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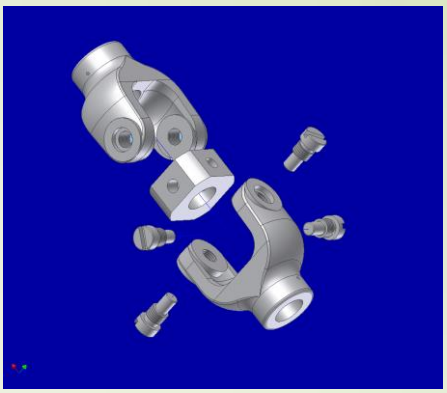


AutoCAD Inventor Advanced

Use CAD to create and display 3D models.

Table of Contents

Topic 1 – Assemblies & Sub-Assemblies:

Required Skills:

On completion of the session, the participants will be able to:

- Create a new assembly file.
- Place parts in the assembly file
- Turn the Degrees of Freedom ON and OFF.
- Place

Required Knowledge:

- Enter the Autodesk Inventor program and navigate the core functions.

Lesson Overview:

The aim of the lesson is to give the operator the necessary skills to create an assembly drawing from existing parts. The lesson will cover creating an assembly using a basic bottom-up assembly design method, placing external components, changing component colours, applying assembly constraints, analysing degrees of freedom and saving design views. In creating an assembly, the lesson will cover:

- Basic bottom-up assembly design.
- Placing external components.
- Changing component colours.
- Using assembly constraints.
- Analysing degrees of freedom.
- Using design view representations.

Assemblies:

An **Assembly** is a group of parts constrained together to show the assembled machine or structure, with all parts, components and fastenings in their functional positions.

A **Sub-Assembly** is a portion of the final product which is assembled separately and later included, usually in module form, to the final assembly. A subassembly may be created in the current assembly or referenced from an external file.

Assemblies and Sub-Assemblies are used to:

- check that the individual parts and components are correct and that the parts will mate correctly,
- check the clearance of moving parts and components,

Assembly drawings can be created using one of three different methods, the Top-Down, Bottom-Up and Middle-Out.

Top-Down:

Top-down is an assembly design method by which the user creates all new parts for the assembly. The user logically partitions the overall product design by creating a hierarchy of subassemblies starting at the top of the assembly tree. The assembly is built up using simple parts with the finer details being added as the assembly takes shape.

Bottom-Up:

Bottom-up is an assembly modelling method that combines existing parts and subassemblies to create an assembly model beginning at the bottom of the assembly tree. The assemblies are created using commercially available or standard parts that have been created on their own drawing files.

Middle-Out:

Middle-out is an assembly design method incorporating both the top up and bottom down methods that is carried out by incorporating existing parts and creating new parts to build the assembly model. This approach is the most common design method and allows the operator to move up or down the assembly tree.

For ease of delivery, this lesson will concentrate on the Top-Down method using parts created and stored in a folder called Ram. The Bottom-Up and Middle-Out methods will be included in future lessons.

Creating an Assembly:

A new Assembly or Sub-Assembly file is created using the Assembly button in the New File dialog box. Once the new file has been created, the graphics area is displayed in which existing parts are placed or new parts created. The assembly mode displays ten tabs on the Ribbon menu consisting of Assembly, Design, Model, Inspect, Tools, Manage, View, Enhancements, Vault and Get Started.

Adding Parts to an Assembly:

The Assembly tab is the normally the most frequently used tabs in the Assembly area and contains the buttons and commands necessary to place (or insert) parts into the file, and place them so they will not move, or move laterally or rotate in relation to another part. The Assembly tab features eight separate panels; Component, Position, Manage, iPart/iAssembly, Productivity, Work Features, Begin and Convert.

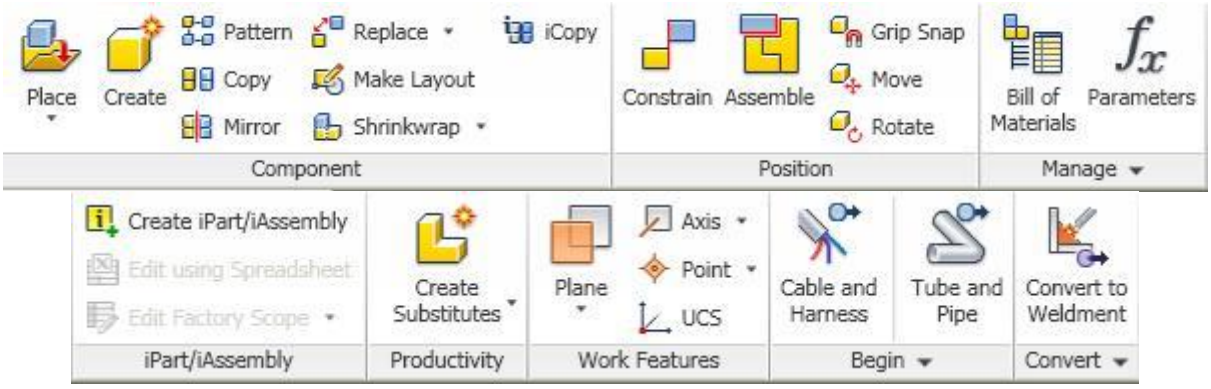


Figure 1.1

In this topic, existing component parts will be placed into the drawing using the Place Component icon in the Component panel with the parts then being assembled using the Constraints icon in the Position panel. Other buttons such as copied, patterned (arrayed) or mirrored will be covered in future topics.

The steps used to create an assembly are:

- Place the external parts in the new assembly file.
- View the degrees of freedom of the components.
- Constrain the components to restrict movement as necessary.
- Save the design view.

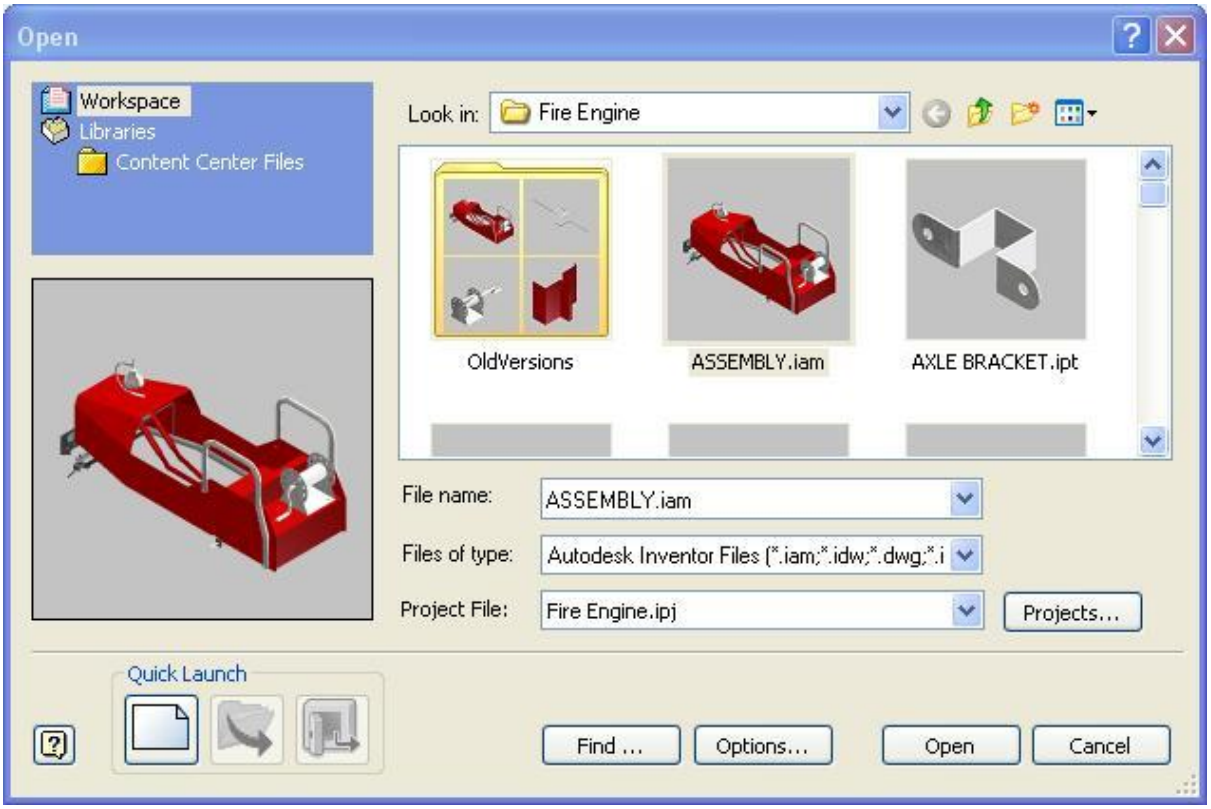
To reduce the number of parts and constraints in a complex assembly, various sub-assemblies which are saved as separate files and all brought together in a main assembly.

Place Component

The Place Component button is used to place (or insert) existing components into an assembly. The first component in an assembly is automatically positioned with its origin coincident with the assembly coordinate origin and becomes the grounded

part as indicated by the symbol in the Browser. Additional components are positioned with the cursor, attached at the component's centre of gravity.

Entering the command displays the Place Component dialog box.



The component is located by expanding the drives and folders in the "*Look in*" box to display the part (and/or sub-assembly) files. The part file is selected and the Open box clicked to place the part on the drawing. Multiple components can be simultaneously placed by holding down the CTRL key and left clicking on the components; to complete the command, click the Open button.

Tutorial Exercise 1-1:

Activate the Topic 1 project and then create a new assembly drawing file in your work area called Hinge Assembly and place the 2 Hinge Plates and the Pin using the existing parts file located the Topic 1 folder on network drive.

Procedure:

1. Enter the Inventor program by clicking on the AutoCAD Inventor icon.
2. Click the Projects button then locate the project file called Topic 1 in your work area.
3. Select the project called Topic 1, then click on **Apply**, and then **DONE**.
4. Click the New button in the Get Started tab, then **Metric**, and then the Standard (mm.iam) . Button.
5. Click **OK** to create the new file .

Place Plate 1 as the Base part.

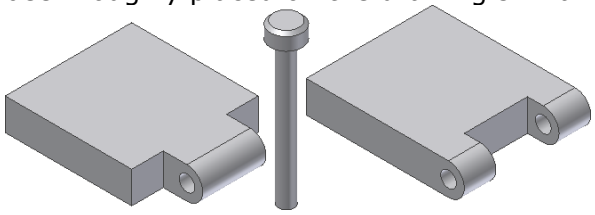
6. Click the Place Component button in the Assembly Panel.
7. Select the file called Plate 1 and then click **Open**.
8. Place the part anywhere in the graphics window by clicking the Left Mouse Button (LMB) once.
9. Click the Right Mouse Button (**RMB**) and select then click **Done**.
   Notice the part has been placed on the drawing.

Place Plate 2 as a separate part.

10. Click the Place Component button in the Assembly Panel.
11. Select the file called Plate 2 and then click **Open**.
12. Place the part anywhere in the graphics window by clicking the **LMB** once.
13. Click the **RMB** and select then click **Done**.
    Notice 2 parts now appear on the drawing.

Place the Pin as a separate part.

14. Click the Place Component button in the Assembly Panel.
15. Select the file called Pin in the Hinge folder.
16. Click **Open** and then click the **LMB** to locate the component.
17. Click the **RMB** and select **Done**.
    The 3 parts have been roughly placed on the drawing similar to that shown below.



Degrees of Freedom:

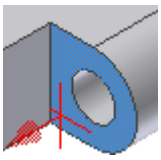

The Degrees of Freedom symbol appears on the part and is displayed by clicking the View tab and selecting the Degree of Freedom button; the Degree of Freedom symbol indicating the planes and axes the part can be moved is shown near the centroid of each part


Each part in an assembly has six degrees of freedom. It can move along X, Y, and Z axes (translational freedom)  and rotates around the X, Y, and Z axes (rotational freedom)  . As the part is constrained to other parts the number of allowable movements displayed is reduced. When a component is fully constrained in an assembly, it cannot move or rotate in any direction as all of its degrees of freedom are removed.

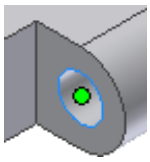
Assembly Constraints:

Activating the Constraint button displays the Place Constraints dialog box which has the Assembly, Motion, Transitional and Constrain Set tabs.

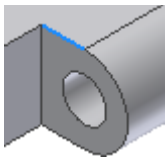
The selected geometry is usually a component face, but select curves, planes, edges, or points for a mate constraint, or faces and planes for a flush constraint can be selected as shown in the following Figures.



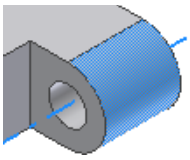
Face selected



Centre of hole selected



Edge selected



Axis or Face

Assembly Tab:

Assembly constraints remove degrees of freedom between selected components. Adaptive components (to be covered in **Error! Reference source not found.**) may allow the resizing or a change shape when constraints are applied. The assembly tab specifies the type of constraint (Mate, Flush, Angle, Tangent or Insert) and illustrates the solution used to position components before the constraint is applied. The constraint type can be changed and constraints placed when the dialog box is open. When a face, curve, or point is selected, an arrow shows the default direction of the solution. Click constraint icons on the dialog box to change solution.

Mate/Flush:

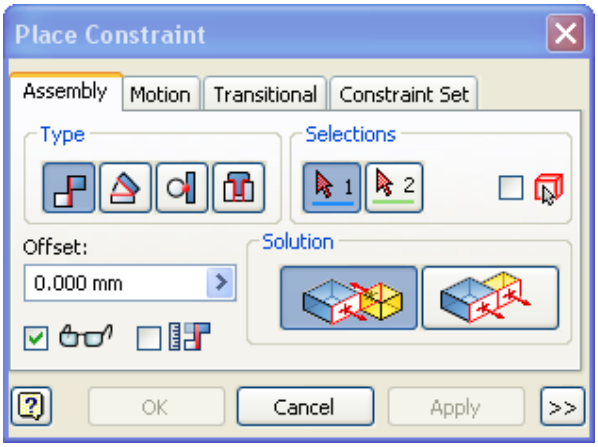
The Mate constraint positions selected faces parallel to one another, with faces coincident or aligns parts adjacent to one another with faces flush. The faces may be offset from one another. A mate constraint can be quickly placed with no offset by holding down the Alt key and dragging the assembly component into position.

- Selecting Mate  in the Solution area allows surfaces to be mated together.
- Flush  places the faces in line with each other. The distance between the faces is controlled by typing the distance in the Offset box.

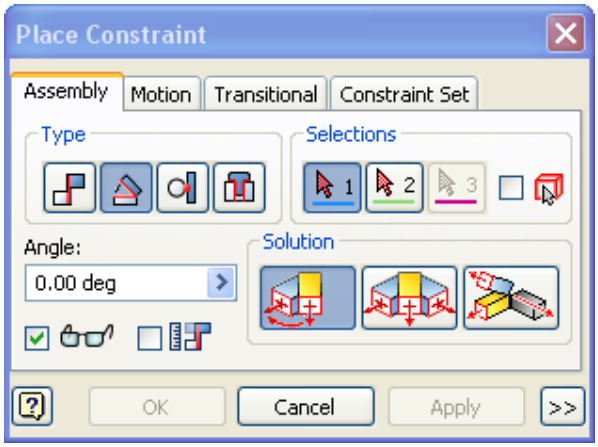
Angle:

The Angle constraint positions edges or planar faces on two components at a specified angle to define a pivot point by removing one degree of freedom in rotation or two degrees of angular rotation between planar surfaces. Three solution options (Directed Angle, Undirected Angle and Explicit Reference Vector) are available depending on the arrangement of the parts.

- The Directed Angle  solution always applies the right-hand rule. Does not support user specified limits and resting position.
- Undirected Angle  allows either orientation, thus resolving situations where component orientation flips during a constraint drive or drag. Supports user specified limits and resting position.
- Explicit Reference Vector  defines direction of Z axis vector (Cross Product) by adding a third pick to the selection process. Reduces tendency of angle constraint to switch to an alternate solution during a constraint drive or drag. This solution is the default. Supports user specified limits and resting position.



Mate/Flush Constraint



Angle Constraint

Tangent:

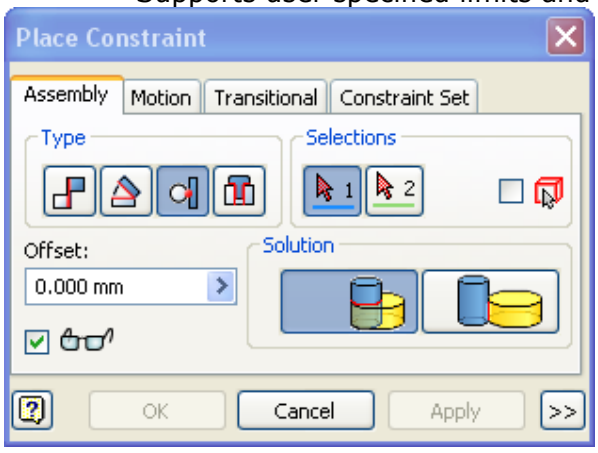
The Tangent constraint causes faces, planes, cylinders, spheres, and cones to contact at the point of tangency. Tangency can be inside or outside a curve, depending on the direction of the selected surface normal. A tangent constraint removes one degree of linear translation. Between a cylinder and a plane, it removes one degree of linear freedom and one degree of rotational freedom.

- Inside Positions  - the first selected part inside the second selected part at the tangent point. Supports user specified limits and resting position.
- Outside Positions  - the first selected part outside the second selected part at the tangent point. Outside tangency is the default solution. Supports user specified limits and resting position.

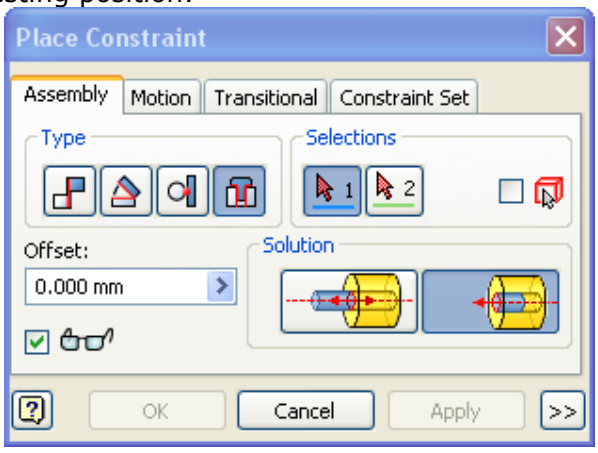
Insert:

The Insert constraint is a combination of a face-to-face mate constraint between planar faces and a mate constraint between the axes of the two components. The Insert constraint positions a bolt shank in a hole. The shank aligns with the hole and the bottom of the bolt head mates with the planar face. A rotational degree of freedom remains open.

- Opposed  reverses the mate direction of the first selected component. Supports user specified limits and resting position.
- Aligned  reverses the mate direction of the second selected component. Supports user specified limits and resting position.



Tangent Constraint

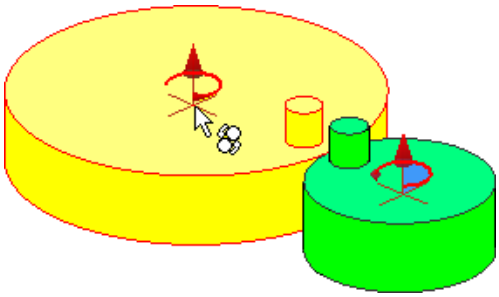


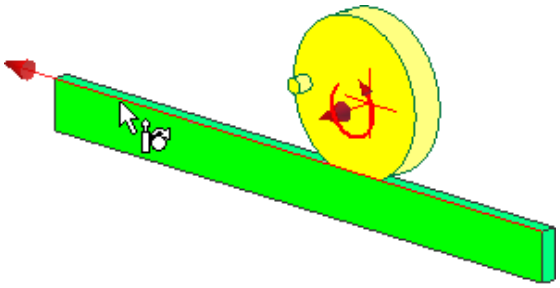
Insert Constraint

Motion Tab:

Motion constraints specify the intended motion between assembly components; because they operate only on open degrees of freedom, they do not conflict with positional

constraints, resize adaptive parts, or move grounded components. Motion constraints are shown in the browser and when clicked, or the cursor hovers over the browser entry, constrained components are highlighted in the graphics window. Drive constraints are not available for motion constraints however, parts that are constrained using motion constraints will drive according to the direction and ratio specified.



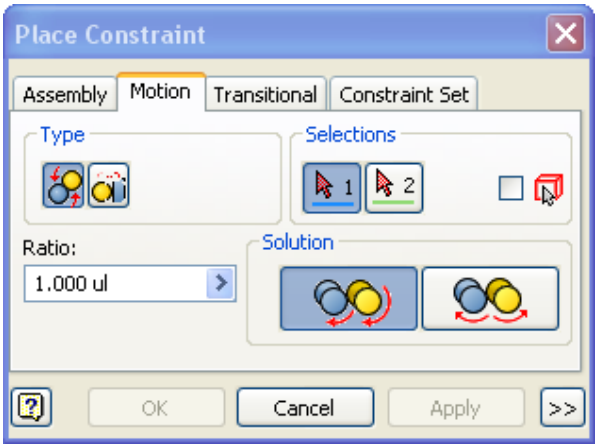


Rotation:

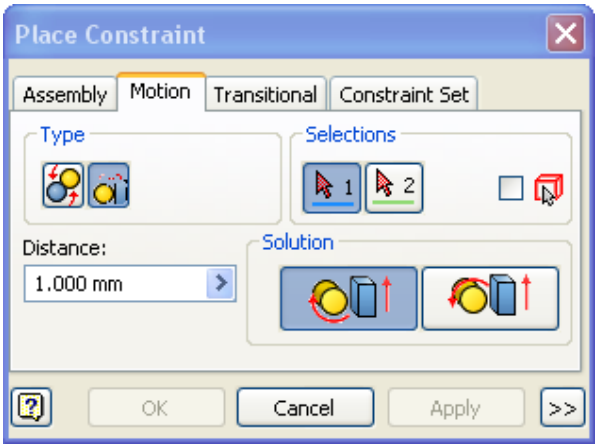
The Rotation constraint specifies that the first selected part rotates in relation to another part using a specified ratio. It is typically used for bearings, gears and pulleys.

Rotation-Translation:

The Rotation-Transition constraint specifies that the first selected part rotates in relation to translation of another part using a specified distance. It is typically used to show planar motion, such as a rack and pinion.



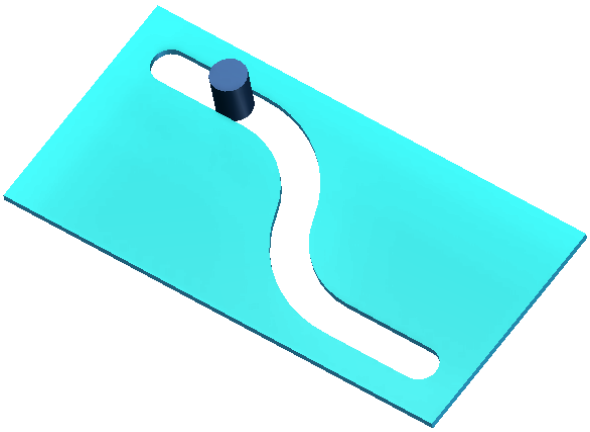
Rotation

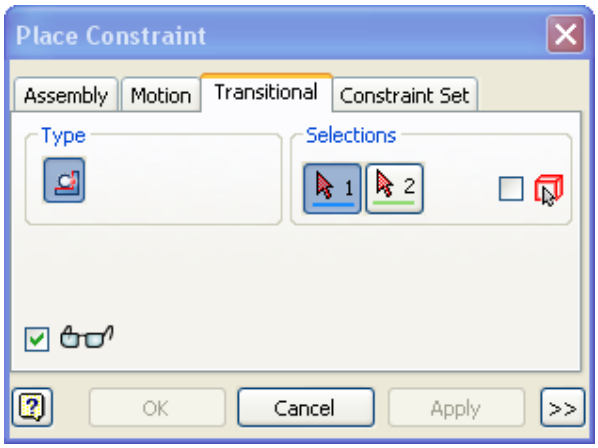


Rotation-Transition

Transitional Tab:

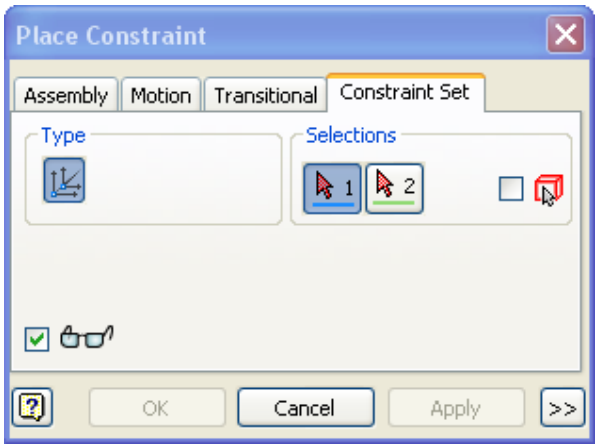
A transitional constraint specifies the intended relationship between (typically) a cylindrical part face and a contiguous set of faces on another part, such as a cam in a slot. A transitional constraint maintains contact between the faces as the component is slid along open degrees of freedom.





Constrain Set Tab:

Constraint Set enables two UCSs to be constrained together. The UCS can be selected in part or assembly file.



Tutorial Exercise 1-2:

Using the parts placed in Tutorial Exercise 1-1, constrain the 2 Hinge Plates together so the tongue fits into the groove and then constrain the Pin through the holes to complete the assembly. Use an offset of 0 between all components.

Procedure:

Mate the tongue of Plate 1:1 with the groove of Plate 2:1.

1. Activate the Place Constraints dialog box by clicking on the Place Constraints button in the Position panel on Assembly tab.
2. In the Assembly tab, select the Mate button.
3. Click the Mate box. *The dialog box temporarily disappears to allow the mating geometry to be selected*.
4. Select the face of the tongue on Plate 1:1 as shown in Figure 1.2.
5. Select the face of the groove on Plate 2:1 as shown in Figure 1.3.
6. Check that the Offset is 0.0 and then click **Apply** to execute the constraint.

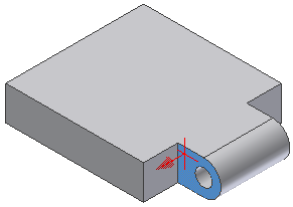


Figure 1.2

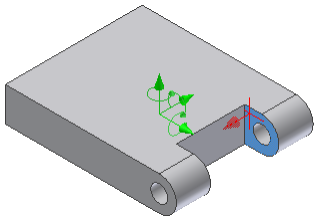


Figure 1.3

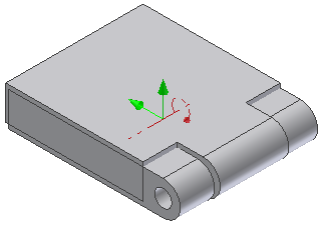


Figure 1.4

If the components do not appear similar to that shown in Figure 1.4, undo the constraint and reselect the faces in their correct sequence, i.e. Plate 2:1 1st followed by Plate 1:1 2nd using the Mate option (and not the Flush option).

Add an angle constrain between the 2 Plates. For clarity, separate the 2 parts by left clicking on Part 2:1 and dragging up a short distance.

7. Click the Angle button.
8. Select the top edge of Plate 1:1 as shown in Figure 1.5.
9. Select the bottom edge of Plate 2:1 as shown in Figure 1.6
10. Change the angle to **-45** and then click on the Undirected Angle solution.
11. Click **Apply** to execute the constraint.

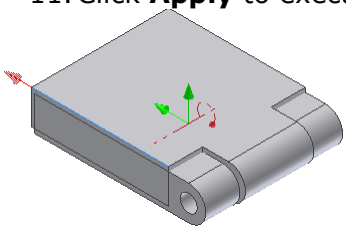


Figure 1.5

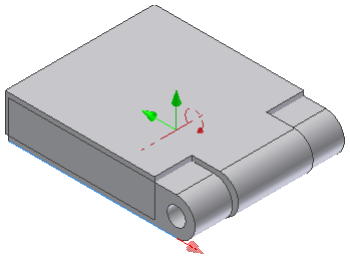


Figure 1.6

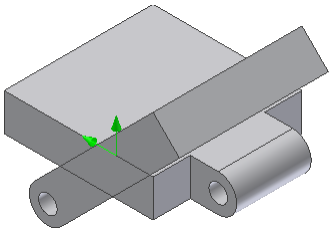


Figure 1.7

The components may not appear as that shown in Figure 1.7, undo the constraint and reselect the faces in their correct sequence, i.e. Plate 1:1 1st followed by Plate 2:1 2nd; the angle may need to be 45 depending on the sequence of edges selected.

Position the 2 holes so the axis the holes are aligned.

12. Click the Mate button.
13. Move the cursor over the hole on Plate 1:1 until the axis is indicated as shown in Figure 1.8 and select using the **LMB**.
14. Move the cursor over the hole on Plate 2:1 until the axis is indicated as shown in Figure 1.9 and select using the **LMB**.
15. Check that the Offset is 0.0 and then click **Apply** to execute the constraint.

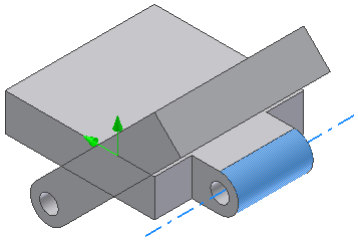


Figure 1.8

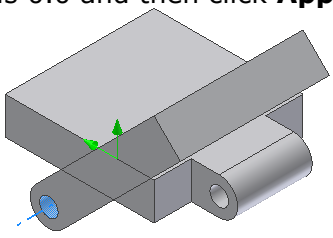


Figure 1.9

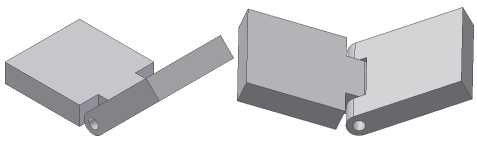


Figure 1.10

The components should appear as that shown in Figure 1.10

Constrain the Pin along the axis of the hole.

16. In the Assembly tab, click **LMB** on the Mate button
17. Click the Mate box. The dialog box temporarily disappears to allow the mating geometry to be selected.
18. Select the shaft of Pin:1 as shown in Figure 1.11.
19. Select the hole through Plate 2:1 as shown in Figure 1.12.
20. Check that the Offset is 0.0 and then click **Apply** to execute the constraint.

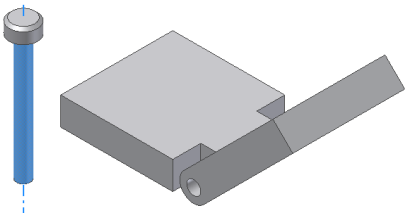


Figure 1.11

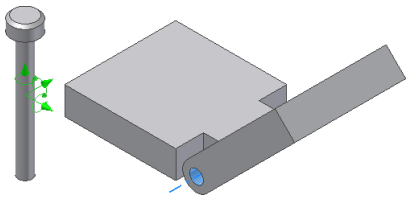


Figure 1.12

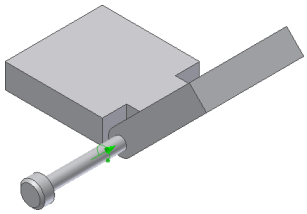


Figure 1.13

The components should appear as that shown in Figure 1.13

Constrain the Pin along the axis of the hole.

21. In the Assembly tab, click **LMB** on the Mate button then click Mate .
22. Select the underside of the head on Pin:1 as shown in Figure 1.14.
23. Select the face of Plate 2:1 as shown in Figure 1.15.
24. Check that the Offset is 0.0 and then click **Apply** to execute the constraint.
25. Click **Cancel** to exit the Constraints dialog box.

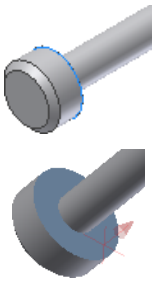


Figure 1.14

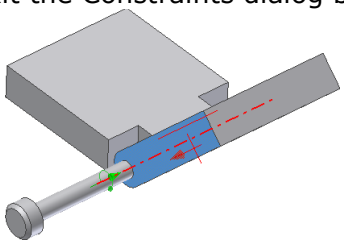


Figure 1.15

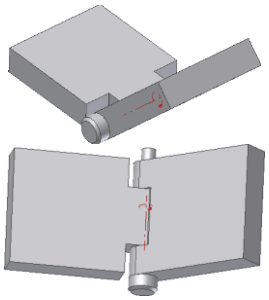


Figure 1.16

The components should appear as that shown in one of the images of Figure 1.16.

Change the view angle to look directly down on Plate 1:1 then show a 5mm gap between the Pin's head and the face of Plate 2:1

26. Left click on the View Face button in the Navigation Bar then left click on the top surface of Plate 1:1.
27. Expand the features for the Pin 1:1 in the Browser by left clicking the sign ⊞.
28. Right click **RMB** on the last Mate constraint (Mate:4) and then selecting **Edit**.
29. In the Edit Constraint dialog box, change the Offset from 0.000 to **5** and click **OK**.
30. The assembly should appear like that shown in Figure 1.

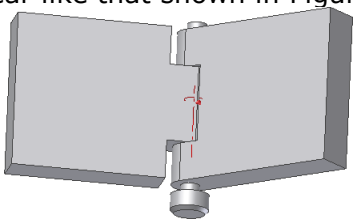
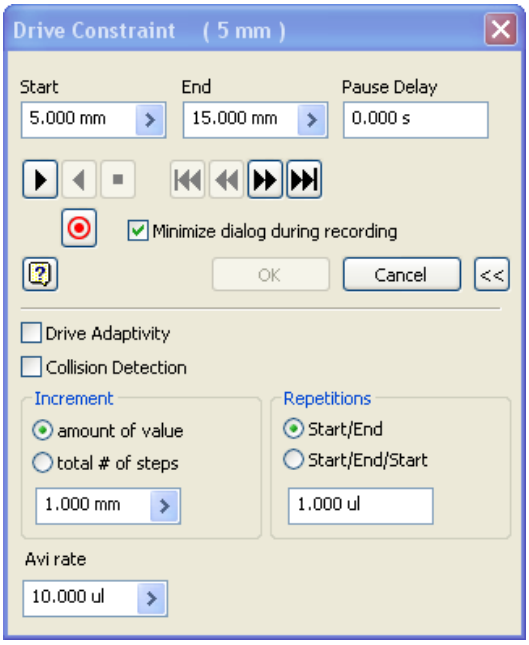


Figure 1

31. Save the work in your work area by clicking the Save icon.

Driven Constraints:



Mechanical motion can be simulated by driving a constraint through a sequence of steps. After the component has been constrained, the Drive Constraint tool can be used to animate the various parts by incrementally changing the value of the constraint. For example, a component can be rotated by driving an angular constraint from zero to 360 degrees. The Drive Constraint tool is limited to one constraint, but additional constraints can be driven by using the Equations tool to create algebraic relationships between the various constraints.

Drive Constraints can also be used to detect the interference between components; when an interference or collision occurs, the motion stops, allowing the component proportions or positions to be adjusted.

The Drive Constraints command is accessed in the Browser by expanding the part and then right clicking over the constraint to be made driven. The **Drive Constraint** is selected from the menu to display the Drive Constraint dialog box. The constraint's offset is shown in brackets next to the dialog box's name in millimetres or degrees.

NOTE: To locate the drive constrain at a later date, it is recommended the name of the constraint be renamed to include the words **Driven Constraint** added before the name of the drive constraint.

Start:
Sets start position of the offset or angle. The value may be entered, measured, or set to a dimensioned value. The default value is the defined offset or angle.

End:
Sets end position of the offset or angle. The value may be entered, measured, or set to a dimensioned value. The default is start value plus 10.

Pause Delay:
Sets delay between steps in seconds. The default is 0.0 which displays a smooth moving continuous operation.

& 

The Expand 

button expands the dialog box to display additional options while the Reduce 

button reduces the size of the dialog box to display the main options only.

Control Buttons:

Advance and reverse the drive sequence. The dialog box remains open while the drive sequence plays. The values may be changed any time the sequence is paused or stopped.

Forward:

Forward drives the constraint forward and is not available unless both Start and End boxes have values. The forward play may be resumed after a Stop.

Reverse:

Reverse drives the constraint in reverse and is not available unless both Start and End boxes have values. The reverse play may be resumed after a Stop.

Pause:


Pause temporarily stops the constraint drive sequence. It allows the values to be edited and forward or reverse play to be resumed, advanced a step at a time, or advanced to beginning or end.

Minimum:


Minimum returns the constraint to the starting value and resets the constraint driver. The minimum option is not available unless the constraint driver has been run.

Step Reverse:


Single Step Reverse, reverses the constraint driver one step in the sequence. The option is not available unless the drive sequence has been stopped.

Step Forward:


Single Step Forward advances the constraint driver one step in the sequence. The option is not available unless the drive sequence has been stopped.

Maximum:


Maximum or Go to End advances the constraint sequence to the end value.

Record:


The record option begins to capture the frames at the specified rate for inclusion in an animation or video.

Drive Sequence Settings:

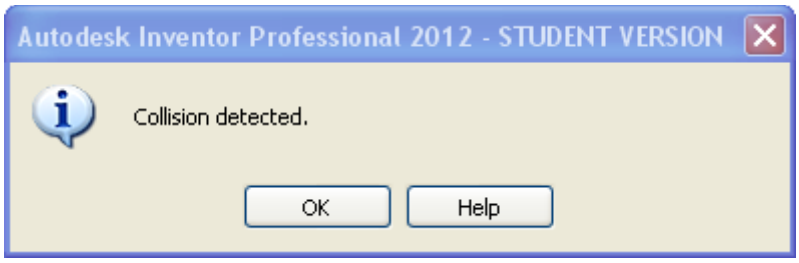
The Drive Sequence setting set a precise amount of motion along a vector or about an axis and is used to simulate motion or test a range of motion for a constrained component.

Drive Adaptability:

The Drive Adaptability is activated by selecting the check box to adapt components while maintaining the constraint relationship, if needed.

Collision Detected:

Select check box to drive the constrained assembly until a collision is detected. When an interference is detected, it is displayed and its constraint value shown. The value at collision is displayed in brackets next to the dialog box name.



Increment:

Amount of Value specifies that the increment will be the value specified in the edit box. Default is 1.0. Start/End/Start drives the constraint from the start value to the end value and then in reverse to the start value. *Number of cycles* completed within one repetition is dependent on the Edit Box value. *Edit box* sets the value of each increment or the number of steps. Value may be measured, entered, or as dimensioned.

Repetitions:

Start/End drives the constraint from the start value to the end value and resets at the start value. *Start/End/Start* drives the constraint from the start value to the end value and then in reverse to the start value. Number of cycles completed within one repetition is dependent on the Edit Box value. *Edit box* sets the number of repetitions for Start/End and the number of cycles for Start/End/Start. Each movement is considered one cycle.

Avi Rate:

Specifies the increments at which a "snapshot" is taken for inclusion as a frame in a recorded or video animation.

Tutorial Exercise 1-3:

Continue with the assembly created in Tutorial Exercises 1-1 and 1-2 and add a drive constraint between the Plate 1:1 and Plate 2:1 starting at -90° and ending at 90° to the Angle:1 constraint allowing 3 repetitions to the animation.

Procedure:

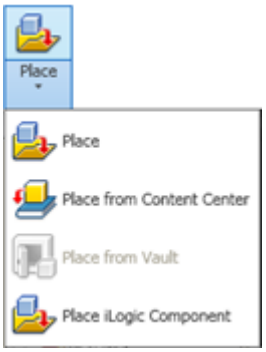
1. Expand the hierarchy of Plate 1:1 if not already expanded by clicking on the ⊞ indicator in the Browser.
2. **RMB** on the Angle:1 (-45.00 deg) constraint button and select **Drive Constraint** from the Context menu.
3. In the Drive Constraint dialog box, change the start value to **-90** and the End value to **90**.
4. Expand the options by clicking the Expand << button.
5. Change the Repetitions to **3** and **Start/End/Start**.
6. Change the Increment to **2**.
7. Click the Forward ▶ button to view the motion in action.
8. Click **Apply** when the drive constraint works correctly.

Modify the drive constraint by changing the Start value to 120.

9. Change the Start value from 90 to **120**.
10. Left click on the Collision Detection box to activate.
11. Click the Forward ▶ button to view the motion in action.
    Plate 2:1 should automatically stop when an angle of 90 is reached and a warning message displayed.
12. Click the Reverse ▶ button to view the motion in action the opposite direction.
    Plate 2:1 should automatically stop when an angle of -90 is reached.

Fastenings Library:

Inventor contains a vast library of fasteners, steel shapes, bearings and other parts required when working with mechanical drive systems that are used in assembling or creating components and can only be used in the Assembly mode.



The Library is located by expanding the Place button in the Component panel of the Assembly tab and selecting Place from Content Centre  which displays the Place from Content Centre dialog box as shown in Figure 1.17. A list of component types is listed in the left column; each component can be expanded by clicking on the ⊞ sign to the left of each component type.

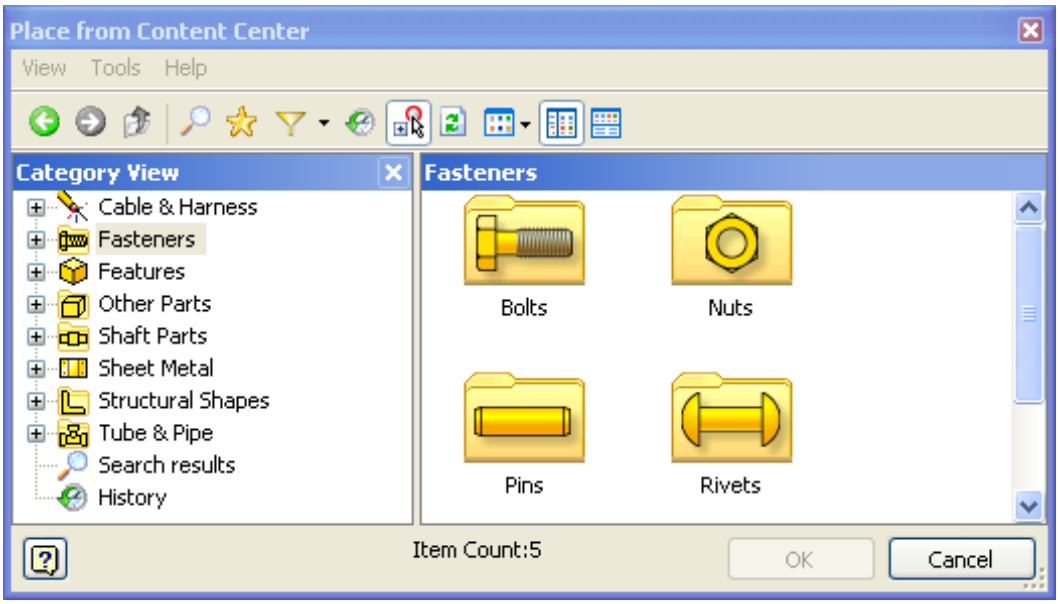


Figure 1.17

Expanding the Fasteners indicates a list of standard fastener types (bolts, nuts, washers etc) as shown in Figure 1.18. Expanding Bolts allows the different types of bolts (Countersunk, Hexagonal Head, Round Head etc) to be selected as shown in Figure 1.19 while further expanding the Hex Head indicates further types of bolts as shown in Figure 1.20.

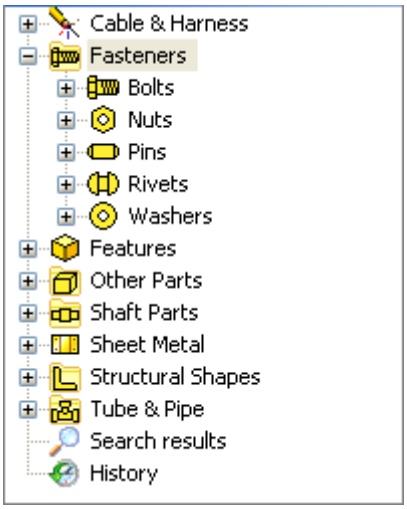


Figure 1.18

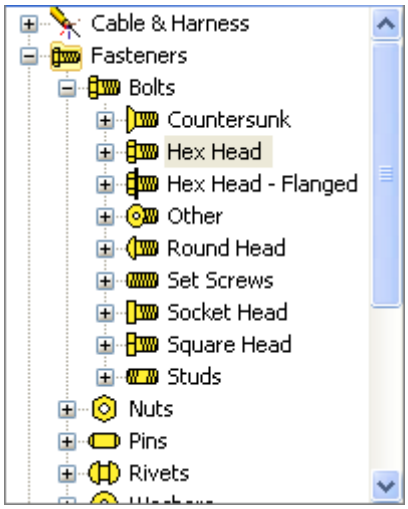


Figure 1.19

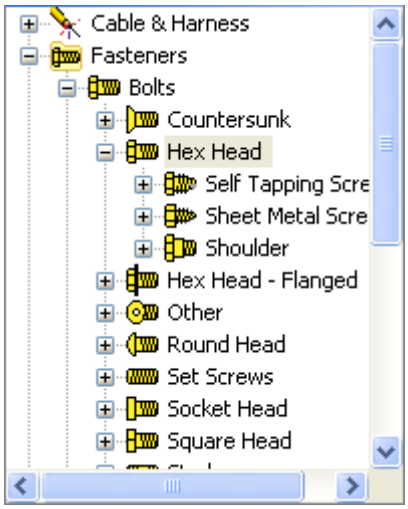


Figure 1.20

The right column of the Place from Contents Centre dialog box contains a list of the different folders and parts (Figure 1.21)

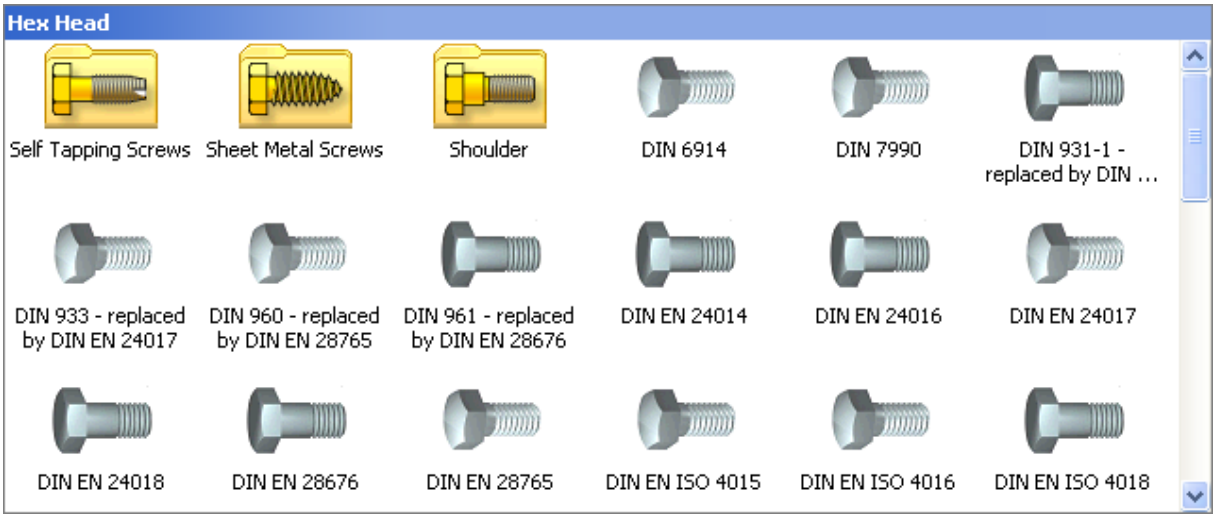
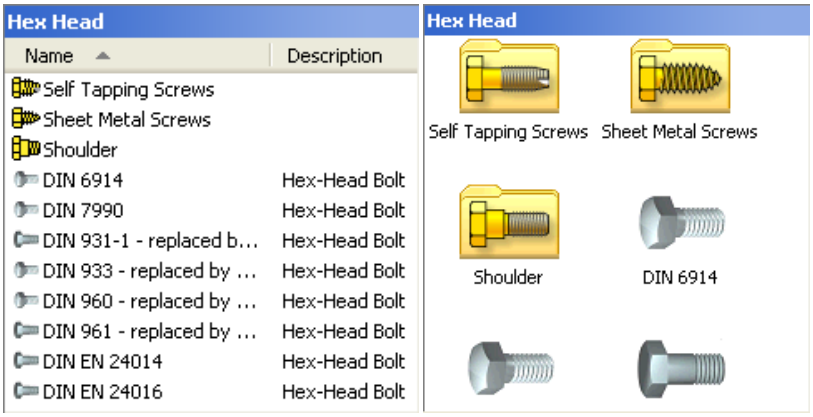


Figure 1.21

Library Panel Options:

The library Panel offers several options available on the toolbar.

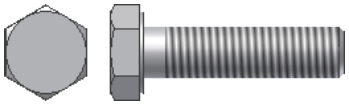
	Back	Back displays the previous viewed page.
	Forward	Forward displays the next page
	Parent	Parent displays the parent (folder) page which is the page one level up in the structure of the current catalogue.
	Search	Search is used to define a search property and search for corresponding standard parts across all registered catalogues.
	Favourites	Favourites displays a list of favourite standard parts or catalogue folders. You can add or remove items in the Favourites list.
	Standards Filter	Standards filter controls which standards are displayed in the Content Library browser, and used in Library search
	History	History displays a list of parts recently copied or moved using i-drop.
	Auto Drop	controls the default placement method for Content Centre components. Switch on to use Auto Drop as the default method for placing Content Centre components in assemblies. Switch off to use Content Centre placement
	Refresh	Restores the contents centre.
	Text, Icon/List	Text switches to the text view mode for displaying the Library items. Icon/List switches the icon view mode for displaying the Library items.

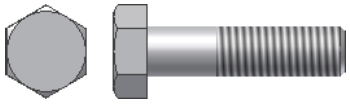


	Tree View	Switches on or off the Tree View pane.
	Table View	Switches On/Off the Table View panel.

Examples of Fastenings:

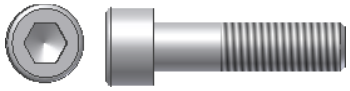
Screws & Threaded Bolts:

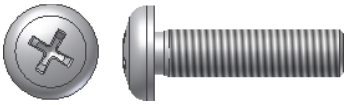

Hexagonal Head Machine Screw

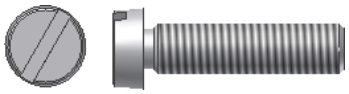

Hexagonal Head Bolt

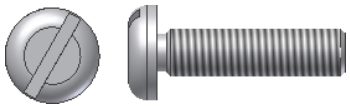

Countersunk-Slotted Head Machine Screw


Countersunk-Phillip Head Machine Screw


Socket Head Cap Screw


Round Head Machine Screw


Fillister Head Machine Screw


Pan Head Machine Screw


Flat Point Set Screw

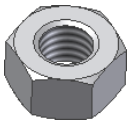

Dog Point Set Screw

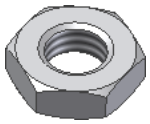

Cup Point Set Screw

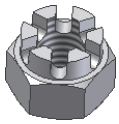

Cone Point Set Screw

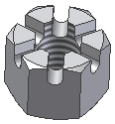

Stud

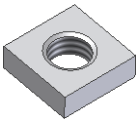
Nuts:

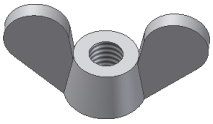

Hexagonal

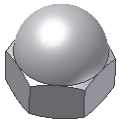

Locknut

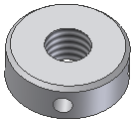

Castle


Slotted


Square

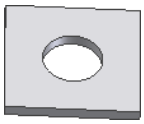

Wing

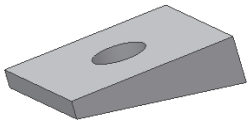

Acorn


Round

Washers:


Plain

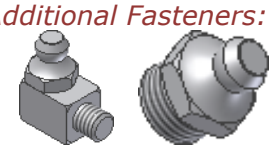

Square

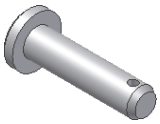

Taper

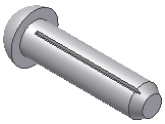

Spring

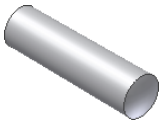

External Tooth

Additional Fasteners:

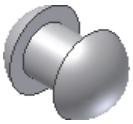

Lubrication Fittings

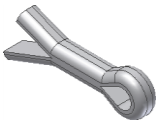

Clevis Pin

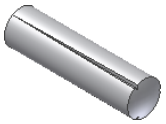

Grooved Stud Drive


Taper Pin


Plug


Rivet


Cotter Pin


Solid Pin

Inserting Parts into Assembly:

Parts can be placed in an assembly using manual or automatic methods. Turning the Auto Drop button OFF allow parts to be placed manually while turning the feature ON allows the part to be inserted automatically. Parts placed automatically are constrained to a surface and determine other locations on the same plane/surface.

Manual Method:

To manually place a part the Auto Drop must be turned OFF. On clicking the Place from Content Centre button in the Assemble tab the Place from Content Centre dialog box is first displayed. The type of fastener or part is expanded in the Browser until the required fasteners are displayed in the viewing column. On selecting the correct fastener from those displayed, a dialog box similar to that shown in Figure 1.22 is displayed. The required diameter and thread is then selected and when the Apply button is clicked, the drawing is returned where the fastening can be placed and then constrained accordingly.

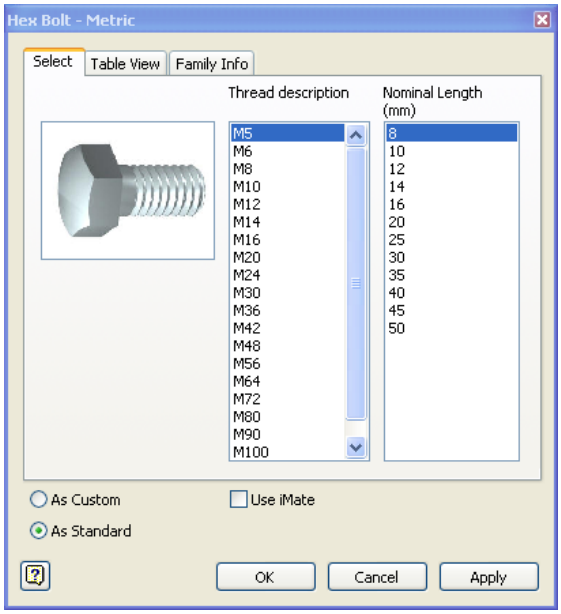


Figure 1.22

Automatic Method:

Auto Drop can be used to place bolts washers, clevis pins, bearings, and circlips. To automatically place a fastening the Auto Drop must be turned ON. On clicking the Place from Content Centre button in the Assemble tab the Place from Content Centre dialog box is first displayed. The type of fastener is expanded in the Browser until the required fasteners are displayed in the viewing column. On selecting the correct fastener from those displayed (similar to Figure 1.22), the placement icon appears in the graphics area; the operator is then required to select the cylinder for the fastener to mate and the surface for the head to sit upon.

The Auto Drop toolbar is then displayed where the length of the fastener can be increased or decreased to suit.

	Multiple Insert	Inserts and gives a count of the number of fasteners into multiple patterned holes.
	Change Size	Select to insert and mate the member. Opens the Content Center Family dialog box where you can select a member of the needed size. Click this command to stop AutoDrop (the same as Done).
	Bolted Connection	Opens the Bolted Connection Generator using the selected bolt and the placement (either By hole or Concentric, depending on the target geometry). Click this command to stop AutoDrop (the same as Done). This command is available in AutoDrop only for specific bolt categories that the Bolted Connection Component Generator supports. Also, either select or preselect an appropriate target.
	Apply	The Apply command finishes the current insertion and Auto Drop continues with the given family. You can insert several sizes during one command. The default member is always the last one dropped.
	Finish	Used to finish the current insertion and stop AutoDrop

Auto Drop Tool

Auto Drop Toolbar

Drag to Increase/Decrease Length

Auto Drop cursor and tooltips

	Question	Either no target geometry is preselected or selected, or the preview size is not updated yet.
	Check	Target geometry is preselected or selected and the updated member fits the target geometry.
	Cross	Target geometry is preselected or selected and the updated member fails to fit the target geometry.

Bolted Connection:

The Bolted Connection opens the Bolted Connection Components Generator, a powerful and useful design tool which allows the selection and placement of a variety of fasteners and associated hardware (including holes) at a single or multiple locations. A higher learning feature of the generator allows the design calculation to select an appropriate bolted connection after specifying the required working load. The strength calculation performs a check of bolted connection (for example, pressure in the thread and bolt stress during joint tightening and operation).

The Bolted Connection Components Generator is displayed by selecting the Bolted Connection button  as shown in Figure 1.23. The Calculation tab calculates bolted connection data based on selections in the Design tab and include data such as loads, material, joint properties etc. The Fatigue Calculation includes loads and design parameters etc.

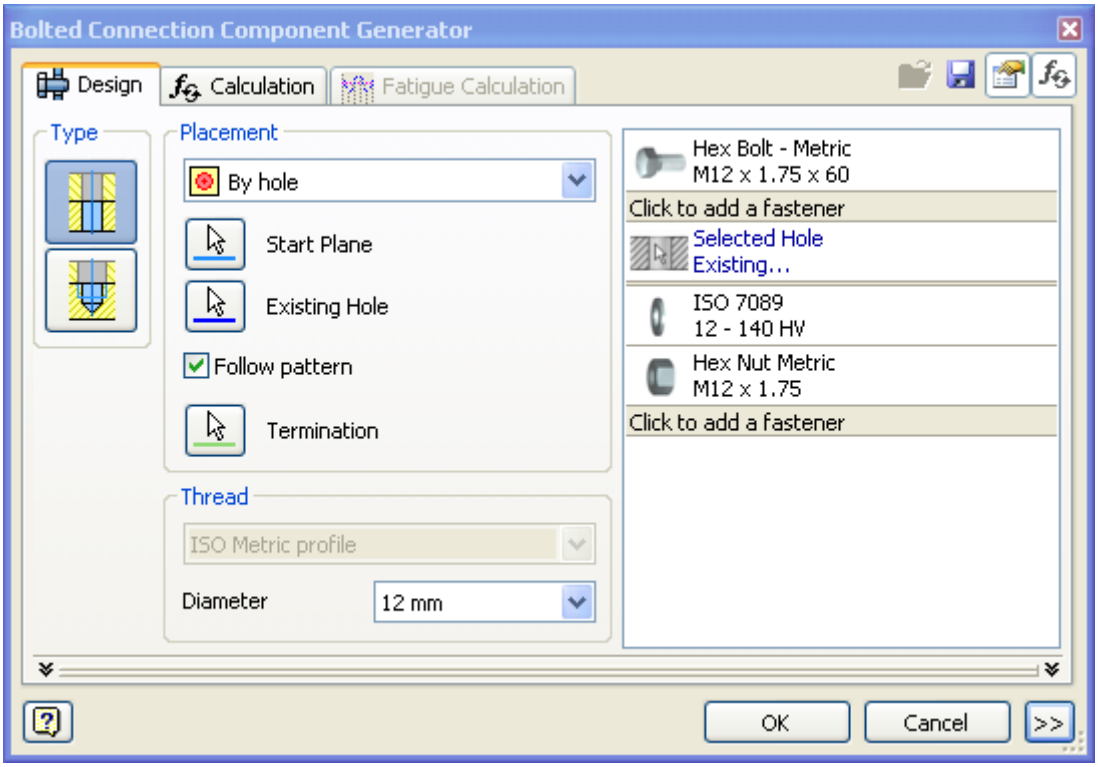


Figure 1.23

Type:

The Type option allows holes to be created from drilled through  or blind  holes. When the Blind connection option is selected, the termination is defined as the start plane of the blind hole.

Placement:

The placement options define the placement of the bolted connection with respect to position (points, concentric, linear, by hole), start and termination planes. Activating the Follow Pattern box allows multiple fastenings to be placed.

Fastener Types:

The individual fastening components are selected using the right side area of the dialog box by selecting **Click to add a fastener**. Selected fasteners can be edited  or deleted  by clicking on the respective button.

Thread:

Allows a specific thread specification to be selected.

Tutorial Exercise 1-4:

Open the assembly files called "Shaft Assembly" in the Topic 1 folder and add a M12 Hexagonal head bolt with nut and washer to the rectangular flange, and a M10x1x20 Socket Head Cap Screw to the circular flange in their respective holes.

Procedure:

Place the Hexagonal Head Bolt.

1. Open the drawing file called **Shaft Assembly** in the Topic 1 folder in your work area.
2. Expand the Place button in the Components panel on the Assembly and select **Place From Content Centre** .
3. In the left column of the dialog box expand **Fasteners** then **Bolts** and **then** click on **Hex Head** to display the Hexagonal Head bolts in the right column.
4. Locate and select double left click on the bolt called **Hex Bolt - Metric**. *The dialog box disappears*.
5. Select a hole on the rectangular flange of the part called SHAFT TRANSITION.
6. Select the face of the SHAFT TRANSITION. *The Auto Drop dialog is displayed.*
7. Check the Insert Multiple option is selected. *A count of 16 should be indicated*.
8. Select the **Bolted Connection** button.
9. On the prompt in the Bolted Connection Component Generator dialog box to select a termination, select the underside of the Shaft Base flange.
10. Check the **Follow Pattern** is selected. *A tick should appear in the box, click the box to make it active.*
11. Click on the bottom **Click to add a fastener**.
12. Select the washer **ISO 7089** from the list displayed.
13. Click on the bottom **Click to add a fastener**.
14. Scroll down and select **Hex Nut Metric**.
15. Click **Apply** and then **OK**.

Place the Socket Head Cap Screw.

16. Click the **Place From Content Centre** button.
17. In the left column of the dialog box expand **Fasteners** then **Bolts** and **then** click on **Socket Head** to display the Socket Head bolts in the right column.
18. Locate and select double left click on the bolt called **Forged Socket Head Cap Screw - Metric**. *The dialog box disappears*.
19. Select a hole on the circular flange of the part called SHAFT ROUND.
20. Select the face of the SHAFT TRANSITION. *The Auto Drop dialog is displayed.*
21. Check the Insert Multiple option is selected. *A count of 16 should be indicated*.
22. Select the **Bolted Connection** button.
23. Check the **Follow Pattern** is selected. *A tick should appear in the box, click the box to make it active.*
24. Click the **Blind Connection Type** box and select the hole in the circular flange of the Shaft Transition.
25. Select the plane (joint) between the Shaft Transition and Shaft Round.
26. Click **Apply** and then **OK**.

Skill Practice Exercises

Skill Practice Exercise 1-1

Assemble the components provided in the Monorail folder as shown in Figure 1.24 with the grooves in the Wheel mating with the groove and shoulder of the Axel.

Save the drawing in your work folder as MEM30004-SP-101.

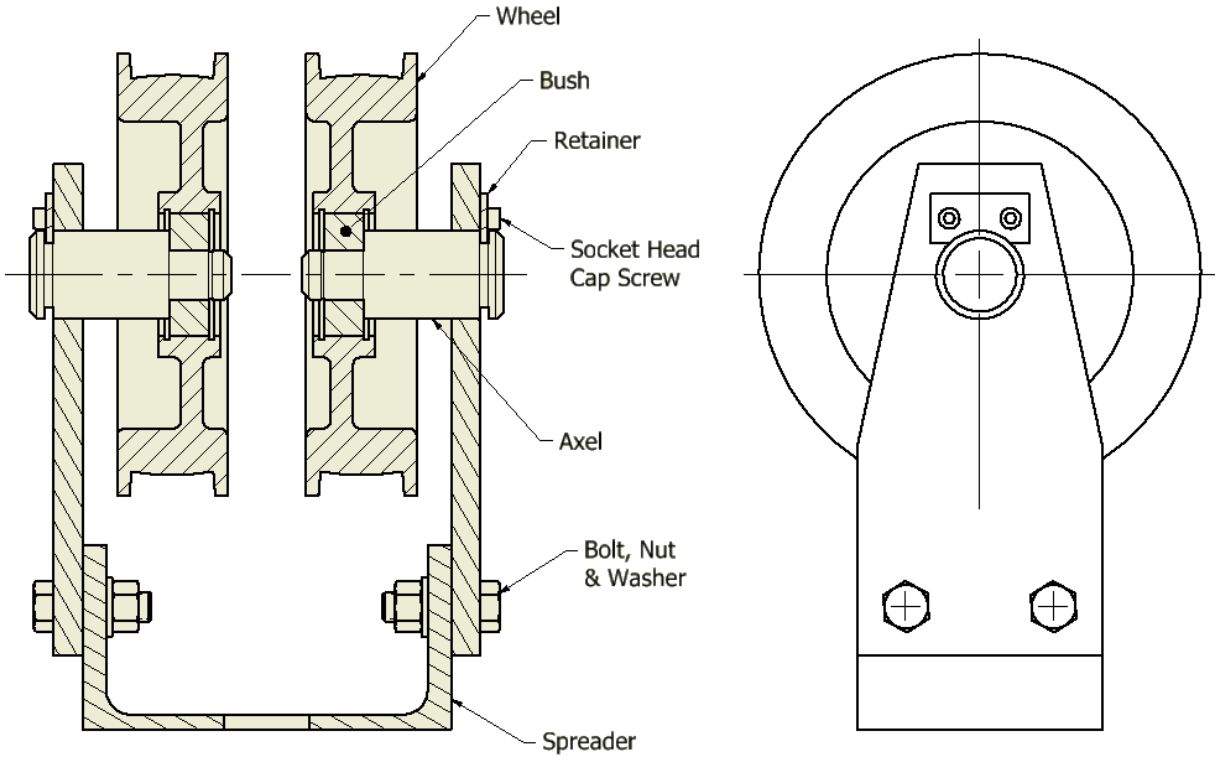


Figure 1.24

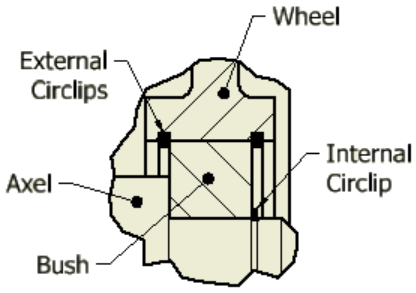
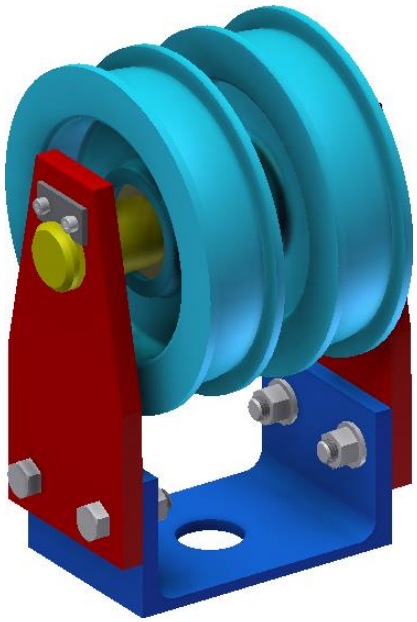


Figure 1.25



Add metric bolts, nuts and washers to secure the Spreader to the Side Plates and Socket Head Cap Screws to secure the Retainer to the Side Plates.

Internal circlips are to be provided in the grooves to locate the Bush while an external circlip is used to prevent the Axel from moving as shown in Figure 1.25.

Skill Practice Exercise 1-2

Assemble the components provided in the Submersible Pump folder as shown in Figure 1.26. Place the fastenings as indicated.

Save the drawing in your work folder as MEM30004-SP-102.

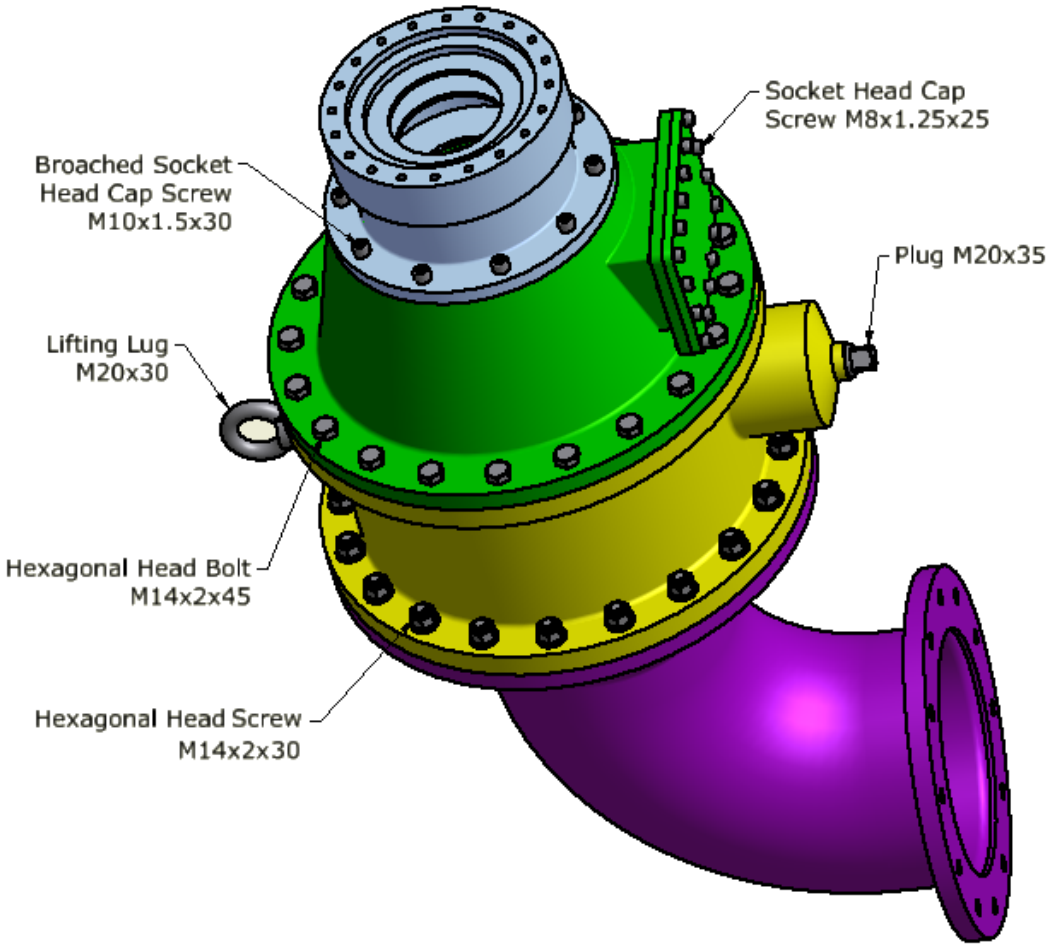


Figure 1.26