an existing line or plane. The **Start** and **End** fields control the length of the line. You also have the options to create a **Mirrored extent** and **Reverse direction**.

**Angle/Normal to curve** creates a Line at an angle or Normal to a selected curve. The curve must have a point created on it to position the line. You select the **Curve** followed by a **Support** plane for the line to be positioned on and finally a location **Point** for the Start point of the line. You then enter the **Angle** followed by the **Start** and **End** length for the line. Further options allow you to **Mirror** the line about the Start point. Project the **Geometry** onto the **support** plane. Create the line **Normal to** the **Curve**. **Reverse Direction** of the line.
**Tangent to curve** allows you to create a line tangent to a selected curve and an existing element.

**Normal to surface** creates a line that is Normal to a selected surface. You will have to select a Point on the surface to indicate the start point for the line. Entering values in the **Start** and **End** field controls the length of the line.

**Bisecting** creates a line that bisects to existing lines through their intersection point.

**Note:** You can use MB3 over certain fields on the Line Definition panel to create reference elements such as points on the fly.
Creating Plane in 3D Space

Create plane in 3D space.

As mention earlier Planes are created in an Openbody with the exception of the three default Planes XY, YZ and ZX at the top of the Specification Tree, which define the origin of the CATPart as the intersection of the three planes. These planes cannot be edited or repositioned.

Planes are represented graphically by a wireframe square.

Select the icon to display a Plane Definition panel. There are eleven options available via the Plane Type: tab:

1. Offset from plane (default Option)
2. Parallel through point
3. Angle/Normal to plane
4. Through three points
5. Through two lines
6. Through point and line
7. Through planar curve
8. Normal to curve
9. Tangent to surface
10. Equation
11. Mean through points

1. Offset from plane creates a Plane that is offset from an existing plane or a planar face of a feature. You have to select a Reference plane as a datum for the offset and enter an Offset value. There are the options to Reverse the Direction of the offset and Repeat object after OK. You can drag the Offset size and Move the plane by using the green arrow.
2. **Parallel through point** creates a plane offset from an existing plane or face, which is positioned by selecting a Point.

![Parallel through point](image)

3. **Angle/Normal to plane** creates a plane that is at an **Angle** or **Normal** to an existing Plane or Face. You have to select a **Rotation axis** which, can be either a line or a linear edge of an exiting feature, a **Reference** Plane or Face and an **Angle** for the new plane. Other options are **Normal to plane** and to **Repeat object after OK**.

![Angle/Normal to plane](image)

4. **Through three points** allows you to create a Plane by selecting three Points. the selected elements can either be points vertices of an existing feature.

5. **Through two lines** creates a Plane through two selected Lines.

6. **Through point and line** creates a Plane through a selected Point and Line.

7. **Through planar curve** creates a Plane parallel to a selected Planar curve.
8. **Normal to curve** allows you create a Plane that is Normal to a selected curve. Select a Planar Curve and a Point on the curve to position the new Plane. If a Point is not selected then a default Middle of curve point is used. You can use MB3 to create a Point on the fly.

![Normal to curve](image)

9. **Tangent to surface** create a Plane that is Tangent is a selected Surface. You have to select a Surface and a Point on the surface to location the Plane. Again use MB3 to create a point on the fly.

10. **Equation** allows you to define a Plane using an equation related to the XYZ Axis of the CATPart.

![Equation](image)

11. **Mean through points** allows you to create a Plane through the Mean position of a series of selected points. The minimum number of point required is three.

![Mean through points](image)
Surface Based Operations Toolbar

Creates features using surface based operations.

Creating a Split

Creates a split using a Plane, Face or Surface.

After selecting the icon you have to select a splitting element which can be a Plane, Face or a Surface. A Split Definition panel will appear with the selected element listed in the Splitting Element field. An orange arrow is displayed on the splitting element, which indicates which portion of the current solid will be kept. Click on the arrow to reverse the direction. Now click OK to complete the command.
A Split Node is added to the Specification Tree.

Note: When using a Face or Surface as a Splitting element it must fully intersect with Solid for the command to be successful.
Thick Surface

Creates a Solid feature by thickening a Face or Surface.

After selecting the icon you must select a Surface to be thickened. A ThickSurface Definition panel will appear with the selected element listed in the Object to offset field. The First and Second Offset fields allow you to enter the thickness value for the Solid from the selected Surface. The orange arrow indicated the direction of the First Offset, which can be reversed by clicking on the arrow or selecting the Reverse Direction button. Click OK to create the solid.

A ThickSurface node is added to the Specification Tree.

Creates a closed solid from a surface. This command is not covered in the foundation course

Sews a Surface onto a feature. As with Close Solid this command is not covered in the foundation course
**Boolean Operations Toolbar**

Boolean Operations allows you Assemble, Add, Remove and Intersect Partbodies together.

- **Assemble**
- **Standard Boolean Operations**
  - Union Trim
- **Remove Lump**

**The Assemble Operation**

**Assemble** Allows you to assemble Multiple Partbodies into an existing Partbody.

Select the Partbody or multiple Partbodies using the **Ctrl** key that you wish to Assemble into an existing Partbody. Select the Assemble icon to display an **Assemble** panel, the **Assemble** field indicates the Partbodies you have selected to assemble. The **After** field contains the name of the Feature or Partbody that is currently the Defined Work Object. Select the Partbody that you wish to assemble to if the **After** field is incorrect and click **OK** to complete the operation.

In this example Partbodies **Pockets** containing a Pocket feature and **Boss** containing a **Pad** feature are to be assembled to **PartBody**.
The result is that Partbodies **Pockets** and **Boss** are attached to the bottom of the **PartBody** Specification Tree. Due to the fact that the **Pockets** Partbody contained a Pocket feature then it was removed from the solid whereas the **Boss** Partbody contained a Pad feature, which was subsequently added to the solid.
Standard Boolean Operations

Creates Standard Booleans **Add**, **Remove** and **Intersect** using selected Partbodies.  

Adds one or more existing Partbodies to another.  
Select the existing Partbodies to Added to another Partbody (remember that you can use the Ctrl key to multi-select). By selecting the Add icon a **Add** Panel will displaying the Partbodies to added in the **Add** field and the current Defined Work Object in the **After** field. Select a Partbody to be added to if different and click **OK** to finish. The selected Partbodies are attached to the Specification Tree of the Partbody displayed in the **After** field with a **Add** node.  

Removes one or more existing Partbodies from another.  
Select the existing Partbodies to be removed from another Partbody. Select the **Remove** icon to display the **Remove** panel. Select a Partbody to be removed from and click **OK** to finish. The selected Partbodies are attached Specification Tree of the Partbody displayed in the **After** field with a **Remove** node.
Intersects one or more existing Partbodies with a selected Partbody.

Select the existing Partbodies to be Intersected followed by the icon to display the Intersect panel. Select a Partbody to be intersected with and click OK. The selected Partbodies are attached to the Specification Tree.

The Union Trim Operation

Allows you to Union and Trim one Partbody to another.

Select the Partbody to be Unioned and Trimmed then select the Union Trim icon to display the Trim Definition panel. If you click OK then the current Defined Work Object and the selected Partbody will be Unioned or added together with no trimming. If you click on the Faces to remove field and then select a portion of the Partbody to be trimmed back to the nearest face on the Defined Work Object, the remaining portion will be Unioned or added. The reverse will happen if you select Faces to keep. The trimmed Partbody is attached the Define Work Object Partbody Specification Tree with a Trim node. There is no Multi-select option with this command.
The Remove Lump Operation

Allows you to remove unwanted Faces from a Feature.

Select the Feature that you wish to remove a Face from followed by the Remove Lump icon. A Remove Lump Definition panel will be displayed, you can click on the Faces to remove field and then select a face to remove. Click on the OK button to complete the command.

The Specification Tree will have a Trim node attached to it.

Note: If the selected face is connected to the main feature then whole feature will be removed or trimmed.
Dress-Up Features Toolbar

Creating Fillet Features

Allows you to create solid Fillet features.

Select the Fillet icon to display the Fillet Definition panel. Click on the More button to see all options.

Creates fillets on **Edges** or **Faces** of an existing feature.
The following options are available:

**The Radius** field allows you to specify the size of the Fillet Radius.

**Objects to fillet** lists the elements you select to fillet which can either be edges or faces of an existing feature.

The **Trim Ribbons** check box is used in conjunction with **Tangency** Propagation to trim fillets that overlap. In some instances the creation of overlapping fillets will fail unless this option is selected.

The **Propagation** toggle allows you to limit the propagation of the fillet to a single edge (**Minimal**) or if the edge is tangent to adjacent edges then with the **Tangency** option selected the fillet will propagate along all edges joined to the selected edge.

The **Edge(s) to keep** option allows you to maintain edges adjacent to a fillet to create a rolled edge fillet.

**Limiting element** can be used to limit the length of the fillet along an edge by selecting a limiting element.

After selecting the Edge(s) or Face(s) to be fillet together with the required options click **OK** to create the fillet.

The following are examples of different edge fillets.

- A single **Edge** selected fillet with **Minimal Propagation**.
- A single **Edge** selected fillet with **Tangency**.
- A Face selected **Edge** fillet.
An edge fillet around a boss with **Edge(s) to keep**

The same edge fillet without **Edge(s) to keep** selected

Overlapping fillets created with **Tangency Propagation** and **Trim ribbons** selected

Limiting Element

A single edge selected fillet with a **Limiting element** selected

In all cases a **EdgeFillet** Node is added to the Specification Tree
Creates a Variable radius fillet along the **Edges** of an existing feature.

Select the icon to display the **Variable Edge Fillet** definition panel and click on the **More** button to view the following options:

The **Radius** field allows you to specify the size of the Fillet Radius.

**Objects to fillet** list the edges you have selected to fillet.

The **Propagation** toggle allows you to limit the propagation of the fillet to a single edge (**Minimal**) or if the edge is tangent to adjacent edges then with the **Tangency** option selected the fillet will propagate along all edges joined to the selected edge.

The **Trim Ribbons** check box is again used in conjunction with **Tangency** Propagation to trim fillets that overlap.

The **Points** field lists the radius change/control points.

**Variation** allows you to specify whether the variable fillet is created by **Cubic** or **Linear** mathematic rules.

The **Edge(s) to keep** as with edge filleting this option allows you to maintain edges adjacent to a fillet to create a rolled edge fillet.

The **Circle Fillet** check box forces the fillet cross section to be normal to a selected **Spine**.

**Limiting element** can be used to limit the length of the fillet along an edge by selecting a limiting element.

After selecting the edge(s) and your options click **OK** to create the fillet.

A **EdgeFillet** Node is added to the Specification Tree although the Icon is different from the standard edgefillet.
The following is an example a Variable Radius along a single edge of a feature using Cubic ruling.

After selecting the edge to be filleted two constraints appear at the vertices of the edge to indicate the size of the radius at that point.

By double clicking on a constraint a **Parameter Definition** panel will appear which will allow you the change the value of the constraint and once you click **OK** the constraint change will be applied.

Now click **OK** on the Variable Edge Fillet definition panel to create the **Cubic** ruled variable radius fillet.
The following is an example a Variable Radius edge fillet with a change point mid way along it length. The point was created using **Point on curve** from the **Reference Elements** toolbar.

After selecting the edge to be filleted select the **Point** field followed by the created point. A third constraint will now appear to allow you change the size of the fillet.
Creates a fillet between two selected Faces. This command is not covered in the Foundation Course.

Creates a TriTangent Fillet on an existing feature.

Select the Tri Tangent icon to display the the TriTangent Fillet Definition panel and click on the More button to view all options:

The **Faces to fillet** field indicates the faces that you have select to fillet between. **Note:** Two Faces must be selected.

The **Face to remove** field indicates the face that you selected to be removed.

**Limiting element** again is used to limit the length of the fillet along an edge by selecting a limiting element.
The following is an example of a Tri Tangent Fillet.
Creating Chamfer Features

Creates Chamfers on Edges or Faces of an existing feature.

Select the Chamfer icon to display the Chamfer Definition panel. The following options are available:

- **Mode** allows you to select both a **Length** and **Angle** or two Lengths to define the Chamfer.
- **Length 1** and **Angle/Length 2** define the values for the chamfer.
- **Object(s) to chamfer** indicates the elements that have been selected to chamfer.

As with filleting, the **Propagation** toggle allows you to limit the propagation of the chamfer using **Tangency** or **Minimal**.

The Reverse check box reverses the **Length** and **Angle/Length** directions of the chamfer. Clicking on the orange arrow displayed on the selected Edge or Face can also perform this.

**Note: The arrow indicates the direction of the Length 1 Constraint**

After selecting the required elements and options click **OK** to create the chamfer. At this point, a Chamfer Node is added to the Specification Tree.

The following is an example of a single Edge chamfer using **Length** and **Angle** values.
Creating a Draft Feature

Creates draft feature on an existing faces

Select the icon to display the Draft Definition panel and then click on the More button to view all options:

- The Draft Type can be either a Standard or Variable Draft.
- The Angle field defines the draft angle for the feature.
- Face(s) to draft indicates the faces that you selected to draft.
- The Neutral Element is a selected Plane or Face on which the true Cross-sectional profile of the selected feature is maintain and the Draft Angle is struck off a plane which is normal to the element.
- The Pulling Direction is indicated by a orange arrow an controls in which direction in the resulting feature can be remove from a mould. Select the arrow to reverse the direction.
- The Parting Element allows you to limit the draft on the selected faces or the draft can be mirrored about an element by selecting the Draft both sides check box. You can either use the Neutral element as the parting element by selecting the Parting = Neutral check box or you use a user defined parting element by selecting the Define parting element check box and then select a parting element i.e. a Plane.

Select OK to create the Draft feature.
The following is an example of the 10° Draft Angle with the bottom face of the solid selected as the Neutral element.

Note: The Draft is applied to all adjacent Faces if they are Tangent to the selected Face to be Drafted.

The following is an example of the 10° Draft Angle with a Plane selected as the Neutral and Parting element. The draft is only applied to one side.
The following is an example of a Variable Draft on a Face which varies from 10° to 15° and back to 10° using a point on the edge of the draft face as a change point.

A Draft Node is added to the Specification tree below the currently Defined Work Object.
Creates a Draft feature using a Reflect Line.

Select the icon to display the **Draft Reflect Line Definition** panel followed by the **More** button to view all options:

The **Angle** field allows you to enter the Draft angle.

**Face(s) to draft** indicates the Faces you have selected to be Drafted.

**Pulling Direction** again controls the direction of the Draft Angle.

The **Parting Element** when selected using the **Define parting element** check box allows you project the draft onto an existing Plane or Face.

Again a Draft Node is added to the Specification Tree.

Creates a Variable Draft Feature. This is the same option that is available on the **Draft Deinition** panel.

This icon is not on the Drafting Toolbar but is available in the Part Design Workbench and used to create Advance Draft Features on multiple Faces and is not covered in the Foundation Course.
Creating a Shell Feature

Creating a Shell feature using selected Faces.

Select the icon to display the **Shell Definition** panel.

The following option are available: -

The **Default inside thickness** and **Default outside thickness** controls the cross sectional thickness of the **Shell** based on the outer faces of the currently select Work Object or Partbody.

The **Faces to remove** field indicates the Faces you have selected to be removed.

The **Other thickness faces** (Optional) allows you to apply different thickness on other faces.

After selecting the face(s) to be removed and the required thickness values click **OK** to create the Shell Feature.

In the following example the top face is selected as the face to be removed and the **Default inside thickness** is entered as 5mm with the Default outside thickness left at 0mm.
A Shell Node is added to the Specification tree.
The Thickness Feature

Adds or removes **Thickness** to selected Faces.

After selecting the icon a **Thickness Definition** panel will appear with the following options:

- The **Thickness** field allows you to specify the value for added or subtracting thickness from selected faces.

- The **Default thickness faces** indicate the faces that you have selected to apply the thickness value to.

- **Other thickness faces** (Optional) allows you to specify different thickness values for other faces during the same command.

After selecting the Face(s) to be thickened and entered the thickness value click **OK** to create the feature.

The following is an example of thickness of 100mm being applied to a single face.

A **Thickness** node is added to the Specification Tree.

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Creates a Thread/Tapped Hole Feature based on an existing Hole. This command is not covered in the Foundation Course.
Transformation Features Toolbar

Standard Transformations

Allows you to Translate, Rotate and create a Symmetry on existing Partbody or Feature.

Translates an existing Partbody or Feature in a user-defined direction.

When you select any of the standard transformation commands the following Question panel will appear.

If you select **NO** the command will be aborted. Selecting **YES** will display **Translate Definition** panel. There are only two inputs for the Translation: -

**Direction** controls the direction of the translation and requires you select an element i.e. a Line, Plane or Axis.

The **Distance** field requires you to enter a distance value for the translation.
Clicking **OK** completes the command and a **Translate** Node is added to the Specification Tree in the Partbody containing the currently Defined Work Object.

Rotates an existing Partbody or Feature through an angle about a user-defined Axis.

After selecting **YES** on the **Question** panel a **Rotation Definition** panel appears with the following options:

The **Axis** field indicates the element you have selected to be the axis of rotation.

The **Angle** field allows you to specify the angle of rotation about the **Axis**.

Clicking **OK** completes the rotate command and a **Rotation** node is added to the Specification Tree in the Partbody containing the currently Defined Work Object.

Creates a Symmetry an existing Partbody or Feature using a user defined Symmetry Line or Plane.

Again after selecting **YES** on the **Question** panel a **Symmetry Definition** panel appears with the option to select a **Reference** element that will be used as the Symmetry Line or Plane.

After selecting the **Reference** element click **OK** to completes the command and a **Symmetry** Node is added to the Specification Tree in the Partbody containing the currently Defined Work Object.

**Note:** When the using the above Transformation commands the resulting transformation is applied to the currently Defined Work Object i.e. the node Underlined in the Specification. If this is a Partbody then all the feature in the Partbody appear to be transformed. If the current Work Object is a feature within a Partbody then that feature transformed and not the Partbody.
**Mirror Body**

Allows you mirror a Partbody or Feature about a Face or Plane.

After selecting the icon you must select a Mirror Plane or face to display the Mirror Definition. The select element is displayed in the **Mirroring element** field. The Object to mirror field is deselected as the command is applied to the currently **Define Work Object** which could be a Partbody or a Feature. By clicking OK the mirror command is completed.

Below is an example of a Partbody mirrored about its own base.

A Mirror node is added to the Specification Tree.
Patterns

Allows you to create Patterns of existing Partbodies and Features.

Can be used to create a **Rectangular** Pattern of an existing feature.

Select the feature to be patterned either from the Specification Tree or graphically followed by the **Rectangular Pattern** icon. A **Rectangular Pattern Definition** panel will appear with the following option:

The **First Direction** tab contains the following options:

- **Parameter** field allows you to specify the type of spacing you require. The options can be displayed by clicking on the black down arrow:

  1. **Instance(s) & Length** equally spaces the number of instances entered in the **Instances** field through the distance value entered in the **Length** field.
2. **Instance(s) & Spacing** equally spaces the number of instances entered in the **Instances** field using distance value entered in the **Spacing** field to define the Spacing or Step size.

3. **Spacing & Length** automatically derives the instances by dividing the value entered in the **Length** field by the value entered in the **Spacing** field.

The **Reference Direction** allows you select an element to define the direction for the Pattern i.e. an Edge or Plane. The selected element is displayed in the **Reference element** field. The **Reverse** button reverses the direction.

The **Object to Pattern** displays the element that you have selected to Pattern. The **Keep Specifications** toggle maintains the specification of the selected Object and applies them to the Pattern i.e. in the case of a Hole, Blind, Up to Last, Up to Next, etc.

The **Position of Object in Pattern** allows you to specify where in the Pattern the selected object is placed. The options are:

1. The **Row in direction 1** positions the selected object at an instance position along direction 1 row.
2. The **Row in direction 2** positions the selected object at an instance position along direction 2 row.
3. The **Rotation angle** controls the angular position of the Pattern.

In the **Pattern Representation** portion of the panel, the checking of the **Simplified Representation** box allows you switch off the displaying of instances within the Pattern by selecting the instance centre point before clicking **OK** on the main panel.

To switch a instance back on you must double click on the Pattern Node in the Specification Tree to display the centre point and select it followed by clicking **OK** on the panel.

The **Second Direction** tab allows you to define a second direction for the Pattern and contains the same options that are on the **First Direction** tab.
After selecting a reference element for the direction of the Pattern and the required spacing options click **OK** to create the pattern.

Below is an example of a First Direction Pattern defined by **Instances** and **Spacing**.

A **RectPattern** Node is added to the Specification Tree.

Note: When using any of the Patterning command you can select multiple features with the use of the Ctrl key.
Allows you to create a **Circular Pattern** of an existing Partbody or Feature.

Select the feature to be patterned followed by the rotational pattern icon to display the **Circular Pattern Definition** panel and then click on the More button to view all options.

The **Axial Reference** tab contains the following options:

4. **Complete Crown** equally spaces the pattern through 360°.

5. **Instance(s) & total angle** equally spaces the number of instances entered in the **Instances** field through the angle entered in the **Total angle** field.

6. **Instance(s) & angular spacing** equally spaces the number of instances entered in the **Instances** field through the angle entered in the **Angular spacing** field.

7. **Angular spacing & total angle** automatically derives the instances by dividing the angle value entered in the **Total angle** field by the value entered in the **Angular spacing** field.

The **Reference Direction** allows you select an element to define the Axis of rotation for the Pattern. The selected element is displayed in the **Reference element** field. The **Reverse** button reverses the direction of the angular rotation.

The **Object to Pattern** displays the element that you have selected to Pattern. The **Keep Specifications** toggle maintains the specification of the selected Object and applies them to the Pattern i.e. in the case of a Hole, Blind, Up to Last, Up to Next, etc.
The **Position of Object in Pattern** allows you to specify where in the Pattern the selected object is placed. The options are:

4. The **Row in angular direction** positions the selected object at an angular instance position.

5. The **Row in radial direction** positions the selected object radially.

6. The **Rotation angle** controls the angular position of the Pattern.

The **Rotation of instance(s)** allows you to align instances radially about the axis of rotation of the Pattern by ensuring the **Radial alignment of instance(s)** box is checked.

In the **Pattern Representation** portion of the panel, the checking of the **Simplified Representation** box allows you switch off the displaying of instances within the Pattern by selecting the instance centre point before clicking **OK** on the main panel. To switch a instance back on you must double click on the **Pattern Node** in the Specification Tree to display the centre point and select it followed by clicking **OK** on the panel.

The **Crown Definition** tab allows you to create multiple Circular Patterns concentrically about the Axis of rotation.

The following options are available:

1. **Circle(s) & crown thickness** allows you to specify the number of Circular Patterns in the **Circles(s)** field and the total radial distance from the axis of rotation to the outer pattern in the **Crown Thickness** field.
2. **Circle(s) & circle spacing** allows you to specify the number of Circular Patterns in the **Circles(s)** field and the radial distance between each Pattern in the **Circle spacing** field.

3. **Circle spacing and crown thickness** allows you to specify the radial distance between each Pattern in the **Circle spacing** field and the total radial distance from the axis of rotation to the outer pattern in the **Crown Thickness** field.

After selecting a reference element for the axis of rotation and the required spacing options click **OK** to create the pattern.

The following is an example of Circular Pattern using a **Complete Crown** spacing with 4 **Instances**.

A **CircPattern** Node is added to the Specification Tree.
Allows you to create Patterns of existing Partbodies or Feature by using a User Defined Pattern.

Select the feature to be patterned followed by the User-Defined Pattern icon. A User Pattern Definition panel will appear with the following option:

The **Instances** portion of the panel allows you select Points or a sketch containing multiple Points to be used to position the instances. The selected elements are listed in the **Positions** field. The **Number** field is deselected although you can use MB3 to access a contextual menu to edit the number of instances displayed under the **Add Range** option.

The **Objects to Pattern** portion of the panel displays the feature that you have selected to Pattern in the **Object** field. By default the instances are positioned using their Centre of Gravity Point i.e. the centre of the circle. This may be changed by selecting the **Anchor** field and then you should select a new point or Vertex. **Keep Specification** copies specification of the selected feature to patterned and applies them to the instances.

After selecting the positioning elements and the required options click **OK** to create the pattern.

Below is an example of a User-Defined Pattern based on Points held in a Sketch.
A **UserPattern** Node is added to the Specification Tree together with the defining positioning Sketch.

![Diagram](image)

**Scaling**

Scales the currently Defined Work Object Partbody or Feature.

Select the **Scaling** icon the display the **Scaling Definition** panel. You must select a **Reference** Point, Plane or Planar Surface, which will be used as the origin of the scaling operation. The selected element will be displayed in the Reference field. You can enter a scaling ratio in the **Ratio** field (The default is 1).

Click **OK** to complete the command.

A **Scaling** Node is added to the Specification Tree below the currently Define Work Object or in the current Partbody.
Part Design Constraints Toolbar

You can use constraints to control the distance and orientation of Partbodies and Features within the Part Design Workbench.

Create Constraints using a dialog box

As in the Sketcher Workbench, creates Constraints using a Dialog Box.

Select the Feature(s) to be constrained followed by the Constraint Dialog Box icon. A Constraints Definition panel will appear from which you can select the desired constraints by using the check boxes followed by clicking OK to apply them.
Create Constraints by selecting elements

As in the Sketcher workbench this command creates Geometric and Dimensional Constraints by selecting elements on Features.

After selecting the Constraints icon you can either:

1. Select a single element on a Feature to apply a constraint to. This will result in the default constraint being generated based on the element that is selected. You can also use the contextual menu MB3 to apply different constraints i.e. Vertical or Horizontal Dimensions, Distance, Angle, Parallelism, etc. If you apply a constraint to the element that is controlled by a feature generated constraint i.e. the extrusion length of a Pad, then a reference constraint will be created.

2. Select two elements to apply a constraint between them. Again the contextual menu can be used to apply different constraints.

The Constraints are added to the top of the Specification Tree.
Using Compass Manipulation

Another way to reposition or manipulation geometry is to use the Compass that displayed in the top right corner of the graphics display.

Using MB1 select the red square on the Compass and Drag and Drop it onto the Face of a Feature. The Compass will attach itself to the Face and will change colour to green.

If you select the bottom element of the compass and drag it the attached feature or Partbody will move with it.
By selecting one of the Axis elements you can drag the compass and the attach feature along that Axis.

Selecting one of the arc elements of the compass allows you to rotate the compass and the attached Feature around the origin of the compass in the Plane of the arc.

By selecting the Point on the top of the ‘Z’ Axis allows you to rotate the compass and the attached Feature freely about the compass origin.
If you double click on the compass itself a Parameters for Compass Manipulation panel will appear. This panel allows you to manipulate the compass position and orientation by entering values in the relevant fields.

![Parameters for Compass Manipulation Panel]

When you have finished with using the compass to position it back on the graphic window select the red Point on the compass using MB1 and drag it onto the graphic area. The compass will then snap back to the top right corner of the window.

**Note:** When attempting to use the compass to manipulate features and Partbodies a warning panel may be displayed informing you that there are constraints or other feature controlling the position and orientation. In this case the compass can not be used.
Creating an User Defined Axis System

When a new CATPart is created a default Axis System is created positioned on the origin of the part at the intersection of the default XY, YZ and ZX Planes. This Axis System is named the **Absolute Axis System** and is displayed in Specification Tree and has a node attached to the Axis Systems node.

Graphically the **Absolute Axis System** is made of an Origin point and X, Y and Z Axis Lines together 3 symbols that define the XY, YZ and ZX planes of the Axis.

Unlike the three default Planes the **Absolute Axis Systems** position and orientation can be altered by the user. Therefore it is worth bearing in mind that any sketches or elements defined using this axis are not necessarily fixed at the origin of the part. Also this axis can be deleted.

There may be occasions when you will need to define your own axis, to do this select the following icon on the bottom menu bar.

An **Axis System Definition** panel will appear with the following options: -

The **Axis System type** field allows you select the following: -

1. A **Standard** axis, which allows you select an **Origin** point by selecting a point or by using **MB3** over the **Point** field you can enter co-ordinates and create reference geometry. The X, Y and Z **Axis** fields allow you to select element to define the relevant axis or again you can **MB3** over the relevant field to enter **co-ordinates** or an angle of **rotation**. All co-ordinate position and angular orientations are relative to the currently active Axis System, which will be highlighted, orange on
the Specification Tree and is displayed graphically in a solid line font. If no Axis System is present in the part then the default planes and their intersection are used as the reference for the new axis.

2. An **Axis Rotation** axis defines the axis by defining an **Origin** Point, a direction for one of the **axis** (X, Y or Z), a **Reference** element and **Rotation** angle to define the second axis. Again you can use MB3 over the selection fields to enter co-ordinates or create reference geometry.

3. An **Euler Angle** axis defines the axis based on an **Origin** Point and **X**, **Y** and **Z Angles**, again is based on the currently active Axis System.

The following is an example of a how to create a Standard user defined axis who’s position and orientation is based on the **Absolute Axis System**.

After selecting the Axis System icon use MB3 over the origin field to display the contextual menu. Select **Coordinates...** to display a **Origin** panel which will allow you to enter the **X**, **Y** and **Z** values for the origin of the axis based on the origin of the currently active Axis. After entering the co-ordinate values click **Close** to close the **Origin** panel.

Leave the **X**, **Y** and **Z Axis** fields with **No Selection** displayed and click **OK** to create the axis. The axis is created with the **X**, **Y** and **Z** Axis aligned in the same orientation as the currently active Axis. The new Axis is now set to the active Axis.