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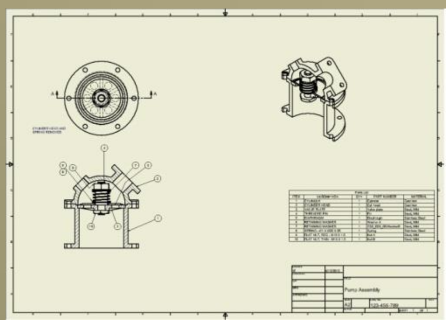


AutoCAD Inventor Basic

Use CAD to create and display 3D models.

Table of Contents

Topic 1 – Inventor Basics:

Required Skills:

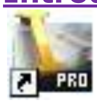
On completion of the session, the participants will be able to:

- Enter the Autodesk Inventor program and navigate the core functions.
- Modify the grid and sketch settings.
- Use sketching tools to create a profile of an object.
- Add constraints to a profile in a sketch.
- Dimension a sketch.
- Zoom and pan the sketch.
- Produce profiles of various shapes using sketching, constraint and dimensioning techniques.

Required Knowledge:

- Enter the Autodesk Inventor program and navigate the core functions.
- Modify the grid and sketch settings.

Introduction to Autodesk Inventor:



Autodesk's Inventor is a parametric solids modelling product used by Engineers and Draftspersons to create and detail components. Those who have used Mechanical Desktop will find Inventor strangely familiar as many of the buttons and procedures from Desktop appear in Inventor, but they are packed into an entirely new, non-AutoCAD user interface. Although Inventor doesn't run in AutoCAD, it can be used to create AutoCAD drawing files, if desired.

The Inventor program is entered by clicking on the AutoCAD Inventor  icon on the Desktop panel. After loading the screen is shown similar to that in Figure 1.1 below.

Figure 1.1

From within the AutoDesk Inventor program, standard Engineering parts, Sheetmetal components, Weldment and Assemble drawings can be created together with Animated Presentations.

Projects:

Inventor is capable of creating complex 3D assemblies. The assemblies are made by combining various part files that are either custom to that assembly, or that are shared across many different assemblies. This collection of files must be organized in such a way to make it easy for the designer to use. Inventor does this by creating what's called *Projects*.

A Project organizes the data by creating a *Workspace* which is a series of directories (folders) that all the custom parts and drawings are located for a given assembly. Some Workspace directories may be stored locally on the designer's computer, while others may be stored on a server allowing others to work on them at the same time.

Often there are many times when part files like fasteners or bearings tend to be reused across many assemblies; these part files are stored in folders that are referred to as *Libraries*. Since Library parts are shared across many different Projects, they are not allowed to be edited.

Inventor supports two types of project file, Single User Projects and Vault Projects. Vault projects are used to collaborate on projects with multiple designers. Common files are stored in a vault and are never accessed directly. Each designer has a personal project that defines where the files are copied for viewing and editing. The vault also maintains version history of files as well as additional attributes.

On selecting the Projects button the Projects dialog box is displayed as shown in Figure 1.2 with the options to create a New project or look for an existing project using the Browse button.

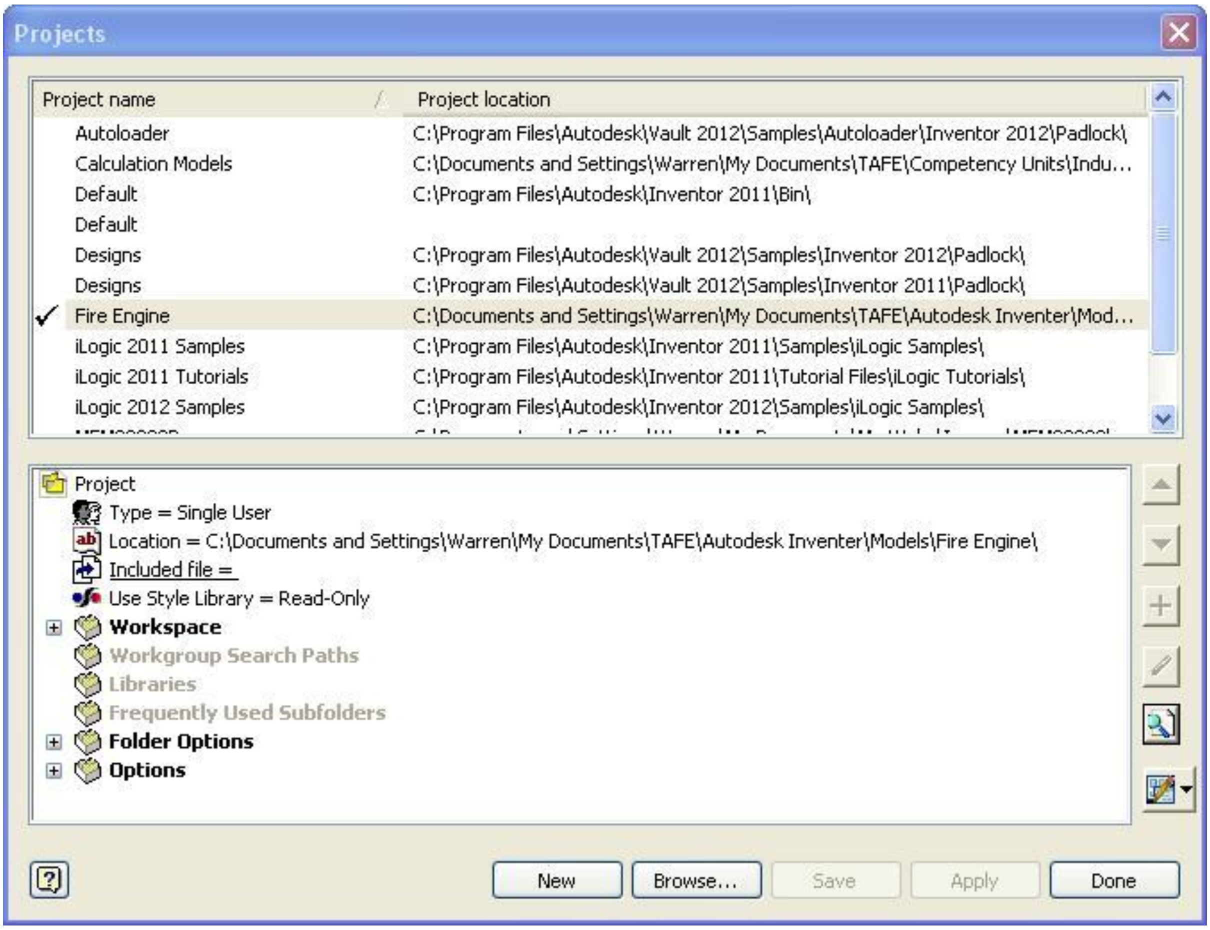


Figure 1.2

Project files are identified by the .IPJ file extension.

New:
Selecting the New button displays a series of Inventor Projects dialog boxes that are used to setup the project file.

- Step 1 - Figure 1.3: Sets either the Vault or Single User. ALL of your projects will be Single User.
- Step 2 - Figure 1.4: Sets the location of the Workplace folder and Project name.
- Step 3 - Figure 1.5: Sets the location of the libraries to be used.

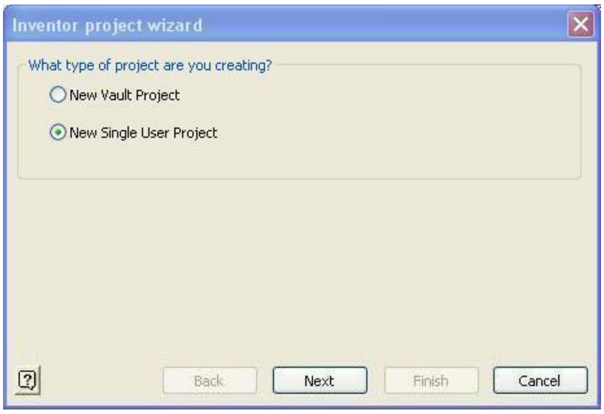


Figure 1.3

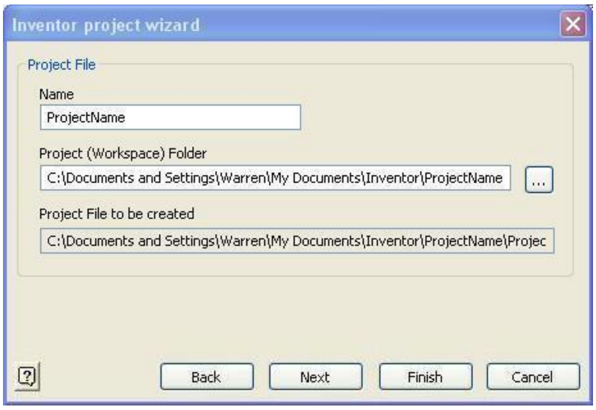


Figure 1.4

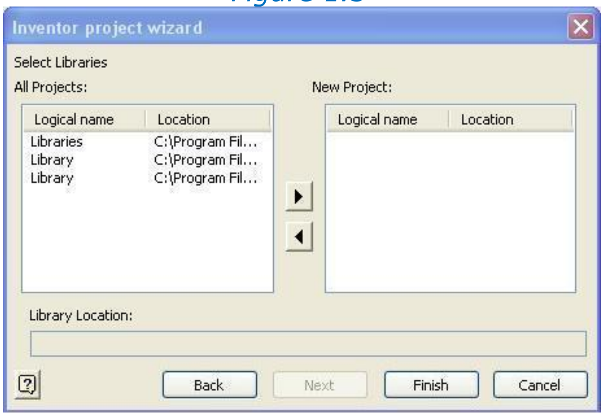


Figure 1.5

Browse:
The browse button is used to look for an existing project file created in drives and folders using the Choose Project File dialog box shown in Figure 1.6.

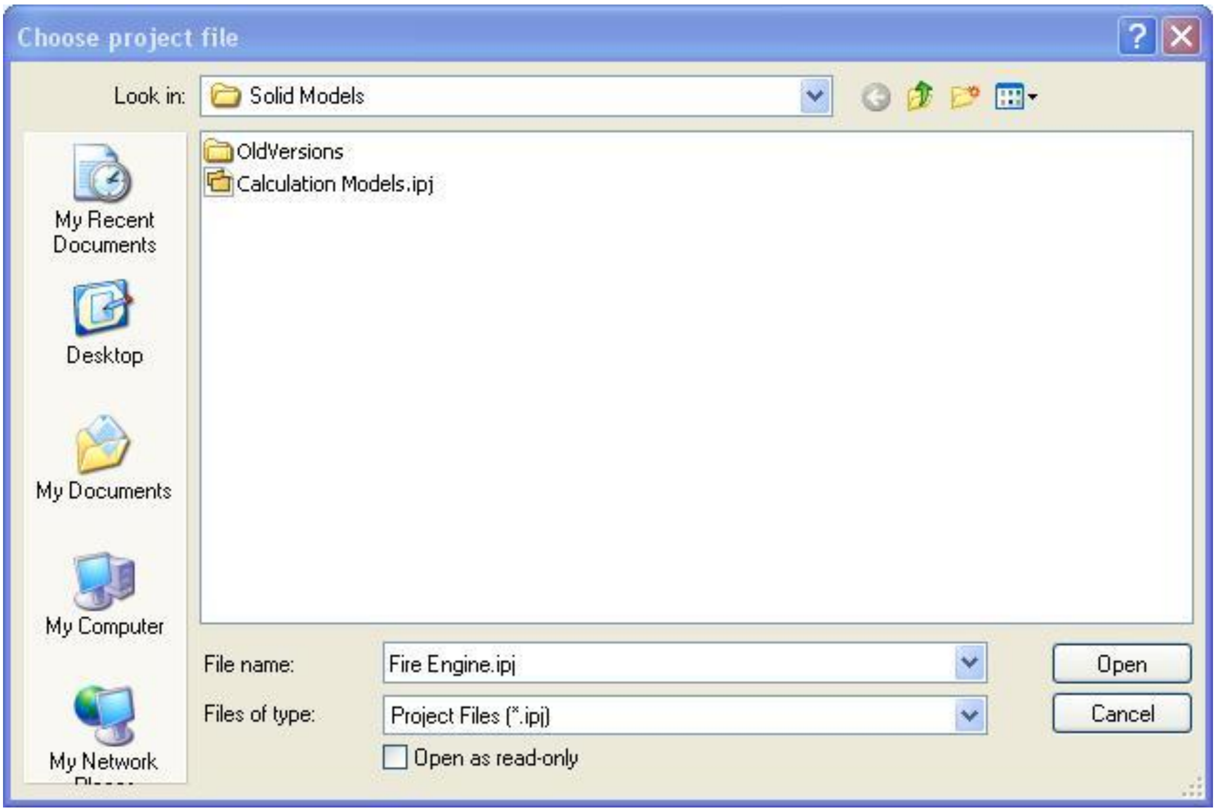


Figure 1.6

Once a project has been located, selected and the Open box checked, the operator is returned to the Projects dialog box Figure 1.2.

The project is made current by one of two methods:

- Double left-clicking on the project name.
- Click once on the project name then click Apply and then Done.

New:

The New button allows new files to be created using the New File dialog box in Figure 1.7. The different file types include Engineering Parts, Assemblies and Sub-Assemblies, Detail, Assembly and Exploded Isometric Drawings, Weldment Assemblies, and Animated Presentations. The dialog box contains 4 tabs containing various preset templates and menus.

Figure 1.7

Standard Parts:

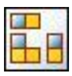
A variety of engineering components can be created using the Standard button or sheetmetal shapes using the Sheetmetal button. A sheet metal file is an extension of the part modelling environment and contains specific sheet metal commands to support the creation of sheet metal parts. Part files are identified by the .IPT file. Extension.

It is important to remember that each part file contains only one part; a separate part file for each part in the assembly.

Assemblies:

The Assemblies button is used to create engineering assemblies and sub-assemblies. A weldment assembly file is an extension of the assembly environment and contains specific weldment commands to support the creation of weldments. Assemblies are identified by the .IAM file. Extension.

Drawings:



Creates an Inventor drawing file for the preparation of detail and assembly drawings. Drawings are identified by the .IDW file. Extension.

Presentations:

Standard.ipn creates an animated presentation. An assembly presentation contains specific presentation commands. Presentations are used to develop exploded views, animations, and other stylized views of an assembly to help document the design. Presentations are identified by the IPN file. Extension.

Open:

The open button is used to open an existing inventor file though the Open dialog box. Files can be located using the Look in box to locate drives and folders with the results being displayed in the Preview area.

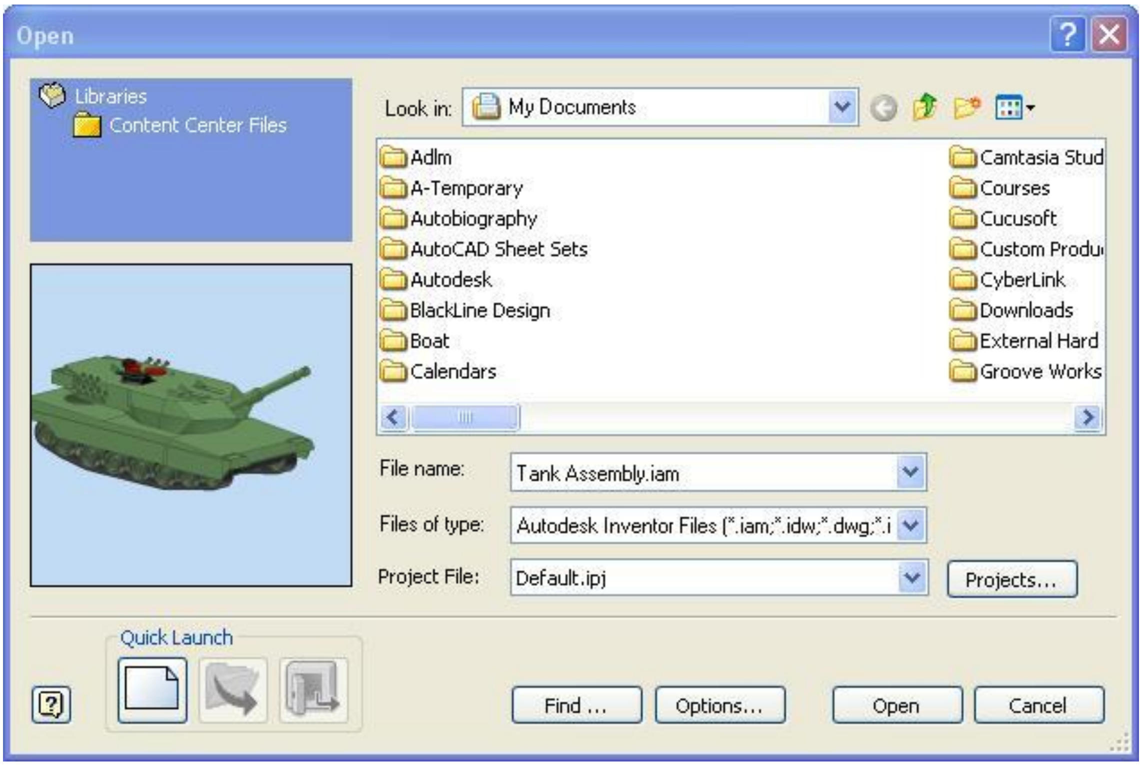
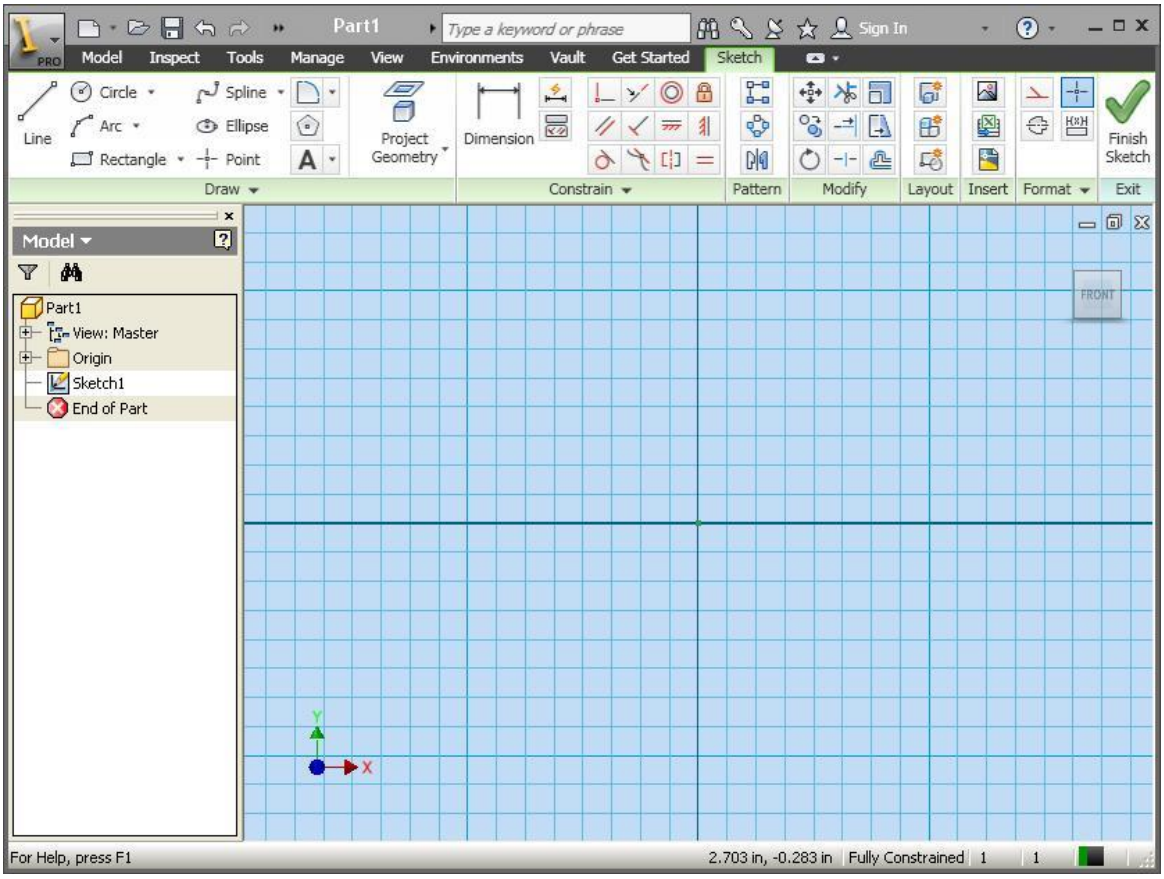


Figure 1.8

Drawing Interface:

On entering the New, the following screen is displayed and contains the ribbon menu, browser, graphics screen and context menus.



Sketches must be finished with closed loop profiles before they can be extrude. The Sketch mode is exited by clicking the Finish Sketch button at the right end of the ribbon.

Ribbon Menu:

The ribbon is organized into the Model, Inspect,Tools, Manage, View, Environments, Vault, Get Started and Sketch tabs. Each tab contains a series of panels appropriate for the tasks.

Part, assembly, and drawing files open at the same time in which case, the ribbon changes to accommodate only the environment of the file in the active window.

To find the locations of commands on the ribbon, use the search field at the top of the application menu. In search results, click a command to see a tooltip and a path to the location on the ribbon, and to launch the command.

All commands are entered using the Buttons, in the Panels on the Ribbon Menus or by right clicking the pointing device and select one of the options displayed in the menu box. The only time the keyboard is used is to enter notes and dimensions.

Figure 1.9 shows the Sketch Panel while Figure 1.10 shows the Model Panel and have been divided into two parts for clarity.

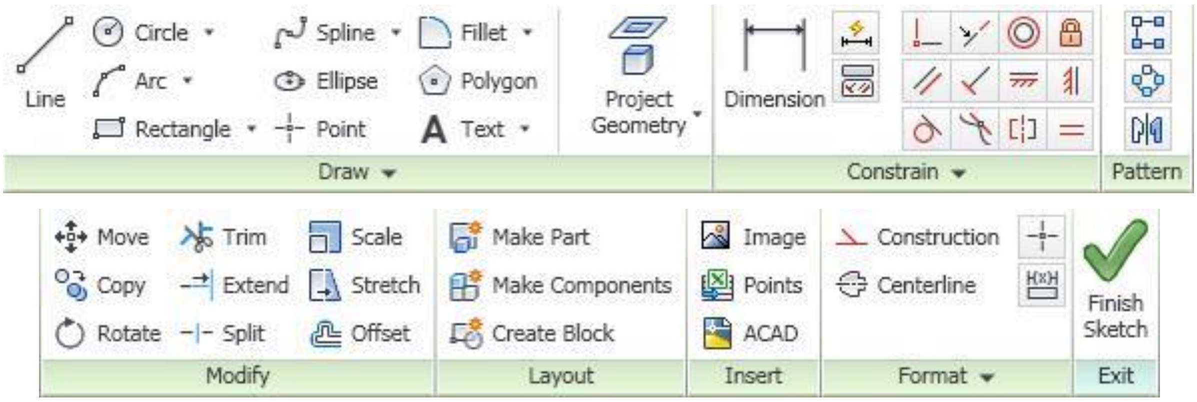


Figure 1.9 Sketch Panel

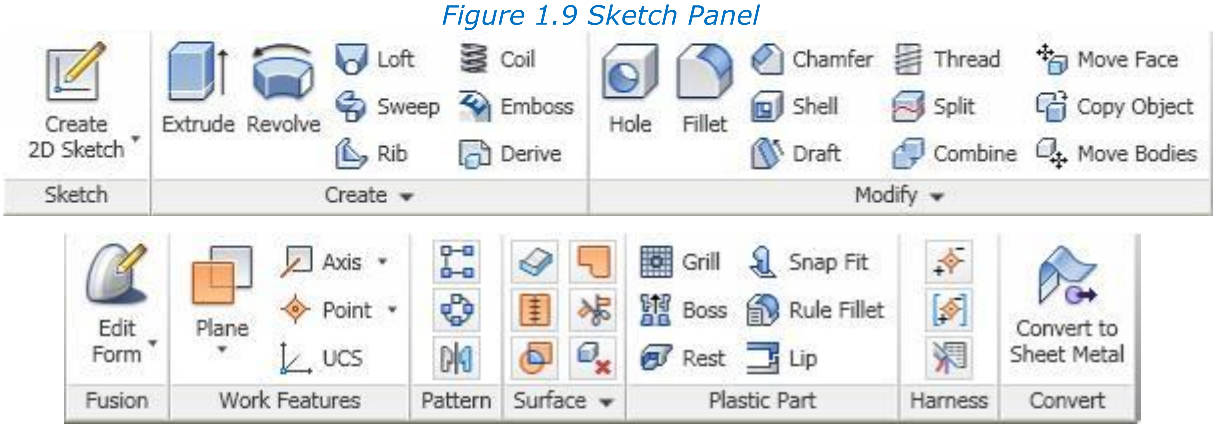


Figure 1.10 Model Panel

When a new part is created, the graphics screen automatically displays the Sketch mode with the Sketch Panel displayed on the ribbon.

Sketches must be finished with closed loop profiles before they can be extrude. The Sketch mode is exited by clicking the Finish Sketch button at the right end of the ribbon.

Browser:

The browser shows the hierarchical structure of parts, assemblies, and drawings. The browser is unique to each environment, and always displays the information for the active file.

Turn visibility of the browser on and off with the Browser check box located on the ribbon:

View tab ➤ Arrow Windows panel ➤ User Interface drop-down list ➤ Select Browser.

The Browser can be docked on the left or right sides of the graphic area.

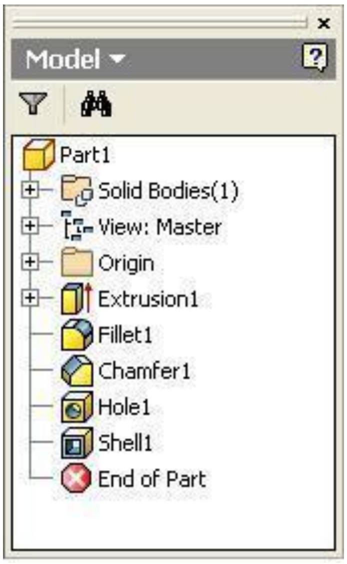


Figure 1.11 Browser

Graphics Screen:

The graphics window displays by default whenever a file is open. If more than one file is open, each file appears in its own graphics window. The window containing the file you are working on is called the active window.

Click the cursor inside any displayed graphics window to make it the active window. Alternatively, to activate a window to work on, click the file in the Switch drop-down list. View tab ➤ on the ribbon, Windows panel, on the Switch drop-down list, select the file. When more than nine files are open, the More Windows option appears at the bottom of the Switch drop-down list. Click More Windows to open the Select Window dialog box, where you can activate another file.

Multiple graphics screens can be arranged in a variety of ways:

- Overlap or cascade windows. Select View tab ➤ Windows panel ➤ Cascade.
- Arrange windows side-by-side, horizontally or vertically. Click View tab ➤ Windows panel ➤ Tile. The number of open files determines the horizontal or vertical ordering of the display.
- View the active part model or assembly in a separate window, but with a different zoom ratio or view orientation. Select View tab ➤ Windows panel ➤ New. The new window displays the currently open file. You can change the zoom ratio or view orientation. To view both windows simultaneously, use Cascade, or Tile.

When working in Inventor:

- When a command requires further steps in the process, a message is displayed at the far bottom left indicating the next step. If no command is active, the message Ready, or, Press F1 for more help displays on the far bottom left.
- On 32-bit operating systems, the capacity meter displays on the far right, and provides information about the current state of memory consumption. The capacity meter is not available on 64-bit operating systems.
- If more than one document is open, the commands Cascade, Arrange, and Open Documents, display at the bottom of the graphics window.

Sketches must be finished with closed loop profiles before they can be extrude. The Sketch mode is exited by clicking the Finish Sketch button at the right end of the ribbon.

Context Menu:
In the Autodesk Inventor software, context menus provide a list of common functions that are active for the current selection.

The Context menus are accessed by right-clicking in the graphics screen. The menus are available for a specific selection, and when there is no selection, each environment provides a context menu.

Some of the selections on the context menus are not available at all times.

Figure 1.12 show the Context menu displayed on the graphics screen while working in the Sketch mode while Figure 1.13 shows a Context menu displayed while working in the model mode.

Figure 1.12 2D Sketch Mode

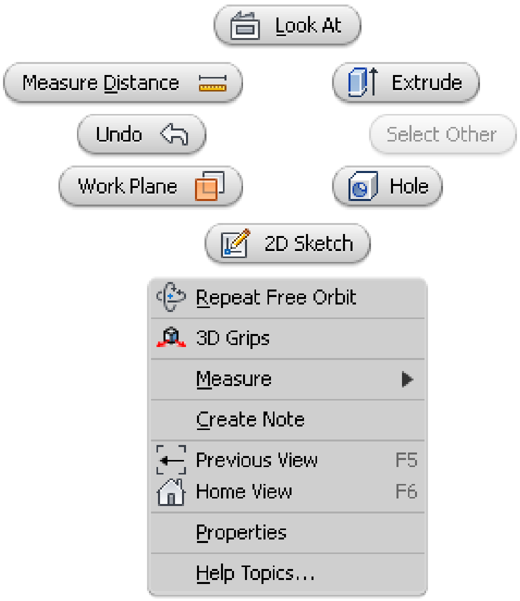


Figure 1.13 Model Mode

Tutorial Exercise 1-1:

Enter the Inventor Program create a project in your Tutorial Exercises work area called Tutorials and then create a new part file using the metric standard template.

Procedure:

1. Enter the Inventor program by clicking on the AutoCAD Inventor icon.

Create a Project.

2. Select the Projects button.
3. In the dialog box click **New**.
4. Click the **New single user project** and then click **Next**.
5. In the Project Name box type **Tutorials**.

6. In the Project (workshop) folder, click on the **Browse for project location** button.
7. Expand the drives and folders to locate your work area.
8. Create a new folder in the work area called Tutorial Exercises by clicking the **Make new folder** button, type **Tutorial Exercises** and then click **OK**.
9. Click **Finish**.

Make the Project current.

10. Select Tutorials and click **Apply**.
11. Click **Done**.

Create a new part file.

12. Select the New icon.
13. Click the **Metric** tab, then **Standard (mm).ipt** and then click **OK**.
14. Click on the Finish Sketch button on the Ribbon Menu.
15. Save the work in your work area as **TUT 1-1** by clicking the Save icon.

Adjusting the Settings:

The settings and options are controlled within the Inventor program using the Document Settings and Application Options buttons in the Tools tab on the Ribbon.

Application Options:

Application Options controls the look and feel of the program. Selecting the Application Options button displays the Application Options dialog box which contains 14 tabs on two lines; they are Content Centre, Drawing, Sketch, Part, iFeature, and Notebook on the top line and General, Save, File, Colours, Display, Hardware, Prompt and Assembly on the bottom line. The main two tabs when learning the program are the Colours and Sketch.

Colours Tab:

The Colours tab, Figure 1.14, allows the operator to adjust the background colour to suit their viewing comfort by the use of one of the nine available colour schemes. The background can be set to 1 colour, gradient or a background image.

The gradient and image backgrounds can slow down some operations such as Orbit because they use additional graphics memory. A solid, single colour background has the least impact on the computer's system resources.

Sketch Tab:

The Sketch tab, Figure 1.15, sets the preferences for sketching. The main area that you may need to constantly change is the Display which controls the visibility of the grid and axes. The various items are turned ON and OFF by clicking the feature to add or remove the tick; the selected item is visible when the tick is in the box.

There is a case for turning the grid and axes OFF, however, the grid and axes are critical for placing the part/component on the datum point. Parts created off the datum point can be harder to place in an assembly.

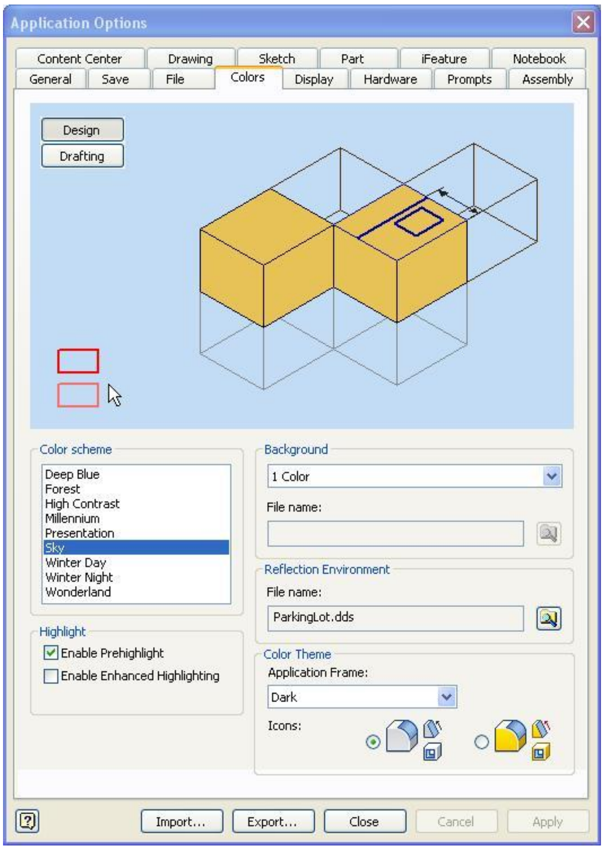


Figure 1.14

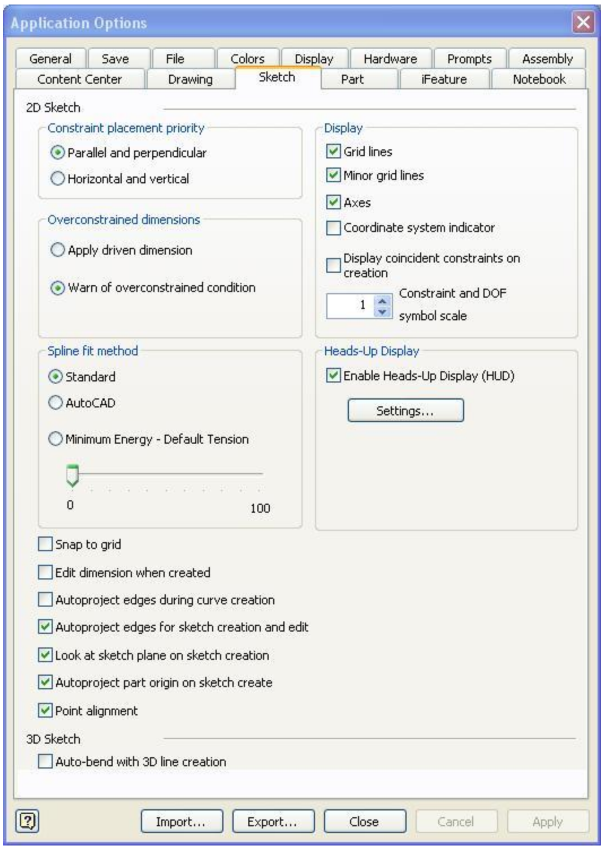


Figure 1.15

Document Settings:

The Document Settings dialog box controls the settings in individual files. Various tabs control settings for the active document. The tabs available are Standard, Units, Sketch. Modelling, Bill of Materials, and Default Tolerance. The main tabs to focus on when learning the program are the Units, Figure 1.16, and Sketch, Figure 1.17.

Units Tab:

The Units tab, Figure 1.16, controls the physical units of the file. Length can be changed from Metric to Imperial, and measured in millimetres or metres, feet or inches. Angles can be set to operate as degrees or radians. The precision of both length and angles can be set to the required number of decimal points while the dimensions can be displayed as values, name, an expression or tolerance.

Sketch Tab:

The Sketch tab, Figure 1.17, adjusts the grid settings to make it easier to draw the sketch by setting the units, Sketch, Modelling and Tolerance settings. The snap spacing sets the spacing between snap points to help with precision when sketching in the active part or drawing; the settings for the two axes can have different values.

The grid display sets the spacing of the lines in the grid display for the active part or drawing. The grid consists of two types of lines, major shown as darker lines and minor which are more numerous and lighter in colour.

The snaps per minor setting sets the distance between minor grid lines relative to the specified snap distance. For example, if the X snap distance is set at 5, and specify 2 snaps per minor, the minor lines are spaced 10 apart. The major every minor lines setting sets the number of minor lines to appear between major lines.

The auto-bend radius sets the default radius for corner bends that are automatically placed on 3D lines as they are drawn.

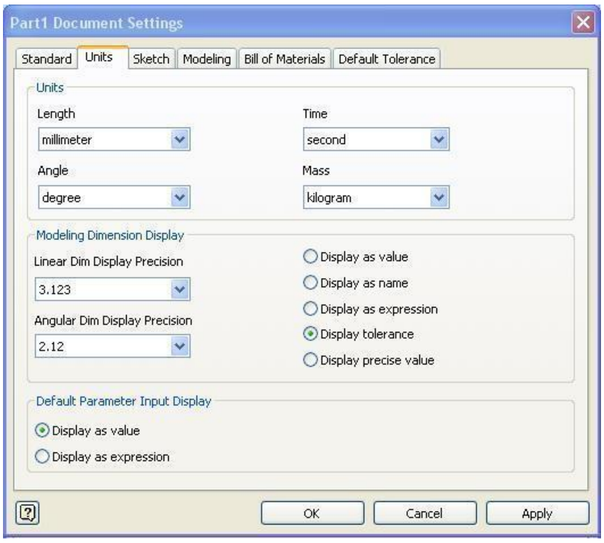


Figure 1.16

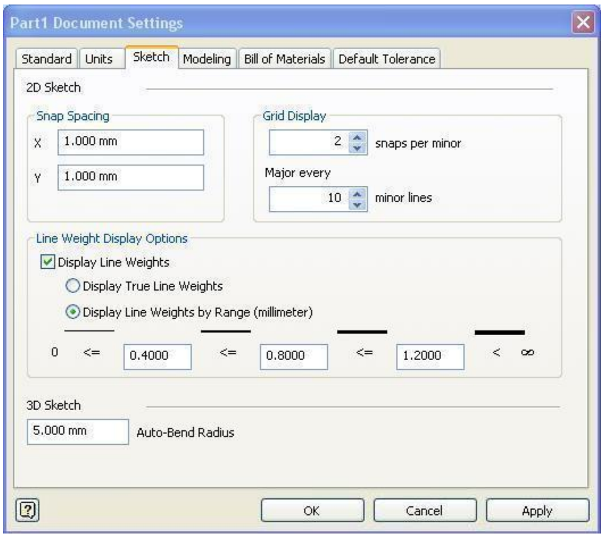


Figure 1.17

Tutorial Exercise 1-2:

Change the following settings: Linear precision to 2 decimal places; Angular precision to 0.1; X/Y snap spacing to 2 mm; Snaps per minor line to 2; Major line every 5 minor lines. Change the background colour to Sky. Display the grid, axes, and minor lines and make sure the snap to grid is selected.

Procedure:

Change the Document Settings.

1. Click the **Tools** tab on the Ribbon menu to display the Options panel.
2. Click the **Document Settings** tab and then the **Units** tab.
3. Expand the "Linear Dim Display Precision" box and select **2.12**.
4. Expand the "Angular Dim Display Precision" and select **0.1**.
5. Click the **Sketch** tab.
6. Type **2** in the X & Y Snap Spacing boxes.
7. Change the Snaps per Minor in the Grid display to **2**.
8. Change the Minor Lines in the Grid display to **5**.
9. Click **OK** to exit the dialog box.

Change the Application Options.

10. Click the **Application Settings** tab.
11. Click the **Colours** tab and then click on **Sky**.
12. Expand the Background box and select **1 Colour**.
13. Click the **Sketch** tab and check the boxes besides "Grid", "Axes", "Minor Lines" and "Snap to Grid" to place a tick in the box to activate the option.
14. Click **Apply** and then **Close** to exit the dialog box.
15. Save the file by clicking the Save icon.

Navigation of the Drawing:

The methods of navigation consist of three methods for viewing the object or assembly from different directions; they are the View Cube, Steering Wheel and the Navigation Bar. An operator may use all three methods but most will become comfortable in using only one; the three methods provide basically the same result.

The visibility of the navigation tools can be turned ON and OFF using the Ribbon.

Ribbon Menu ➤ View tab ➤ User Interface ➤ *Click on the View Cube or Navigation Bar as required*.

View Cube:

The View Cube allows the operator to change between standard planes and isometric views. The Front, Rear, Top, Bottom, Left and Right views look directly onto the view and are set by clicking on one of the sides of the View Cube as shown in Figure 1.18. Isometric views are viewed by clicking on one of the six corners of the box as shown in Figure 1.19, while two sides can be viewed by clicking on an edge, Figure 1.20.

The left and right arrows shown in Figure 1.18 allow the view to be rotated 90°.

Clicking the Home  symbol returns the

Figure 1.18

Figure 1.19

Figure 1.20

Steering Wheel and Navigation Bar:

The Steering Wheel Figure 1.21, and Navigation Bar Figure 1.22, combines many of the common navigation tools into a single interface and contains the Zoom, Orbit, Pan, Rewind, Centre, Look, Up/Down and Walk wedges. All tools work by moving the cursor over the wedge and left-clicking.

Figure 1.21

Figure 1.22

Zoom:

The Zoom tool works by dragging the wheel into the position that the part is to be zoomed about; when the left button is held down on the Zoom wedge, the pivot symbol is displayed and by moving the mouse about the screen, the object zooms In or OUT.

Orbit:

Orbit allows the part to be viewed from any angle; when the left mouse button is held

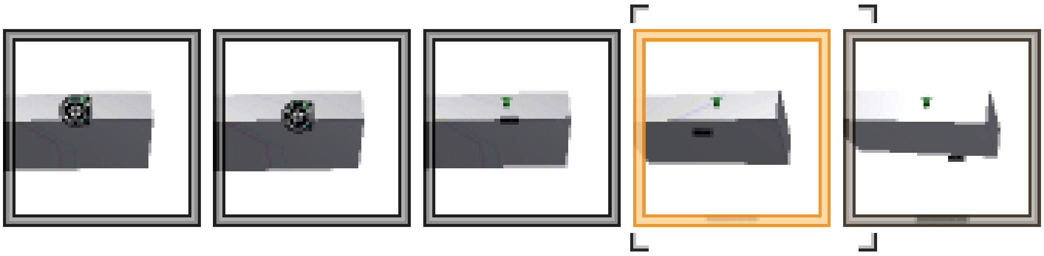
down over the Orbit wedge the object can be pivoted about the pivot symbol. The pivot point is calculated based on the centre of the extents of the selected objects.

Pan:

Changes the cursor to a four-directional arrow used to drag the view in the graphics window. The drawing can also be panned by holding down the center wheel (if fitted) on the pointing device and moving the device up, down, left or right

Rewind:

The Rewind function allows the automatically saved previous views to be selected and displayed from the animation history. The images appear as thumbnails in the graphics area; by holding down the left button and moving the cursor over the different images, the view is displayed on the screen.



Centre:

The Centre tool defines the centre of the current view of the part by bragging the cursor over the part. The sphere indicates the point below the cursor within the part is the centre of the view; when the mouse button is released, the model is cantered on the sphere.

Look:

The Look tool rotates the current view vertically and horizontally; when rotating the view, the line of sight rotates about the current eye position.

Up/Down:

Unlike the Pan tool, the UP/Down tool is used to adjust the height of the current viewpoint along the model's Z axis. To adjust the vertical elevation of the current view, drag up or down. As the cursor is dragged, the current elevation and the allowed range of motion is displayed on a graphical element called the Vertical Distance indicator.

The Vertical Distance indicator has two marks that show the highest (Top) and lowest (Bottom) elevation possible for the view. While changing the elevation with the Vertical Distance indicator, the current elevation is shown by the bright orange indicator, while the previous elevation is shown by the dim orange indicator.

Walk:

The Walk tool allows the operator to navigate through a model as if walking through it. Once the Walk tool is started, the Centre Circle icon is displayed near the centre of the view and the cursor changes to display a series of arrows. To walk through the model, drag the cursor in the direction in which it is to be moved.

Face View:

The Face view views the faces of a part from a selected plane. The view is set by left-clicking on the face to be viewed.

2D Sketches:

The starting point of a design in Autodesk Inventor is to create a Sketch using 2D drawing tools (Buttons), which are similar to the AutoCAD command icons, to produce a 2D Sketch on a Sketch Plane, normally the XY Plane. All sketches MUST form a closed loop; that is each entity (lines, arcs etc) must be joined with no gaps or a smaller entity drawn over a larger entity.

There are 2 methods that can be used to create a sketch:

Method 1 – Accurately draw a line by clicking at the start point, moving the cursor in the direction for the line and then typing the distance on the keyboard before pressing the Enter key as shown in Figure 1.23.

Method 2 - create rough sketches to create the designs, and then fine tune them by adding parametric constraints and dimensions.

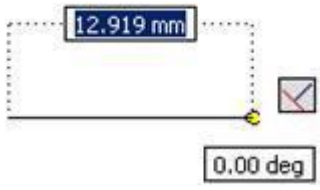


Figure 1.23

When using this method Sketch close to the size and shape of the object, use simple sketches, and then add features to refine the shape. Constrain all geometry that will remain fixed, starting with the smallest elements. The first task is to create the base sketch by drawing some lines. Start at the origin of the grid (in the centre of the part window); as the lines are drawn, watch the lower right of the screen for information about the position of the cursor, and the length of the line being drawn.

The General Dimension and Constraints tools are then used to constrain the sketch which can then be turned into parts or sheetmetal objects. As 2D geometry is created on the Sketch Plane, it not only finds key data such as end-points and mid-points, but the program automatically draws temporary dashed lines when new geometry aligns to existing geometry. It also infers constraints such as parallel and perpendicular. The latter is particularly impressive because, unlike with most systems, Inventor will recognize a perpendicular condition even if the lines are not at exact vertical and horizontal angles.

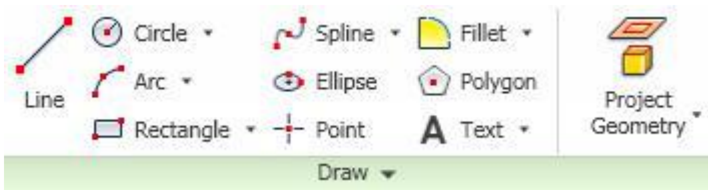
When drawing an element (line, arc etc) on the 2D Sketch Plane, the cross hairs of the cursor will indicate that the element is co-incident with the previously drawn element by indicating a "yellow" dot, when the cursor is positioned at the exact endpoint or midpoint of the element, a "green" dot will be indicated. Clicking the left mouse button will pick a button and/or draw an element from that point; clicking the right button displays a "Pop Menu" that allow several options depending on the stage of the command as shown in Figure 1.12 2D Sketch Mode and Figure 1.13 Model Mode.

The steps in creating a part are:

- Draw the sketch with lines and arcs.
- Add constraints.
- Dimension the shape.
- Finish the sketch.

Draw Buttons:

All entities for creating a sketch are found in the Sketch Panel on the Ribbon and are shown in Figure 1.24. As an element is drawn in the graphics area, it may be automatically constrained into position; the constraint symbol displayed near the cursor indicated the type of constraint being applied – constraints will be covered later in this lesson.



Error! Reference source not found.*Figure 1.24*

Line:

A line is the geometry consisting of a straight curve bounded by two endpoints. The line button on the Sketch buttonbar chains line segments together and creates arcs tangent or perpendicular to existing curves. Segments and arcs are automatically joined by coincident constraints at their endpoints. The lines are created on the graphics screen by picking a point using the Left Mouse Button (LMB).

Circle:

Center-point circle - Creates a circle defined by a center point and a point on its radius. The first click sets center point; the second click specifies the radius. A tangent constraint is applied if the second point is on a line, arc, circle, or ellipse.

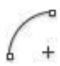
Tangent circle – Creates a circle tangent to three lines on its circumference. The first click sets the tangent point for the circle and the first line. The second click sets the tangent point for the circle and the second line. The third point sets the diameter of the circle (tangent to the third line).

Arc:

Three-point arc - Creates an arc defined by two endpoints and a point on the arc. The first click sets the first endpoint, the second sets the other endpoint (chord length), and the third point sets the arc direction and radius.

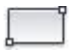
Tangent arc - Creates an arc from the endpoint of an existing curve. The first click (on the endpoint of the curve) sets the tangent endpoint. The second point sets the end of the tangent arc.

Center point arc - Creates an arc defined by its center point and two endpoints. The first click sets the center point, the second specifies the radius and start point, and the third point completes the arc.



Rectangle:

Two Point rectangle A rectangle created by clicking two points (diagonal corners) to set length and width. Two Point rectangles are aligned with the sketch coordinate system.



Three Point rectangle A rectangle whose length and width is created by specifying three points. First and second points set the length and direction of the first side. The third point sets the length of the adjacent side. A three-point rectangle can have any alignment.

Spline & Bridge Curve

Spline - 2D and 3D splines are curves of constantly changing radius that pass through a series of fit points which can be partially or fully constrained.

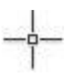
Bridge Curve - A Bridge Curve is used in a 2-D sketch to create a smooth (G2) spline that connects two selected sketch curves. Bridge Curve is typically used to create adjoining geometry.

Ellipse:

The Ellipse creates an ellipse from a centre point, a major axis, and a minor axis. The first click sets the centre point, the second click sets the direction and length of the first axis, and the third point can be any point on the ellipse.

Point:

Points are used as construction commands to help position sketched geometry. Center points look like tiny cross-hairs. Sketch points are deleted when the coincident geometry (line, spline) is deleted.



Fillet & Chamfer:

Fillet – Places an arc of a specified radius, tangential on perpendicular or parallel lines, concentric arcs, intersecting and nonintersecting arcs, elliptical arcs, splines, or between arcs and lines. Multiple fillets may be created in a single command with all fillets having the radius specified for the first fillet created; when the first radius is modified, all related fillets are automatically updates.

Chamfer – The Chamfer tool places a chamfer (or bevel) at the corner or intersection of

two lines or two nonparallel lines. The chamfer may be specified as equal distances, two unequal distances, or a distance and an angle.

Polygon:


The Polygon tool is used to create polygons containing 3 to 120 sides.

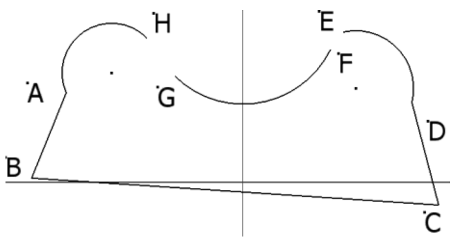
Text & Geometry Text:

Text – Text is added to a sketch when the text is to be embossed or engraved onto the surface of the part.

Geometry Text – Geometry Text is used to create and align text to sketch geometry.

Tutorial Exercise 1-3:

Produce a sketch of the shape opposite. DO NOT let the ends of the arcs touch each other. DO NOT worry about the lines being horizontal or vertical or about the dimensions as the sketch will be used in Tutorial Exercises 1-4 and 1-5 to complete the sketch.



Procedure:

Draw the straight lines at Points A, B, C, & D.

1. Click on the Line button  in the Sketch panel.
2. Click on any point on the screen to create the beginning of the line at Point A.
3. Move the cursor down and to the left and then click to create the line ending at Point B.
4. Move the cursor to Point C and click.
5. Move the cursor to Point D and click.
6. Right click and select **Done**.

Draw the arcs at Points D to E, F to G & A to H.

7. Click on the Three Point Arc  button in the Sketch panel.
8. To select the 1st end of the arc, move the cursor over Point D and click when the dot at the centre of the cursor shows green.
9. Move the cursor to the left and up and click to create the other endpoint of the arc at Point E.
10. Move the cursor to create an approximate shape and left click.
11. Move the cursor to Point F similar to that shown and click to start the arc.
12. Move the cursor down and to the left to create the arc's endpoint at Point G.
13. Move the cursor to create an approximate shape and left click.
14. Move the cursor over Point A and click when the dot at the centre of the cursor shows green.
15. Move the cursor up ad to the right to create the arc's endpoint at Point H.
16. Move the cursor to create an approximate shape and left click.

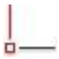
Constraints:

A constraint defines how the entities within a sketch can change shape or size. Geometric constraints control the shape and relationships among sketch lines and arcs. Dimensional constraints control the size of sketch geometry.

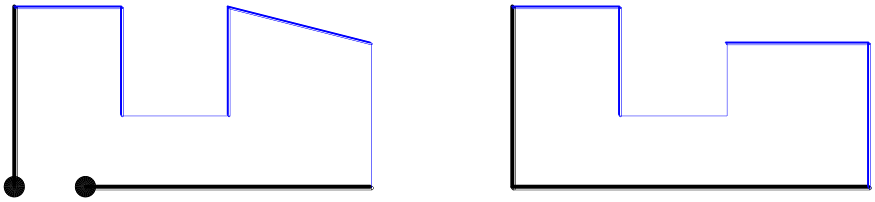
12 geometric constraint buttons are located on the Constraints Panel as shown Figure 1.25.

Figure 1.25

Coincident:



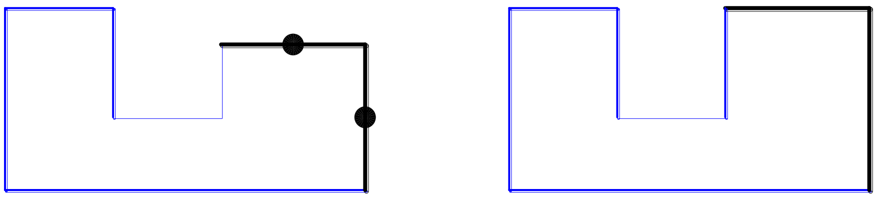
The Coincident constraint constrains two points together or one point to a curve. When the constraint is applied to the centre points of two circles, arcs, or ellipses, the result is the same as that of a concentric constraint.



Before After

Colinear:

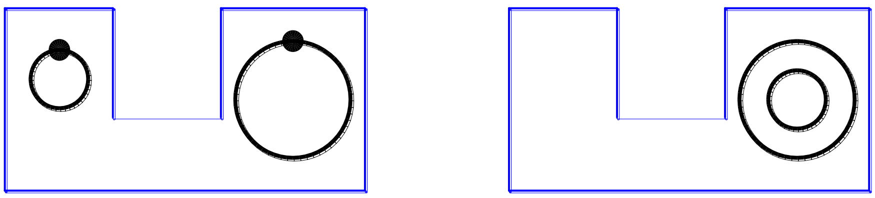
Causes two lines or ellipse axes to lie along the same line.



Before After

Concentric:

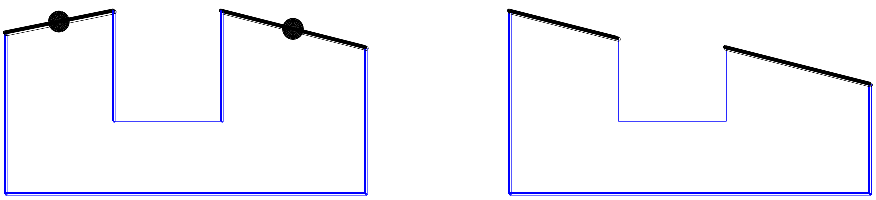
Constrains two arcs, circles, or ellipses to the same centre point. The result is the same as that of a coincident constraint applied to the centre points of the curves.



Before After

Parallel:

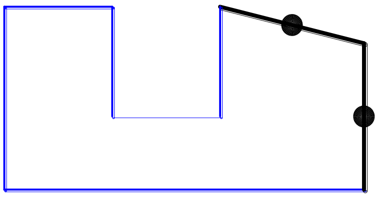
The parallel constraint causes two or more lines or ellipse axes to be constrained parallel to one another. In a 3D sketch, the parallel constraint is parallel to x, y, or z part axes unless it is manually constrained to selected geometry.



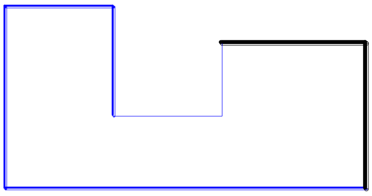
Before After

Perpendicular:

Causes selected curves or ellipse axes to lie at right angles to one another. To add a perpendicular constraint to a spline, the constraint must be applied endpoint to endpoint between the spline and the other curve.



Before

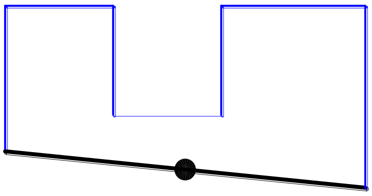


After

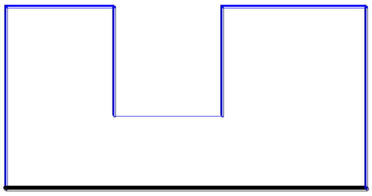
Horizontal:

Causes lines, ellipse axes, or pairs of points to lie parallel to the X axis of the sketch

coordinate system. A sketch point is automatically created on a midpoint when it is constrained. Two points (curve ends, center).



Before

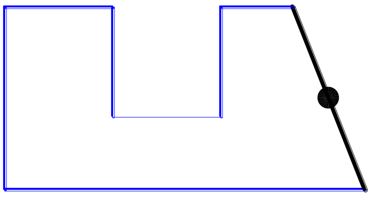


After

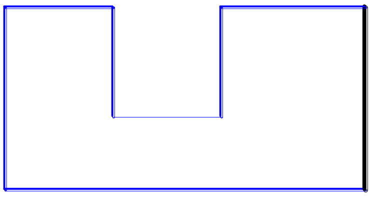
Vertical:

Causes lines, ellipse axes, or pairs of points to lie parallel to the Y axis of the sketch

coordinate system. A sketch point is automatically created on a midpoint when it is constrained. Two points (curve ends, center)



Before

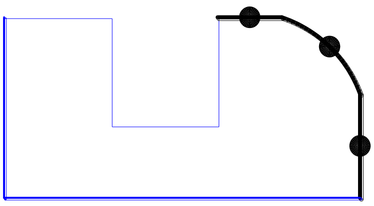


After

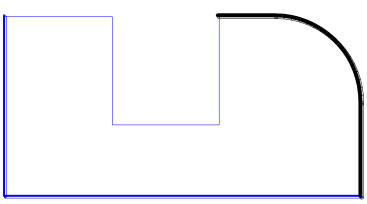
Tangent:

Constrains two curves to be tangent to one another, even if they do not physically share

a point (in a 2D sketch). Tangency is commonly used to constrain an arc to a line. A tangent constraint is required to specify how to end a spline that is tangent to other geometry.

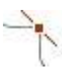


Before

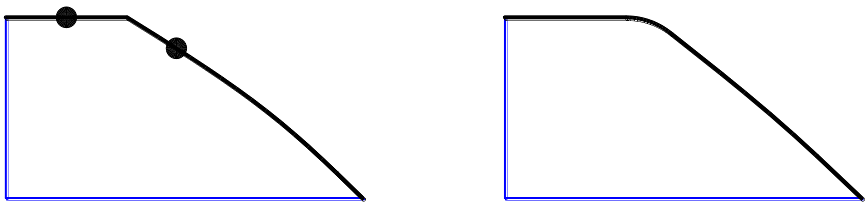


After

Smooth:



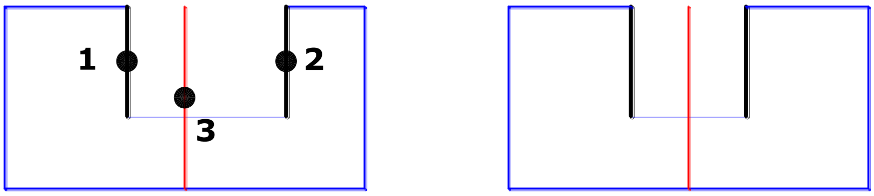
The Smooth command is used to create a curvature continuous (G2) condition between a spline and another curve, such as a line, arc, or spline. The Smooth constraint is available in a 2D or 3D sketch or a drawing sketch.



Before After

Symetric:

Causes lines and arcs to become aligned symmetrically about a selected line. Symmetry  constraints are added to the selected geometry. Selected lines and arcs are constrained about a line to create a symmetrical shape.

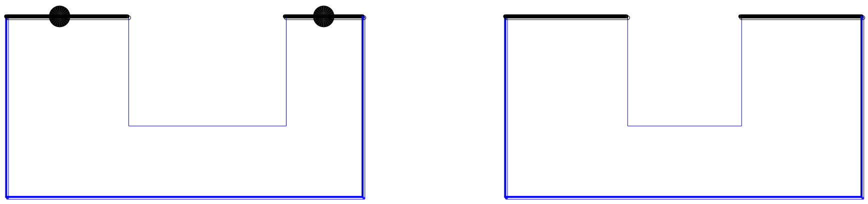


Before After

In the example above, Line 1 would be selected first and Line 2 second. Line 3 becomes the centreline about which all other lines are placed and needs to be selected only the once.

Equal:

Resizes the selected arcs and circles to the same radius or selected lines to the same  length.

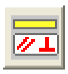


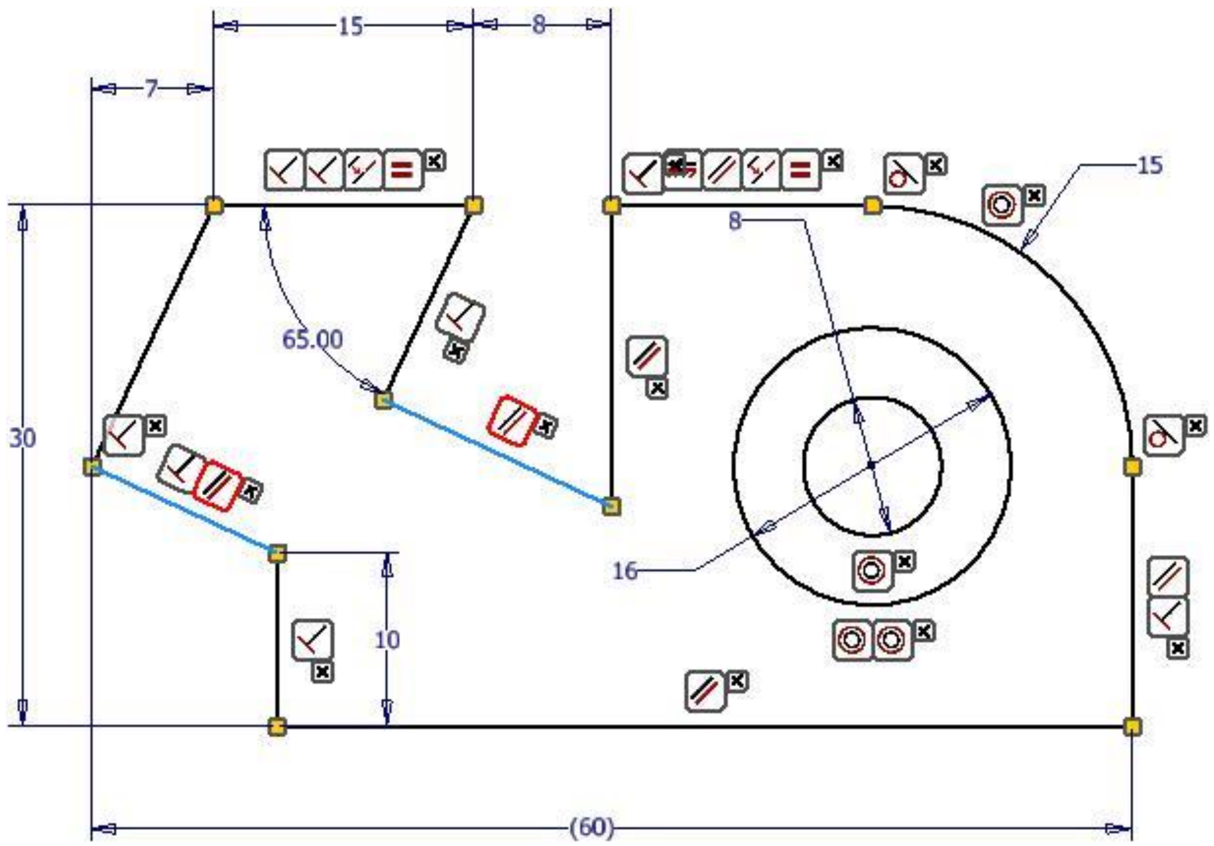
Before After

Fixed Point (Locked):

The fixed constraint causes points or curves to be constrained to a fixed location relative  to the sketch coordinate system. Projected (reference) geometry cannot be fixed. If the sketch coordinate system is moved or rotated, fixed curves or points move with it.

Displaying Constraints:

 The Display Constraints button is used to Show Constraints tool on the 2D Sketch or 3D Sketch toolbar to view constraints for individual lines and curves. The display for all elements can be displayed by right clicking the pointing device and selecting **Display all Constraints** or by pressing the function key **F8**. To hide all constraints, right-click, and then select **Hide All Constraints** or by pressing the function key **F9**.



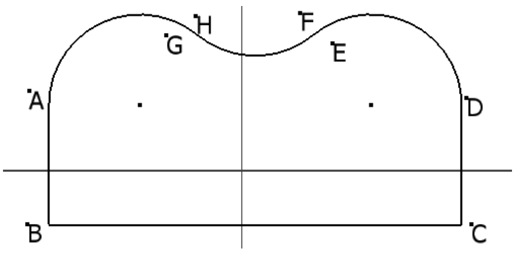
Individual groups of constraints can be hidden by clicking on the Hide Constraint button at the end of the group.

Deleting Constraints:

A constraint can be deleted by moving the cursor over the particular constraint, right click the pointing device and then select **Delete**. When a constraint is deleted from one element, the constraint on the corresponding element is also deleted. Moving the cursor over a constraint highlights the both elements.

Tutorial Exercise 1-4:

Continuing with Tutorial Exercise 1-3, add constraints to the sketch so the final shape looks similar to that shown:



Procedure:

Constrain the arcs to each other.

1. Expand the constraints in the 2D Sketch panel by clicking the triangle next to the Perpendicular button and then select the Coincident button**.**
2. Select the end of the left arc at Point H.
3. Select the end of the center arc at Point G. The arcs are joined but are NOT tangential to each other.
4. Select the end of the right arc at Point E.
5. Select the end of the center arc at Point F. The arcs are joined but are NOT tangential to each other.

Constrain the arcs and straight lines to form tangents.

6. Expand the constraints in the 2D Sketch panel by clicking the triangle next to the Coincident button and then select the Tangent button.
7. Select the straight line AB.
8. Select arc AH that connects to the straight line. *The line and arc are now tangential.*
9. Select the straight line CD.
10. Select arc DE that connects to the straight line. *The line and arc are now tangential.*
11. Select the arc DE.
12. Select the arc FG. The arcs are now tangential.
13. Select the arc AH.
14. Select the arc FG. The arcs are now tangential.

Constrain lines AB & CD vertical.

15. Expand the constraints in the 2D Sketch panel by clicking the triangle next to the Tangent button and then select the Vertical button.
16. Select the straight line AB.
17. Select the straight line CD.

Constrain line BC horizontal.

18. Expand the constraints in the 2D Sketch panel by clicking the triangle next to the Vertical button and then select the horizontal button.
19. Select the straight line BC.

Constrain the lines AB & CD to the same length and the arcs to the same radius.

20. Expand the constraints in the 2D Sketch panel by clicking the triangle next to the horizontal button and then select the Equal button.
21. Select the straight line AB.
22. Select the straight line CD.
23. Select the arc AH.
24. Select the arc FG.
25. Select the arc DE.
26. Select the arc FG.

Constrain the lines AB & CD so the sketch is symmetrical about the centerline.

27. Expand the constraints in the 2D Sketch panel by clicking the triangle next to the Equal button and then select the Symmetric button.
28. Select the straight line AB.
29. Select the straight line CD.
30. Select the unnamed centerline.

Save the work.

31. Save the file in your work area as TUT1-3.

Dimensioning the Sketch:

Dimensions are added to the sketch to control the size of a part and can be expressed as numeric constants, as variables in an equation, or in parameter files. Parametric dimensions resize the geometry when a dimension valve is changed; the ability of the sketch being driven by the dimensions (and constraints) reduces the need for sketching exact lengths and angles etc. When a sketch dimension is edited, its position adjusts as the sketch geometry updates. As the sketch is orbited, the dimension text remains parallel to the bottom of the screen to allow easy reading as shown in Figure 1.26.

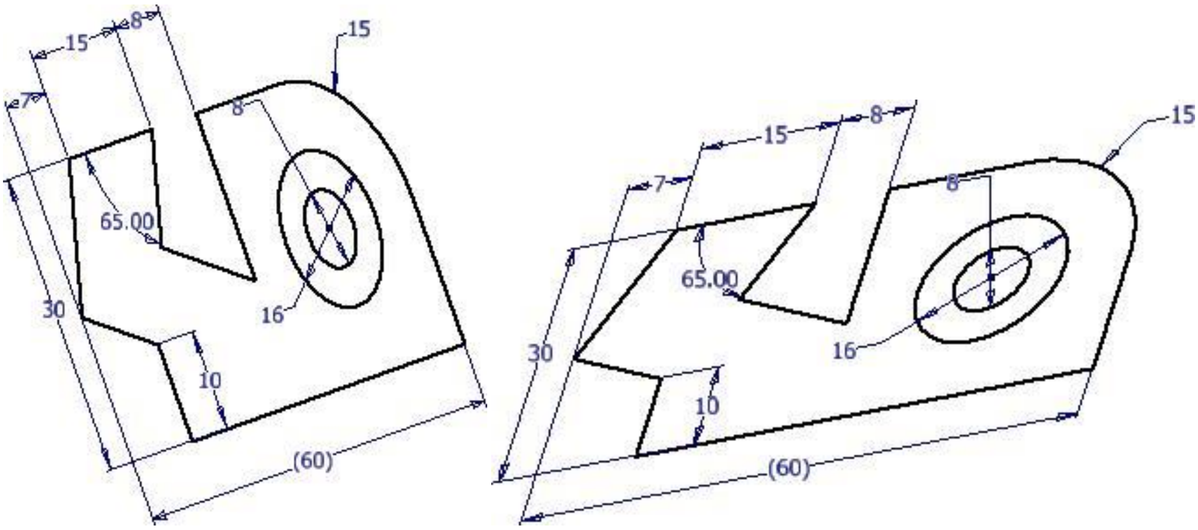


Figure 1.26

General Dimension:

The General Dimension button is used on the Sketch buttonbar to add dimensions to a sketch. Dimensions can be added to the sketch by several methods, and depending on the entity selected. Dimensions can be added by:

- Selecting the geometry from the graphics window and dragging the displayed dimension to a position near the sketch.
- Linear dimensions for curves are added by selecting the curve and then dragging the displayed dimension to a position near the sketch.
- Linear dimension between two points, two curves, or a curve and a point, are added by clicking at each endpoint of the line or curve and then dragging the displayed dimension to a position near the sketch.
- Radial or diametric dimensions are added by clicking arc or circle. Right-click and select Radius for an arc or circle or Linear for a centerline.
- To add angular dimensions, select two curves or lines and then position the dimension.
- An aligned dimension is placed on the sketch by selecting the points or line, and then clicking the right mouse button and selecting "Aligned" from the options in the menu.

When the dimension is added to the drawing the Edit Dimension dialog box is displayed. The desired dimension is typed directly into the dialog box, and applied to the sketch by

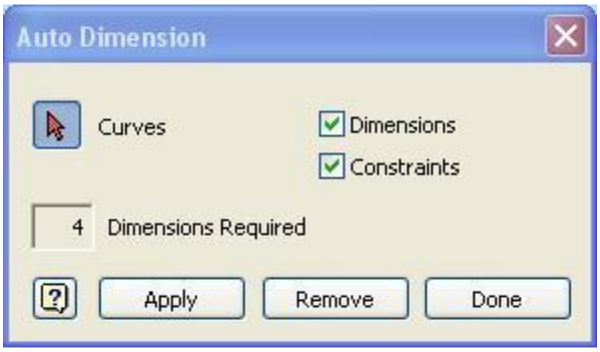
clicking the "tick" displayed in the lower right corner of the dialog box or pressing the **Enter** key.

Editing Dimensions:

Dimensions can be edited simply by moving the cursor over the dimension in the Sketch Mode and then double left clicking on the dimension. The Edit Dimension dialog box is displayed where the new dimension can be directly typed.

Automatic Dimension:

The Automatic Dimension button adds dimensions and fully constrains the sketch and is used in addition to the General Dimension button to place critical dimensions. The program remembers which dimensions are placed with the General Dimension and Automatic Dimension buttons so that specifically added dimensions are not replaced by automatic dimensions. Using the Automatic Dimensioning button displays the Auto Dimension dialog box.



Curves – Selects geometry to dimension.

Dimensions – Default is On. Applies automatic dimensions to the selected geometry. Clear the check mark to exclude dimensions.

Constraints – Default is On. Applies automatic constraints to the selected geometry. Clear the check mark to exclude constraints.

Dimensions Required – Shows number of constraints and dimensions required to fully constrain the sketch. If either Dimensions or Constraints are excluded from the solution, the number is removed from the total shown.

Rules for Dimensioning:

Place dimensions on large elements before small ones to minimize distortion. First define large elements that tend to determine sketch size. Dimensioning small elements first might restrict overall size. Delete or undo a dimension if it distorts the shape of the sketch.

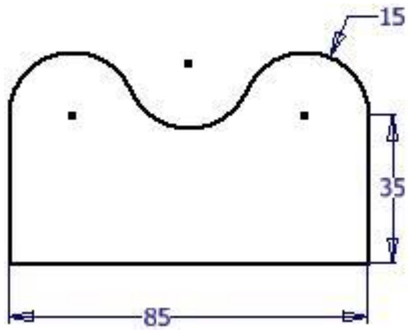
If a feature sketch is likely to change size, consider leaving some of its elements underconstrained. For example, in sketching a slot, it may be a better to leave its length undimensioned. The slot can then adapt to the required size the part is placed in an assembly.

Use both geometric constraints and dimensions as some constraint combinations may distort underconstrained portions of the sketch. If distortion occurs, undo the last constraint placed and consider using a dimension or a different constraint combination.

At times when the shape of the sketch becomes unmanageable, or if the entity is always "over constrained" but it you cannot constrain it to it's correct length or dimension, it is better to delete the entity and redraw it. Any constraint or dimension should be removed and the new entity constrained with little trouble – we hope.

Tutorial Exercise 1-5:

Continue with Tutorial Exercise 1-4 and add the dimensions where shown. Due to the number of constraints forming the shape, only 3 dimensions are required.



Procedure:

1. Click the Dimension  button.
2. Select line BC. The dimension is displayed.
3. Drag the cursor down and click to place the dimension. *The Edit Dimension dialog box is displayed*.
4. Type **85** in the dialog box and click the green tick or press the **Enter** key.
5. Select line CD. The dimension is displayed on the drawing.
6. Drag the cursor right and click to place the dimension. *The Edit Dimension dialog box is displayed*.
7. Type **35** in the dialog box and click the green tick or press the **Enter** key.
8. Select line CD. The dimension is displayed on the drawing.
9. Select arc DE. The dimension is displayed.
10. Drag the cursor to the side and click to place the dimension. *The Edit Dimension dialog box is displayed*.
11. Type **15** in the dialog box and click the green tick or press the **Enter** key.
12. Select line AB.
13. Select the FG.
14. Drag the cursor up and click to place the dimension. The Edit Dimension dialog box is displayed with the dimension 42.5 (half of 85).
15. Click the green tick or press the **Enter** key and the following warning is displayed.

Over Constraining the Sketch:

Inventor tracks all constraints and dimensions, and detects when the constraint or dimension being placed, duplicates, coincides or shows inconsistency with the existing dimensions and constraints used on the sketch.

Detection is indicated as shown in Figure 1.27 for constraints and Figure 1.28 for dimensions.

Figure 1.27

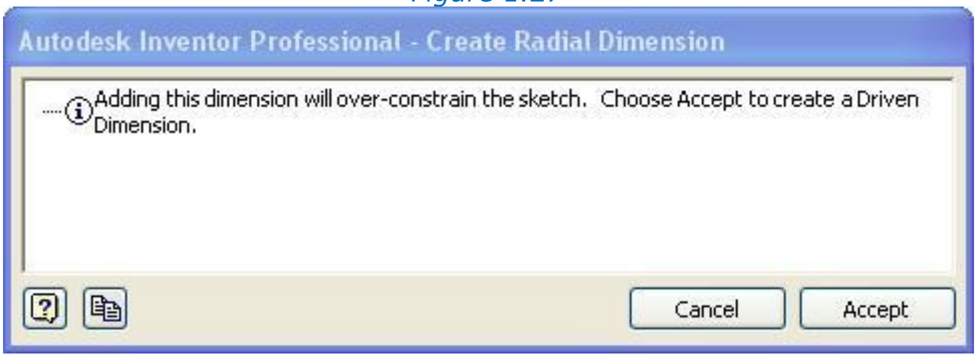


Figure 1.28

Over constrained dimensions can be displayed on the drawing as "Reference Dimensions" and appear inside brackets as shown in Figure 1.29 below. Reference do not control the sketch (or model) size and are intended for checking only when the accumulative or overall or location dimensions are not indicated but included with the sketch dimensions and constraints..

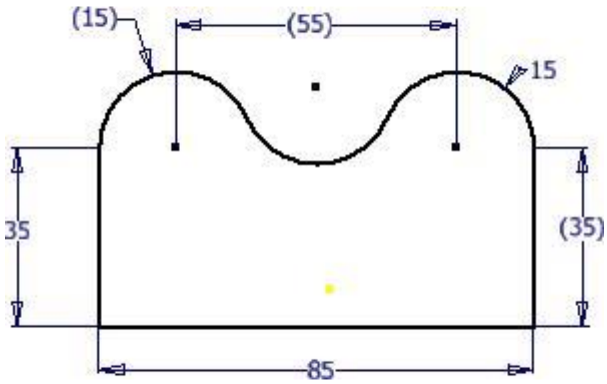


Figure 1.29

Completing the Sketch:

A sketch must be "Finished" by clicking the Finish Sketch button on the Sketch panel or right clicking the pointing device and selecting **Finish 2D Sketch** before it can be saved. The sketch must always be finished before any other procedure can be executed.

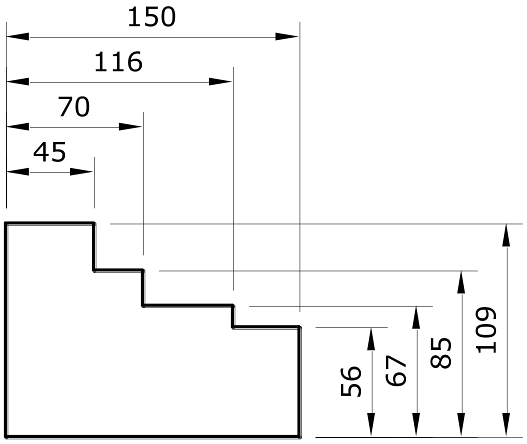
Skill Practice Exercises

Skill Practice Exercise 1-1

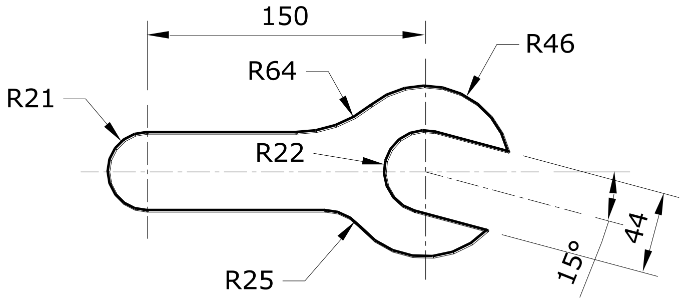
Create a fully constrained and dimensioned sketch of the 10 mm thick Stepped Block. Save the file in your Skill Practice project folder as MEM30004-SP-101.

Skill Practice Exercise 1-2

Create a fully constrained and dimensioned sketch of the 10 mm thick Spanner. Save the file in your Skill Practice project folder as MEM30004-SP-102.



Skill Practice Exercise 1-1



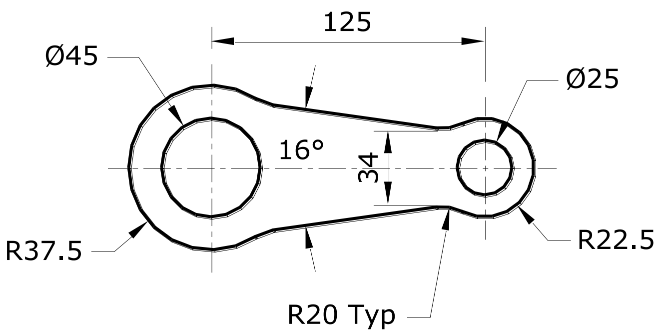
Skill Practice Exercise 1-2

Skill Practice Exercise 1-3

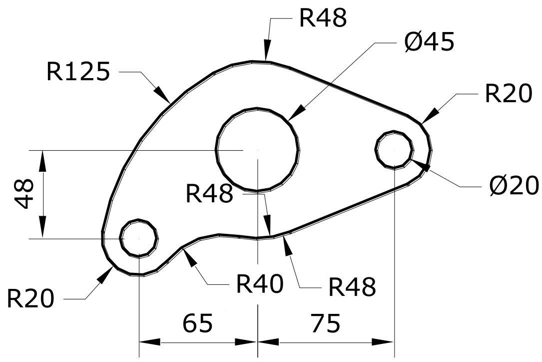
Create a fully constrained and dimensioned sketch of the 10 mm thick Crank Arm. Save the file in your Skill Practice project folder as MEM30004-SP-103.

Skill Practice Exercise 1-4

Create a fully constrained and dimensioned sketch of the 10 mm thick Rocker Arm. Save the file in your Skill Practice project folder as MEM30004-SP-104.



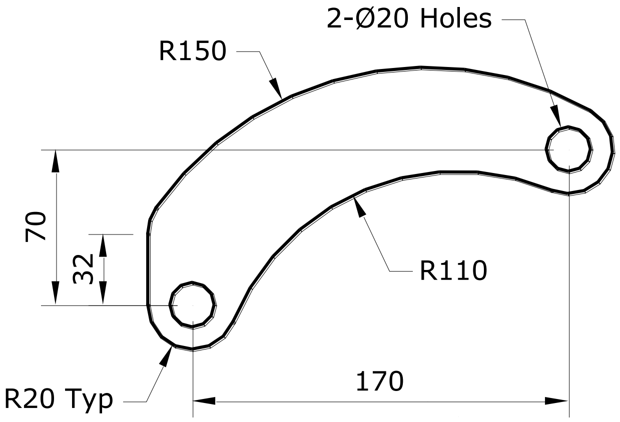
Skill Practice Exercise 1-3



Skill Practice Exercise 1-4

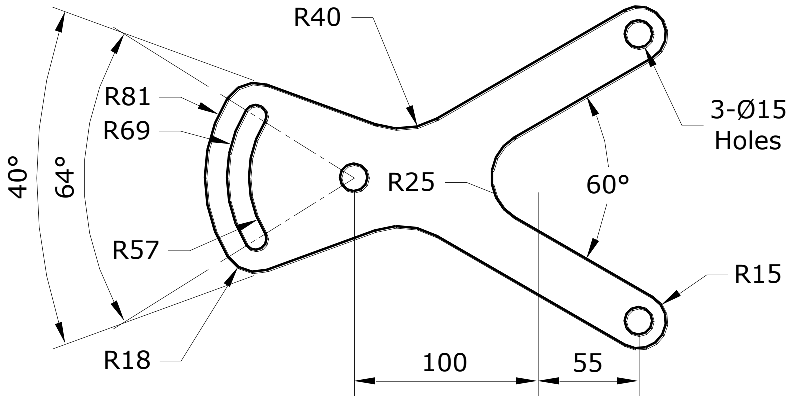
Skill Practice Exercise 1-5

Create a fully constrained and dimensioned sketch of the 12 mm thick Hook Hanger. Save the file in your Skill Practice project folder as MEM30004-SP-105.



Skill Practice Exercise 1-6

Create a 3D model of the Adjustable Y-Clamp which is 18 mm thick using Autodesk Inventor; save the file to Skill Practice project folder area as MEM374-SP-106.



Skill Practice Exercise 1-7

Create a 3D model of the 3 mm thick Head Gasket using Autodesk Inventor; save the file to your Skill Practice project folder as MEM30004-SP-107.