DraftSight[®]

Getting Started Guide







Welcome to the DraftSight Getting Started Guide.

For beginners and experienced CAD users alike, this guide has more than 180 pages of fundamental skills necessary for the efficient use of DraftSight. Chapters one through four cover functionalities for all users, and chapter 5 details the powerful, time-saving features that are only available in the Enterprise and Professional Pack Offerings for DraftSight. Discover more about each of these premium offerings below.

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INTRODUCTION

When you complete this lesson, you will be able to:

- Understand the basic capabilities of DraftSight.
- Start a DraftSight session.
- Identify the principal components of the DraftSight user interface.
- Identify the graphics area, layout tabs, command window, command prompt, and status bar.
- Identify the default toolbars in the user interface: Main Menu, Standard, Draw, Modify, Layer, Properties and Entity Snap.
- Understand the use of the default Palettes.
- Manipulate toolbars by moving them and turing them on and off.

Using this Book

This book is designed to be used by the learner alone, without the aid of an instructor. This is a basic training guide for new DraftSight users. It is designed for users with little or no CAD experience.

This guide will cover the fundamental skills necessary for efficient use of DraftSight and will provide a strong foundation for advancement.

Experienced users may also benefit from using this guide by understanding the menu system and dialog used by DraftSight.

This guide is organized in a chronological manner that will allow you to build on your knowledge as you progress through the course.

What is DraftSight?

DraftSight is a drafting automation software tool. A coordinate system determines each point of a drawing surface or entity unambiguously. DraftSight uses the Cartesian coordinate system consisting of three coordinate axes. Axes are arranged orthogonally, crossing at the origin. All axes use the same unit of measurements.

Conventions Used in this Book

This manual uses the following typographical conventions:

Convention	Meaning
17 Click Line.	The steps in the lessons are numbered in sans serif bold.
Bold	DraftSight commands and tools appear in this style. For example, Clicking the Line icon S means choose the Line tool on the Draw toolbar.
<u>م</u> ا	The \dashv symbol means press the Enter key on the keyboard.
Command prompt	Prompts and information in the command window and Command prompt are displayed in this font. For example: Specify next point»
Bold	Information you type into the command window appears in this font. For example: Specify next point» 200,100 ↓
Typewriter	File names are in this font. For example, Lesson 1-1

TIP:

Start a DraftSight Session

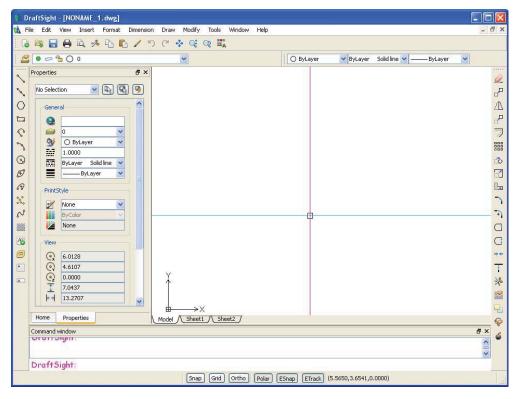


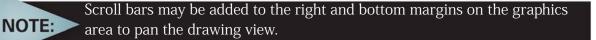
1 Start a DraftSight session.

From the Windows Start menu, click **Start**, **All Programs**, **Dassault Systemes**, **DraftSight**.

If you created the DraftSight icon on your desktop, double-click the icon to start a DraftSight session.

DraftSight automatically assigns a generic name, **NONAME_0.dwg** to a new drawing. Each subsequent file that is open assigns another name by increasing the numeric value by 1.





DraftSight User Interface

The default DraftSight drawing screen contains the Main Menu, Standard toolbar, Modify toolbar, Draw toolbar, Layers toolbar, Properties toolbar, the Graphics area, command window, Status bar, and Tool Palettes.

Tool tips appear on each icon as you move the mouse cursor over the icon. You can resize the graphics area by click and drag at the edges of the window, or by resizing or closing the Tool Palettes, default toolbars, or command window.

Main Menu

The Main Menu is displayed by default. The default menu items for an active drawing document are: **File**,

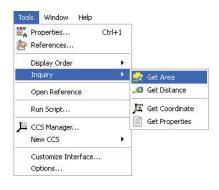
	Drat	tSigh	t - [NO	NAME_	1.dwg]						
D	File	Edit	View	Insert	Format	Dimension	Draw	Modify	Tools	Window	Help

Edit, View, Insert, Format, Dimension, Draw, Modify, Tools, Window, and Help.

NOTE: Menu content varies depending on the menu file that is loaded -The standard menu or individualized menus.

Drop-down Menu

Drop-down menus are available from the Main Menu at the top of the screen. To select any of the available commands, move your cursor over an item and click. Commands followed by a black arrow will display an additional list of related options.



Context Menu

Right-click in the graphics area displays a Context Menu. The Context Menu remembers your last command. If no command is active, the Context Menu displays commonly used commands.

	Repeat OPTIONS	
-%	Cut	Ctrl+X
6	Сору	Ctrl+C
1	Copy with Reference Point	Ctrl+Shift+C
ß	Paste	Ctrl+V
5	Undo	Ctrl+Z
C	Redo	Ctrl+Y
\$	Pan	
Q:	Zoom	
	Options	
	Drafting Options	

TIP:

Keyboard Shortcuts

Some menu items indicate a keyboard shortcut like this:

Windows conventions for shortcuts such as:

- Ctrl+O for File Open
- Ctrl+S for File, Save
- Ctrl+X for Cut
- Ctrl+C for Copy

In addition, you can customize DraftSight by creating your own shortcuts.

File	Edit View	Insert	Format	Din
6	New		Ctrl+N	
	Open		Ctrl+O	
×	Close			
	Save		Ctrl+S	
	Save As		Ctrl+Shift+	-s
	Export			э н
	Drawings Now			
	Publish eDrawing	s		
	Page Layout Man	ager		
	Sheet			۲
D.	Print Preview			
8	Print		Ctrl+P	
	Error Check			
*	Clean			
	Recover			
	Exit		Ctrl+Q	

G 📪 🗖 🖨 Q 🥕 🗗 🛍 🦯 🍏 C 🚸 Q Q

Standard Toolbar

Toolbars contain groups of related

command icons for quick access. The Standard toolbar is located (docked) beneath the Menu bar. The Standard toolbar allows you to quickly access the most frequently used commands. You can customize the toolbar by adding or removing icons.

The available default tools on the Standard toolbar are:

- **New**: Creates a new drawing file.
- **Open**: Opens an existing drawing file.
- **• Save**: Saves the drawing file.
- Print: Prints a drawing to a plotter, printer or file.
- **Print Preview**: Shows how the drawing will look when it is printed or plotted.
- **Cut**: Copies entities to the Windows Clipboard and removes the entities from the drawing.
- **Copy**: Copies entities to the Windows Clipboard.
- Paste: Inserts data from the Windows Clipboard.
- **Property Painter**: Applies the properties of a selected entity to other entities.
- **Undo**: Reverses the most recent action.
- **Redo**: Reverses the effects of the previous **Undo** or **U** command.
- **Dynamic Pan**: Moves the view in the ViewPort.
- **Dynamic Zoom**: Zooms to increase or decreases the apparent size of entities in the ViewPort.
- **Zoom Previous**: Zooms to display previous view.
- ■ Properties: Controls properties of existing entities.

Hold the mouse pointer over an icon momentarily, and a tool tip appears describing the icon.



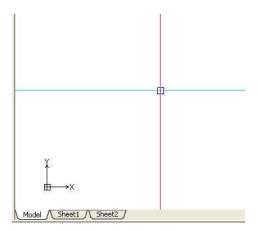
INTRODUCTION

DraftSight

Graphics Area

The graphics area is the part of the desktop where drawing entities are created and modified. Multiple drawings can be opened simultaneously.

Each drawing and each view is represented in its own window. The individual windows can be tiled, cascaded, or enlarged to fill the graphics area.



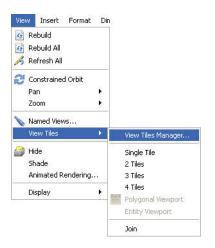
For clarity, the graphics area in this book is displayed in white. Default colors can be modified from **Tools**, **Options**, **System Options**, **Element Colors** from the Main Menu

	🕀 General	
<u>~~</u> /	🖃 Display	
File Locations	E Screen options	
[7]		
	Element Colors	
System Options	Command Window Background Command Window Text Cursor X Cursor Y Cursor Z ESnap Cue	Color:
Drawing Settings	Reference Guides Model Background Model Crosshair Sheet Background	O White

View Tiles

The **View Tiles** command divides the graphics area into multiple windows. Each tile can contain a unique view of the drawing. This enables you to view different areas of the drawing, whether they are plan views in different detailing grades, or the model from different view points. You can work in only one window at a time, the current window.

Click **View**, **View Titles** from the Main Menu to view your options.



Graphics Cursor / Crosshairs

Outside the graphics area, the cursor assumes the typical Windows pointing shape. Within the ____

graphics area, the mouse cursor appears as crosshairs during an active command.

The crosshairs act as the pointer in this application. The crosshairs indicate the pointer position in the graphics area. As with the coordinate symbol, the pointer symbol also changes when the drawing view changes from a default plan view (viewed from above) to an isometric view.

Cartesian Coordinate System / Origin

A coordinate system determines each Point of a drawing surface or entity unambiguously. DraftSight uses the Cartesian coordinate system consisting of three coordinate axes. Axes are arranged orthogonally, crossing at the origin. All axes use the same measurements.

All drawings are based on a Cartesian coordinate system in which perpendicular axes are used: the X, Y axes. The intersection of the two coordinate axes forms a Point call the origin.

All axes originate at the origin of the coordinate system. The X axis and Y axis define a horizontal plane and is located in the lower left corner of the graphics area.

The Coordinate symbol is a visual reference to the drawing. This symbol indicates the position of the axes of the coordinate system and provides orientation of the alignment of the coordinate system. The directions of the arrows displayed on the axes identify the positive side of the coordinates.

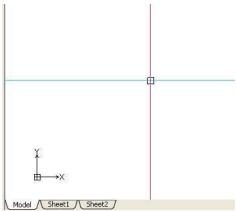
Coordinate System

The Coordinate System (CCS) icon is known as the coordinate symbol. By default, the CCS symbol and origin is displayed in the graphics area. To control the display of the coordinate system icon click **View**, **Display**, **CCS Icon** from the Main Menu or type **csicon** at the Command prompt in the command window.

: csicon.⊣

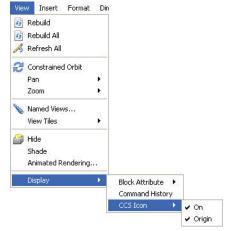
Specify option»

- All: By default, the CSicon command affects only the current view. The All option applies changes to the coordinate system icon in all views. You can choose from No origin, Off, On or Origin.
- **No origin**: Displays the coordinate system icon at the lower left corner independent from the origin of the current coordinate system.
- **Off**: Hides the Coordinate system icon.



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3	14		8		39	-
			32		8	53
8	12	28	191	22	20	23
4	32	8	5	1		13
	1.	81	8	a.		10
٠¥		8	3	13	8	12
	8		85	12	40	23
Ľ	14	××.	84	64	20	23





- **On**: Displays the Coordinate system icon.
- **Origin**: Displays the coordinate system icon at the origin (0,0) of the current coordinate system. If the origin is outside of the display, the icon is positioned in the lower left corner.

Model Tab and Sheet Tab

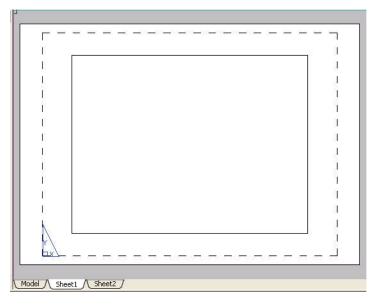
By default, the desktop contains three tabs on the bottom left labeled:

- Model
- Sheet1
- Sheet2

Select the desired page by clicking the appropriate tab, which displays your drawing either as a model or as a layout sheet in your drawing.

You change the drawing environment between the Model Tab (for drawing entities) and Sheet Tabs (for layouts for printing/plotting).

Create your design in the Model (Model tab), and then create layouts to print/plot your drawing on a Sheet (Sheet tab).



Work with multiple Sheets (layouts) using the **Sheet** command to perform additional options.

Command window : sheet

Default: Activate

NOTE:

Options: <u>2</u> to list, <u>Activate</u>, <u>Copy</u>, <u>Delete</u>, <u>New</u>, <u>Rename</u>, <u>SAveas</u>, or <u>Template</u> Specify option»

Command Window

The command window provides the ability to enter commands and communicate between you and the drawing.

Command window
: _RECTANGLE
Options: <u>C</u> hamfer, <u>E</u> levation, <u>Fillet</u> , <u>T</u> hickness, line <u>W</u> idth or Specify start corner »

The command window is located at the bottom section of the screen and provides status information and prompts you for the next step.

Press $\langle F2 \rangle$ to display the command history in a separate window. When the command window dialog box is displayed, you can scroll through the command history. Press $\langle F2 \rangle$ again to close the command window dialog box.

Options: Segments, Undo, Enter to exit or	1
Specify next point»	
Options: Segments, Undo, Close, Enter to exit or	
Specify next point»	
Options: Segments, Undo, Close, Enter to exit or	
Specify next point»	
Options: Segments, Undo, Close, Enter to exit or	
Specify next point» «Cancel»	

You can move the floating command window anywhere on the screen and resize its width and height by dragging a side, bottom, or corner of the window.

🚺 Command Window	
Specity next point»	
Options: <u>S</u> egments, <u>U</u> ndo, <u>C</u> lose, <i>Enter to exit</i> or	
Specify next point» «Cancel»	

You must press the **Enter** key (\Box) after typing a command or entering a command option. The command window continues to inform you of the type of responses(s) that you must enter, until the command is either completed or terminated.

Each command (**Line**, **Circle**, etc.) has its own series of prompts. The prompts in the Command prompt area that appear when a particular command is used in one situation may differ from the prompts or sequence or prompts when invoked in another situation.

There are three way to terminate a command:

- Complete the command and return to the : prompt. Specify next point» Options: Segments, Undo, Close, Enter to exit or
- Command window Specify next point» Options: Segments, Undo, Close, Enter to exit or Specify next point» Options: Segments, Undo, Close, Enter to exit or Specify next point» c
- Press the **Esc** key to terminate the command.

command in progress.

- : line Options: <u>S</u>egments, *Enter to continue from last point* or **Specify start point**» «Cancel»
- Enter another command from one of the menus or toolbars, which automatically cancels any Specify start Options: Some

Specify start point» Options: <u>Segments, U</u>ndo, *Enter to exit* or

Specify next point» «Cancel» : _CIRCLE

Options: <u>3P</u>oint, <u>2P</u>oint, <u>T</u>tr, <u>TTT</u>, <u>Enter to exit</u> or **Specify center point**»

Status Bar

The Status bar is located beneath the command window and is divided into three sections: Tool tip display, Drafting tools, and Current coordinate display.



- **Tool tip display**. The left hand side of the status bar displays tool tips. If you move the cursor over toolbar icons, a brief description of the corresponding command or function displays in the tool tips area. Descriptions of commands selected from any of the pull-down menu items also display in this area.
- Drafting tools. Drafting tools make it possible to create and modify entities more easily and accurately. Each of these drafting tools can be toggled On when needed and toggled Off when not. When used appropriately, these commands provide the power, speed, and accuracy associated with 2D CAD drafting. The following drafting tools are available:
 - Snap (Snap Grid)
 - Grid Grid (Grid Display)
 - Ortho (Orthogonal Drawing Mode)
 - Polar (Polar Guides)
 - ESnap (EntitySnap)
 - **ETrack** (Entity Tracking Guides)
- **Current coordinate display.** The information field at the right of the buttons shows the X, Y, and Z coordinates of the crosshair (cursor) position.

Snap (Snap Grid)

The **Snap** command provides an invisible grid in the graphics area. With Snap on, the cursor can only select Points at designated (X, Y) coordinates.

Use the **Snap** command to determine the distance between snap Points.

Snap spacing may or may not be equal to the Grid spacing.

You can also use an Isometric style of snap. Use isometric snap to create 2D isometric drawings representing 3D entities.

Grid (Grid Display)

The **Grid** command is used to display a visible pattern of dots. With **Grid** on, a latticedpoint grid is displayed in the graphics area to help visualize distances, angles, and entity relationships.

The grid is a drawing tool and is not part of the drawing. It will not be printed or plotted. The Grid spacing may or may not be equal to the Snap spacing.

Ortho (Orthogonal Drawing Mode)

The **Ortho** command limits the cursor movement so that it moves parallel to the axes of the current coordinate system. With **Ortho** on, you can only enter Points located parallel to these axes. The **Ortho** command makes it easier to draw and place parallel or colinear Lines on entities.

Polar (Polar Guides)

The **Polar** command draws at exact angles at fixed intervals, starting from 3 o'clock (or East), and at additional angle increments. With **Polar** on, the angular direction and the distance from the last Point selected will be tracked and displayed according to the settings.

ESnap (EntitySnap)

The **ESnap** command provides the ability to detect and snap to geometrically significant Points on drawing entities, for example; end points, intersections, and center Points. Snapping to a significant Point provides an exact position for drawing and editing commands.

The **ESnap** command opens the Pointer Control settings of the Drafting Options dialog box so that you can set EntitySnap.

ETrack (Entity Tracking Guides)

The **ETrack** command helps you draw entities at specific angles or in specific relationships to other entities. With **ETrack** on, temporary alignment guides help you create entities at precise positions and angles. **ETrack** works with **ESnap** and polar angle settings. You must set an EntitySnap before you can track from an ETrack Point.

While **Grid**, **Snap**, **Ortho**, **Polar** and **ESnap** commands do not crerate entities, They make it possible to create them more easily and accurately.

Palettes

NOTE:

Palettes are non-fixed areas on the left or right side of the graphics area which provides the ability to display or manage drawing entities. The available Palettes are:

- Properties Palette: Allows you to view and change properties of entities in a drawing. The palette displays the entity's properties including the coordinates and data that define its geometry. If more than one entity is selected, the palette displays only those properties common to all entities in the selection, such as Layer, LineColor, LineScale, LineWeight and Hyperlink. If the common properties are not identical in all of the selected entities, the corresponding drop-down list or edit field of the palette displays <<Varies>>.
- References Palette: Lets you control and manage Referenced drawings or image files. When you open a drawing that contains Referenced drawings, they are displayed in their current state. When multiple people work on a project over a network, it might be necessary to update the external references. The program reloads the specified drawings in their most recent saved states.



 Home Palette: Contains centralize links to online and web resources, design elements, and the latest in CAD news. The two default sections are: Community and Latest News.

You can display or hide the Tool Paletts at any time to increase your graphics area. Tool Palettes can also be resized or minimized.

Zoom Commands

NOTE

The **Zoom** command is like a zoom lens on a camera. You can increase or decrease the viewing area, although the actual size of the entities remains constant.

As you increase the visible size of entities, you view a smaller area of the drawing in greater detail as though you were closer. DraftSight provides you with numerous view options on the Zoom toolbar and the View drop-down menu. The available default tools on the Zoom toolbar are:

- Zoom Center: Allows you to specify a center Point for the new view and magnification value or height.
- **Zoom Dynamic**: Allows you to zoom in and out of the drawing window at real-time.
- Zoom Previous: Allows you to undo the last zoom operation. You can restore up to ten previous displays.
- **Zoom Window**: Allows you to show a selected part of a drawing in the largest possible scale.
- **Zoom Selected**: Calculates the boundary of the area that contains the specified entities and zooms in or out so the entities are visible on the screen.
- **Zoom Factor**: Allows you to zoom the display by a scale factor. This is helpful when working with sheets or when plotting or printing the drawing.
- **Zoom In**: When you zoom in, the view of the drawing enlarges.
- **Zoom Out**: When you zoom out, the view of the drawing reduces.
- **Zoom Bounds**: Displays the entire drawing even if only a potion of the drawing has entities. Zoom Bounds zooms to the bounds of the drawing if no entities extend beyond these bounds. If entities extend beyond the defined bounds, they are included in the calculation of the boundaries in which to zoom.
- **Zoom Fit:** Displays the drawing with all its elements as large as possible. Unlike Zoom Bounds, Zoom Fit ignores drawing bounds. Zoom Fit includes entities on Layers that are turned off but does not include those on frozen Layers.



Draw Toolbar

N N 0 □ C N 0 Ø R % N Ⅲ B 0 □ □

The default Draw toolbar contains all

commands that relate to drawing 2D geometrical entities. You can use these drawing tools to create new entities such as Lines, Arcs and Circles.

The available default tools on the Draw toolbar are:

- **Line**: Creates straight Line segments.
- **Infinite Line**: Creates a set of one or more Lines that extend infinitely in both directions from an origin. Use InfiniteLines to create frameworks or grids for use in drawing construction.
- O Polygon: Creates an equilateral closed PolyLine. A polygon is a closed linear PolyLine segment consisting of between 3 and 1024 equal length sides.
- **Rectangle**: Creates a rectangular PolyLine segment. Draw rectangles with any size or orientation in the plane with square, filleted, or chamfered corners.
- PolyLine: Creates PolyLines. A PolyLine consists of connected Lines or connected arc segments. You can create PolyLines with segments of different widths and fill settings. The default segment type for a PolyLine is a linear segment.
- Arc: Creates an arc segment. Construct arcs of any length or radius. All Arcs are drawn from a start Point in the positive horizontal direction.
- **Circle**: Creates a Circle segment. The default method for drawing a Circle is to specify a center point and a radius.
- Ellipse: Creates an Ellipse segment. An Ellipse has a center like a Circle, but has a longer radius along its major axis and a shorter radius along its minor axis.
- **Ellipse Arc**: Creates an Ellipse Arc segment. An Ellipse has a center like a Circle, but has a longer radius along its major axis and a shorter radius along its minor axis.
- Point: Creates a Point, usually used as reference in drawings. A Point is a reference to an pattern holding the x and y coordinates of the Point.
- Spline: Creates quadratic or cubic Spline curve. A Spline is a smooth curve that is fit to a set of Points. You can create a non-uniform rational B-Spline or (Nurbs) curve for any number of fit Points. You can also fit the Spline to the specified Points to within a specified tolerance value.
- **Hatch/Fill**: Fills a specified boundary with a pattern. Hatch patterns can greatly increase the clarity and legibility of a drawing. You can fill enclosed areas or selected entities in drawings with different types of hatch patterns or a solid hatch fill.
- **Make Block**: Defines a Block from selected entities. A Block is a collection of entities bound together as a single entity.
- Region: Converts an entity enclosing an area into a Region. Transforms an entity that forms a closed shape into a Region. You can use the command to combine all entities of any closed loop such as closed PolyLines (including polygons and donuts), Circles, Ellipses, and closed Splines into one entity.
- Note: Creates paragraph text entities that have the same text formatting applied to each line of text in the paragraph.
- SimpleNote: Displays text on screen as it is entered. Create Text lines of any position, orientation, justification, height and TextStyle.

Modify Toolbar

Use the Modify toolbar to modify the entities in your drawing.

The available default tools on the Modify toolbar are:

- **Delete**: Removes entities from the drawing.
- Copy: Duplicates drawing entities by selecting drawing entities and placing a copy of them at a specified position. The duplicated entities are exact copies of the selected entities including all entity properties such as Layer, LineStyle, LineColor, and LineWeight, as well as the outline of the entities. Based on one selection set, you can create multiple copies.
- Mirror: Copies selected entities onto the opposite side of a specified mirror line. To define the mirror axis, specify two Points.
- P Move: Repositions drawing entities within the coordinate system without changing their orientation or size.
- Offset: Creates parallel shapes of Lines, PolyLines, Circles, Arcs, Ellipses, Elliptical Arcs, Splines, rays, and InfiniteLines. A copy of a selected entity is placed at a specified distance from the original entity. The original entity stays in position.
- **Pattern**: Creates multiple copies of selected entities in a rectangular matrix or a polar, circular pattern. The created entities inherit all properties of the original entities such as Layer, LineStyle, LineColor, and LineWeights.
- Rotate: Rotates entities through an angle, about a base point. The base point is the Point about which rotation occurs, typically a corner or a center of an entity. If the base point is the corner of a selected entity, the entity rotates on that corner. If the base point is the center of a selected entity, the entity stays in the same relative position, but spins about the base point.
- **Scale**: Enlarges or reduces entities while keeping proportions the same.
- Stretch: Changes the position of an entities nodes, while retaining their relation to other nodes in the entity. Stretching enlarges or reduces entities in only one direction, unlike scaling. Use stretching also to move single entities inside of a contour.
- **Chamfer**: Allows you to connect two Lines with a beveled corner. The selected entities do not have to intersect, and they can overlap. If the entities overlap, they will be trimmed to create the beveled corner.
- Fillet: Allows you to create rounded corners by replacing parts of two Lines with an arc. The filleted entities do not have to intersect, and they can overlap. If the entities overlap, they are trimmed to create the rounded corner and the Lines to be filleted are connected to the fillet arc.
- **G** Split at Point: Breaks an entity at a specific Point.
- Split: Erases a part of an entity between two Points you select. Use Split as a construction tool or to create a gap between two parts of an entity to place text or a symbol. Use the command with Lines, Circles, Arcs, polyl ines, Ellipses, Splines, InfiniteLines and Rays, but not with Rich Lines.

- Weld: Merges two entities into a single entity. You can merge Lines, open PolyLines, Arcs, Elliptical Arcs or open Splines. You can also transform Arcs to Circles and Elliptical Arcs to Ellipses.
- **Extend**: Allows you to edit entities that have common boundary edges. Extend expands parts of entities to the intended boundary edge, letting you close polygons precisely or extend entities to the proper boundary edge. As a drawing entity expands, they do not change direction. A Line remains a Line, and an Arc remains an Arc.
- **Trim**: Cuts drawing entities precisely at the edges defined by one or more entities. Use Lines, InfiniteLines, rays, PolyLines, Circles, Arcs, or Ellipses as cutting edges to trim the selected entity. Blocks and text cannot be trimmed or used as cutting edges. On sheets, viewtiles can be cutting edges.
- **Edit Hatch**: Modifies hatch patterns, solid color fills or gradient color fills. You can then add additional entities to the set of entities to be hatched or filled.
- Display Order: Changes the drawing and printing order of any entity in the drawing database. You can move entities to the 'front' or 'back' of the sort order. You can also order entities relative to other entities, that is, above or below a selected entity.
- **Edit PolyLine**: Edits PolyLines or PolyLine vertices by opening a closed PolyLine or closing an open PolyLine. Joining a PolyLine with other drawing entities such as Lines, Arcs or PolyLines. You can also change the width of the whole PolyLine and also edit the vertices of individual PolyLine segments.
- **Explode**: Breaks a complex entity into its component entities. You can explode Blocks and other complex entities like PolyLines, Hatches and Dimensions.

EntitySnap Toolbar

EntitySnap tools are available on the

EntitySnap toolbar. They allow for precise drawing within DraftSight. For example, a rectangle can be drawn from the exact endpoint of an existing Line by simply selecting the **Snap to Endpoint** tool, or you can draw a Line from the center point of a Circle using the **Snap to Center** tool.

The available default tools on the EntitySnap toolbar are:

- **Snap From**: Locates a Point offset from a reference Point within a command.
- **Interference Point**: Creates a temporary Point used by Esnap.
- **Snap to Visual Intersection**: Snaps to the apparent intersection of two entities.
- **Snap to Intersection**: Snaps to the intersection of two entities.
- **Snap to Endpoint**: Snaps to the closest end point of an entity.
- **Snap to Extension**: Snaps to the phantom extension of an Arc or Line.
- **• Snap to Midpoint**: Snaps to the mid point of an entity.
- **Snap to Center**: Snaps to the center of an Arc, Circle, Ellipse or Elliptical Arc.
- Snap to Quardant: Snaps to a quadrant point of an Arc, Circle, Ellipse or Elliptical Arc.

- Snap to Tangent: Snaps to the tangent of an Arc, Circle, Ellipse, Elliptical Arc or Spline.
- Snap to Insert: Snaps to the insertion point of entities such as Text, Blocks or Attributes.
- Snap to Node: Snaps to a Point entity
- Snap to Parallel: Snaps parallel to a specified Line.
- **Snap to Perpendicular**: Snaps to a Point perpendicular to an entity.
- **• C Snap to Nearest**: Snaps to the nearest Point on an entity.
- **• Snap to None**: Suppresses running EntitySnap for the selection.
- **ESnap Settings**: Sets running EntitySnap modes.

Layers Toolbar

Use the Layers toolbar to create new Layers,

designate active Layers, and set layer properties.

The available default tools on the Layers toolbar are:

- **EXAMPLE SET UP: EXAMPLE 1 Layer Set UP: EXAMPLE 1 Controls Layer and layer properties.** The Layer command creates and controls Layer and layer properties.
- Show/Hide: Shows or hides a layer on the drawing. When a layer is shown, it is visible. Hidden Layers are invisible. You cannot select entities on a hidden layer.
- Freeze: Suppresses or un-suppresses a layer on the drawing. When you Freeze a layer, you cannot select or modify entities on the layer. You cannot suppress (Freeze) the active layer.
- Lock/Unlock: Protects or un-protects a selected layer on the drawing. When a layer is protected, you cannot edit the layer. You can only edit unprotected Layers.
- • LineColor: Displays the LineColor of the layer assigned layer LineColor.
- **Layer name**: displays the active name layer.

Layer Tools Toolbar

Use the Layer Tools toolbar to create new

Layers, designate active Layer, and set layer properties.

The available default tools on the Layer Tools toolbar are:

- 🗃 Hide Layer: Entities on hidden Layers are invisible. you cannot select entities on hidden Layers.
- Show All Layers: All Layers in the drawing become visable except those that are suppressed.
- **Freeze Layer**: When a layer is frozen, it is protected and invisible.
- **•** *i* **Example 7 Constant C**
- **General Content Lock Layer**: When you lock a layer, you cannot select or modify entities on the layer.
- **Unlock Layer**: When you unlock a layer, you can edit entities on the layer.
- **Isolate Layer**: Use the Isolate Layers command to isolate Layers by hiding Layers. You cannot edit entities on these Layers.



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- Activate layer: You can activate a layer by specifying an entity on that layer.
- **Entity to Activate Layer**: This will change the Layer of the selected entity to the active layer.
- Change Entity's Layer: This will change the Layer of selected entities to that of a selected entity.
- **Layer Delete**: Deletes the layer of a selected entity.
- **Restore Layer's State**: This will undo previous changes made to layer settings.

Click **Format**, **Layer** from the Main Menu to display the Layers Manager dialog box.

Click **Format**, **Layer Tools** from the Main menu to display the available Layer tools.



Properties Toolbar

NOTE:

The Properties toolbar provides the following menu options:

- LineColor. You can assign LineColors to drawing entities directly or through Layers. The LineColor command lets you determine the color of new entities.
- **LineStyle**. Use the **LineStyle** command to load and set LineStyles. Choose a LineStyle to display on the active segment of the drawing.



ByLayer	Solid line 💙
ByBlock	
ByLayer	Solid line
Continuo	usSolid line
Other	

DraftSight

 LineWeight. Determines that the entity inherits the LineWeight associated with its layer. LineWeight defines how thick or thin an entity appears. The LineWeight of entities does not change when you zoom in or out.

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The default layer is called 0. It is automatically generated by the program when a new drawing is created.

Mouse Buttons

NOTE

It is important to understand the basic functions of the mouse buttons. It is recommended that you use a three button mouse with DraftSight.

- Left mouse button: The left-mouse button is used to select menus and icons, or a graphic entity. A single click is used to select icons, menus and entities.
- Middle mouse button (scroll wheel): The middle-mouse button is used to zoom in or zoom out in the graphics area. Also, pressing the scroll wheel will allow you to pan the drawing.
- Right mouse button: The right-mouse button is used to bring up the Context menu in the graphics area or additional available options during the execution of a command.

Repeat TOOLBARS	
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Command window

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Cancel or End Command

The **Esc** key is used to cancel the command in the Command prompt. For some commands, the **Esc** key is used to exit the command. If a command is active, you can also right-click in the graphics area and click either **Enter** or **Cancel** from the Context menu.

Enter: Confirms default options of the command or ends entity selection.

Cancel: Ends a command.

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Help

DraftSight has a comprehensive help function that is designed to assist you. The DraftSight help screen is displayed in its own window with two menu tabs. Click **Help**, **Help** a from the Main Menu to view the comprehensive

DraftSight help screen. You can also activate the help menu by pressing the < F1 > key on the keyboard.

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Properties

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Specify Toolbars

Toolbars:

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Manipulating Toolbars

Move

You can reposition toolbars and set visibility. Each toolbar has a drag

handle. Drag and drop toolbars to convenient positions in the application window. You can also dock them below the Main Menu or at the left and right margins of the application desktop.

€ 2

Display

Toolbars are available for most DraftSight tools. Named toolbars assist you in performing specific drawing tasks.

Only the most frequently used toolbars are displayed by default.

Right-click the Main Menu or a docked toolbar. Click **Toolbars**. The **Select Toolbars** dialog box is displayed.

Check or uncheck the desired toolbars to be displayed.

Click the **Customize** button to display the **Customize Commands** dialog box. This box provides the ability to customize a toolbar, or to create your own custom toolbar.





Modify

Properties
 Command window

Lighting ✓ Home

Tutorial

MAIN

Lock Location

Toolbars... Customize...

Options Toolbar References

*

Tools

Lesson 1 QUICK START

When you complete this lesson, you will be able to:

- Create a new drawing
- Set the drawing Unit System and Drawing Boundary
- Set the Snap and Grid
- Utilize the Line, Arc, and Circle Draw tools
- Utilize the Save, Save As, and Open a drawing command
- Apply the Window Select and Cross Selection methods
- Delete a single entity and multiple entities
- Manipulate EGrips in an entity
- Move a single entity and multiple entities
- Understand relative and absolute coordinates
- Use the Command window, Command prompt and Status bar
- Print / Plot a drawing
- Apply the Pan and Zoom commands
- Correct drafting mistakes

Quick Start

CAD systems in general create designs using basic geometric entities. Many of the constructions used in designs are based on two-dimensional planar geometry. In this section, you will use the **Line**, **Arc**, and **Circle** Draw tools to create geometric entities.

Straight Lines and Circles are used in most technical designs. Triangles and polygons are planar figures bounded by Lines. Splines and Ellipses can be created by connecting arcs with different radii.

Start a DraftSight Session



1 Start a DraftSight Session. From the Windows Start menu, click Start, All Programs, Dassault Systemes, DraftSight.

If you created the DraftSight icon on your desktop, double-click the icon to start the DraftSight session.

Creating a New Drawing

2 Create a New Drawing

Click the **New** icon **i** on the Standard toolbar. The **Select Template** dialog box is displayed.



Drawing Template

TIP:

Drawing templates act as a basis for your finished work and set up the required drawing environment. Using templates saves time because you can reuse the elements and settings of an already existing drawing.

3 Select a Default Drawing Template.

Select standard.dwt from the Specify Template dialog box.

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4 Click Open.

Set the Drawing Environment

Set **System Units**, drawing **Boundary**, **Snap** and **Grid**. These settings are stored in your default drawing template.

Set Drawing Unit System

The **Unit System** section of the **Options - Drafting Settings** dialog box, provides the ability to set: **Base Angle**, **Length Type** and **Precision**, **Angle Type** and **Precision**, as well as **Units scale** for the drawing.

Five different units systems are available:

- Scientific
- Decimal
- Engineering
- Architectural
- Fractional

The default units system is the Decimal system and can be applied to either English (inches) or metric values (millimeters).

5 Set Linear Length Precision.

The **Unit System** section of the **Options** - **Drawing Settings** dialog box allows you to set the linear and angular unit formats.

Click **Format**, **Unit System...** from the Main Menu. The **Unit System** portion of the **Options** - **Drawing Settings** dialog box is displayed.

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Click **0.00** for **Precision** in the **Length** section of the dialog box.

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6 Set Linear Units Scale.

The **Units scale** section of the **Unit System** dialog box allows you to set the linear and angular unit formats.

Click **Millimeters** for **Block units format** in the **Units scale** section of the dialog box.

Click OK.

Set Drawing Boundary

The Drawing Boundary defines the extent of the grid display and restricts the graphics area. The Drawing Boundary can be helpful for printing/plotting; if you set your graphics area so that it scales directly to the default drawing sheet.

Drawing Boundary does not affect the display on the screen. Boundaries are expressed as a pair of Points in the World Coordinate System (WCS), a lower left and an upper right limit.

For example, to set limits for an A4 (metric) vertical sheet size, set the lower left as 0,0 and upper right as 210,297.

Usually drawing boundaries are relative to the paper size to be used when printing/ plotting a drawing.

Default drawing sheet sizes - Millimeters are:

- A4 210 x 297
- A3 297 x 420
- A2 420 x 594
- A1 594 x 841
- A0 841 x 1189

Default drawing sheet sizes - Inches are:

- A 8.5 x 11.0
- B 11.0 x 17.0
- C 17.0 x 22.0
- D 22.0 x 34.0
- E 34.0 x 44.0

In the next section, set the drawing boundary to use a horizontal A4 default drawing sheet.

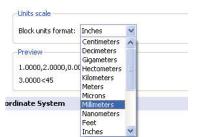
7 Set Drawing Boundary.

Click Format, Drawing Boundary from the Main Menu.

Default: (0.00,0.00) Options: <u>OF</u>f, <u>ON</u> or Specify lower left corner» ↓

Default: (12.00,9.00) Specify upper right corner» **297,210**,





NOTE: Press the **Enter** key to accept the default command, complete the command or continue to the next step. Press the **Enter** key again to repeat the command

Use the command window to view the transcript of the commands entered, so you can read or reconstruct your working steps, including keyboard inputs.

Command window

Default: (0.00,0.00) Options: <u>OF</u>f, <u>ON</u> or Specify lower left corner»

Specify lower left corner» '_DRAWINGBOUNDS

Set Snap and Grid

TIP:

The Snap is an invisible grid in the graphics area. With Snap activated, the cursor can only select Points positioned directly on the snap grid. When you Point to the drawing, startpoints, endpoints, center points, and other specific Points are exactly on a Point of the snap grid.

The Grid is a pattern of evenly spaced dots that serve as a visual distance reference. The Grid is not part of the drawing file and does not appear in the printed output.

You can set the Grid to span the graphics area or limit it to the extents of the drawing

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The Snap and Grid follow the axis of the coordinate system.

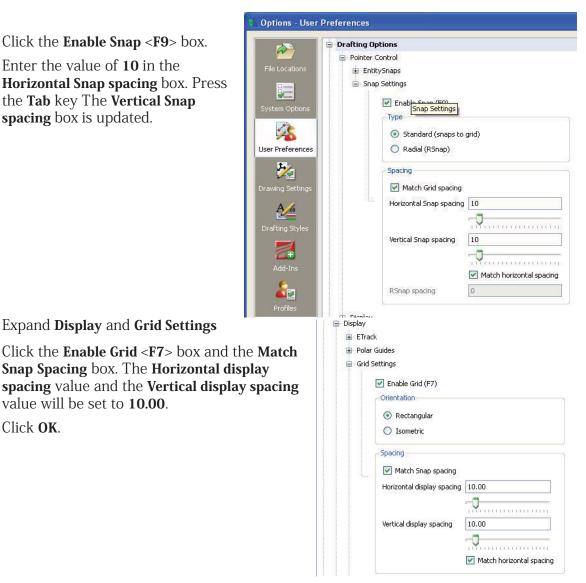
8 Set Snap and Grid.

NOTE:

Click **Tools**, **Options** from the Main Menu.

The **Options** dialog box is displayed.

Click the **User Preferences** tab. Expand **Drafting Options**, **Pointer Control** and **Snap Settings**.



Zoom Bounds

The **Zoom Bounds** (a) tool provides the ability to adjust the display so that all entities in the drawing are displayed to be as large as possible.

9 Display the Drawing with the New Boundary. Drawing boundaries need to be used to adjusted to the view.

Select **View, Zoom, Bounds** if from the Main Menu.



NOTE: You can also select the **Zoom Bounds** icon on the Zoom toolbar if it is active, or type the following at the command prompt:

: Zoom.⊣

Default: Dynamic

Options: <u>B</u>ounds, <u>C</u>enter, <u>D</u>ynamic, <u>F</u>it, <u>P</u>revious, <u>SE</u>lected, *specify a scale factor (nX or nXP)* or

Specify first corner>> $B \lrcorner \lrcorner$

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Line Command

The Line Scommand provides the ability to draw a straight Line segment between two defined Points. Each segment in a series of connected Lines is a separate entity. This book only focuses on the 2D drawing abilities of DraftSight.

There are four ways to define the length and position of a Line:

- 1. Randomly select Points.
- 2. Set a specific value for the Snap function and select Points using the Snap spacing values.
- 3. Enter the absolute coordinate values for the start and endpoints.
- 4. Enter the relative coordinate values and specify the starting Point and the length and direction of the Line.

In the next section, use two dimensional (X,Y) coordinates, and apply the ${\bf Line}$ - ${\bf Snap}$ Point method.

10 Apply the Line Command.

Click the Line icon 🔨 on the Draw toolbar.

Options: <u>S</u>egments or

Specify start point>>

Click a **grid point** six grid spaces up and six grid spaces to the right of the origin.

Options: <u>S</u>egments, <u>U</u>ndo, *Enter to exit* or

Specify next point>>

Click a grid point horizontally to the right six grid spaces.

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Options: <u>S</u>egments, <u>U</u>ndo, <u>C</u>lose, *Enter to exit* or Specify next point>>

Click a grid point vertically upward one grid space.

Options: <u>S</u>egments, <u>U</u>ndo, <u>C</u>lose, *Enter to exit* or Specify next point>>

QUICK START

Click a grid point horizontally to the right two grid spaces.

Options: <u>Segments</u>, <u>Undo</u>, <u>C</u>lose, *Enter to exit* or Specify next point>> Click a **grid point** vertically upward four grid spaces.

Options: <u>Segments</u>, <u>Undo</u>, <u>C</u>lose, *Enter to exit* or Specify next point>> Click a **grid point** horizontally to the left two grid spaces.

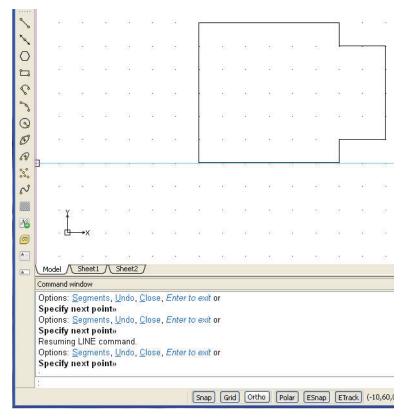
Options: <u>S</u>egments, <u>U</u>ndo, <u>C</u>lose, *Enter to exit* or Specify next point>>

Click a **grid point** vertically upward one grid space.

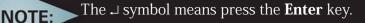
Options: <u>Segments</u>, <u>Undo</u>, <u>C</u>lose, *Enter to exit* or Specify next point>> Click a **grid point** horizontally to the left six grid spaces.

Options: <u>Segments</u>, <u>Undo</u>, <u>C</u>lose, *Enter to exit* or Specify next point>> Click the start **grid point**.





Options: <u>S</u>egments, <u>U</u>ndo, <u>C</u>lose, *Enter to exit* or Specify next point>>↓



To close the Line you can also right-click and select Enter.



To undo a Line segment, press **Ctrl + Z** or type **Undo** at the Command Prompt.

11 Save the Drawing.

Click the **Save** icon **I** on the Standard toolbar.

File name:	lesson 1-1	*	Save
Save as type:	R2007-2009 Drawing (*.dwg)	~	Cancel

Enter file name: Lesson 1-1.

Click Save.

12 Hide the Grid.

You can hide or show the grid.



Click the **Grid** button on the Status bar.

<Grid off> appears in the command window.

13 Show the Grid.

Click the Grid button from the Status bar.

<Grid on> appears in the command window.

Note:

Snap

Also you can choose to use the function keys.

Use $<\!\!F7\!\!>$ for the Grid toggle and $<\!\!F9\!\!>$ for the Snap toggle.

14 Close the Active Drawing.

Click File, Close from the Main Menu.

Open an Existing Drawing

1 Open a Drawing.

Click the **Open** icon **(a)** on the Standard toolbar.

Select Lesson 1-1 from the list.

Click Open.

Delete Command

Single Entity

The **Delete** command, located on the Modify toolbar, removes a selected entity while creating, editing, or detailing your drawing.

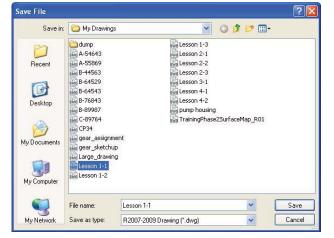
2 Delete an Entity.

Click the **Delete** icon *Q* on the Modify toolbar.

Specify entities>>

1 found, 1 total

Click the right Vertical Line in the drawing. The selected Line is highlighted by turning dotted. 



Speci fy entities>>.J

You can also reverse the order of operation by clicking the Line first and displaying the EGrips, then choosing the Delete icon.

Arc Command

NOTE:

The Arc S command provides the ability to construct arcs of any length or radius. You can also append Arcs to other Line, PolyLine, or Arc entities. All Arcs are drawn from the start Point in the positive horizontal direction.

The available Arc commands are provided later in this section.

3 Draw a Center Point Arc.

Click the Arc icon 🕥 on the Draw toolbar.

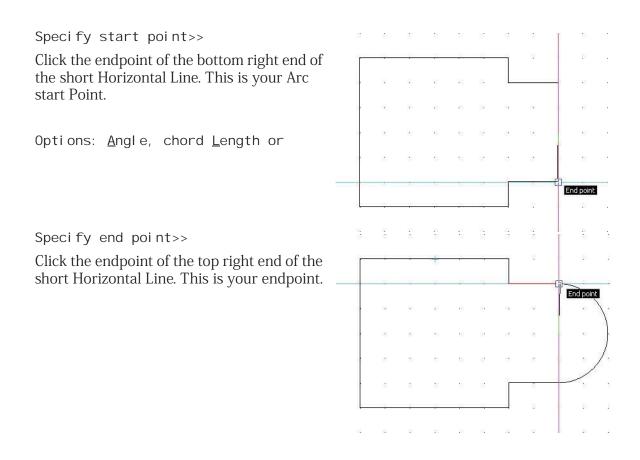
Options: <u>C</u>enter or

Specify start point>> \mathbf{c} ,

Specify center point>>

Click a grid point centered between the two Horizontal Lines.

	35	12	2	12	88	8	10	12	81	
22		- 22	4	÷	22	52	Ĩ	2 9	75	
10	ŀ	99.	12) 12)	a.	15	20	5		90	
85		64	19	1	10	85	84	19. IG	22	
ii -	-	<u>G</u>	÷	æ	τ <u>ε</u>	ti.	- Tel		0	
12	ł	22	<i>3</i> 2	5	12	12		82 A.I	25	
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8	_	-	4	÷	22	2		22 SI	75	
8	1	72	a.	12	20	29	62	12 1	40	



NOTE: If Auto-Save is on, DraftSight will save a back-up file of your drawing.

Circle Command

The **Circle (a)** command provides the ability to use several methods to create Circles of any size. The available Circle commands are provided below.

Circles may be defined by:

- A center point and either a radius or diameter.
- Two or three Points.
- Defining the tangents to two existing Lines or Arcs and then defining a radius value.
- Defining three tangent entities.

Command window

:_CIRCLE Options: <u>3Point</u>, <u>2Point</u>, <u>Ttr</u>, <u>TTT</u>, *Enter to exit* or **Specify center point**» Options: <u>Diameter or</u> **Specify radius**»

QUICK START

DraftSight

4 Draw a Circle.

4	Click the Circle icon (a) on the Draw toolbar.	8	12	13	8	81	85	35	81	8	8	10
	on the blaw toolbar.	20	×						1	a 1	1	10
	Options: <u>3P</u> oint, <u>2P</u> oint, <u>T</u> tr, <u>TTT</u> ,	28		32	25	10	10	84			<u> </u>	
	<i>Enter to exit</i> or	65		9 x	14	e.	19	8	14	9 B	80 10	$\sum_{i=1}^{n}$
	Specify center point>>	51		10	.tc	đa	tin.	i.	63	q] .	
	Click the center point of the Arc.	52		22	15	55		82		a 1]
		22		2	77	10	12	32	34			12
		10	5							s a 1	1 13	20
	Options: <u>D</u> iameter or	15	62	72	2	ās.	20	81	62	12	- 20	25
	Specify radius>>	82	82	12	10	81	85	35	88	10 10	2	8
	Click one grid point horizontally to the right.	10	8							55 B	e - 11	2
		13		59.			25	84		<u> </u>	-	25
		65		0 x	335	82	89	29	88			1.
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		52		2	15	55		82			1].
		12		2	27		12	101	3 <u>1</u>		_/	2
		20	_							5 S		22
5	Save the Drawing. Click File, Save As from the Main Menu.							Fi	New	0	nsert Fo	ormat
	Enter File name: Lesson 1-2.								Open.	8		
	Click Save.								Save			
									Save A Export			

Use the **Save As...** command to save the drawing under a different name or in a different location

NOTE:

DraftSight

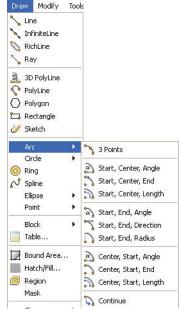
Besides using the Draw toolbar to create an Arc or Circle, you can also select various Draw commands from the drop-down menu in the Main Menu. The available **Circle** commands are:

- **Center, Radius:** click the graphics area to define the center point of the Circle. Click the graphics area to define the Circle radius, or enter the radius in the command window.
- **Center, Diameter:** click the graphics area to define the center point of the Circle. Click the graphics area to define the Circle diameter, or enter the diameter in the command window.
- **2 Points:** click two Points in the graphics area that define the Circle diameter.
- **3 Points:** click three Points in the graphics area that define Points on the Circle.
- **Tangent, Tangent, Radius:** click a Point on each of two entities that are tangent to the Circle to be created in the graphics area. Click a Point in the graphics area to define the radius or enter the radius at the command window.
- **Tangent, Tangent, Tangent:** click a Point in the graphics area on each of three entities that are tangent to the Circle to be created in the graphics area. This option is useful when inscribing the Circle within a regular polygon.

When you specify a point coordinate as the diameter of a circle, DraftSight calculates the distance from the center of the circle to the specified point.

The available Arc commands are:

- **3 Points:** click the graphics area to define the start point of the Arc. Click the endpoint to define the Arc.
- Start, Center, Angle: click the start point and center point in the graphics area, and then enter the value of the Arc tangent angle from its chord.
- **Start, Center, End:** click the start point, the center point, and the endpoint in the graphics area.
- Start, Center, Length: click the start point and center point in the graphics area, and then enter the value of the Arc chord length. Click the required Points in the graphics area or enter the option values at the command line prompt.
- Start, End, Angle: click the start point and endpoint in the graphics area, and enter the positive or negative value of the angle from the chord to the tangent to the start point.



• Start, End Direction: click the start point, endpoint, and tangent direction from the start point in the graphics area. Click the required Points

Draw	Modify	Tools	
	ifiniteLine ichLine		
Po Po) PolyLine blyLine blygon ectangle ketch		
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Ci	rcle	•	🔦 Center, Radius
Sp El	ng bline lipse bint	•	Center, Diameter 2 Points 3 Points
	ock able	•	 Tangent, Tangent, Radius ₹ Tangent, Tangent, Tangent

NOTE:

in the graphics area or enter the option values at the Command Line prompt.

- Start, End, Radius: click the start point and endpoint in the graphics area, and then enter the value of the Arc tangent angle from its chord.
- **Center, Start, Angle:** similar to the Start, Center, Angle command except the first Point selected is the center point or the Arc rather than the start point.
- **Center, Start, End:** similar to the Start, Center, End command except the first Point selected is the center point of the Arc rather than the start point.
- **Center, Start, Length:** similar to the Start, Center, Length command except the first Point selected is the center point of the Arc rather than the start point.
- **Continue:** click a Line, PolyLine, or Arc entity in the graphics area. Click the Arc endpoint in the graphics area.

Entity Selection

Many of DraftSight's modify and drafting commands prompt you to select one or more entities. When you select one or more entities, DraftSight highlights them by displaying dashed Lines (if no command is active, the Lines will also show the EGrips (blue blocks). There are many different ways of selecting entities. We will discuss and apply:

Window Select and Crossing select.

Window Select

The Window select method allows you to select more than one drawing entity in a defined area at a time. You define this area by specifying two corner points. Only the entities that lie completely within the rectangle will be selected.

6 Window Select Entities.

Click a position above the top Horizontal Line. Speci fy opposi te corner: Drag downward and to the right to create a rectangular area as illustrated.

20

Click a position below the bottom Horizontal Line to the right of the Arc. The selected entities inside the window are displayed with dashed Lines and blue squares (EGrips).

7 Undo the Window Selection. Press Esc or Enter to exit.

Crossing Select

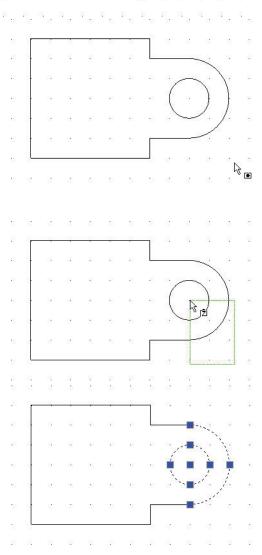
The Crossing select method allows you to select all entities that are contained completely or partially within the rectangular area. You define this area by specifying two corner points.

8 Window Cross Select Entities

Click a position below the bottom Horizontal Line to the right of the Arc.

Specify opposite corner:

Drag upward and to the center point of the Circle.



Click the center point of the Circle. All entities that are inside the window and all entities that crosses are selected window and are displayed with dashed Lines and blue squares.

9 Undo the Selection.

Press Esc or Enter to exit.

Delete Command

Multiple Entities

The Delete *command* provides the ability to remove selected entities while creating, editing, or detailing drawings.

10 Turn Snap Off.

Click the **Snap** button from the Status bar. The command window displays:

<Snap Off>.

Snap Grid Ortho Polar ESnap ETrack

N	With Snap on, the cursor can only select Po dierectly on the Snap Grid.	oints j	pos	itic	one	d							
1	Click Delete on the Modify toolbar. Specify entities>>												
2	Click the circumference of the Arc and Circle in the drawing. The selected Line becomes highlighted by turning dotted		-										
	1 found, 2 total		L				2		2		8	2	* *
	Specify entities>>↓		9. 27 - 3					20 27		8	1	а В 1	8
	The entities are removed.					20 52 20			н н н				
3	Save the Drawing. Click the Save icon 📄 on the Standard toolbar.					2 2	е н т		10 10	-	12 68 25	е и н и н и	

Move Command

The **Move** P command repositions drawing entities within the coordinate system without changing the orientation or size.

Multiple Entites

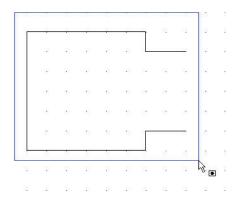
1 Move Multiple Entities.

Click the **Move** icon 🗗 on the Modify toolbar.

Specify entities>>

Window select the drawing entities. Click a position above the top Horizontal Line to the left of the Vertical Line.

Specify opposite corner: Drag downward and to the right.



QUICK START

	Click a position below the bottom Horizontal Line. The selected entities inside the window are displayed.					1 			2 3 7 8 8.	2 11		ATT 100 100
	7 found, 7 total	•			к) к)		20 20	а , к. (6 69 6 69		6 22	10
	Specify entities» , Default: <u>D</u> isplacement: Specify from point>>						5 5 5 11		12 22 14 12 25 15 25			AC 10 00
	Click the lower left corner end point. Options: <i>Enter from point as displacement</i> or Specify destination>>									8 33 8 8		20 20 10 20 20
	Move three grid units to the right. Move three grid units upward.	: :: (3 En:	d point				 	2 22 10 - 21	13 	3 10	8
	Click a position to move all selected entities.				85	8	0	15				0 2
Single	Entity.		·									2
1	Select a Single Entity.				72 75 75			22 22 51	i			
			2	Ø			•			2		5
	Click the bottom Horizontal Line.					10 a	1		1 ²⁵ 1	n N		
	Click the Move icon P on the Modify toolbar.					1 S 1 1				33		10 20 E
	Default: <u>D</u> isplacement Options: <u>D</u> isplacement or Specify from point>>									2. 2. 2.		
								Ι.				•

Click two grid units below the mid point of the bottom Horizontal Line. Options: Enter from point as displacement or Specify destination>> Move the selected horizontal linear segment three grid units downward as illustrated. _dh Click a position to move the selected entity. 2 Save the Drawing. Click the **Save** icon 🔲 on the Standard toolbar. 3 Close the Drawing. Click File, Close from the Main Menu. L.....

To use the **Undo** command, click the Undo icon on the Standard toolbar or type **U** at the command prompt.

Use the **Undo** command to withdraw the last command. You can repeat undo until all steps in the undo list have been canceled, indicated by the message Nothing to undo in the Command prompt.

Creating a New Drawing

NOTE

TIP:

1 Create a New Drawing

Click the **New** icon **(a)** on the Standard toolbar. The **Select Template** dialog box is displayed

	8 Q	» G	ß	1	5
New 🗢 😘	0 0				

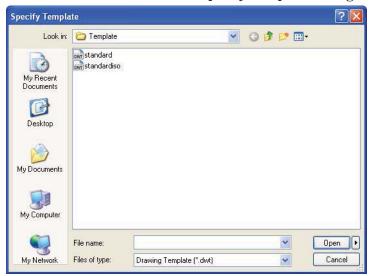
R

You can also click **File**, **New** from the Main Menu to create a new drawing.

Drawing Template

Drawing templates act as a basis for your finished work and set up the required drawing environment. Using templates saves time because you can reuse the elements and settings of an already existing drawing.

2 Select a Default Drawing Template. Select standard.dwt from the Specify Template dialog box.



3 Click Open.



Set the Drawing Environment

Set **System Units**, drawing **Boundary**, **Snap** and **Grid**. These settings are stored in your default drawing template.

Set Drawing Unit System

The Unit System section of the Options - Drafting Settings dialog box, provides the ability to set: Base Angle, Length Type and Precision, Angle Type and Precision, as well as Units scale for the drawing.

Five different units systems are available:

- Scientific
- Decimal
- Engineering
- Architectural
- Fractional

The default units system is the Decimal system and can be applied to either English (inches) or metric values (millimeters).



4 Set Linear Length Precision.

The **Unit System** section of the **Options** - **Drawing Settings** dialog box allows you to set the linear and angular unit formats.

Click Format, Unit System... from the Main Menu.

The Unit System portion of the Options - Drawing Settings dialog box is displayed.

2	 Behavior Display 	
ile Locations	Points Unit System	
rstem Options	Base angle Length Type: Decimal V Precision: 0.000 V 0.00 0.0	
awing Settings	Angle 0.0000 0.0000 0.0000 0 0.00000 0 0.000000 0 0.0000000 0 0.0000000 0 0.0000000 0 0.0000000	
Add-Ins	Block units format: Millimeters Preview 1,2,0 3<45	
	Coordinate System	

Click **0.00** for **Precision** in the **Length** section of the dialog box.

5 Set Linear Units Scale.

The **Units scale** section of the **Unit System** dialog box allows you to set the linear and angular unit formats.

Click **Millimeters** for **Block units format** in the **Units scale** section of the dialog box.

Units scale
Block units format: Inches
Centimeters
Cen

Click OK.

Set Drawing Boundary

The Drawing Boundary defines the extent of the grid display and restricts the graphics area. The Drawing Boundary can be helpful for printing/plotting; if you set your graphics area so that it scales directly to the default drawing sheet.

Drawing Boundary does not affect the display on the screen. Boundaries are expressed as a pair of Points in the World Coordinate System (WCS), a lower left and an upper right limit.

For example, to set limits for an A4 (metric) vertical sheet size, set the lower left as 0,0 and upper right as 210,297.

Usually drawing boundaries are relative to the paper size to be used when printing/ plotting a drawing.

Default drawing sheet sizes - Millimeters are:

- A4 210 x 297
- A3 297 x 420
- A2 420 x 594
- A1 594 x 841
- A0 841 x 1189

Default drawing sheet sizes - Inches are:

- A 8.5 x 11.0
- B 11.0 x 17.0
- C 17.0 x 22.0
- D 22.0 x 34.0
- E 34.0 x 44.0

In the next section, set the drawing boundary to use a horizontal A4 default drawing sheet.

6 Set Drawing Boundary. Click Format, Drawing Boundary from the Menu bar.



Default: (0.00,0.00) Options: <u>OF</u>f, <u>ON</u> or Specify lower left corner» ↓

Command window Specify lower left corner» '_DRAWINGBOUNDS

Default: (0.00,0.00) Options: <u>OF</u>f, <u>ON</u> or **Specify lower left corner**»

Default: (12.00,9.00) Specify upper right corner» **297,210**,

Set Snap and Grid

The Snap is an invisible grid in the graphics area. With Snap activated, the cursor can only select Points positioned directly on the snap grid. When you Point to the drawing, startpoints, endpoints, center points, and other specific Points are exactly on a Point of the snap grid.

The Grid is a pattern of evenly spaced dots that serve as a visual distance reference. The Grid is not part of the drawing file and does not appear in the printed output.

The Snap and grid follow the axis of the coordinate system.

You can set the Grid to span the graphics area or limit it to the extents of the drawing.

Too	İs	Window	Help
-	Pre	operties	Ctrl+1
è	Re	ferences	
	Dis	play Order	÷
	In	quiry	•
	Op	en Referenc	e
	Ru	n Script	
厚	СС	S Manager	
	Ne	w CCS	•
	Cu	stomize Inter	rface
	Op	tions	

NOTE: The shap and grid follow the axis of the coordina

7 Set Snap and Grid.

Click Tools, Options from the Main Menu.

The **Options** dialog box is displayed.

Click the User Preferences tab.

Expand Drafting Options, Pointer Control and Snap Settings.

Click the **Enable Snap** <**F9**> box.

Enter the value of **10** in the **Horizontal Snap spacing** box. Press the Tab key The **Vertical Snap spacing** box is updated.



Expand Display and Grid Settings

Click the **Enable Grid** <**F7**> box and the **Match Snap Spacing** box. The **Horizontal display spacing** value and the **Vertical display spacing** value will be set to **10.00**.

Click OK.

Display		
🗄 ETrack		
😟 Polar Gu	iides	
😑 Grid Set	tings	
	Enable Grid (F7)	
C'	Orientation	
	💿 Rectangular	
	🔘 Isometric	
ja ka	Spacing	
	🗹 Match Snap spacing	
	Horizontal display spacing	10.00
	Vertical display spacing	10.00
		-0
		Match horizontal spacing

Zoom Bounds

The **Zoom Bounds** (a) tool provides the ability to adjust the display so that all entities in the drawing are displayed to be as large as possible.

8 Display the Drawing with the New Boundary. Drawing boundaries need to be used to adjusted to the view.

Select View, Zoom, Bounds 🔍 from the Main Menu.

Absolute and Relative Coordinates

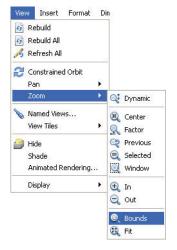
In this manual, we will only deal with the 2D functionality of DraftSight.

All drawings are based on a Cartesian coordinate system in which two perpendicular axes are used: the X and Y axis. All axis originate in the origin Point of the coordinate system. The X axis and Y axis define a horizontal plane.

In the Cartesian coordinate format, a Point is defined by its distances in the XY plane. These distances are called the XY coordinates of a Point. When you draw in two dimensions, you specify Points only on the XY plane. In CAD, the fixed Cartesian coordinate system is called the World Coordinate System (WCS). New drawings are based in the WCS.

Enter coordinates as absolute or relative values.

To specify the Point coordinates with respect to the previous Point (relative coordinate input), type the @ character in front of the coordinate values.



Absolute Coordinates

To specify absolute coordinates:

Type of Coordinate	Input	Example
Cartesian	Х,Ү	30,60
Polar	distance <angle< td=""><td>75<45</td></angle<>	75<45

Enter a command requiring coordinates, for example:

:Line.∟

Specify the X and Y coordinates separated by a comma. 30,60

Specify subsequent coordinates to complete the command. 35, 82

Relative Coordinates

Type of Coordinate	Input	Example
Cartesian	@deltaX,deltaY	@30,60
Polar	@relative_distance <angle< td=""><td>@75<45</td></angle<>	@75<45

To specify relative coordinates: Enter a command requiring coordinates. Specify the first Point. **0,0**

To specify the second Point with respect to the first Point, type the @ character in front of the coordinate values. @30,60

Entering Keyboard Input

When you enter data using the keyboard, your data is precise. With keyboard input, you can construct drawing entities with exact dimensions or with exact placement in the coordinate system.

Values for distances, straight Lines, and other linear quantities are specified in drawing units. Angle values are specified in degrees.

Methods and syntax to use in this book:

- Interactive method: use the cursor to select on the graphics area
- Absolute coordinate (Format: X, Y) method: enter the X and Y coordinates to locate the Point on the active coordinate system relative to the origin.
- Relative rectangular coordinates (Format: @X,Y) method: enter the X and Y coordinates relative to the last Point
- Direct Distance method: specify the second Point by moving the cursor to indicate direction. Enter distance.

NOTE:

- Relative polar coordinates (Format: @Distance<angle) method: enter a distance and angle relative to the last Point.
- Periods: enter periods for decimal points.
- Commas: enter commas to separate the values for the axes, for example 0,2.1
- < (less than): use the < (less than) sign before angle values, for example, <30 represents a 30 degree angle.

You can type the Command name **Line**, or click the Line icon on the Draw toolbar.

```
Example of an input sequence using the keyboard:
```

: **Line**↓ Options: <u>S</u>egments or Specify start point>> **40,40**↓

Options: <u>Segments</u>, <u>Undo</u>, <u>Enter to exit</u> or Specify next point>> **120,40** \downarrow

Options: <u>S</u>egments, <u>U</u>ndo, *Enter to exit* or Specify next point>> **120,80** ↓

Options: Segments, Undo, Enter to exit or Specify next point>> \downarrow

Command	window
: line	
Options:	Segments or
Specify	start point» 40,40
	Segments, Undo, Enter to exit or
Specify	next point» 120.80
Options:	Segments, Undo, Enter to exit or
Specify	next point»

NOTE: Commands in DraftSight are not case sensitive.

Draw a Rectangle Using Absolute Coordinates

Draw a 60,40 rectangle using the **Line** icon \searrow on the Draw toolbar starting at the coordinate **30,30** using absolute coordinates.



A rectangle drawn using the **Rectangle** command is considered to be a single entity.

If the same size rectangle were drawn using the **Line** command, the resulting rectangle will be considered four individual unrelated Lines.

9 Draw the First Shape.

Click the **Line** icon 🔪 on the Draw toolbar.

Enter coordinates in the Command prompt. View the command window area, near the bottom of the graphics area.

Options: Segment or Specify Start point>> 30,30, Options: Segments, Undo, Enter to exit or Specify next point>> 90,30, J Options: <u>Segments</u>, <u>U</u>ndo, Enter to exit or Model Sheet1 Sheet2 Specify next point>>90,70, J Command window _LINE Options: Segments, Enter to continue from last point or Specify start point» 30,30 Options: <u>Segments</u>, <u>Undo</u>, <u>Enter to exit</u> or Specify next point» 90,30 Options: Segments, Undo, Close, Enter to exit or Options: Segments, Undo, Enter to exit or Specify next point» 90,70 Specify next point>>30,70, Options: Segments, Undo, Close, Enter to exit or Specify next point» 30,70 Options: Segments, Undo, Close, Enter to exit or Specify next point» c Options: Segments, Undo, Close, Enter to exit or Specify next point>>c→

The **C** option in the command closes the profile and ends the command.

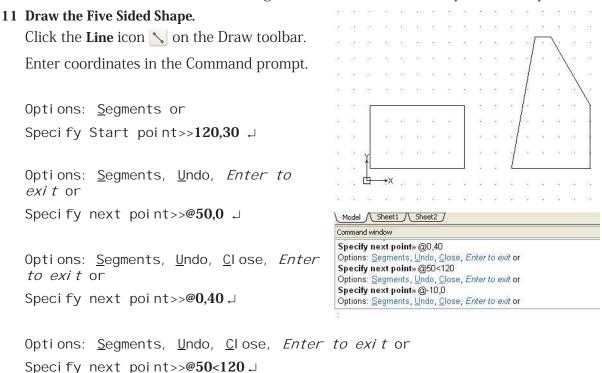
10 Save the Drawing.

Click **File**, **Save As...** from the Main Menu. Enter File name: Lesson 1-3. Click **Save**.

File	Edit	View	Insert	For
a •	lew			
	Dpen			
	lose			
	Save			
2	Save As	ine.		
E	Export			

Draw the Second Shape Using Relative Coordinates

Draw a five sided shape using the **Line** icon **S** on the Draw toolbar starting at the absolute coordinate **120,30** and using relative coordinates to complete the shape.



Options: <u>S</u>egments, <u>U</u>ndo, <u>C</u>lose, *Enter to exit* or Specify next point>>@-10,0 ↓

Options: Segments, Undo, Close, Enter to exit or Specify next point>> $c \downarrow$



If you enter an incorrect coordinate, press the **U** key to **Undo** the last coordinate in the command.

Draw the Third Shape Using Relative and Absolute Coordinates

Draw the third shape using the **Line** icon S on the Draw toolbar starting at the absolute coordinate **30,120** and using relative coordinates and absolute coordinates to complete the shape.

12 Draw the Third Shape.

Click the Line icon 🔨 on the Draw toolbar.

Enter coordinates at the Command prompt.

Options: <u>S</u>egments or Specify Start point>>**30,120** ↓

Options: <u>S</u>egments, <u>U</u>ndo, *Enter to exit* or Specify next point>>@**50,0** ↓

Options: <u>S</u>egments, <u>U</u>ndo, *C*lose, *Enter to exit* or Specify next point>>@**0,20** ↓

Options: <u>S</u>egments, <u>U</u>ndo, *C*lose, *Enter to exit* or

Specify next point>>@40<0.

Options: <u>S</u>egments, <u>U</u>ndo, *C*lose, *Enter to exit* or

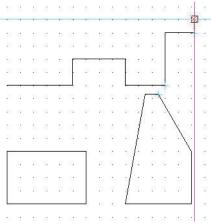
Specify next point>>@0,-20 \downarrow

Options: <u>S</u>egments, <u>U</u>ndo, *C*lose, *Enter to exit* or Specify next point>>@**30**<**0**.↓

Options: <u>S</u>egments, <u>U</u>ndo, *C*lose, *Enter to exit* or Specify next point>>@40<90.↓

Options: <u>S</u>egments, <u>U</u>ndo, *C*lose, *Enter to exit* or Specify next point>>@**20**<**0**.↓

Options: <u>S</u>egments, <u>U</u>ndo, *C*lose, *Enter to exit* or Specify next point>>@**30**<**90**.↓



Options: Segments, Undo, Close, Enter to exit or Specify next point>>@-40,0.↓ Options: Segments, Undo, Close, Enter to exit or Specify next point>>@-20,-20, Options: <u>S</u>egments, <u>U</u>ndo, *C*lose, Enter to exit or Specify next point>>@-20,20, Options: <u>Segments</u>, <u>U</u>ndo, *C*lose, Enter to exit or Specify next point>>@-25,0_ →X · 曲 Options: Segments, Undo, Close, Model / Sheet1 / Sheet2 / Enter to exit or Command window Specify next point>>c→

13 Save the Drawing.

Click the **Save** icon 🗟 on the Standard toolbar.

Print Command

The **Print** Geommand provides the ability to print or plot an entire drawing sheet, or just a selected area of the sheet. You have the option to print in black and white (the default) or in color.

There are no complicated printing commands. Click the **Print** icon 🔒 on the Standard toolbar or type **Print** at the Command prompt. The **Print** - **Model** dialog box is displayed.

Use the **Page Setup** section to set a previously defined print style or create a named print style based on what you set up in the dialog box.

Print - Model		
Page layout options Quick print Use previous settings Manual setup Printer/plotter Name: ((corp-printa)(HPLJ8100N-OM1) Printer: HP LaserJet 8100 Series PCL Location: OM, 1st Flr Notes: OM, 1st Flr Paper size	Properties	Preview 11.00° g g g g g g g g g g g g g
Letter Print scale Fit to paper size Scale: User-defined 1 Inches = 1.347595 units Scale LineWeight to print scale	Print range All geometry Drawing boundary Named view Specify Specify Window > Current view Print only area within specified window 	Additional Options Print Preview

Model Tab / Sheet Tabs

The DraftSight software provides the ability to work on your drawing in two different environments: Model or Sheet.

You can print your drawing from Model to any scale by specifying the print setup in the **Print** - **Model** dialog box.

Sheet, is a environment used for arranging various views of what was drawn in your Model. It represents the paper size which you set in the drawing environment.

	8	100
	-15	8
	82	12
Y	81	35
1	12	82
Ľy→X	8	82
Model Sheet1	Sheet2	J .
Command window		

You can apply different scales and specify different visibility for Layer in each floating ViewPort.

You can switch between Model and Sheet by selecting the appropriate tab provided at the bottom of the graphics area. Model can be accessed from the Model Tab. Selecting one of the available Sheet Tabs will access the sheet.

By default, two sheet tabs are created: Sheet1 and Sheet2. You can rename these and if necessary create up to 255 additional sheet tabs. The names of the sheet tabs must be unique.

- 14 Click the sheet1 tab This will open Sheet1
- **15 Right click on the Sheet1 name** This will open the Sheet Manager Dialog box.



16 Click the **Print...** option. Set your options.

Print - Sheet1		X
Page layout options Quick print Use previous settings Manual setup Printer/plotter Name: \(corp-printa\)+IPLJ4100N-Train1 Printer: HP LaserJet P4010_P4510 Series PC Location: Training dept by Dave Pancoast, 1st Notes: Training dept by Dave Pancoast, 1st	: Fir	Preview 11.00" Image: state sta
Paper size Letter	×	Additional Options Print Preview
Print scale Fit to paper size Scale: 1:1 I I I I I I I Scale LineWeight to print scale	Print range All geometry Sheet Named view Specify Specify Current view Print only area within specified window	✓ OK X Cancel W Help

Page setup options:

 Quick print: Prints exactly what is on screen with the system default printer, default paper size, Fit to paper size selected and Print range set to Current view.

Page layout options -
🔘 Quick print

Use previous settings
 Manual setup

- Use previous settings: Repeats printing with the same setting you used for the previous printout.
- Manual setup: Allows you to determine the print settings.

Printer/plotter:

 Name: Allows you to select an output device. Printers and plotters available on your system are listed. PDF, JPG, PNG and SVG are also listed. These built in plotters print to a file in the specified format.

Printer/plo	ccer	
Name:	None	Properties
Printer:	None	
Location:	Not applicable	
Notes:	The Sheet will not be printed unless a new printer is selected.	

 Properties: Allows you to set up the printer or plotter as needed in the dialog box provided by the printer's driver.

Select the **Help** button to explore the **Print**- **Model dialog** box in detail.

17 Select the Cancel button to return to the graphics area.

Pan Commands

TIP.

The most basic functions that change drawing views are the **Pan** and **Zoom** commands. The **Pan** tools move the visible portion of the drawing in a specified direction. The **Pan** tools do not change the position or the magnification of drawings entities.

۸ 🖧 🖶 🖨 🗛 🖈 🖒 🖊 5 4 Q Q 🖉 💿 🗢 🖕 🔿 o Dynamic Pan Repeat PRINT Insert Format Dimension Draw 🚱 Rebuild 5 6 % Cut Ctrl+X 🗿 Rebuild All Copy Ctrl+C 🦂 Refresh All 눩 Copy with Reference Point Ctrl+Shift+C 陷 Paste Ctrl+V 🔁 Constrained Orbit 💠 Dynamic S Undo Ctrl+Z Zoom 🔹 Point CH Redo Ctrl+Y 💠 Pan 💊 Named Views... 📀 Left View Tiles Q Zoom 🔶 Riaht 🔐 Hide 🐴 Up Options... Drafting Options.. Shade 📀 Down

Pan allows you to view entities previously not visible because they were out of the viewing area. When

panning, the scale of the view does not change, as it does when zooming.

```
NOTE: When you use the Pan tool, it as if the camera, not the scene is moving.
```

Access various **Pan** tools from the Main Menu, drop-down menu in the Standard toolbar, or right-click in the graphics area.

- **Dynamic Pan**: allows you dynamically pan by moving the mouse pointer.
- **Point Pan**: use point pan to move the view of the drawing in any direction by pointing a displacement (vector). the size of the view remains the same.
- You define the displacement by identifying two Points. these Points determine a vector giving the distance and direction of the pan.

- Left Pan: when you pan left, the visible part of the drawing moves left.
- **Right Pan**: when you pan right, the visible part of the drawing moves right.
- **Up Pan**: when you pan up, the visible part of the drawing moves up.
- **Down Pan**: when you pan down, the visible part of the drawing moves down.

18 Apply Dynamic Pan.

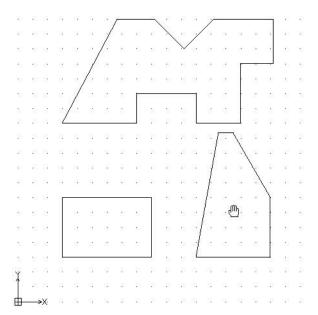
Click the **Dynamic Pan** icon 🔹 on the Main Menu. The Pan icon of a hand is displayed.

Press **Esc** or **Enter** to exit.

Position the **Pan** icon in the graphics area near the second drawing as shown.

Click and drag to the left.

Click and drag upwards.



The scroll bars on the right and bottom of the graphics area, when activated can also be used to adjust the viewing for the drawing.

19 Exit Dynamic Pan.

NOTE:

Press Esc or Enter to exit.

20 Close the Active Drawing. Click **File, Close** from the Main Menu.



NOTE: Dimensions are inseted in Lesson 4.

Correct Drafting Mistakes

DraftSight provides many tools to correct drafting mistakes. Correct mistakes using the following commands:

- Delete: deletes any entity from your drawing. Use the Delete command to remove selected entities while creating, editing, or detailing drawings. Click the Delete icon on the Modify toolbar or type Delete at the Command prompt.
- Undo: use the Undo command to withdraw the last command. You can repeat undo until all steps in the undo list have been canceled, indicated by the message Nothi ng to undo in the Command prompt. Click the Undo icon on the Standard toolbar or type U at the Command prompt.
- Redo: the Redo command reverses a previous Undo operation. You can use the Redo command only if no other command has changed the drawing database since the last Undo command. Click the Redo icon con the Standard toolbar or type Redo at the Command prompt.
- Undelete: use the Undelete command, like the Undo command, to restore entities that have been deleted by mistake. Also use Undelete to restore entities after creating a Block using the MakeBlock command. Type Undelete at the Command prompt.

Lesson 2 DRAFTING TUTORIALS

When you complete this lesson, you will be able to:

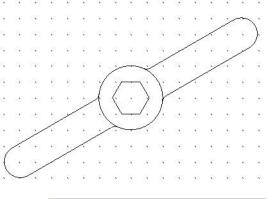
- Create three new drawings with multiple Layers
- Use absolute coordinates the command prompt and status bar
- Utilize the Line, Circle (R & Ttr), Polygon, Rectangle, SimpleNote, Spline, and PolyLine Draw commands
- Apply the Copy, Rotate, Move, Trim, Fillet, Mirror, Chamfer, Offset, Edit PolyLine, and Pattern (circular & linear) Modify commands.
- Create a new Layer with properties
- Manage Layers in a drawing
- Move Entities to a different Layer

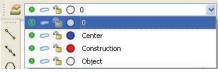
Drafting Tutorials

During this lesson, you will create three drawings by inputting absolute coordinates at the Command prompt and the graphics area.

In the first drawing you will:

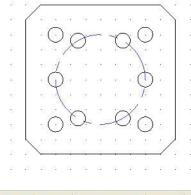
- Create three new Layers with custom properties: Object, Centerline, and Construction.
- Use the Circle, Polygon, Line, Rectangle, SimpleNote, Spline and PolyLine commands on the Draw toolbar.
- Apply the Copy, Rotate, Trim, Fillet, Mirror, and Edit PolyLine commands on the Modify toolbar,

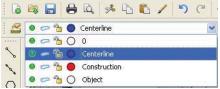




In the second drawing you will:

- Create three new Layers with custom properties: Object, Construction and Centerline.
- Use the **Rectangle**, and **Circle** commands on the Draw toolbar.
- Apply the **Pattern** command to create a circular and linear pattern along with the **Chamfer** command on the Modify toolbar.



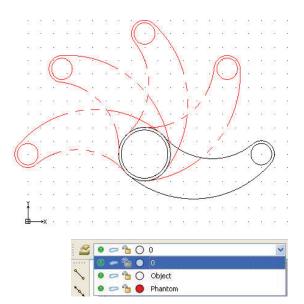


In the third drawing you will:

- Create two new Layers with custom properties: Object and Phantom.
- Use the **Circle** (**r**), and **Circle** (**Ttr**) commands on the Draw toolbar.
- Apply the Trim, Offset, and Pattern (circular) commands on the Modify toolbar.

Start a DraftSight Session

 Start a DraftSight session. From the Windows Start menu, click Start, All Programs, SolidWorks Corporation, DraftSight.



Create a New Drawing

2 Create a new drawing.

Click the **New** icon **a** on the Standard toolbar. The **Select Template** dialog box is displayed.

Drawing Template

- **3** Select a Default Drawing Template. Select standard.dwt from the Specify Template dialog box.
- 4 Click Open.

NOTE: Double-clicking on the template name will also open the file.

Set the Drawing Environment

Drawing Unit System

The default units system is the Decimal system and can be applied to either English (inches) or metric values (millimeters).

5 Set Linear Length Precision.

Click Format, Unit System... from the Main Menu.

The Unit System section of the Options - Drawing Settings dialog box is displayed.

Select 0.00 for Length Precision.

6 Set Linear Units Scale.

The **Unit System** section of the **Options** - **Drawing Settings** dialog box allows you to set the linear and angular unit formats.

Click **Millimeters** for **Block units format** in the **Units scale** section of the dialog box. Click **OK**.

Drawing Boundary

Use a horizontal A4 default drawing sheet.

1 Set Drawing Boundary. Click Format, Drawing Boundary from the Main Menu. Default: (0.00, 0.00) Options: <u>OF</u>f, <u>ON</u> or Specify lower left corner»,J

Default: (12.00,9.00) Specify upper right corner» **297,210** ↓

Set Snap and Grid

In this section the Grid equals the Snap.

1 Set Snap and Grid.

Click Tools, Options from the Main Menu.

The **Options** dialog box is displayed.

Click the **User Preferences** tab.

Expand Drafting Options, Pointer Control and Snap Settings.

Click the Enable Snap <F9> box.

Enter the value of **10** in the **Horizontal Snap spacing** box. Press the **Tab** key The **Vertical Snap spacing** box is updated.

Expand Display and Grid Settings.

Click the **Enable Grid** <**F7**> box and the **Match Snap Spacing** box. The **Horizontal display spacing** value and the **Vertical display spacing** value will be set to **10.00**. Click **OK**.

Zoom Bounds

2 Display the Drawing with the New Boundary.

Note: Click **View**, **Zoom**, **Bounds (a)** on the Main Menu.

NOTE: You can also select the **Zoom Bounds** icon on the Zoom toolbar if it is active or type the following at the command prompt:

Zoom.⊣

Default: Dynamic Options: <u>B</u>ounds, <u>C</u>enter, <u>D</u>ynamic, <u>F</u>it, <u>P</u>revious, <u>SE</u>lected, *specify a scale factor (nX or nXP)* or Specify first corner>> **B**.



DraftSight

Format Dimension Dra

🚄 Layer...

Layer Tools

Line Weight...

🞯 Line Color...

Print Style...

Dimension Style...
Point Style...

RichLine Styles... Table Style... Text Style... Drawing Boundary

Unit System...

🗐 Rename...

Working with Layers

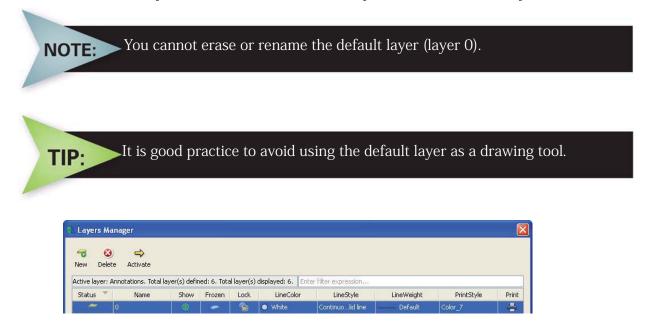
Working with Layers is analogous to drawing manually using Layers of see-through media. Parts or portions - these might be functional portions, such as Construction Lines, - of one complete drawing are drawn on different 'media' that, imposed on top of one another, form the entire drawing.

Layers are useful in separating entities, such as parts or sub-assemblies, or structural components, such as pipe sizes or building materials. They also serve to separate elements such as Construction Lines, Centerlines, Outlines, and Dimensions, or items that are to be placed in a group, such as foliage in an architectural drawing.

When you use Layers, you have a clearer construction of the drawing and better control over the drawing.

Layer 0: The Default Layer

The default Layer is called 0. It is automatically generated by DraftSight when a new drawing is created. This is your active Layer which is shown, un-suppressed and unlocked. The color 7 (white on a black background, or black on a white background) and the LineStyle Continuous are automatically established for this Layer.



Layer Properties

Each Layer has a **Status**, **Name**, **Show**, **Frozen**, **Lock**, **LineColor**, **LineStyle**, **LineWeight**, **PrintStyle Print**, and **Description** associated with it when it is created. Anything drawn on a Layer will be drawn with that Layer's properties.

There are other Layer properties that allow you to switch between different states.

These are Show/Hide, Frozen/UnFrozen, and Lock/Unlock.

The Layer and its properties are saved with the drawing.

Layer properties include:

- Status: one Layer in each drawing is the active Layer. In the Layer Properties Manager this is indicated in the Status column. All new entities are placed on this Layer.
- Name: each Layer has a distinctive name to identify it. Names can contain letters, numbers, or special characters like under scores and dollar signs.
- **Show/Hide**: An advantage of working with Layers

is that it aids in visualizing the part or drawing. You can improve the clarity of a drawing by choosing Layer properly and hiding drawing entities. The show icon indicates whether the Layer is shown or hidden. You can turn each Layer on or off independently.

- Frozen: These properties control the visibility and the influence of entities within the Layer. A Frozen Layer is invisible and protected and thus impossible to change. A Frozen Layer is not included when you regenerate a drawing. When working with complex drawings, freezing unneeded Layers can increase performance. When you un-freeze a Layer, it becomes visible and you can make changes to it. You cannot suppress the active Layer of a drawing.
- Lock/Unlock: You can lock Layers so the entities on the Layer cannot be changed. The locked state prevents unintentional modification. When a Layer is locked, you cannot change, add, or delete entities

from the Layer. You must change the status to unlocked if you want to work with the entities on this Layer. You can draw new entities on locked Layers, but it is not possible to work with them further. A locked Layer may be designated active, visible or invisible, or frozen or unfrozen.

- LineColor: When you start a new drawing, new Layers are automatically assigned the LineColor white. When you create a new Layer, the Layer adopts the LineColor of the new Layer. Entities you draw on the Layer appear in the assigned LineColor except those to which you assigned a LineColor with the LineColor command.
- LineStyle: A default LineStyle is assigned to a Line. If you do not choose another LineStyle, the LineStyle Continuous is automatically assigned. The chosen LineStyle is displayed in the Properties Palette.



Frozen	Lock 🔻	LineColor	LineStyle
-	8	😑 Red	CENTER
0	8	O White	Continuolid line
0	2	O White	Continuolid line



LineStyle	LineWeight	PrintStyle
Continuolid line -	—— Default	Color_7
Continuolid line -	Default	Color_7
CENTER	Default	Color_1
Continuolid line CENTER HIDDEN Other		

DRAFTING TUTORIALS

- LineWeight: You can set the LineWeight of the entities on the Layer. LineWeight affect the printed drawing only. The chosen LineWeight is displayed in the Properties Palette.
- **PrintStyle:** You can assign individual PrintStyles to Layers. The chosen PrintStyle is displayed in the Properties Palette.
- Print: You can turn Layers on or off for printing / plotting.
- **Description:** You can enter a Layer description.

Manage Layers

LineWeight PrintStyle – Default Y Default 🔥 Color_7 - 0.00 mm - 0.05 mm Color_1 0.09 mm 0.13 mm 0.15 mm 0.18 mm 0.20 mm - 0.25 mm - 0.30 mm 0.35 mm

Use the **Layers Manager** icon store to create new Layers, designate active Layers, and set Layer properties.

S C C C			~	
O ByLayer 💌	ByLayer Soli	d line 💌 🖳	— ByLayer	*

	Total layer(s) de	fined: 1. Tot	al layer(s) d	lisplayed:	: 1.	Enter filter expre	ssion			
itus 🍸	Name	Show	Frozen	Lock	LineColor	LineStyle	LineWeight	PrintStyle	Print	Description
⇒ 0				-	White	Continuolid line	Default	Color_7	-	
1						JIII 6			•	Edit Filters.
						ini c				

Creating New Layers

You can create a new Layer any time. For efficiency, it is recommended that you plan the Layers to be used in advance and create them in a systematic manner.

To create a new Layer using the Layers Manager, click the **New Layer** icon 🔫. Type a Layer name, or press **Enter** to accept the existing default Layer name.

Changing the Active Layer

To make a Layer active (current), select the Layer name from the **Name** cell, and then click **Activate** located at the top of the dialog box. Click **OK** from the pop-up box.



Layers Manager

3

Delete

Activate

Name

Act New Layer (Alt+N) ver(s) defined: 1. Total layer(s) displayed:

Show Frozen

Lock

-

New

Status

Renaming a Layer

To rename a Laver in the Laver Properties Manager. double-click the **Name** cell of the Layer you want to rename. Type the **new name**, then press **Enter**.

Controlling Layer Behavior

Show/Hide: when a Layer is shown, it is visible. Hidden Layers are invisible. To show or hide a Layer in the Layer Properties Manager, doubleclick the **Show** icon in the Show cell that you want to change.

Frozen/UnFrozen: in addition to turning the Layer off, you can freeze Layers. The Layers that are Frozen will not be visible in the view window, nor will they be printer. Frozen Layers are similar to Layers that are off (hidden). However, Layers that are simply hidden still go through screen regeneration each time the system regenerates your drawing, whereas the Layers that are suppress are not considered during screen regeneration.

Lock/Unlock: controls the ability to edit entities on a Layer. When a Layer is locked, it is protected but visible. When a Layer is unlocked, it is unprotected and visible. To lock or unlock a Layer in the Layers Manager, double-click the **Lock** cell you want to change.

Setting the LineColor

New Layers are automatically assigned the LineColor white. Entities you draw will appear in the assigned LineColor for that Layer except those to which you assigned a LineColor with the LineColor command.

To set a LineColor in the Layers Manager, double-click the LineColor cell you want to change. Select a color from the dropdown menu or click **Specify Color** for additional LineColor options.

Setting the Layer LineStyle

A LineStyle is a graphical symbol that has specific settings that make the Lines appear differently. Entities you draw will appear in the assigned LineStyle except those to which you set to another LineStyle.

۲	-	1	🕘 White
0	0	-	🔴 Red

LineColor

O White

O White

• Red

Blue
Cyan
Green

O White O Yellow

🍈 Magenta Red

Lock

Show

Frozen

LineColor

LineStyle

Continuo...lid line

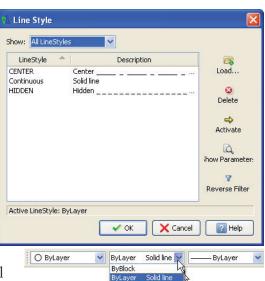
Continuo...lid line

CENTER ...___



	N L	ayers Ma	anager			
	Ne	i 🕄 w Dele	te Ac	⇔ tivate		
	Act	ive layer:	0. Total l	ayer(s) defir	ned: 2. Tota	al layer(s)
	S	atus 💌)	Name	Show	Frozen
		4	0		•	0
		2	Layer1		•	0
Status	*	Na	ame	Show	Frozen	Lock
⇒		0		۲	-	1
9		Object		0	0	1
0	6	Phantom		0	0	9

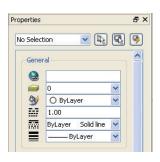
To set a Layer LineStyle, double-click the LineStyle cell and select the desired LineStyle or select Other, Load to load a new LineStyle from the LineStyles Manager dialog box.



Continuous...Solid line

CENTER Other...

By setting the **LineStyle** to **ByLayer** in the properties toolbar all elements of this Layer will be drawn with the LineStyle that is set. Elements of the Layer that were drawn with a different LineStyle than the default LineStyle are not affected by the change.



Setting a Layer LineWeight

By default, DraftSight assigns the LineWeight of the currently selected Layer in the Layer list to the newly created Layer. To set the Layer LineWeight, double-click the **LineWeight** cell and select the desired option.



NOTE: The display on the screen will not change. The LineWeight will only be visible on a printed drawing.

Setting PrintStyle

PrintStyle is a collection of properties settings (such as **LineColor**, **LineStyle** and **LineWeight**) that can be assigned to a Layer or to individual entities. The chosen PrintStyle type is displayed in the Properties Palette.

1	None	×
	ByColor	

DraftSight allows you to turn Printing / Plotting on or off for visible Layers. Example, if a Layer contains Construction Lines that need not be printed, you can specify that the Layer is not

printed. If you turn off printing for a Layer, the Layer is displayed but is not printed.

To change the setting for the printing of Layers, double-click the icon corresponding to the Layer name located in the **Print** column as shown.

Setting Description

Double-click inside the desired cell under the **Description** column to enter a Layer description.

Setting Filtering and Display Options

You can create filters that contain Layers you select. Grouping Layers this way establishes group filters. Two Layer group filters are predefined:

- All
- All Used Layers

Create Three New Layers for the Drawing

In this section, you will create three new Layers called Object, Construction and Centerline. As well as apply custom properties.

3 Create the first new Layer.

Click the Layers Manager icon 📓 on the Layers toolbar

The Layers Manager dialog box is displayed.

Note: You can also click Format, Layer from the Main Menu to display the Layers Manager dialog box.



Click the **New Layer** icon. 🔫 The new default name is Layer1.

Rename this layer **Object**

Vew Dele	te Activate				
Diam'r Law	(*** · ***)				
INEW Lay	er (Alt+N)				7055
ctive layer.	er (Alt+N) o. nocamayer(s) de	fined: 2. Tot	al layer(s) c	lisplayed	: 2.
Status	er (Alt+N) o. iocana ver(s) de Name	fined: 2. Tot Show	al layer(s) c Frozen	lisplayed Lock	: 2. LineColor
	n resources	In states and	I December 1		



Double-click inside the Layer1 Name cell in order to rename if it is not active.

Object is the new Layer name.

DraftSight

The Object Layer will inherit all of the properties of the default Layer 0.

Rename this Layer **Construction**.

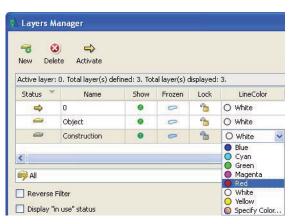
Construction is the new Layer name.

5 Set the Second Layer LineColor. New Layers are automatically assigned the LineColor white.

Double-click the **LineColor** cell for the Construction Layer.

Click Red.

vew Dele					
Active layer:	0. Total layer(s) de	fined: 2. Tot	al layer(s) c	lisplayed	: 2.
Active layer: Status 🍼	0. Total layer(s) de Name	fined: 2. Tot Show	al layer(s) c Frozen	lisplayed Lock	: 2. LineColor
	1 70.00000	In a successful			



O White

O White

Red

Continuo...lid line

Continuo...lid line

Continuous 🛛 Sc 😒

Continuo...lid line Other...

6 Create the Second Layer LineStyle. Create a MM HIDDEN LineStyle.

Double-click the **LineStyle** cell from the Construction Layer.

Click Other. The LineStyles Manager dialog box is displayed.

Click the Load icon.

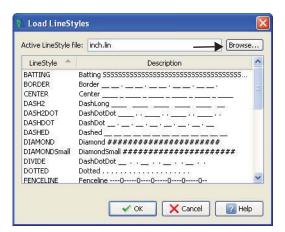
The Load LineStyles dialog box is displayed

LineStyle 🔶	Description	
Continuous S	Solid line	Load
		0
		Delete
		ے
		Activate
		D.
		ihow Paramete
		8
		Reverse Filte

DraftSight

DRAFTING TUTORIALS

Note: The default LineStyle file: inch.lin. Click the **Browse** button.

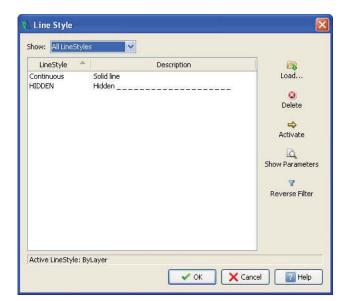


Select MM.LIN from the Specify LineStyle File dialog box.		Specify LineStyle File	
Click Open from the Specify LineStyles File dialog	g box.	My Recent Documents	
Select HIDDEN from the Load LineStyles dialog	LineStyle 🔺	Description	
box.	FENCELINE1 FENCELINE2	Fence circle00000- Fence square[]	
Click OK .	FENCELINE21 FIVELINE	Fence square 45[][][][FIVELINE =5=5=5=5=5=5=5=5=5=5=	

LineStyle 🐣	Description	^
ENCELINE1	Fence circle00000-	
ENCELINE2	Fence square[][][][][]	
ENCELINE21	Fence square 45[][][][][
FIVELINE	FIVELINE =5=5=5=5=5=5=5=5=5=5=5=5=5=5=5=5=5=5=5	
GAS_LINE	Gas lineGASGASGASGAS	-
HIDDEN	Hidden	
HOT_WATER	Hot water line HW HW HW	-
PHANTOM	Phantom	
5INUS	SINUS ๛๛๛๛๛๛๛๛๛๛๛๛๛๛๛๛	10
5TAR4	Star4	
5TAR4Rot45	Star4R45	
TRACK	Tracks - - - - - - - - - - - - - - - - - - -	~

NOTE: Use the scroll bar on the right to view items in the dialog box.

HIDDEN is added to the **LineStyles** dialog box.

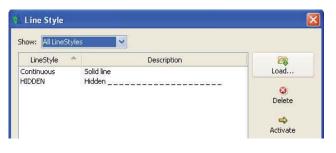


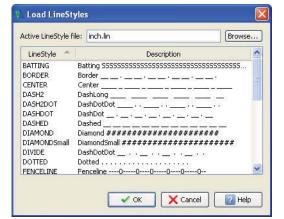
7 Create the Second Layer LineStyle. Create a MM CENTER LineStyle.

Click the Load icon.

View the **Load LineStyles** dialog box. The default LineStyle file: inch.lin.

Click the **Browse** button.





Select MM.LIN from the **Specify LineStyle File** dialog box. Click Open from the Specify LineStyles File dialog box



Load...

3 Delete

Click **CENTER** from the **Load LineStyles** dialog box.

Click OK.

LineStyle 🐣	Description	^
	atting 555555555555555555555555555555555555	
	enter	
DASH2 Da	ashLong	
DASH2DOT Da	ashDotDot	-
	ashDot	
	ashed	
DIAMOND Di	amond ########################	
DIAMONDSmall Di	amondSmall ########################	
DIVIDE Da	ashDotDot	
DOTTED DO	otted	
FENCELINE Fe	enceline00000	~

Description

CENTER is added to the **Line Styles** dialog box.

Click **OK**.

- 8 Click OK from the Line Style Dialog Box.
- **9** View the Available New LineStyles Double-click inside the LineStyle cell.

View the HIDDEN and CENTER LineStyle from the drop-down menu.

Line Style

Show: All LineStyle

Center

Solid line

Hidden ___

LineStyle

CENTER

HIDDEN

Continuous

Status 🍸	Name	Show	Frozen	Lock	LineColor	LineStyle	LineWeigh
⇒	0	•	0	1	O White	Continuolid line –	Default
<i>a</i>	Object		0	1	O White	Continuolid line –	Default
-	Construction		0	-	Red	Continuous Sc 💙 –	Default
						Continuolid line HIDDEN CENTER Other	IN

10 Set HIDDEN for LineStyle.

Click HIDDEN in the LineStyle cell for the Construction Layer.

HIDDEN is the new the **LineStyle** for the Construction Layer. Click **OK**.

11 Save the Drawing.

Click the **Save** icon 🗟 on the Standard toolbar.

Enter File name: Lesson 2-1.

Click Save.

12 Create the third new Layer.

Click the Layers Manager icon 📓 on the Layer toolbar.

The Layers Manager dialog box is displayed.

Click the **New Layer** $rac{1}{rac{3}}$ icon. The new default name is Layer 1.

Rename this Layer **Centerline**.

Centerline is the new Layer name.

New Dele	te Activate				
Active layer:	0. Total layer(s) de	fined: 4. Tot	al layer(s) c	lisplayed:	: 4.
Status 🔻	Name	Show	Frozen	Lock	LineColor
4	0	•	0	1	O White
1000000000	1 10.00000046	•	0 0		O White O White
⇒	0	•	1.5	1	

Vew Dele	· · · ·				
Active layer:	0. Total layer(s) de	fined: 4. Tot	al layer(s) c	lisplayed	: 4.
Status 🔻	Name	Show	Frozen	Lock	LineColor
Status 🍼	Name 0	Show	Frozen	Lock	LineColor
Status ×	0.00000	Show	Frozen		NUL STREET
Status ×	0	•	0	1	O White

Lock

-

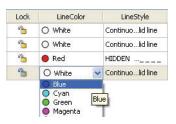
LineColor

O White

13 Set the Third Layer LineColor.

Press the **Enter** key.

New Layers are automatically assigned the LineColor white. Double-click the **LineColor** cell for the **Centerline** Layer. Click **Blue**.



LineWeight

- Default

14 Set the Third Layer LineStyle.

Double-click the **LineStyle** cell for the Centerline Layer.

Click **CENTER**.

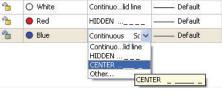
CENTER is new the **LineStyle** for the Centerline Layer.

15 Add a Description.

Double-click inside the **Description** cell for the Centerline Layer.

Enter Customer XYZ.

Press the Enter key.



LineStyle

Continuo...lid line

PrintStyle	Print	Description
Color_7		
Color_7		
Color_1	-	
Color_5	8	Customer XYZ

NOTE: At this time, the default Layer is active.

16 Activate the Object Layer.

Double-click the icon in the **Status** box next to the Object Layer. A

➡ is displayed in the Status box.

Object is now the current Layer. Click **OK**.

View the status of the Layers toolbar and the Properties Palette.

	3 ⇒					
New De	lete Activate					
Active layer	: Object. Total layer	(s) defined: 4	i. Total laye	r(s) displ	ayed: 4.	Enter filter ex
Status	Name	Show	Frozen	Lock	LineColor	LineStyle
9	0	•	0	1	O White	Continuolid line
					CALCULATION OF A CALCUL	N PARA STREET,
⇒	Object	0		- 19	White	Continuolid line
∂	Object Construction	© 0	• 0	1 12	 White Red 	Continuolid line Continuolid line





Create the First New Drawing

Draw a Circle

In this section, use the **Circle, Center, Radius** command from the drop-down menu. Click the graphics area to define the center point of the Circle or enter an absolute coordinate position at the Command prompt. Click the drawing to define the Circle radius, or enter the radius value at the Command prompt.

Draw	Modify	Tools	
	finiteLine :hLine		
Po Po) PolyLine IlyLine Ilygon ectangle etch		
Ar	c	•	
Cir	rcle	•	🕢 Center, Radius
Ell	ng Iline ipse Iint	•	Center, Diameter 2 Points 3 Points
	ock ble	•	 Tangent, Tangent, Radius ₹ Tangent, Tangent, Tangent

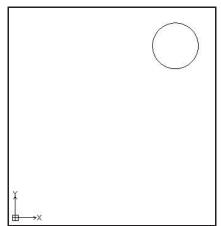
NOTE: You can also use the **Circle** icon on the Draw toolbar.

1 Draw a 20mm Radius Circle as the First Entity.

Click **Draw**, **Circle**, **Center**, **Radius** from the Main Menu. Options: <u>3Point</u>, <u>2Point</u>, <u>Ttr</u>, <u>TTT</u> *Enter to exi t* or

Specify center point»140,150 \lrcorner

Option: <u>D</u>iameter or Specify radius»**20**.⊣



Draw a Hexagon

Draw a hexagon using the **Polygon** command. A Polygon is a closed linear PolyLine segment consisting of between 3 and 1024 equal length sides.

1 Draw a six Side Hexagon.

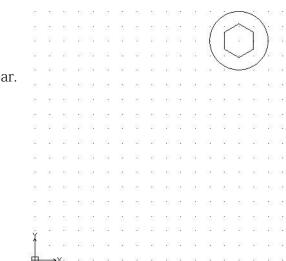
Click **Draw**, **Polygon** from the Main Menu. Default: 4 Enter number of Sides» 6 ↓

Options: <u>S</u>ide Length or Specify center point» Click the center point of the Circle in the graphics area.

Default: COrner Options: <u>CO</u>rner, or <u>S</u>ide Specify distance option»s \neg Specify distance» **10** \neg

2 Save the Drawing.

Click the **Save** icon 🗟 on the Standard toolbar.







Polar ESnap ETrack

Command window

<Snap Off>

Snap Grid Ortho

Copy Command

Use the **Copy** a command to duplicate drawing entities by selecting drawing entities and placing a copy of them at a specified position. The duplicated entities are exact copies of the selected entities including all entity properties such as Layers, LineStyles, LineColors, and LineWeight.

1 Copy the Existing Circle to the Right of the Drawing.

2 Turn Snap Off.

Click the **Snap Snap** button on the Status bar. The command window displays: <Snap Off>.



3 Click the Copy Icon P on the Modify Toolbar.

Specify entities»

Click the circumference of the Circle. The Circle is highlighted by turning dotted.

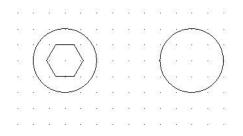
1 found, 1 total

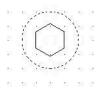
Specify entities»⊣

Default: Displacement Options: <u>D</u>isplacement or Specify from point» Click the center point of the Circle in the graphics area.

Options: *Enter to use first point as displacement* or Specify second point» **220,150.**

Default: Exit Options: <u>E</u>xit, <u>U</u>ndo or Specify second point: ↓





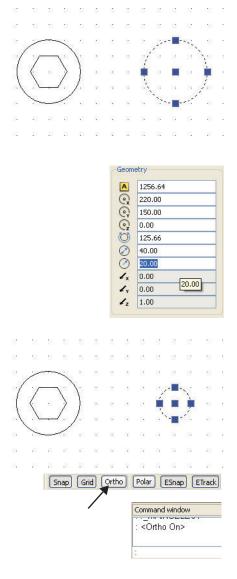
Modify the Diameter of the Copied Circle

Use the Properties Palette to modify the diameter of the copied Circle. The Properties Palette allows you to view and change drawing entity properties. The Properties Palette is located on the right side of the graphics area.

4 Modify the Circle Diameter.

Click the circumference of the copied Circle.

Delete 20.00 for Radius in the Properties Palette.



Enter **10**.

Press Esc or Enter to exit.

5 Save the Drawing. Click the Save icon 🗟 on the Standard toolbar.

Draw Two Parallel and Horizontal Lines

Turn Ortho mode on.

Click the **Ortho** ortho button on the Status bar.

The command window displays:

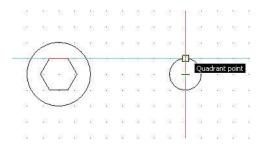
<Ortho On>.

Orthogonal mode limits the cursor movement so that it moves parallel to the axes of the coordinate system. You can only enter points located parallel to these axes. Orthogonal mode makes it easier to draw and place parallel or colinear Lines on entities.

6 Draw the First Linear Segment.

Click the Line icon 💫 on the Draw toolbar.

Options: <u>Segments</u> or Specify start point» Click the top quadrant point of the small Circle.



Options: <u>S</u>egments, <u>U</u>ndo *Enter to exit* or Specify next point»

Draw a Line from the top quadrant point to a point within the large Circle as illustrated.

Click a point.

Options: <u>S</u>egments, <u>U</u>ndo *Enter to exit* or Specify next point»J

7 Draw the Second Linear Segment.

Click the **Line** icon 🔨 on the Draw toolbar.

Options: Segments or Specify start point» Click the bottom quadrant point of the small Circle.

Options: <u>Segments</u>, <u>U</u>ndo *Enter to exit* or Specify next point»

Draw a Line from the bottom quadrant point to a point within the large Circle as illustrated.

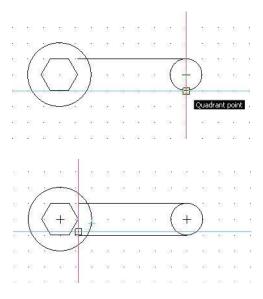
Click a point.

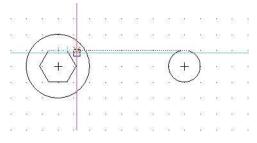
Options: Segments, Undo Enter to exit or Specify next point»↓

Trim Command

The **Trim** \aleph command provides the ability to cut drawing entities precisely at the edges defined by one or more entities.

Lines, **InfiniteLines**, **Rays**, **PolyLines**, **Circles**, **Arcs**, or **Ellipses** can act as cutting edges and as a result of the command execution be trimmed. **Blocks** and **text** cannot be trimmed or used as cutting edges. In a sheet the entities of type ViewPort can also be cutting edges. When you select a PolyLine as cutting edge, the command trims at its centerline.





8 Trim the Top and Bottom Horizontal Lines Back to the Large Circle.

Click the **Trim** icon **i** on the Modify toolbar. Specify cutting edges...

Options: *Enter to specify all entities* or Specify entities» Click the circumference of the large Circle.

1 found, 1 total

Options: *Enter to specify all entities* or Specify entities»J

Options: <u>C</u>rossing, <u>CR</u>ossline, <u>P</u>roject, <u>E</u>dge, e<u>R</u>ase, *Shift + select to* extend or

Specify segments to remove»

Click the bottom Horizontal Line inside the large Circle. The inside bottom Line is trimmed.

Options: <u>C</u>rossing, <u>CR</u>ossline, <u>P</u>roject, <u>E</u>dge, e<u>R</u>ase, *Shift + select to extend* or Specify segments to remove»

Click the top Horizontal Line inside the large Circle. The inside top Line is trimmed

Options: <u>C</u>rossing, <u>CR</u>ossline, <u>P</u>roject, <u>E</u>dge, e<u>R</u>ase, *Shift + select to extend* or Specify segments to remove»↓

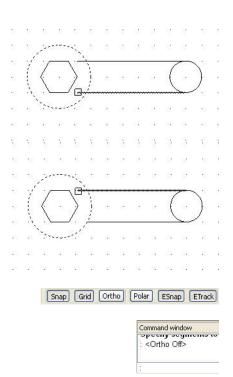
9 Turn Ortho Mode Off.

Click the **Ortho** ortho button on the Status bar.

Fillet Command

The **Fillet** Command allows you to create rounded corners by replacing part of two Lines with an Arc. The

12	80	-	\$3)	-	10	32	1	13	32	13	35	83	ŝ
53	53	53	5		8	8		12	1	32	32	13	;
10	14	1.00		··	8	8		12	1	82	32	10	;
53	1	1		1	1					1	\sim	1	ţ
63		<	0	>	}		1	at a	8	()	÷
67	À.,	. <u>`</u>		/	; <u>'</u>						~	· ·	÷
8	43	·	E	30			•		1	0	8	13	÷
42	133	10	÷		33) (1)	12		3	14	35	33	23	ţ
22	22	8	<u> 4</u> 1	44	ф.	2	12	12	82	8	101	22	ł,



filleted entities do not have to intersect, and they can overlap. If the entities overlap, they will be trimmed (by default) to create the rounded corner. In any case, the Lines to be filleted are connected to the fillet arc.

Applying a fillet radius of 0 will result in creating sharp corners instead of rounded. You can hold down the Shift key on the keyboard while selecting the entities to override the current fillet radius with a value of 0.

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In the next section, draw two fillets between the Horizontal Lines and the large Circle.

10 Create the First Fillet.

Click the **Fillet** icon n the Modify toolbar.

Mode = TRIM, Radius = 0.00. Set the Radius to 5mms. Options: <u>M</u>ultiple, <u>P</u>olyline, <u>R</u>adius, <u>T</u>rim mode, <u>U</u>ndo or Specify first entity»r,J Default: 0.00 Specify radius»5J

Options: <u>M</u>ultiple, <u>P</u>olyline, <u>R</u>adius, <u>I</u>rim mode, <u>U</u>ndo or Specify first entity»

Click the bottom Horizontal Line.

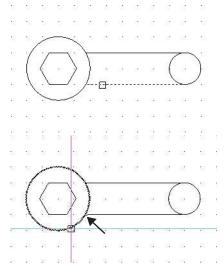
Options: Shift + select to apply corner or Specify second entity»

Click circumference of the large Circle below the bottom Horizontal Line.

View the fillet in the graphics area.

11 Create the Second Fillet Entity.Click the **Fillet** icon n the Modify toolbar.

Mode = TRIM, Radius = 5.00



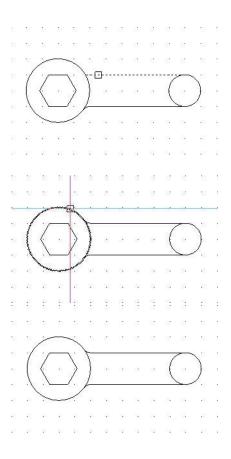
NOTE: Pressing Enter on the keyboard will recycle the previous command.

Options: <u>M</u>ultiple, <u>P</u>olyline, <u>R</u>adius, <u>T</u>rim mode, <u>U</u>ndo or Specify first entity» Click the top Horizontal Line.

Options: <u>M</u>ultiple, <u>P</u>olyline, <u>R</u>adius, <u>T</u>rim mode, <u>U</u>ndo or Specify first entity»

Click circumference of the large Circle above the top Horizontal Line.

View the fillet in the graphics area.



NOTE: Since we set the fillet radius to 5 earlier, it is not necessary to enter a new radius unless the value were to change.

Mirror Command

The **Mirror** \triangle command allows you to copy selected entities onto the opposite side of a specified Mirror Line. To define the mirror axis, specify two points.

NOTE: By default, the **Mirror** command retains text direction when entities are mirrored.

12 Apply the Mirror Command.

Click the **Mirror** icon \triangle on the Modify toolbar.

Specify entities»

Window select all entities in the graphics area.

Click a position to the top left of the drawing.

Click a position below the bottom right of the drawing. The selected entities inside the graphics area are displayed. 7 found, 7 total Specify entities»,J • Specify start point of mirror line» Click the top quadrant point of the large Circle as illustrated. Specify end point of mirror line» Click the bottom quadrant point of the large Circle. Accept the default setting. Default: No Confirm: Delete source Quadrant point enti ti es? Specify Yes, or No»↓

13 Save the Drawing.

Click the **Save** icon 🔚 on the Standard toolbar.

Rotate Command

Use the **Rotate** solution command to rotate entities through an angle, about a base point. By default, rotation is counterclockwise.

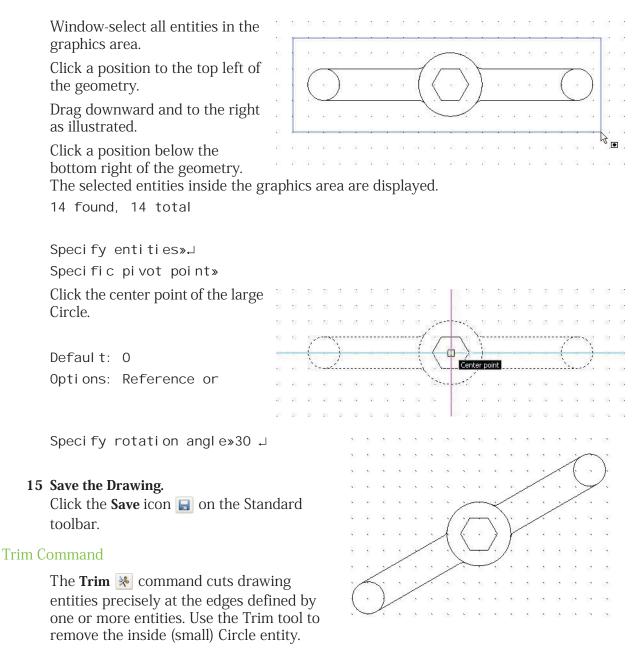


When you rotate, set the Orthagonal mode to constrain the angle of rotation to 90 degree intervals.

14 Rotate the Geometry 30 Degrees

Click the **Rotate** icon 🔊 on the Modify toolbar.

Active positive angle in CCS: DIRECTION+counterclockwise BASE=0 Specify entities»



16 Trim the First Circle Entity.

Click the **Trim** icon 💥 on the Modify toolbar.

Specify cutting edges...

Options: *Enter to specify all entities* or Specify entities»

Click the bottom Line on the left side and the top Line on the left side.

1 found, 2 total

Options: Enter to specify all entities or Specify entities»,J Options: <u>Crossing</u>, <u>CR</u>ossline, <u>P</u>roject, <u>E</u>dge, e<u>R</u>ase, <u>U</u>ndo, *Shift + select* to extend or Specify entity to remove»↓ Click right side of the circumference of the small Circle that is on the left side of the entity as shown. Specify entity to remove»↓ **17** Trim the Second Circle Entity. Click the **Trim** icon 💥 on the Modify toolbar. Specify cutting edges... Options: Enter to specify all entities or Specify entities» Click the bottom Line on the right side and the top Line on the right side. Л 1 found, 2 total Options: Enter to specify all enti ti es or Specify entities», J Click left side of the circumference of the small Circle that is on the right side of the entity as shown Options: <u>Crossing</u>, <u>CR</u>ossline, <u>Project</u>, <u>Edge</u>, e<u>R</u>ase, <u>U</u>ndo, *Shift* + *select* to extend or Specify entity to remove»J 18 Save the Drawing. Click the **Save** icon 🔚 on the Standard toolbar.

Draw Entities on the Centerline Layer

In this next section, draw a **Line**, **Circle**, and **Rectangle** on the Centerline Layer. Only display the Centerline Layer.

19 Open the Layers Manager Dialog Box.

Click the **Layers Manager** icon **Solution**. The **Layers Manager** dialog box is displayed. At this time, all Layers are displayed: Default (0), Object, Construction, and Centerline.

Activate the Centerline Layer

Double-click the **Status** icon *e* in the Centerline Layer **Show** cell.

Turn the Default Layer (0) Off.

Double-click the **Show** icon • in the default (0) **Show** cell.

Turn the Construction Layer Off.

Double-click the **Show** icon **()** in the Construction **Show** cell.

Turn the Object Layer Off

Double-click the **Show** icon **•** in the Object **Show** cell.

Click \mathbf{OK} from the Layer Manager dialog box.

View an empty work space.

20 Save the Drawing.

Click the Save icon \blacksquare on the Standard toolbar.

21 Draw a Linear Segment.

Click the Line icon 📐 on the Draw toolbar.

Options: <u>S</u>egments or Specify start point»**50,50**↓

Options: <u>S</u>egments, <u>U</u>ndo, *Enter to exit* or Specify next point»**160,50**.J

Options: <u>S</u>egments, <u>U</u>ndo, *Enter to exit* or Specify next point»,J

The blue **LineStyle** *CENTER* is displayed on the Centerline Layer.

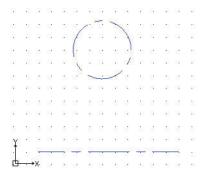
Layers M	anager			
New Dele Active layer: Status	te Activate Centerline, Total la Name	yer(s) define Show	d: 4. Total l Frozen	layer(s) Lock
<u></u>	0	Driow O	0	- COCK
9	Object	0	0	-
9	Construction	0	0	-
⇒	Centerline	0	-	1

New Dele	te Activate			
Active layer:	Centerline. Total la	yer(s) define	d: 4. Total l	ayer(s)
MEDINEROD	1 12/2/00/27/2	110120293397		
Status 🍼	Name	Show	Frozen	Lock
Status *	Name 0	Show	Frozen	Lock
Status ·	1.00000	Show	Frozen	Lock
Status	0	Show	0	Lock

23	82	12	22	22	22	12	22	13	3	23	3	23	4
80	8		8		a.		æ	0		63		63	
20	92 -	- 27		1000	+	17	1	-		-	14		4
33	13		10					:23		:25	8	53	3
20	85	13	12	13		13	12	32	52	22	53	32	52

22 Draw a Circle Entity.

Click the **Circle** icon (a) on the Draw toolbar. Options: <u>3Point</u>, <u>2Point</u>, <u>Itr</u>, <u>TTT</u>, *Enter to exi t* or Specify center point ***100,130**. Options: <u>D</u>iameter or Specify radius ***d**. Specify diameter ***45.5**.



The blue **LineStyle** *CENTER* is displayed on the Centerline Layer.

Rectangle Command

The **Rectangle** command creates a closed PolyLine in a rectangular shape. A PolyLine is a drawing entity that consists of connected Lines or connected Arc segments. You can specify the **area**, **dimensions** or **rotation** parameters. You can also control the type of corners on the rectangle.

23 Draw a Rectangle Entity.

Druw a Rectangle Littly.	
Click the Rectangle icon ኳ on the	
Draw toolbar.	
Options: <u>C</u> hamfer, <u>E</u> levation,	
<u>F</u> illet, <u>T</u> hickness, line <u>W</u> idth	
or	
Specify start corner»180,120,	
	Y
Options: <u>A</u> rea, <u>D</u> imensions,	
Rotation or	
Specify opposite corner»240,50.J	

24 Save the Drawing.

Click the **Save** icon 📕 on the Standard toolbar.

SimpleNote Command

The SimpleNote 🖾 command creates a single line text entity of any position, orientation, justification, and style. When you issue the command, you are prompted with a dialog box that will enable these options.

25 Apply the SimpleNote Command.

The SimpleNote command displays text in the graphics area as it is entered.

Click the **SimpleNote** icon 📼 on the Draw toolbar.

ext: RECTANGLE								
			本路					
Insertion point	Insertio	n orientatio			Options			
X: 190.00		Left	Center		Style:	Standard	~	A
Y: 40.00	Тор	0	0	0	Angle:	0		
z: 0.00	Center		0	0	Height:	0.2000		
Select in graphics area	Bottom Upper	0	0	0				
		O Align	O Fit	O Center				

26 Complete the Insert SimpleNote Dialog.

Input the word **RECTANGLE** in the **text**: field.

Use Insertion point

X: 190

Y: 40

Change Height: to 5.

Click OK.

27 Save the Drawing.

Click the **Save** icon 🗟 on the Standard toolbar.

Draw a Spline on the Construction Layer

In this next section, you will draw a **Spline** on the Construction Layer. The default Layer, Object Layer and Construction Layer are not shown at this time.

28 Turn the Centerline Layer Off.

Click the drop-down arrow from the Layers Manager.

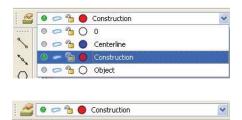
Click the **Show** icon in the *Centerline* row.

Click **OK** from the pop-up dialog box.

Make the Construction Layer active.

Click the drop-down arrow from the **Layers Properties Manager**.

2	🗕 📨 👈 🔵 Centerline	~
	🛛 🗢 🔭 🔿 0	
S.	🔊 🖛 📸 🔵 Centerline	
2/	💿 🗢 🔭 🔴 Construction	
Ľ	🔍 🗢 造 🔘 Object	



Click **Construction**.

The Construction Layer is displayed in the graphics area. The Default Layer, Object Layer and the Centerline Layer are Off.

29 Save the Drawing.

Click the **Save** icon 🗟 on the Standard toolbar

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Command window		M	odel		sect		ieetz	<i>,</i>																						8
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Spline Command

The **Spline** \swarrow command provides the ability to draw a smooth curve that is fit to a set of points. You can create a non-uniform rational B-Spline or NURBS curve for any number of fit points. You can also fit the Spline to the specified points to within a specified tolerance value.

30 Draw a Spline Entity.

Click the **Spline** icon 🖍 on the Draw toolbar.

Specify start fit point»150,60.↓

Specify next fit point»185,80,J

You can either click the graphics area or enter successive points to continue the Spline or enter F to define a fit tolerance and then click or enter other Spline points. In the next section, enter a fit tolerance of **.2mm**.

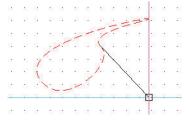
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Options: <u>C</u>lose, <u>E</u>it tolerance, <u>Enter for start tangent</u> or Specify next fit point»f.J Default: 0.00 Specify fit tolerance».2.J Options: <u>C</u>lose, <u>E</u>it tolerance, <u>Enter for start tangent</u> or Specify next fit point»100,40 .J Options: <u>C</u>lose, <u>E</u>it tolerance, <u>Enter for start tangent</u> or Specify next fit point»115,25.5 .J Options: <u>C</u>lose, <u>E</u>it tolerance, <u>Enter for start tangent</u> or Specify next fit point»115,25.5 .J

Specify tangency»

Click a position inside the graphics area as illustrated. Your drawing may differ depending on the position that you select in the graphics area.



31 Delete the Spline Entity. Select the Spline entity.

Click the **Delete** icon **Q**.

32 Save the Drawing. Click the Save icon 🕞 on the Standard toolbar.



The fit tolerance applies to all fit points until you change it. Setting the fit tolerance to zero forces the Spline to pass through the fit points.

PolyLine Command

The **PolyLine ©** command creates an equilateral (edges with equal length) closed PolyLine. A PolyLine is a drawing entity that consists of connected Lines or connected Arc segments. The number of sides can vary from 3 (which forms an equilateral triangle) to 1024.

You can create PolyLines with segments of different widths and fill settings. The default segment type for a PolyLine is a Linear Segment.

33 Draw a PolyLine Segment on the Construction Layer

Click the **PolyLine** icon 📀 on the Draw toolbar.

Options: *Enter to continue from last point* or Specify start point»**50,50**,J

Options: <u>Arc</u>, <u>Halfwidth</u>, <u>Length</u>, <u>Undo</u>, <u>Width</u>, <u>Enter to exit</u> or Specify next vertex»100,50,

Options: <u>Arc</u>, <u>Halfwidth</u>, <u>Length</u>, <u>Undo</u>, <u>Width</u>, <u>Enter to exit</u> or Specify next vertex»130,25,

Options: Arc, <u>H</u>alfwidth, <u>L</u>ength, <u>U</u>ndo, <u>W</u>idth, <u>Enter to exit</u> or Specify next vertex»180,25,

Options: Arc, <u>H</u>alfwidth, <u>L</u>ength, <u>U</u>ndo, <u>W</u>idth, <u>Enter to exit</u> or Specify next vertex»**200,50**.

Options: <u>Arc</u>, <u>H</u>alfwidth, <u>L</u>ength, <u>U</u>ndo, <u>W</u>idth, <u>Enter to exit</u> or Specify next vertex»250,50.

Options: <u>Arc</u>, <u>Halfwidth</u>, <u>Length</u>, <u>Undo</u>, <u>W</u>idth, <u>Enter to exit</u> or Specify next vertex»**250,100**.

Options: <u>Arc</u>, <u>Halfwidth</u>, <u>Length</u>, <u>Undo</u>, <u>Width</u>, <u>Enter to exit</u> or Specify next vertex»230,130.

Options: <u>Arc</u>, <u>Halfwidth</u>, <u>Length</u>, <u>Undo</u>, <u>Width</u>, <u>Enter to exit</u> or Specify next vertex»200,130,

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Options: <u>A</u>rc, <u>H</u>alfwidth, <u>L</u>ength, <u>U</u>ndo, <u>W</u>idth, *Enter to exit* or Specify next vertex»180,100,J

Options: <u>A</u>rc, <u>H</u>alfwidth, <u>L</u>ength, <u>U</u>ndo, <u>W</u>idth, *Enter to exit* or Specify next vertex»**130,100**, □

Options: Arc, <u>H</u>alfwidth, <u>L</u>ength, <u>U</u>ndo, <u>W</u>idth, *Enter to exit* or Specify next vertex»100,130,

Options: <u>Arc</u>, <u>H</u>alfwidth, <u>L</u>ength, <u>U</u>ndo, <u>W</u>idth, <u>Enter to exit</u> or Specify next vertex»75,130 \lrcorner

Options: <u>Arc</u>, <u>H</u>alfwidth, <u>L</u>ength, <u>U</u>ndo, <u>W</u>idth, <u>Enter to exit</u> or Specify next vertex»**50,100** $_{}$

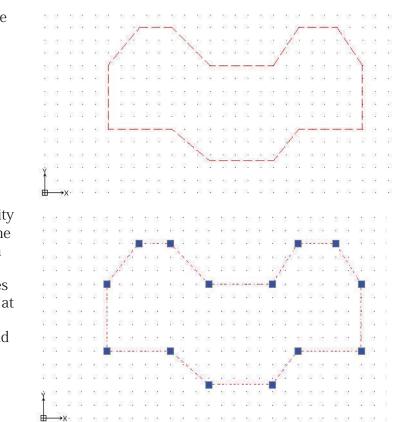
OOptions: <u>Arc, H</u>alfwidth, <u>L</u>ength, <u>U</u>ndo, <u>W</u>idth, <u>Enter to exit</u> or Specify next vertex $c \downarrow$

34 Save the Drawing.

Click the **Save** icon **a** on the Standard toolbar.

Click any segment of the entity in the graphics area. View the EGrips. The EGrips display a single entity. Remember, EGrips are selectable handles (displayed as small squares) at the end points, center, vertices, insertion points, and other geometric points of entities.

35 Remove the EGrips. Press **Esc** or **Enter** to exit.



Edit PolyLine Command

The **Edit PolyLine** command provides the ability to edit PolyLines, PolyLine vertices, and three-dimensional polygon meshes in several ways. PolyLines can be modified in the following ways:

- Open a close PolyLine or close an open PolyLine
- Join a PolyLine with other drawing entities such as Line, Arc or PolyLine
- Modify the width of the whole PolyLine
- Edit individual PolyLine segments by editing their control points (vertices)

Modify the PolyLine Width

In the following section, modify the width of the PolyLine from .2mm to 5mm.

36 Apply the Edit PolyLine Command.

Click the **Edit PolyLine** icon icon the Modify toolbar.

Options: <u>M</u>ultiple or

Specify polyLine»

Click a **Line segment** of the polyline.

Options: <u>D</u>ecurve, <u>E</u>dit vertex, <u>F</u>it, <u>J</u>oin, <u>L</u>engthen, <u>O</u>pen, <u>S</u>pline, <u>U</u>ndo, <u>W</u>idth, or e<u>X</u>it

Specify option» w_{\prec}

Specify new width»5.↓

Options: <u>D</u>ecurve, <u>E</u>dit vertex, <u>F</u>it, <u>J</u>oin, <u>L</u>engthen, <u>O</u>pen, <u>S</u>pline, <u>U</u>ndo, <u>W</u>idth, or e<u>X</u>it Specify option»**x** J

37 Undo the Modification.

Click the **Undo** icon 🕤 on the Standard toolbar.

38 Turn the Construction Layer Off.

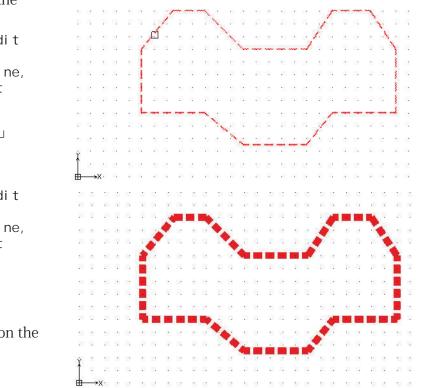
Click the drop-down arrow from the **Layers Properties Manager**.

Click the **Show** icon **•** in the **Construction** row.

Click **OK** from the pop-up dialog box.

A gray **Show** icon **o** is displayed.

2	🔍 🗢 🎦 🔴 Construction	*
	o 🗢 🛅 🔿 0	
S	💿 🥏 🎦 🔵 Centerline	
R	🐵 🛥 🖀 🔴 Construction	
~	💿 🥏 🎦 🔿 Object	



Return and Activate the Object Layer.

Return to the Object Layer.

39 Show the Object Layer. Click the drop-down arrow from the Layers Properties Manager.



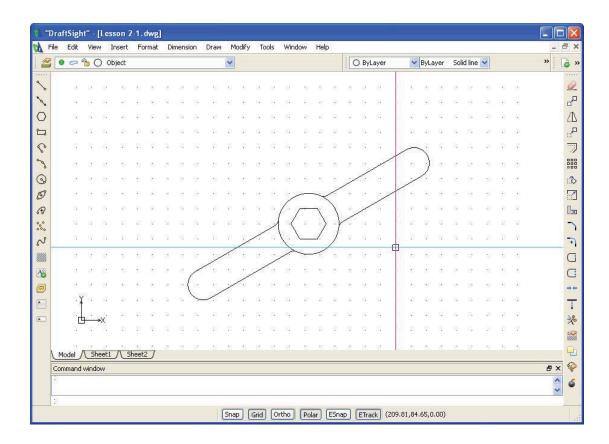
Click the **Show** icon 🔍 in the Object Layer row. A green **Show** icon 🔍 is displayed.

40 Activate the Object Layer.

Click **Object**. The Object Layer is active.

41 Save the Drawing.

Click the **Save** icon 🗟 on the Standard toolbar.



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Create a Second New Drawing

Create three new Layers. Use the **Rectangle**, and **Circle** commands.

Apply the **Pattern** command to create a Circular and Linear pattern along with the **Chamfer** command.

Create an Object Layer and a Centerline Layer. Set custom Layer properties.

1 Create a New Drawing. Click the New icon 🐻 on the Standard toolbar.

Drawing Template

2 Select a Default Drawing Template. Double-click standard.dwt from the Select Template dialog box.

Set the Drawing Environment

Drawing Unit System

The default units system is the Decimal system and can be applied to either English (inches) or metric values (millimeters).

3 Set Linear Length Precision.

The **Unit System** section of the **Options** - **Drawing Settings** dialog box allows you to set the linear and angular unit formats.

Click Format, Unit System... from the Main Menu.

The **Unit System** portion of the **Options** - **Drawing Settings** dialog box is displayed.

Click 0.00 for Precision in the Length section of the dialog box.

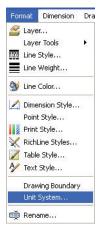
4 Set Linear Units Scale.

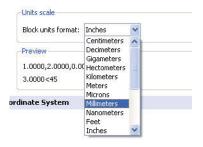
The **Units scale** section of the **Unit System** dialog box allows you to set the linear and angular unit formats.

Click **Millimeters** for **Block units format** in the **Units scale** section of the dialog box.

Click OK.

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- C)	÷		K	D.	0	
	8	82	3	3	1	85	• /
1-	12	102	137	- 20	- 22	122	





Set Drawing Boundary

Use a horizontal A4 default drawing sheet.

5 Set Drawing Boundary. Click Format, Drawing Boundary from the Main Menu.

Default: (0.00,0.00) Options: <u>OF</u>f, <u>ON</u> or Specify lower left corner» ↓

Default: (12.00,9.00) Specify upper right corner» **297,210.**↓

Set Snap and Grid

Set the **Snap** and **Grid**. In this section the Grid equals the Snap.

6 Set Snap and Grid.

Click **Tools**, **Options** from the Main Menu.

The **Options** dialog box is displayed.

Click the User Preferences tab.

Expand Drafting Options, Pointer Control and Snap Settings.

Click the **Enable Snap** <**F9**> box.

Enter the value of **10** in the **Horizontal Snap spacing** box. Press the **Tab** key The **Vertical Snap spacing** box is updated.

Expand Display and Grid Settings

Click the **Enable Grid <F7>** box and the **Match Snap Spacing** box. The **Horizontal display spacing** value and the **Vertical display spacing** value will be set to **10.00**.

Click OK.

Zoom Bounds

7 Display the Drawing with the New Boundary.

Drawing boundaries need to be used to adjusted to the current view.

8 Select View, Zoom, Bounds 🔍 from the Main Menu.

9 Save the Drawing.

Click the **Save** icon 🗟 on the Standard toolbar.

Enter File name: Lesson 2-2.

Click Save.

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	Laye	r	
	Laye	r Tools	F.
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≡	Line	Weight	
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Tools	Window	Help	
A Pro	operties		Ctrl+1
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Ind	quiry		•
Op	en Referer	nce	
Ru	n Script		
co طر	S Manager		
Ne	w CCS		•
Cu	stomize Int	erface	
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Create New Layers for the Drawing

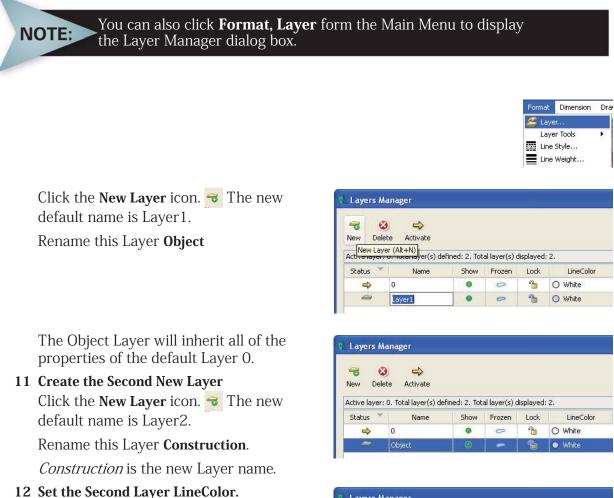
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In this section, you will create three new Layers called Object, Construction and Centerline. As well as apply custom properties.

10 Create the First New Layer.

Click the Layers Manager icon 📓 on the Layers toolbar

The Layers Manager dialog box is displayed.



New Layers are automatically assigned the LineColor white.

Double-click the **LineColor** cell for the Construction Layer.

Click Red.

Layers M	anager				
New Dele Active layer:	te Activate 0. Total layer(s) de	fined: 3. Tot	al layer(s) o	lisplayed	: 3.
Status 🔻	Name	Show	Frozen	Lock	LineColor
4	0		0	2	O White
<i>—</i>	Object		0	-	O White
0	Construction	0	0	ъ	O White
<					 Blue Cyan Green Magenta
Reverse I	Filter n use" status				 Red White Yellow Specify Color.

Continuo...lid line

Continuo...lid line

Continuous 👘 Sc 🔽

Continuo...lid line

O White

O White

🔵 Red

13 Create the Second Layer LineStyle. Create a MM HIDDEN **LineStyle**.

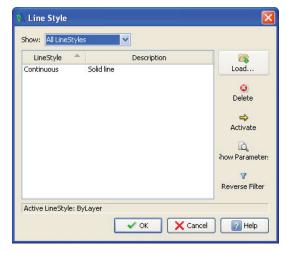
Double-click the **LineStyle** cell from the Construction Layer.

 $C(t_{1}, t_{2}, t_{1}) = C(t_{1}, t_{2}, t_{2}) + C(t_{1}, t_{2}, t_{$

Click Other. The LineStyles Manager dialog box is displayed.

Click the Load icon.

The Load LineStyles dialog box is displayed



The default LineStyle file: inch.lin

Click the **Browse** button.

NOTE:

Active LineStyle f	ile: inch.lin Browse	 ,
LineStyle 🐣	Description	-
BATTING	Batting SSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSS	
BORDER	Border	
CENTER	Center	
DASH2	DashLong	
DASH2DOT	DashDotDot	-
DASHDOT	DashDot	
DASHED	Dashed	
DIAMOND	Diamond ####################################	
DIAMONDSmall	DiamondSmall ###################################	
DIVIDE	DashDotDot	
DOTTED	Dotted	
FENCELINE	Fenceline00000	*

Select **MM.LIN** from the **Specify LineStyle File** dialog box.

Click Open from the Specify LineStyles File dialog box.



Select **HIDDEN** from the **Load LineStyles** dialog box.

Click OK.

LineStyle 🐣	Description	^
FENCELINE1 FENCELINE2 FENCELINE21 FIVELINE GAS LINE	Fence circle0000- Fence square[][][][] Fence square 45[][][][][FIVELINE =5=5=5=5=5=5=5=5=5= Gas lineGA5GA5GA5GA5	
HIDDEN	Hidden	
HOT_WATER PHANTOM	Hot water line HW HW HW Phantom	
SINUS STAR4	SINUS ~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~	-
STAR4Rot45 TRACK	Star4R45 Tracks - - - - - - - - - - - - - - - - - - -	~

HIDDEN is added to the **LineStyles** dialog box.

LineStyle	Descript	ion 🛛 🙀
Continuous HIDDEN	Solid line Hidden	Load Delete Activate Show Parameters Reverse Filter

14 Create the Second Layer LineStyle. Create a MM CENTER **LineStyle**.

Click the Load icon.

View the **Load LineStyles** dialog box. The default LineStyle file: inch.lin.

Show: All LineStyle	s 💉	
LineStyle 🐣	Description	
Continuous HIDDEN	Solid line Hidden	Load
TIDDEN	Hidden	•
		Delete
		4
		Activate

Click the **Browse** button.

ctive LineStyle f	ile: inch.lin Browse	
LineStyle 🔺	Description	^
BATTING	Batting SSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSS	
BORDER	Border	
CENTER	Center	
DASH2	DashLong	
DASH2DOT	DashDotDot	-
DASHDOT	DashDot	
DASHED	Dashed	
DIAMOND	Diamond ####################################	
DIAMONDSmall	DiamondSmall ###################################	
DIVIDE	DashDotDot	
DOTTED	Dotted	
FENCELINE	Fenceline00000	~

Select **MM.LIN** from the **Specify LineStyle File** dialog box.

Click Open from the Specify LineStyles File dialog box

Click **CENTER** from the **Load LineStyles** dialog box.

Click OK.



Specify LineStyle File

Look in: 🛅 Linestyles

R

Load..

3 Delete

CENTER is added to the **Line Styles** dialog box.

Click OK.

- 15 Click OK from the Line Style Dialog Box.
- **16 View the Available New LineStyles** Double-click inside the **LineStyle** cell.

View the HIDDEN and CENTER **LineStyle** from the drop-down menu.

Status 🍸	Name	Show	Frozen	Lock	LineColor	LineStyle	LineWeight
⇒	0	•	0	-	O White	Continuolid line –	Default
<i>a</i>	Object		0	1	O White	Continuolid line –	Default
-	Construction		0	-	🔵 Red	Continuous Sc 🗸 –	Default
						Continuolid line HIDDEN CENTER Other	IN

v

Center

Solid line

Hidden _ _

Description

Line Style

LineStyle

CENTER

HIDDEN

Continuous

Show: All LineStyles

17 Set HIDDEN for LineStyle.

Click $\ensuremath{\textbf{HIDDEN}}$ in the $\ensuremath{\textbf{LineStyle}}$ cell for the Construction Layer.

HIDDEN is the new the **LineStyle** for the Construction Layer.

Click OK.

18 Save the Drawing.

Click the **Save** icon 🔳 on the Standard toolbar.

19 Create the Third New Layer.

Click the Layers Manager icon 📓 on the Layer toolbar.

The Layers Manager dialog box is displayed.

Click the **New Layer** $rac{1}{rac{3}}$ icon. The new default name is Layer 1.

Rename this Layer **Centerline**.

New Dele					
Active laver:	0. Total layer(s) de	fined: 4. Tot	al laver(s) c	lisplaved	: 4.
					(1999)
Status 🍼	Name	Show	Frozen	Lock	LineColor
Status 🎽	Name 0	Show	Frozen	Lock	LineColor O White
Status 🐣	1.670008	Show	I CONTRACT.	Lock	
Status ×	0	Show	0	1	O White

Centerline is the new Layer name. Press the **Enter** key.

- 6					
vew Dele	te Activate				
active layer:	0. Total layer(s) de	fined: 4. Tot	al layer(s) c	isplayed	: 4.
	1.	1102463377	1.000.000.000.000.000.000		a second s
Status 🍸	Name	Show	Frozen	Lock	LineColor
Status 🎽	Name 0	Show	1.000.000.000.000.000.000		a second s
Status 🎽	0.00000	Show ©	1.000.000.000.000.000.000		LineColor
Status 🔻	0	•	Frozen	Lock	LineColor O White

20 Set the Third Layer LineColor.

New Layers are automatically assigned the LineColor white.

Double-click the **LineColor** cell for the **Centerline** Layer.

Click Blue.

21 Set the Third Layer LineStyle.

Double-click the **LineStyle** cell for the Centerline Layer.

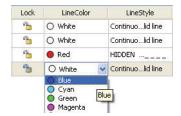
Click CENTER.

CENTER is new the **LineStyle** for the Centerline Layer.

22 Add a Description.

Double-click inside the **Description** cell for the Centerline Layer.

Enter Customer XYZ.



Lock	LineColor	LineStyle	LineWeight
-	O White	Continuolid line –	Default
-	O White	Continuolid line —— Defau	
-	🔴 Red	HIDDEN	Default
-	🔵 Blue	Continuous Sc 💙 –	Default
		Continuolid line HIDDEN	
		CENTER	

PrintStyle	Print	Description
Color_7		
Color_7		
Color_1	-	
Color_5		Customer XYZ

Press the Enter key.

NOTE: At this time, the Default Layer is active.

23 Activate the Object Layer.

Double-click the icon in the **Status** box next to the Object Layer. A

➡ is displayed in the Status box.

Object is now the current Layer. Click **OK**.

View the status of the Layers toolbar and the Properties Palette.

Create the Second Drawing

Rectangle Command

A rectangle drawn using the **Rectangle** command is considered to be a single entity. If the same sized rectangle were drawn using the **Line** command, the resulting rectangle will be four individual unrelated Line entities.

24 Draw a Rectangle Entity.

Click the **Rectangle** icon 🔁 on the Draw toolbar.

Options: <u>C</u>hamfer, <u>E</u>levation, <u>F</u>illet, <u>T</u>hickness, line <u>W</u>idth or

Specify start corner»50,50,

Options: <u>A</u>rea, <u>D</u>imensions, <u>R</u>otation or Specify opposite corner**»150,150**,J



LineColor

O White

• White

Red

O White

LineStyle

Continuo...lid line

Continuo...lid line

Continuo...lid line

Continuo...lid line

To specify the point coordinates with respect to the previous point (relative coordinate input, type the @ character in front of the coordinate value.

Layers Manager

3

Delete

w.

Object

Construction

Centerline

-

New

Status

ω 0

D

Activate

Name

Active layer: Object. Total layer(s) defined: 4. Total layer(s) displayed: 4.

0

0

0

Show Frozen Lock

0

0

0

2

9

2

9

(@100,100 could be used as the relative value)

25 Draw a Circle Entity

Click the **Circle** icon ③ on the Draw toolbar.

NOTE

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Options: <u>3Point</u>, <u>2Point</u>, <u>Itr</u>, <u>TTT</u>, <u>Enter to exit</u> or Specify center point»**70**,**70**,

Options: <u>D</u>iameter or Specify radius»**5**.

26 Turn Snap Off.

Click the **Snap Snap** button on the Status bar.

The command window displays: <Snap Off>.

Pattern Command: Linear Option

The **Pattern** is command provides the ability to create multiple copies of selected entities in a rectangular matrix or a polar, circular pattern. The created entities inherit all properties of the original entities such as Layers, and Layer custom properties.

27 Create a Linear Pattern.

Click the **Pattern** icon **#** on the Modify toolbar.

The **Pattern** dialog box is displayed.

Click the **Linear** button.

Number of elements on Horizontal axis: Enter 2.

Number of elements on Vertical axis: Enter 2.

Spacing between elements on:

Horizontal axis: Enter 60.

Vertical axis: Enter 60.

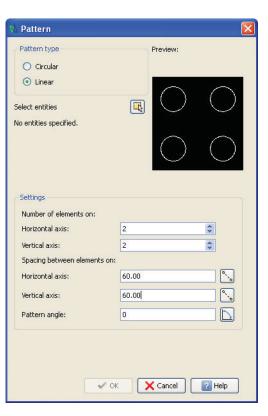
Click the **Select entities** icon.

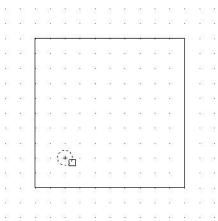
Select the circumference of the Circle on the screen.

1 found, 1 total

Specify entities»,J

Click OK.





Chamfer Command

The **Chamfer** command allows you to connect two Lines with a beveled corner. The selected entities do not have to intersect, and they can overlap. If the entities overlap, they will be trimmed (by default) to create the beveled corner.

Since there are often non-symmetrical chamfers to be drawn, a chamfer distance from the corner of adjacent Lines can be specified for both of the intersecting entities. You can also define the connecting Line via a length and an angle.

8 8	35	1	33	35	35	1	1	10	8	85 B	8	3	8
85 781	22	82	22	1	2	127	33	20	12	<u>195</u> - 3	1	21	3
8 18	Ē	12	12	- 2	30	2		1	8/	12	ĩ	2	3
8 81	8	81	12	92	52	12		. :	18	12	1	8	10
8 8	8	12	\bigcirc	<u>.</u>	12		i.	. 1	\odot	12		3	8
8 8	2	14	5	12	12	$\langle g \rangle$			7	13		3	85
9 8		14	-	×	14			æ	$\hat{\mathbf{x}}_{i}^{i}$	-		8	9
8 (s		8	12		×			80	85	-		68	8
5 32	8	13	3	*	13	82	8			58		2	12
5 32	8	13		8		\sim				28		2	12
8 82		8	(\cdot)		8	8		eð.	\odot	-		27	22
8 3		8	13.	*	3	١.	\$ 3		- 58	12		S.	8
8 8	F	1	132	*	32	100	101	101	8	12		8	8
8 8	22	82	22	4	1	127	23	20	10	25 3	1	21	8
8 84	83	33	32	12	12	8	85	85	12	83 8	8	3	81

28 Create a Chamfer.

Create a 10mm multiple chamfer using the Distance trim mode option.

Click the **Chamfer** icon \frown on the Modify toolbar.

(TRIM mode) Active chamfer Dist1 = 0.00, Dist2 = 0.00

Options: <u>A</u>ngle, <u>D</u>istance, m<u>E</u>thod, <u>M</u>ultiple, <u>P</u>olyline, <u>T</u>rim mode, <u>U</u>ndo or Specify first line» d_{-}

Defaul t: 0. 00

Specify first distance: 10 →

Defaul t: 10.00

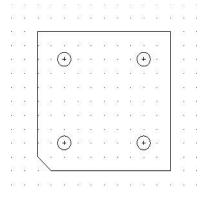
Specify second distance: ↓

Options: <u>A</u>ngle, <u>D</u>istance, m<u>E</u>thod, <u>M</u>ultiple, <u>P</u>olyline, <u>T</u>rim mode, <u>U</u>ndo or

Specify first line≫m⊣

Options: <u>A</u>ngle, <u>D</u>istance, m<u>E</u>thod, <u>M</u>ultiple, <u>P</u>olyline, <u>T</u>rim mode, <u>U</u>ndo or Specify first line»

Click the lower left Vertical Line in the graphics area.

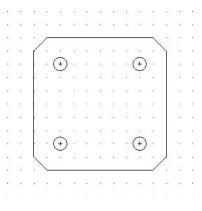


Options: *Shift+ select to apply corner* or Specify second line ↓

Click the lower bottom Horizontal Line in the graphics area.

ptions: <u>Angle</u>, <u>D</u>istance, <u>mEthod</u>, <u>Multiple</u>, Polyline, Trim mode, Undo or Specify first line»

Repeat the above procedure for the three other corners. Press **Esc** or **Enter** to exit.

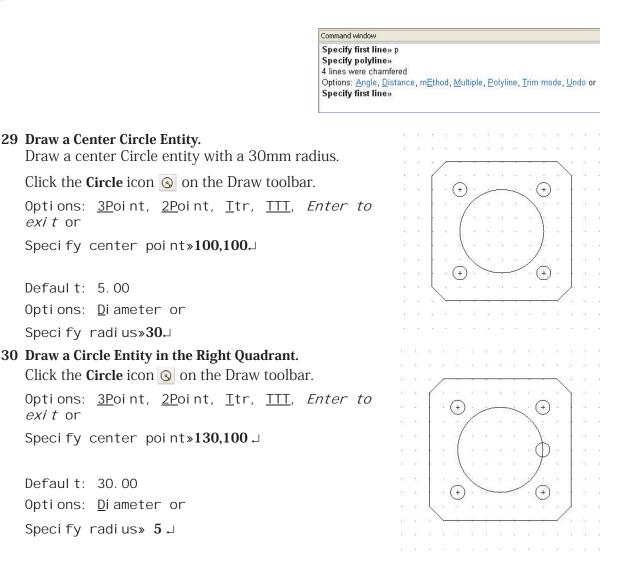




exit or

exit or

Using the PolyLine option, all four corners will be chamfered at once. The purpose of this book is to expose you to different methods and tools.



31 Save the Drawing.

Click the **Save** icon 🔳 on the Standard toolbar.

Pattern Command: Circular Option

The Pattern IIII command creates multiple copies of entities in a pattern (arranged in rows and columns, on a Circle, or on an Arc).

In the next section, you will create a circular pattern.

32 Draw a Circular Pattern.

Click the **Pattern icon m** on the Modify toolbar.

The Pattern dialog box is displayed.

Click the **Circular** button.

Base pattern on: set to Fill Angle and Total Number of Elements

Fill angle: Enter 360

Total number: Enter 6.

Axis point X: Enter 100.00.

Axis point Y: Enter 100.00

Click the **Select entities** icon.

Select the circumference of the small Circle that sits on the large Circle.

1 found, 1 total

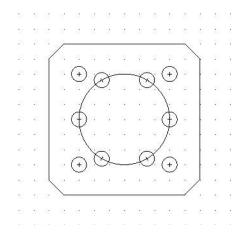
Specify entities»⊣

Click OK.

33 Save the Drawing.

Click the **Save** icon 🔲 on the Standard toolbar.

🕅 Pattern	
Pattern type Circular Linear 	Preview:
Select entities No entities specified	
Settings	
Base pattern on:	Fill Angle and Total Number of Elements
Angle between:	60
Fill angle:	360
Total number:	6
-Element base poin V Use last entity X: 0.00 Y: 0.00 V Orient elements	v selected X: 100 Y: 100 Y: 100



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34 Move the Large Circle to the Centerline Layer. Click the circumference of the large Circle in the graphics area. Click the drop-down arrow from the Layers Manager. (+ Click Centerline. 🔒 🔍 🥕 🛍 🛍 🖊 🍤 (° à Centerline 0 0 ω., 造 🔴 Construction 1 O Object Press Esc or Enter to exit. 35 Save the Drawing. Click the **Save** icon 🔲 on the Standard toolbar. Create the Third New Drawing Create two Layers. Use the Circle (r), and Circle (Ttr) Draw tools.

The **Circle** (**Ttr**) Draw command constructs a Circle from two tangent entities and the specified raduis.

NOTE:

Apply the Trim, Offset, and Pattern (circular) modify commands,

Create an Object and a Phantom Layer. Set custom properties.

1 Create a New Drawing. Click the New icon 🝙 on the Standard toolbar.

Drawing Template

2 Select a Default Drawing Template. Double-click standard.dwt from the Select Template dialog box.

Set the Drawing Environment

Drawing Unit System

The default units system is the Decimal system and can be applied to either English (inches) or metric values (millimeters).

3 Set Linear Length Precision.

The **Unit System** section of the **Options** - **Drawing Settings** dialog box allows you to set the linear and angular unit formats.

Click Format, Unit System... from the Main Menu.

The **Unit System** portion of the **Options - Drawing Settings** dialog box is displayed.

Click 0.00 for Precision in the Length section of the dialog box.

4 Set Linear Units Scale.

The **Units scale** section of the **Unit System** dialog box allows you to set the linear and angular unit formats.

Click **Millimeters** for **Block units format** in the **Units scale** section of the dialog box.

Click OK.

Set Drawing Boundary

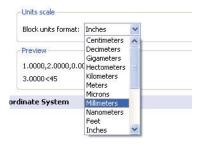
Use a horizontal A4 default drawing sheet.

5 Set Drawing Boundary. Click Format, Drawing Boundary from the Main Menu.

Default: (0.00,0.00) Options: <u>OF</u>f, <u>ON</u> or Specify lower left corner» ↓

Default: (12.00,9.00) Specify upper right corner» **297,210**,







Set Snap and Grid

Set the **Snap** and **Grid**. In this section the Grid and Snap spacing are equal.

6 Set Snap and Grid.

Click Tools, Options from the Main Menu.

The **Options** dialog box is displayed.

Click the **User Preferences** tab.

Expand Drafting Options, Pointer Control and Snap Settings.

Click the **Enable Snap** <**F9**> box.

Enter the value of **10** in the **Horizontal Snap spacing** box. Press the **Tab** key The **Vertical Snap spacing** box is updated.

Expand Display and Grid Settings

Click the **Enable Grid** <**F7**> box and the **Match Snap Spacing** box. The **Horizontal display spacing** value and the **Vertical display spacing** value will be set to **10.00**.

Click OK.

Zoom Bounds

- 7 **Display the Drawing with the New Boundary.** Drawing boundaries need to be used to adjusted to the current view.
- 8 Select View, Zoom, Bounds 🔍 from the Main Menu.
- 9 Save the Drawing.

Click the **Save** icon 🗟 on the Standard toolbar.

Enter File name: Lesson 2-3.

Click Save.

Create New Layers for the Drawing

In this section, you will create three new Layers called Object, Construction and Centerline. As well as apply custom properties.

10 Create the First New Layer.

Click the Layers Manager icon 📓 on the Layers toolbar

The Layers Manager dialog box is displayed.

Click the **New Layer** icon. **G** The new default name is Layer1.

Rename this Layer **Object**

Rew Dele	te Activate				
Active layer.	er (Alt+N) o. notamayer(s) de	fined: 2. Tot	al layer(s) c	lisplayed	2.
Active layer Status	er (Alt+N) o. iocanaver(s) de Name	fined: 2. Tot Show	al layer(s) c Frozen	lisplayed Lock	: 2. LineColo
A	I PALANANANA	LE VINCENTER	1.0000000000000000000000000000000000000		

	s	Window	Help	
A	Pro	perties		Ctrl+1
	Ref	erences		
	Disp	olay Order		ŀ
	Inq	uiry		•
	Ope	en Referer	nce	
	Run	Script		
耳	CCS	5 Manager		
	Nev	V CCS		•
	Cus	tomize Int	erface.	
	Opt	ions	•	←

The Object Layer will inherit all of the properties of the default Layer 0.

11 Create the Second New Layer

Click the **New Layer** icon. **G** The new default name is Layer2.

Rename this Layer **Phantom**.

Phantom is the new Layer **Name**.

12 Set the LineColor.

New Layers are automatically assigned the LineColor white.

Double-click the **LineColor** cell for the Phantom Layer.

Click Red.

13 Create a LineStyle.

Create a MM PHANTOM **LineStyle**. Double-click the **LineStyle** cell from the Phantom Layer.

Click Other. The LineStyles Manager dialog box is displayed.

Click the Load icon

The default LineStyle file: inch.lin.

Click the **Browse** button.

Select MM.LIN for LineStyle. Double-click MM.LIN from the LineStyle File dialog box.

Click **PHANTOM** from the **Load LineStyles** dialog box.

Click **OK**.

PHANTOM is added to the **LineStyles Manager** dialog box.

Click OK.

14 Set PHANTOM for LineStyle.

Click **Phantom** in the **LineStyle** cell for the Phantom Layer.

PHANTOM is the new the **LineStyle** for the Phantom Layer.

Click OK.

15 Save the Drawing.

Click the **Save** icon \blacksquare on the Standard toolbar.

Layers M	anager				
New Dele					
Active layer: Status	0. Total layer(s) de Name	fined: 2. Tot	al layer(s) o Frozen	lisplayed Lock	: 2. LipeColor
Status	1.000000	SHUW	Fruzen		an re coror
₽	0	•	0	1	O White
				P	White



ctive LineStyle f	ile: Data\DraftSight\1.2.55\Linestyles\MM.LIN Brows	e
LineStyle 🐣	Description	^
FIVELINE	FIVELINE =5=5=5=5=5=5=5=5=5=	
GAS_LINE	Gas lineGASGASGASGAS	
HIDDEN	Hidden	
HOT_WATER	Hot water line HW HW HW	
PHANTOM	Phantom	
SINUS	SINUS ๛๛๛๛๛๛๛๛๛๛๛๛๛๛๛๛๛	
STAR4	Star4	
STAR4Rot45	Star4R45	
TRACK	Tracks - - - - - - - - - - - - - - - - - - -	
TRILINE	TriLine =3=3=3=3=3=3=3=3=3=	
XLINE	XLine X-X-X-X-X-X-X-X-X-X-X-X-X-X-X-X-X-X-X-	-
ZIGZAG	Ζία zaa ΛΛΛΛΛΛΛΛΛΛΛΛΛΛΛΛΛΛΛΛ	~

	Other	NTOM
	PHANTOM	
	Continuolid line	r -
🔵 Red	Continuous Sc 🐱	Default
O White	Continuolid line	—— Default
O White	Continuolid line	Default
LineColor	LineStyle	LineWeight

The Default Layer and PHANTOM Layer is On. NOTE: Construct a Circle from Two Tangent Circles and the Radius 16 Draw the First Circle Entity. Draw a Circle from two tangent entities and the radius. Click the **Circle** icon **(S)** on the Draw toolbar. Options: <u>3Point</u>, <u>2P</u>oint, <u>T</u>tr, <u>TTT</u>, *Enter to exit* or Specify center point»90,50, J Options: Diameter or Specify radius»20, J 17 Draw the Second Circle Entity. Command window Options: <u>3Point, 2Point, Ttr, TTT</u>, Enter to exit or : പ Specify center point» 180 50 Options: <u>3Point</u>, <u>2Point</u>, <u>Ttr</u>, <u>TTT</u>, <u>Enter to exit</u> Default: 20.00 Options: Diameter or or Specify radius» 10 Specify center point»180,50 → Options: Diameter or Specify radius»10, J

18 Draw the Third Circle Entity. \therefore

Options: <u>3P</u>oint, <u>2P</u>oint, <u>T</u>tr, <u>TTT</u>, *Enter to exit* or

Specify center point»ttr.J

Specify first tangency»

19 Turn Snap Off.

Click the **Snap Snap** button on the Status bar. The command window displays: <Snap Off>.

Click the first tangent point on the left Circle as illustrated.

Specify first tangency»

Click the second tangent point on the right Circle.

Defaul t: 10.00

: പ

Specify radius»50, J

20 Draw the Fourth Circle Entity.

Options: <u>3P</u>oint, <u>2P</u>oint, <u>T</u>tr, <u>TTT</u>, *Enter to exit* or

Specify center point»ttr.J

Specify first tangency»

Click the first tangent point on the left Circle as illustrated.

Speci fy second tangency» Click the second tangent point on the

right Circle.

Defaul t: 10.00

Specify radius»80,

21 Save the Drawing.

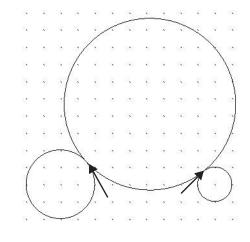
Click the **Save** icon 📕 on the Standard toolbar.

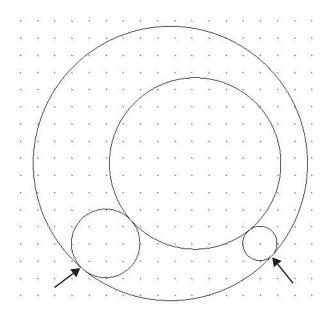
22 Trim the Top Circle.

Click the **Trim** icon 💥 on the Modify toolbar.

Specify cutting edges...

Options: Enter to specify all entities or





Specify entities»,J

Options: <u>C</u>rossing, <u>CR</u>ossline, <u>P</u>roject, <u>E</u>dge, e<u>R</u>ase, <u>U</u>ndo, <u>Shift + select</u> to extend or

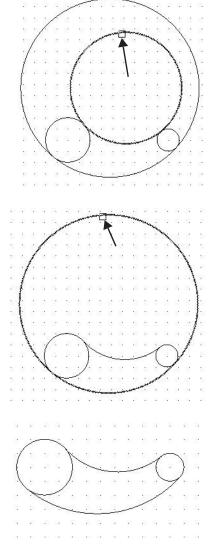
Specify segments to remove»

Click the top section of the 50mm Circle.

Options: <u>C</u>rossing, <u>CR</u>ossline, <u>P</u>roject, <u>E</u>dge, e<u>R</u>ase, <u>U</u>ndo, *Shift + select to extend* or Specify segments to remove»

Click the top section of the 80mm Circle.

Options: <u>C</u>rossing, <u>CR</u>ossline, <u>P</u>roject, <u>E</u>dge, e<u>R</u>ase, <u>U</u>ndo, *Shift* + *select to extend* or Specify segments to remove».



23 Save the Drawing.

Click the **Save** icon 🔚 on the Standard toolbar.

Offset Command

The Offset 🗊 command provides the ability to create parallel shapes of Lines, PolyLines, Circles, Arcs,

Ellipses, Elliptical Arcs, Splines, and InfiniteLines. A copy of a selected entity is placed at a specified distance from the original entity. The original entity stays in position.

24 Apply Offset to the Drawing.

Offset the two inside Circles.

Click the **Offset** icon is on the Modify toolbar.

Default: 1.00

Options: <u>D</u>elete, <u>D</u>istance, destination <u>Layer</u>, <u>T</u>hrough point or Specify distance»2.

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Defaul t: Exit, Options: <u>E</u>xit, <u>U</u>ndo or Specify source entity» Click the circumference of the left Circle in the graphics area.

Default: Exit Options: <u>B</u>oth sides, <u>E</u>xit, <u>M</u>ultiple, <u>U</u>ndo or Specify side for destination» Click inside of the left Circle.

Default: Exit, Options: <u>E</u>xit, Undo or Specify source entity» Click the circumference of the right Circle in the graphics area. Default: Exit Options: <u>B</u>oth sides, <u>E</u>xit, <u>M</u>ultiple, <u>U</u>ndo or Specify side for destination» Click inside of the right Circle.

Specify source entity»↓

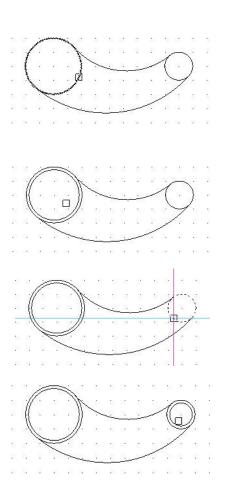
25 Trim the Drawing.

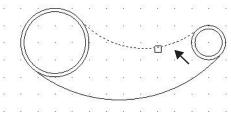
Trim the two inside Circles and two Arcs.

Options: *Enter to specify all entities* or Specify entities» Select the top Arc.

1 found, 1 total

Options: *Enter to specify all entities* or Specify entities»





Select the bottom Arc. 1 found, 2 total Options: Enter to specify all entities or · 🗖 Specify entities»↓ Options: Crossing, CRossline, Project, Edge, eRase, Undo, Shift + select to extend or Specify segments to remove» Click the right side of the left Circle. Options: Crossing, CRossline, Project, Edge, eRase, Undo, Shift + select to extend or Specify segments to remove» Click the left side of the right Circle. Options: Crossing, CRossline, Project,

Edge, eRase, Undo, Shift + select to extend or

Specify segments to remove»J

Pattern Command: Circular Option

The Pattern III command creates multiple copies of entities in a pattern (arranged in rows and columns, on a Circle, or on an Arc). In the next section, create a circular pattern.

26 Draw a Circular Pattern.

Click the **Pattern** icon **m** on the Modify toolbar.

The Pattern dialog box is displayed.

Click the **Circular** button.

Base pattern on: set to **Select Fill Angle and Total Number of Elements**

Fill angle: Enter 180.

Total number: Enter 5.

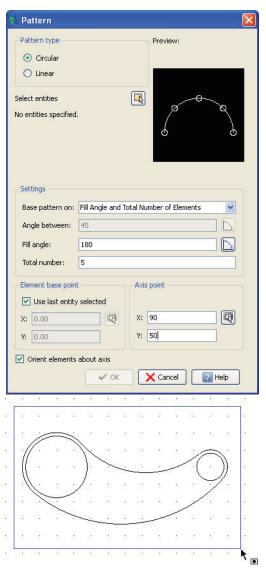
Click the **Select entities** icon.

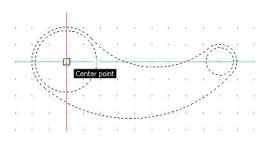
Specify entities» Windows select all entities 6 found, 6 total»

Specify entities»,J

Click the **Axis Point** icon from the **Pattern dialog**

Specify pattern center point» Click the center point of the large Circle in the graphics area.





Click OK.

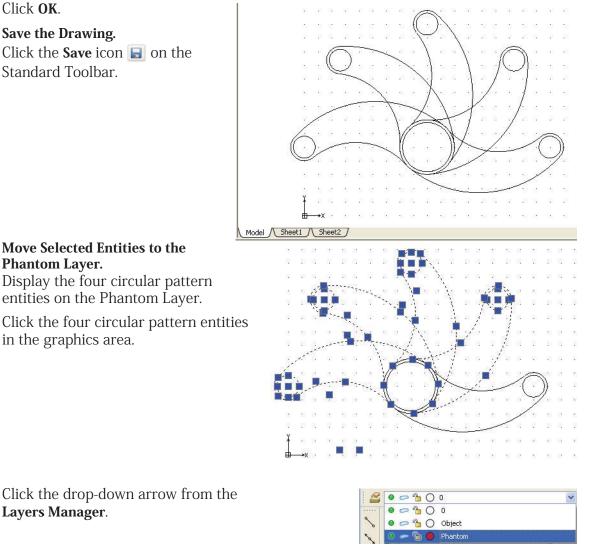
27 Save the Drawing. Click the **Save** icon 🔲 on the Standard Toolbar.

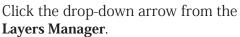
28 Move Selected Entities to the

Display the four circular pattern entities on the Phantom Layer.

Phantom Layer.

in the graphics area.



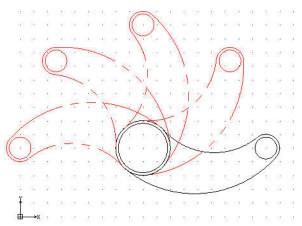


Click **Phantom**.

Press Esc or Enter to exit.

29 Save the Drawing.

Click the **Save** icon 🔲 on the Standard toolbar.



You are finished with this section.

Lesson 3 BASIC DIMENSIONING

When you complete this lesson, you will be able to:

- Create two drawings with custom layer properties
- Use absolute coordinates the command prompt and status bar
- Modify DimensionStyles and precision
- Edit Dimensions and Dimension text
- Measure and rotate an Aligned Dimension
- Apply the Move and Pattern Modify commands
- Insert, Modify and Create a Pattern of CenterMarks
- Apply the Linear, DimensionStyle, Angular, Radius, Aligned, DimensionEdit, DimensionTextEdit, Ordinate, Continue, and CenterMark Dimension commands
- Reset the Coordinate System

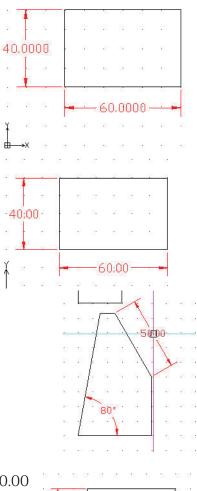
Basic Dimensioning

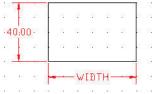
During this lesson, you will rename a drawing, modify an existing drawing and apply Dimensions using absolute coordinates in the Command prompt and the graphics area.

In the first drawing you will:

- Open an existing drawing from Lesson 1(Lesson 1-3).
- Apply the **Save As** command and rename it Lesson 3-1.
- Create a new layer called Dimension with custom properties.
- Apply the DimensionStyle a command and the Linear command to create a linear horizontal and vertical Dimension the Dimension layer.
- Modify the precision of the two Dimensions.

- Apply the **Angular ** command to create an 80 degree Angular Dimension.
- Apply the **Aligned** *include* command to create an Aligned Dimensionon the top right angular Line.
- Apply the **DimensionEdit** a command to modify the 60.00 Linear Dimension to the WIDTH text
- Measure the Aligned Dimensionangle.





BASIC DIMENSIONING

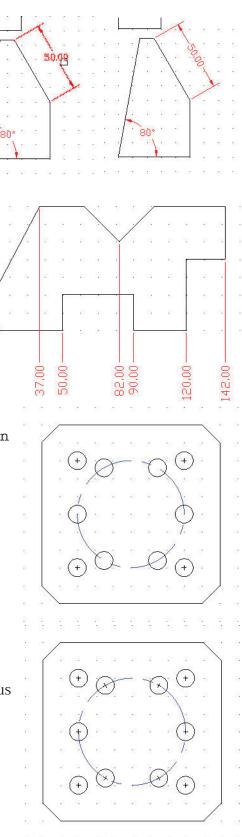
- Apply the **DimensionTextEdit** command to rotate the 50.00 Aligned Dimension text match the angle of an entity.
- Reset the coordinate system (CS) from the default position.
- Apply the **Ordinate** command to apply Dimensions on the top entity.

In the second drawing you will:

Open an existing drawing from Lesson 2 (Lesson 2-2)

00'0

- Create a new layer called Annotations with custom properties.
- Apply the **DimensionStyle** a command to set properties.
- Modify the default CenterMark size.
- Apply the **CenterMark** ④ command and insert CenterMarks on the Annotations layer in the drawing.
- Apply the **Pattern m** command to create a pattern of CenterMarks.
- Activate the Annotations layer.
- Apply the **Radius** (a) command to create a Radius Dimension.
- Modify the default precision setting.



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Start a DraftSight Session

1 Start a DraftSight Session.

From the Windows Start menu, click Start, All Programs, Dassault Systemes, DraftSight.

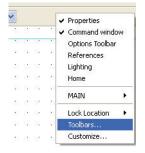
Dimension Toolbar

DraftSight provides a full range of Dimensioning tools and utilities. You can use a variety of Dimensioning tools to display measurements of entities or relations between them in your drawings.

By default, Dimensioning is associative. If you modify the geometry of drawing entities, Dimensions associated to them reflect the changes.

Display the Dimension Toolbar.

2 Right-click the Main menu or a docked toolbar. Click Toolbars.... The Specify Toolbars dialog box is displayed.



Check the **Dimension** box. Click **OK**.

CCS Dimension	Customize
Display Order	😮 Delete
Draw	
Entity Snap	
Inquiry	
Insert Layer Tools	
Layers	
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Modify	
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You can access the Dimension commands from the Dimension pull-down menu. Almost all of the Dimension commands are also found on the Dimension toolbar. Access on-line Help for detail	Dimension Draw Mod Image: Aligned Image: Aligned
information on all commands.	 → Baseline → Continue ◇ Ordinate
	 CenterMark Diameter Radius
	Angular Arc Length Jogged
	Arx Leader
	Align Text
	Relate Dimension
	💾 Rebuild

- Aligned: (ParallelDimension) Creates an aligned Linear Dimension. Aligned Dimensioning is a type of Linear Dimensioning that measures and labels the absolute distance between two Points, regardless of the relative position of the axes. This lets you Dimension Lines, edges, or spaces, that are not located on the principal axes of the coordinate system, that is not parallel to the X or Y axes of the user coordinate system (UCS).
- **Linear**:(LinearDimension) Creates a Horizontal, Vertical, or Rotated Dimension.
- **Baseline**: (BaselineDimension) Creates a series of Parallel Linear Dimensions that share the same baseline. Baseline Dimensioning allows you to read the cumulative Dimension between two end points, extending over a number of individual Dimensions within a chain.

- Continue: (ContinueDimension) Continues a Linear, Angular, or Ordinate Dimension from the second Extension Line of the previous or a selected Dimension. Continuous Dimensioning allows one long Dimension to be broken into shorter segments that add up to the total measurement.
- Ordinate: (OrdinateDimension) Creates ordinate Point Dimensions. The command displays an X or Y coordinate with a Leader Line to create X-datum or Y-datum Dimensioning. The value of the X or Y coordinate is defined by the Point you specify, called feature position.
- CenterMark: (centermark) Creates a Center Mark or Center Lines of Circles and Arcs. Center marking is used to position a mid point mark (+) or broken Lines crossing at the center of a Circle or an Arc.
- Diameter: (DiameterDimension) Creates diameter Dimensions for Circles and Arcs. Unlike the Linear Dimensioning applied on Circles (which also display the diameter), no Dimension Lines are shown. The command creates a single Leader Line to show the relationship between the value and the entity itself.
- Radius: (RadiusDimension) Creates radial Dimensions for Circles and Arcs. Radius Dimensioning is carried out in the same way as diameter Dimensioning, the only difference being that a radius Dimension is calculated instead of a diameter measurement.
- Angular: (AngleDimension) Creates an Angular Dimension. The command lets you create the Dimension of both the interior angle between the two legs and of the exterior angle.
- Arc Length: (ArclengthDimension) Creates an Arc Length Dimension. It measures the distance along an Arc or PolyLine Arc segment. To differentiate Arc Length Dimensions from Linear or Angular Dimensions, an Arc symbol is displayed with the Dimension text.
- **Jogged**: (JoggedDimension) Creates jogged radius Dimensions for Circles and Arcs. Jogged Leader Lines are typically used when a sheet is too small to display the true center point of a Radial Dimension.
- Leader: (leader) Creates Leader Lines that connect annotations to a drawing entity. Creating a Leader is often useful when the Dimension text or annotation does not fit next the drawing entity it measures or describes.
- **Tolerance**: (tolerance) Creates tolerances (datum indicators and Basic Dimension notation).
- Dimension Edit: (EditDimension) Changes the position and the angle as well as the value of Dimension text. In addition, you can modify the orientation of the Dimension Extension Lines.
- Dimension Text Edit: (EditDimensionText) Moves and rotates Dimension text of a single Dimension entity. The default sequence of the command lets you position Dimension text dynamically.
- **Dimension Rebuild**: (RebuildDimension) Applies the stored Dimension system variable settings to selected Dimensions. Unlike text styles, DimensionStyles do not automatically update when the style is changed. Note that the Dimension update permanently overrides DimensionStyles previously applied to the selected entity.

Dimension Style: (DimensionStyle) Creates and modifies DimensionStyles DimensionStyles are the main method used to control the way Dimensions look

Dimension Terminology

Dimension Line

The Dimension Line is normally offset for clarity from the measured feature. The Dimension Line indicates the direction and length of the measured distance. Dimension Lines are usually terminated with arrows. Angular Dimension Lines become Arcs whose centers are at the vertex of the angle.

Arrow

The arrow is a mark at the end of a Dimension Line. The arrow indicates its termination. DraftSight provides the ability to apply various shapes other than arrows in some styles.

Extension Line

Extension Lines, sometimes called witness Lines indicate the offset of the Dimension Line from the measured feature. Normally Extension Lines are perpendicular to the direction of the measurement.

Dimension Text

The Dimension text can consists of words, characters, numbers, and symbols used to display the measured type and value of the Dimension. Dimension text format conforms to the same linear and angular units as the default of the drawing. The text style conforms to the current text style.

Leader

The Leader is a radial Line used to Point from the Dimension text to the Arc or Circle whose diameter or radius is being Dimensioned.

Center Mark

The center mark is used to position a mid point mark (+) or broken Lines crossing at the center of a Circle or an Arc.

Open an Existing Drawing

3 Click the **Open** icon **a** on the Standard toolbar. The **Open** dialog box is displayed.

BASIC DIMENSIONING

4 Double-click Lesson 2-3. The drawing is displayed in the graphics area.

Recent	dump A-54643 We A-55869 We B-44563 We B-64529 We B-64543 We B-76843 We B-89987 Ve C-89764 Ve C-89764		Lesson 1-3 tria Lesson 2-1 tria Lesson 2-3 tria Lesson 3-1 tria Lesson 4-1 tria Lesson 4-2 tria Lesson 4-2 tria TrainingPhas			Preview	7
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E	File name:	Lesson 1-3		~	Open 🕨		

5 Save the Drawing.

Click File, Save As from the Menu bar.

Enter File name: Lesson 3-1.

Click Save from the Save As... dialog box.



Use the **Save As** command to save the drawing under a different name or in a different location.

Create a New Layer

NOTE

In this section, create a new layer called Dimension with custom properties.



6 Create a new Layer.

Click the Layers Manager icon. So The Layers Manager dialog box is displayed.

Click the **New Layer** icon. 🤿 The new default name is Layer 1.

Rename this layer **Dimension**.

Dimension is the new layer name.

BASIC DIMENSIONING

7 Set the LineColor.

Double-click the **LineColor** cell for the Dimension layer.

Click Red.

Status 🍸	Name	Show	Frozen	Lock	LineColor
⇒	0	•	0	-	O White
<i>a</i>	Object	•	0	<u></u>	O White
<i>—</i>	Phantom	0	0	-	🔴 Red
~	Dimension		0	-	O White
					 Blue Cyan Green Magenta Red
					O White O Yellow

NOTE: By default, *Continuous* LineStyle is selected.

8 Activate the Dimension Layer. Click the Dimension layer in the Layers Manager dialog box.

Click the **Activate Layer** button. A yellow arrow is displayed in the Dimension **Status** box.

9 Close the Layers Manager Dialog box. Click OK.

New Dele						
Active layer:	Dimension. Total la	yer(s) define	d: 5. Total l	ayer(s) c	lisplayed: 5.	Enter filter exp
Status 🐣	Name	Show	Frozen	Lock	LineColor	LineStyle
9	0	•	0	1	O White	Continuolid line
<i>a</i>	Defpoints	0	0	-	O White	Continuolid line
9	Object	0	0	-	O White	Continuolid line
9	Phantom	0	0	-	🔴 Red	CENTER
-	Dimension	0	-	4	White	Continuolid line

10 Save the Drawing.

Click the **Save** icon 📕 on the Standard toolbar.

Working with Dimensions

DimensionStyles are the main method used to control the way Dimensions look. You create and manage DimensionStyles by using the **DimensionStyles Manager** dialog box.

Dimension Styles Manager Dialog Box

Use the style name, input and selection dialog to either store the current settings as a Dimensioning style, or to select a predefined style as the default.

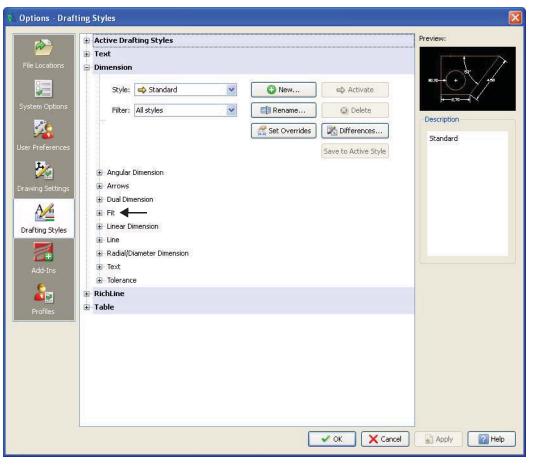
DimensionStyle Command

The **DimensionStyle** command on the Dimension toolbar provides the ability to create and modify DimensionStyles. Remember, DimensionStyles are the main method used to control the way Dimensions look.

11 Set DimensionStyle

Click the **DimensionStyle** icon 🗾 on the Dimension toolbar.

The **Options** - **Drafting Styles** dialog box is displayed with the **Dimensions** option expanded.



Expand the Fit, Dimension scale option.

View the **Fit** options displayed on the right side of the dialog box.

Scale factor: Enter 25.

Click OK.

Fit				
🕒 Geometry				
표 Dimension te:	xt			
Dimension sc	ale			
⊙ Sca	ale factor:	25	-	-
O Sca	ale dimensio	ins acc	ordina t	o sheet

Linear Command

The Linear 🔄 command creates a horizontal, vertical, or rotated Linear Dimension.

One way to use the Linear command is to define two Points of measurement which are the origins of the Extension Lines.

The Command prompt also offers you to select an entity to be Dimensioned. Press Enter (,,) at the prompt of the command. You are then requested to select a drawing entity to be Dimensioned. Lines, PolyLines, Circles and Arcs can be selected. The start and end points of linear entities, or the diameter of the circular entities are evaluated for the measurement.

Com	mand window
1	
Def	ault: Entity
Opt	ions: <u>E</u> ntity or
Sp	ecify first extension line position

Once the Points of measurement are defined, you are prompted to indicate the position for the Dimension Line.

The distance between the two Points is measured along the X axis for a Horizontal Dimensioning and along the Y axis for a Vertical Dimensioning. For oblique measurements the Rotated option is provided.

The measurement is placed as Dimension text based on the definitions of the current DimensionStyle.

Create Two Linear Dimensions

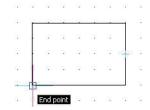
Dimension the rectangle in the graphics area. Create a Horizontal and Vertical Dimension.

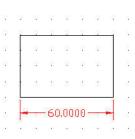
12 Create a Horizontal Dimension.

Click the **Linear** icon 🔄 on the Dimension toolbar.

Default: Entity Options: Entity or Specify first extension line position»

Click the **lower left** end point of the rectangle. Specify second extension line position»





Click the **lower right** end point of the rectangle.

Options: <u>Angle</u>, <u>H</u>orizontal, <u>N</u>ote, <u>R</u>otated, <u>T</u>ext, <u>V</u>ertical or Specify Dimension line position»

Click a **position** about one grid below the bottom Horizontal Line.



The **Note** option provides the ability to modify the Dimension text using the Note editor dialog box.

13 Create a Vertical Dimension.

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TIP:

Press the Enter key (,,) to repeat the command.

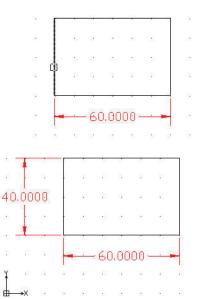
Default: Entity Options: <u>E</u>ntity or Specify first extension line position»↓

Specify entity»

Click the left Vertical Line of the rectangle.

Options: <u>Angle, H</u>orizontal, <u>N</u>ote, <u>R</u>otated, <u>T</u>ext, <u>V</u>ertical or Specify Dimension Line position»

Click a position about two grid spaces to the left of the left Vertical Line. The rectangle now has a Vertical and Horizontal Dimension.



NOTE: The **Angle** option changes the angle of the dimension text.

Modify Dimension Precision Display

The present Vertical and Horizontal Dimensions display four decimal places. Modify the precision of the Dimensions.

🕀 Angular Dimension
Arrows
😥 Dual Dimension
🕀 - Fit
🕀 Linear Dimension 🛛 🚽
🕀 Line
😥 Radial/Diameter Dimension

14 Modify Dimension Precision.

Click the **DimensionStyle** icon is on the Dimension toolbar. The **DimensionStyles Manager** dialog box is displayed.

Expand the Linear Dimensions option.

View the Linear Dimensions options displayed on the right side of the dialog box.

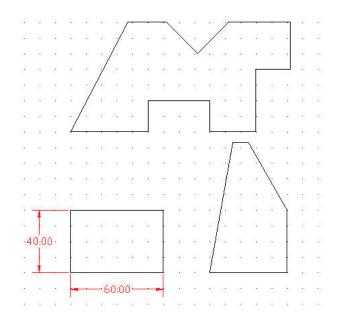
Precision: Select **0.00** from the drop-down menu.

Click OK.

Format:	Decimal	~		
Precision:	0.0000	~		
Fractional display:	0 0.0			
Decimal separator:	0.00			
Round to the nearest:	0.0000			
Prefix:	0.000000			
Suffix:	0.0000000 0.00000000			
Measurement scale:				
Scale factor:	1.00	\$		
Zeroes display:				
🔲 Follow sheet dimensi	ons			
🔲 Hide leading zeroes	📝 Hide if 0'			
Hide trailing zeroes	Ide if 0"			

15 Save the Drawing.

Click the **Save** icon **a** on the Standard toolbar.



Angular Command

The **Angular** \square command provides the ability to Dimension angles in your drawing. The tool creates the Dimension of both the interior angle between the two legs and of the exterior angle using various methods:

- Three Points: (vertex/point/point)
- Between two nonparallel Lines
- On an Arc between the two end points of the Arc, with the center as the vertex
- On a Circle, between two Points on the Circle, with the center as the vertex

Create an Angular Dimension

Create an 80 degree Angular Dimension.

16 Create an Angular Dimension.

Click the **Angular icon** \square from the Dimension toolbar.

Options: *Enter to specify vertex* or Specify entity»

NOTE: Be sure the **SNAP** is set to off. Click the **Snap** button on the status bar.

View the The command window for the display: <Snap Off>. Click the Horizontal Line as illustrated.

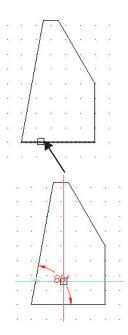
Specify second line»

Click the left bottom Angular Line.

Options: <u>Angle</u>, <u>Note</u>, <u>Text</u> or Specify dimension position» Click a position inside the entity as illustrated.

17 Save the Drawing.

Click the **Save** icon 🗐 on the Standard toolbar.



Aligned Command

The **Aligned** command provides the ability to create an aligned Dimension. Aligned Dimensioning is a type of linear Dimensioning that measures and labels the absolute distance between two Points.

Create an Aligned Dimension

Create an aligned Dimension on the top right Angular Line.

18 Create an Aligned Dimension.

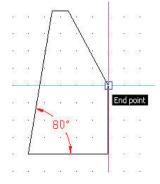
Click the **Aligned** icon 🥑 on the Dimension toolbar.

Default: Entity Options: <u>E</u>ntity or

Specify first extension line position»

Click the bottom right end point of the right side Diagonal Line as illustrated.

Specify second extension line position»



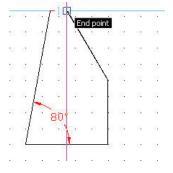
Click the top left end point of the right Diagonal Line as illustrated.

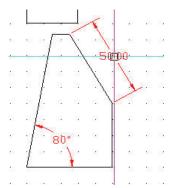
Options: <u>A</u>ngle, <u>N</u>ote, <u>T</u>ext or Specify dimension line position» Place the Dimension.

Click a position above the Line as illustrated.

Edit Dimensions and Dimension Text

DraftSight provides several Dimensioning commands which allows you to edit or format the Dimension text during the creation of the current Dimension.





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DimensionEdit Command

The **DimensionEdit** command provides the ability to replace the Dimension text with new text, rotate the existing text, move the text to a

new position, and if necessary, restore the text back to its home position. Home position is the position defined by the current style. In addition, the provided options allow you to modify the angle of the Extension Lines, normally perpendicular relative to the direction of the Dimension Line (Obl i que) option.

DimensionTextEdit Command

The **DimensionTextEdit** or command provides the ability to move and rotate Dimension text of any single Dimension entity.

The default sequence of the (DimTextEdit) command lets you position Dimension text dynamically.

After calling the command, select a Dimension entity. Then point to the drawing to determine the new position of the Dimension text.

Dimension Text Alignment Options

- The **Angle** option rotates the Dimension text about its center by the angle you specify.
- The **Center** option moves the text to a left justified position within the Dimension.
- The **Reset** option undoes the move of Dimension text and returns it to the original position.
- The **Left** option moves the text to a left justified position within the Dimension.
- The **Right** option moves the text to a left justified position within the Dimension.

Modify the Linear Dimension Text

Modify the 60.00 Linear Dimension to WIDTH text.

19 Modify the Linear Dimension Text.

Click the **DimensionEdit** icon 🔄 on the Dimension toolbar.

Use the **New** option. Default: Home options: <u>A</u>ngle, <u>H</u>ome, <u>N</u>ew, <u>O</u>blique or Specify option»**n**→

Specify new Dimension Text: WIDTH 🚽

Specify dimension(s)»





BASIC DIMENSIONING



1 found, 1 total

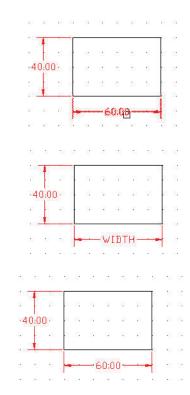
Specify dimension(s)»↓

- 20 Undo the Modified Linear Dimension. : U,J
- **21 Save the Drawing.** Click the **Save** icon **a** on the Standard toolbar.

Measure and Rotate an Aligned Dimension

First measure the Aligned Dimension angle. Rotate the 50.00 Aligned Dimension text to match angle of the entity.

22 Measure the Aligned Dimension Angle. Click Tools, Inquiry, Get Distance from the Main Menu.



Tools	Window Hel	p	
	operties eferences	Ctrl+1	
Dis	splay Order	۰,	
In	quiry	×	💁 Get Area
Open Reference Run Script			🧔 Get Distance
			🗯 Get Coordinate
te co	🛄 CCS Manager		Get Properties

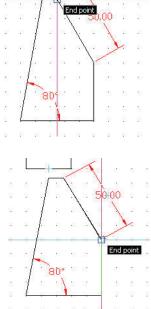
NOTE: Make sure the EnttitySnap function is set to ON. Click the **ESnap** button on the Status bar.

View the command window: <ESnaps On>. Specify start point» Click the top left end point of the Aligned Dimension.

Specify end point»

Click the bottom right end point of the Aligned Dimension.

View the angle in the XY Plane in the command window: Distance = 50.00, Angle in XY Plane = 300, Angle from XY Plane = 0 Delta X = 25.00, Delta Y = -43.30, Delta Z = 0.00



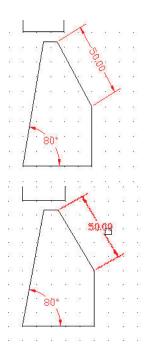
NOTE: Angle in XY Plane is 300.

23 Rotate the Aligned Dimension Text. Click the **DimensionTextEdit** icon 📧 on the Dimension toolbar.

Specify dimension»

Click the **50.00** aligned Dimension text as illustrated.

Default: (174.29,101.34,0.00) Options: <u>Angle, C</u>enter, <u>H</u>ome, <u>L</u>eft, <u>R</u>ight or Specify new text position» **a**.⊣ Specify angle for Dimension Text: **300**.⊣



Move Command

Apply the **Move** P command to reposition drawing entities within the coordinate system without changing their orientation or size.

24 Move the Selected Entities.

Click the **Move** icon P on the Modify toolbar. Specify entities»

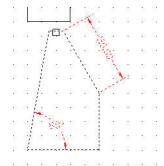
Select all 7 entities as illustrated.

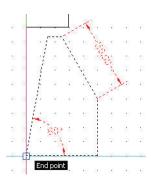
1 found, 7 total

Select entities»⊣

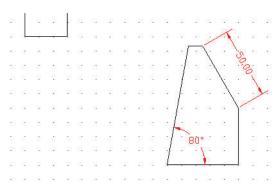
Default: Displacement Options: Displacement or Specify from point: Click the end point as illustrated.

Options: Enter to use from point as displacement or





Specify destination»@100,0.↓



Coordinate System Command

A coordinate system determines each Point of a drawing surface or entity unambiguously. DraftSight uses the Cartesian coordinate system consisting of three coordinate axes. Axes are arranged orthogonally, crossing at the origin. All axes use the same measurements.

Reset the coordinate system (CCS) from the default position. Use the CCS command to set coordinate systems and save, restore, rename, and delete them.

Reset the Coordinate System

25 Reset the Coordinate System.

Set the coordinate system position to the bottom left point of the top entity.

Click Snap On from the Status bar. The command window displays: <Snap On>.

Click ESnap On from the Status bar. The command window displays: <ESnaps On>.

Click Ortho On from the Status bar. The command window displays: <Ortho On>.

```
:
```

```
: CCS 🖵
```

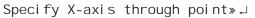
```
Default: World
```

Options: align to Entity, NAmed, Previous, View, World, X, Y, Z, ZAxis or

Specify origin»

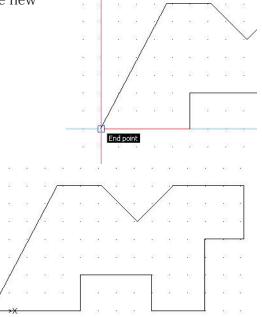
Click the bottom left point as illustrated. View the new coordinate system origin position.

Option: Enter to accept or



26 Save the Drawing.

Click the **Save** icon **I** on the Standard toolbar.



Ordinate Command

The **Ordinate** is command creates ordinate Point Dimensions. The graphics area displays an X or Y coordinate with a Leader Line to create X datum or Y datum Dimensioning. The value of the X or Y coordinate is defined by the Point you specify, called the feature position.

Create Ordinate Dimensions

NOTE:

Create Ordinate Dimensions on the top entity using the new origin position.

27 Create Ordinate Dimensions.

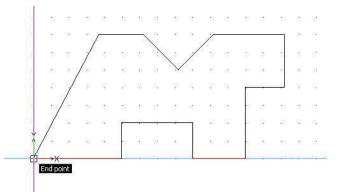
Click the **Ordinate** icon 💩 on the Dimension toolbar.

Make sure that Snap mode is set to OFF. Click **Snap** on the Status bar. The command window will display **<Snap Off**>

Make sure that Ortho mode is set to ON. Click **Ortho** on the Status bar. The command window will disply **<Ortho** on>

Specify datum position» Click the bottom left end point.

options: <u>Angle, Note, Text, X</u> datum, <u>Y</u> datum or Specify dimension position»



Click a position directly under the end point. The command window displays:

Dimension Text: 0.00

Click the **Continue** icon 🕎 on the Dimension toolbar.

Options: Select dimension, Undo or Specify feature position»

Click all the end points from left to right as illustrated.

Options: <u>S</u>elect dimension, <u>U</u>ndo or Specify feature position»↓ Specify dimension»↓

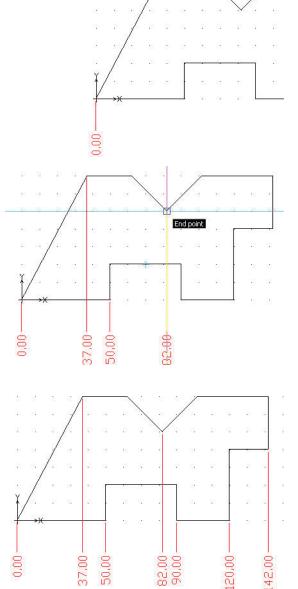
28 Save the Drawing.

Click the **Save** icon **a** on the Standard toolbar.

CenterMarks Command

The **CenterMark** O command provides the ability to mark the center of a Circle or Arc. You can display the center as a center mark (a Point) or as a Center Line (a broken Line crossing the center).

In the next section, open Lesson 2-2 and create a new layer called Annotations. Insert CenterMarks on the Annotations layer in the drawing.



Open an Existing Drawing

- 1 Click the **Open** icon **a** on the Standard toolbar. The **Open** dialog box is displayed.
- 2 Double-click Lesson 2-2.

Create a New Layer

In this section, create a new layer called Annotations. Apply custom properties.

3 Create the New Layer.

Click the **Layers Manager** icon. Solution The **Layers Manager** dialog box is displayed.

Click the **New Layer** icon. 🔫 The new default name is Layer1.

Rename this Layer Annotations.

Annotations is the new layer name.

4 Set the LineColor.

New layers are automatically assigned the LineColor white.

Double-click the **LineColor** cell for the **Annotations** layer.

Click Magenta.

Click OK.

5 Save the Drawing. Click File, Save As from the Menu bar.

Enter File name: Lesson 3-2.

Click Save from the Save As... dialog box.

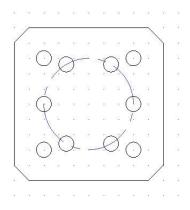
DimensionStyle Command

The **DimensionStyle** command on the Dimension toolbar provides the ability to create and modify DimensionStyles. Remember, DimensionStyles are the main method used to control the way Dimensions look.

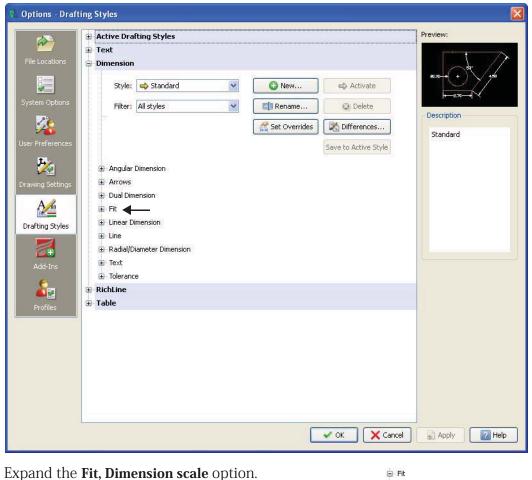
6 Set DimensionStyle

Click the **DimensionStyle** icon is on the Dimension toolbar. The **DimensionStyles Manager** dialog box is displayed.

The **Options** - **Drafting Styles** dialog box is displayed with the **Dimensions** option expanded.







View the **Fit** options displayed on the right side of the dialog box.

Scale factor: Enter 25.

Modify the Default CenterMark Size

Set the Radial Dimension of the CenterMark. 7

Expand Radial/Diameter Dimensions, Center mark display from the Navigation box.

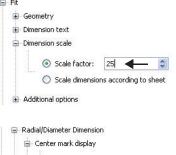
Click As mark on the dialog.

Enter 2.5 for Size.

Click OK .

8 Insert CenterMarks.

Click the **CenterMark** icon 🕘 on the Draw toolbar.



C) None	Size:	
) As mark	2.5	47
C	As centerline		

DraftSight

Specify curved entity»

Click the circumference of the bottom left Circle as illustrated.

: പ

Specify curved entity» Click the circumference of the top left Circle.

: J Specify curved entity» Click the circumference of the top right Circle.

: പ

Specify curved entity»

Click the circumference of the bottom right Circle.

Specify curved entity»

Four Center Marks are displayed.

9 Save the drawing.

Click the **Save** icon 📄 on the Standard toolbar.

Create a Pattern of CenterMarks

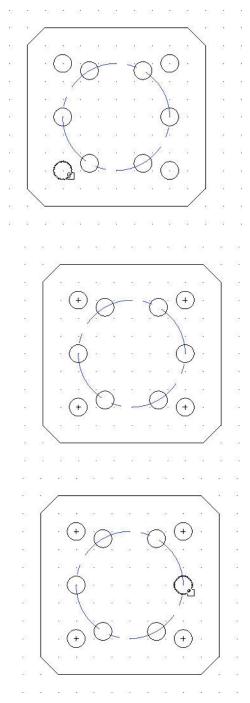
In the next section, create a circular pattern of CenterMarks on the inside Circles of the drawing.

10 Create a Pattern of CenterMarks.

Click the **CenterMark** icon 💿 on the Draw toolbar.

Specify curved entity»

Click the circumference of the right most Circle of the bolt pattern as illustrated. The CenterMark is displayed.



Click the **Pattern** icon **m** on the Modify Pattern toolbar. Pattern type Preview: Circular The Format Pattern dialog box is displayed. 🔘 Linear Set the options. Select entities Click the **Circular** button. Select Entiti No entities specified. Click the Select Entities icon. Select entities» Settings Click the Horizontal Line of the Center Base pattern on: Fill Angle and Total Number of Elements ~ Mark display. Angle between: 60 0 360 Fill angle: Select entities» Total number: 6 Click the Vertical Line of the Center Mark Element base point Axis point display. Use last entity selected X: 0.00 X: 0.00 Q Y: 0.00 Y: 0.00 Select entities», J Orient elements about axis X Cancel 🕜 Help V OK

▶ It may be necessary to use the scroll wheel to zoom into the entity.

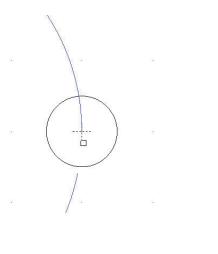
Base pattern on: Fill Angle and total number of Elements

Fill angle: Enter 360.

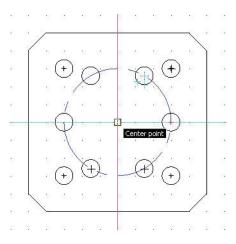
TIP:

Total number: Enter 6.

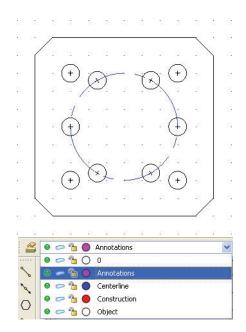
Click the **Axis Point** icon. R The graphics area is displayed. Specify pattern center point»



Click the center point of the large Circle as illustrated. The **Pattern** dialog box is displayed.







View the Axis point X: = 100.00 and the Axis point Y: = 100.00 $\,$

The **Orient elements about axis** box is check by default. This options provides the ability to have the CenterMarks rotate about the large Circle.

Click OK.

11 Activate the Annotations Layer.

Click Annotations from the Layers Manager dropdown menu.

Radius Command

The **Radius** *O* command provides the ability to

create Radial Dimensions for Circles and Arcs. Radius Dimensioning is carried out in the same way as Diameter Dimensioning, the only difference being that a Radius Dimension is calculated instead of a diameter measurement. Create a Radial Dimension for the large Circle.

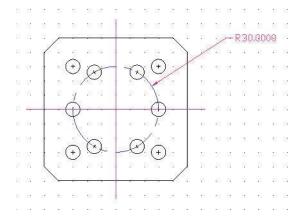
12 Create a Radius Dimension.

Click the **Radius** icon 🙆 on the Dimension toolbar.

Specify curved entity»

Click the circumference of the large Circle.

Options: <u>Angle</u>, <u>Note</u>, <u>Text</u> or Specify dimension position» Click a position diagonally to the right.





13 Save the Drawing.

Click the **Save** icon 🗟 on the Standard toolbar.

14 Modify the Precision.

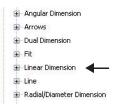
Click the **DimensionStyle** icon 🗾 on the Dimension toolbar.

The **Options** - **Drafting Styles** dialog box is displayed with the **Dimensions** option expanded.

Expand Linear Dimension in the navigation box.

View the Linear Dimensions options displayed on the right side of the dialog box.

Precision: Select 0.00 from the drop-down menu.



Click OK.

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15 Save the Drawing.

Click the **Save** icon 🗟 on the Standard toolbar.

You are finished with this section.

Lesson 4 DRAFTING APPLICATION

When you complete this lesson, you will be able to:

Apply the MakeBlock, Note, and Line Draw commands

- Make and Insert Blocks
- Create and Edit a Note
- Apply the Ring command
- Explode a Block
- Apply the Hatch/Fill command
- Create a Multiview drawing
- Apply a Tolerance

DraftSight

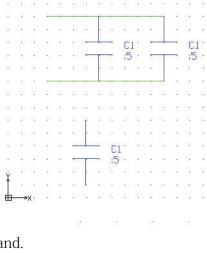
Drafting Application

During this lesson, you will create two new drawings using absolute coordinates in the Command prompt and the graphics area.

In the first drawing you will:

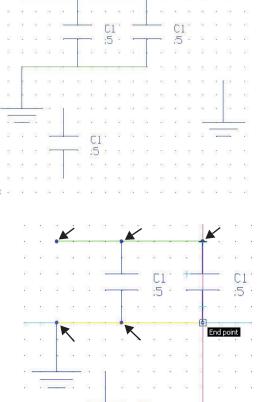
- Create two new layers with custom properties: Symbols and Schematic.
- Activate the Symbols layer.
- Use the **Line** S command to draw the entities.
- Apply the **Note (** command.
- Create a single entity with the **MakeBlock**
- Use the **Insert Block** dialog box.
- Apply the **Insert Block** 🕅 command to insert two Symbol copies.
- Use the **Insert Block** dialog box.
- Activate the Schematic layer.
- Use the Line Schematic layer.
- Activate the Symbols layer. Apply the Line command to draw three Horizontal and Parallel Lines: 30, 20, and 10mm's in length and 5mm's apart. Draw a Vertical Line perpendicular at the mid point to the top Horizontal Line 20mms in length.
- Create a single entity with the **MakeBlock** B command.

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- Use the **Define Block** dialog box.
- Apply the InsertBlock Command to insert the Symbol (Ground).
- Use the **Insert Block** dialog box.

- Apply the **Ring** command to create a ring with an inside diameter of **Omm**, and outside diameter of **2mm**.
- Apply the Explode command. The Explode command provides the ability to explode a symbol into its component entities.
- Edit the Note. Use the **Edit Note** dialog box.



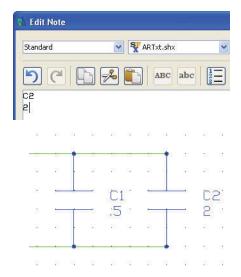
A symbol loses its characteristics when exploded, and only a uniform scaled symbol can be exploded.

In the second drawing you will:

- Create six new layers called: Dimensions, Profile, Construction, Center, Hidden, and Section.
- Apply custom properties.
- Draw six Horizontal InfiniteLines and seven Vertical InfiniteLines.
- Draw three profiles.

NOTE

- Utilize the Line command
- Draw the first profile.



DRAFTING APPLICATION

• Draw the second profile.

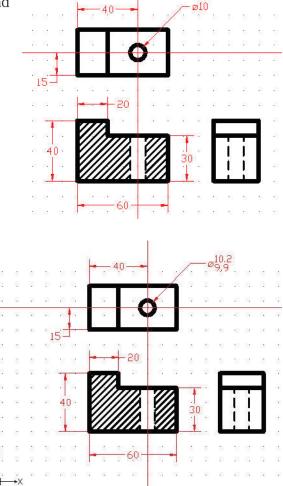
• Draw the third profile.

Draw two Hidden Lines views.

• Apply the Hatch/Fill command.

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Apply a Tolerance to the diameter

dimension to complete the drawing.

Start a DraftSight Session

1 Start a DraftSight session.

From the Windows Start menu, click Start, All Programs, Dassault Systemes, DraftSight.

Create a New Drawing

2 Create a New Drawing.

Click the **New** icon **(a)** on the Standard toolbar. The **Select Template** dialog box is displayed.

Set the Drawing Environment

Drawing Template

- 3 Select a Default Drawing Template. Select standard.dwt from the Specify Template dialog box.
- 4 Click Open.

TIP: Double-clicking on the template name will also open the file.

Drawing Unit System

The default units system is the Decimal system and can be applied to either English (inches) or metric values (millimeters).

5 Set Linear Length Precision.

Click Format, Unit System... from the Main Menu.

The Unit System section of the Options - Drawing Settings dialog box is displayed.

Select 0.00 for Length Precision.

6 Set Linear Units Scale.

The **Unit System** section of the **Options** - **Drawing Settings** dialog box allows you to set the linear and angular unit formats.

Click **Millimeters** for **Block units format** in the **Units scale** section of the dialog box. Click **OK**.

Drawing Boundary

Use a horizontal A4 default drawing sheet.

 7 Set Drawing Boundary. Click Format, Drawing Boundary from the Main Menu. Defaul t: (0.00, 0.00)
 Options: <u>OF</u>f, <u>ON</u> or

Default: (12.00,9.00) Specify upper right corner» **297,210**,J

Specify lower left corner»,J

Set Snap and Grid

Set the Snap to 5mms. Set the Grid to 10mms.

8 Set Snap and Grid.

Click **Tools**, **Options** from the Main Menu.

The **Options** dialog box is displayed.

Click the User Preferences tab.

Expand Drafting Options, Pointer Control and Snap Settings.

Click the Enable Snap <F9> box.



Enter the value of **5** in the **Horizontal Snap spacing** box.

Press the Tab key The **Vertical Snap spacing** box is updated.

🗹 Enable Snap (F9)	
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Standard (snaps to)	grid)
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🔲 Match Grid spacing	
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Expand the **Display, Grid Settings** option

Click the **Enable Grid** <**F7**> box.

Enter **10** in the **Horizontal display spacing** box. Press the **Tab** key. The **Vertical display spacing** box is updated. Click **OK**.

- 9 Display the Drawing with the New Boundary.
- 10 Click View, Zoom, Bounds 🔍 on the Main Menu.

11 Save the Drawing.

Click the **Save** icon 🔲 on the Standard toolbar.

Enter File name: Lesson 4-1.

Click Save.

Create two new Layers for the drawing

In this section, create two new layers called: Symbols and Schematic. Apply custom properties.

12 Create the First New Layer.

Click the Layers Manager icon 📓 on the Layers toolbar

The Layers Manager dialog box is displayed.

Click the **New Layer** icon. **The new default name is Layer** 1.

Rename this layer Symbols

13 Set the LineColor.

New layers are automatically assigned the LineColor white.

Double-click the LineColor cell for the Symbols layer.

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Click Blue.

Accept the default LineStyle: Continous and options.

14 Create the Second New Layer.

Click the **New Layer** icon. **The new default** name is Layer2.

Rename this Layer **Schematic**.

Schematic is the new layer name.

15 Set the LineColor.

Double-click the **LineColor** cell for the Schematic layer.

Click Green.

Accept the default LineStyle: Continous.

16 Activate the Symbols Layer.

Click inside the **Status** cell of the Symbols layer.

Click the **Activate Layer** icon. A gold arrow is displayed in the Symbols **Status** cell. Click **OK**.

ener **or**.

Draw a capacitor symbol

Draw multiple entities on the Symbols layer. Apply the **Line** , and **Note** commands. Use the **Edit Note** dialog box to to create a note with a **5mm** height, left justify to the right of the drawing.

Create a single entity with the **MakeSymbol** command. Use the **Define Symbol** dialog box. Apply the **Insert**, **Symbol** command from the Main Menu to insert two Symbol copies. Use the **Insert Symbol** dialog box.

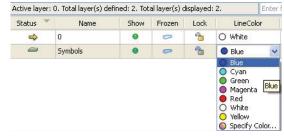
17 Apply the Line Command.

Draw two Horizontal Parallel Lines 20mms long and 10mms apart.

Click the Line icon 🔨 on the Draw toolbar.

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Options: Segments, or
Specify start point>>50,30,J
Options: <u>S</u>egments, <u>U</u>ndo, Enter to exit or
Specify next point>>70,30,J
↓
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Options: Segments, Enter to continue from last point or



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New Dele	te Activate				
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Active layer: Status 🍼	Symbols, Totanaye Name	r(s) defined: Show	3. Total lay Frozen	/er(s) dis Lock	played: 3. LineColor
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┢	→X ·	81 83	2 3	2 3	2 K	1	8.8	N 12

```
Specify start point>>70,40
Options: Segments, Undo, Enter to exit or
Specify next point>>50,40. ↓
Right-click Enter.
Draw two Vertical Lines perpendicular to the midpoint of the two Horizontal Lines.
: പ
Options: Segments, Enter to continue from last point or
Specify start point>>60,30,
Options: Segments, Undo, Enter to exit or
Specify next point>>60,10,
Right-click Enter.
: പ
Options: Segments, Enter to continue from last point or
Specify start point>>60,40,
Options: Segments, Undo, Enter to exit or
Specify next point>>60,60,
```

Right-click **Enter**.

18 Save the Drawing.

Click the **Save** icon 🔲 on the Standard toolbar.

Note Command

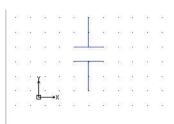
The **Note** command provides the ability to enter multiple Lines of text that are treated as a single entity. In the next section, use the **Edit Note** dialog box to create a note with a 5mm height, left justify to the right of the drawing.

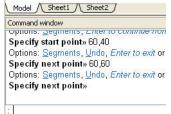
19 Create a Note.

Click the **Note** icon 🛅 on the Draw toolbar.

Active TextStyle: "Standard" Text height: 0.20

Specify first corner>>80,40, J





DraftSight

Options: Angle, Height, Justify, Line spacing, textSTyle, Width or Specify opposite corner>> $h \!$

default: 0.20 Specify text height>>5.J

Options: <u>Angle, H</u>eight, <u>J</u>ustify, <u>L</u>ine spacing, text<u>ST</u>yle, <u>W</u>idth or Specify opposite corner>>100, $20 \downarrow$

The Edit Note dialog box is displayed.

Enter C1.

Press the **Enter** key.

Enter .5

Click OK from the Edit Note dialog box.

Return to the graphics area.

20 Save the Drawing.

Click the **Save** icon 🔚 on the Standard toolbar.

Blocks

A Block is a collection of entities bound together as a single entity. After you create a Block, you can insert it whenever you need it in a drawing. When you use the **MakeBlock** is command, the symbol is recognized in the current drawing only. The **MakeBlock** is command lets you write a symbol to an external drawing file for use in any drawing. In the next section, create a Block named Capacitor.

21 Apply the MakeBlock Command.

Create a symbol named Capacitor.

Click the **MakeBlock** icon **B** on the Draw toolbar. The **Block Definition** dialog box is displayed.

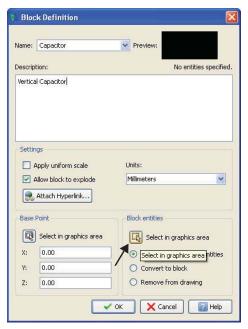
Name: Enter Capacitor.

Description: Enter Vertical Capacitor.

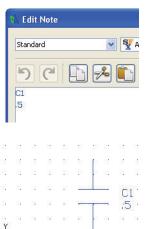
Units: Select Millimeters.

Block entities: Click the **Select in graphics area** icon.

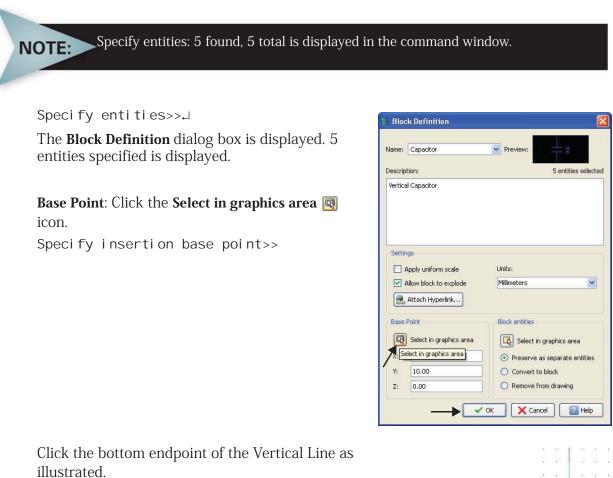
Specify entities>>



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Window-select the capacitor geometry and the note.



The **Block Definition** dialog box is displayed.

Click OK.

22 Insert the Block.

Insert the Capacitor symbol twice above the original symbol.

Click **Insert**, **Block** from the Main menu.



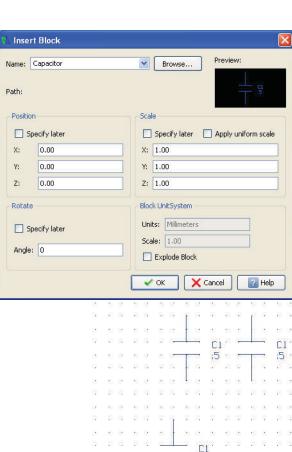
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🖹 Rel	ference Im	age	
Ob	ject		

The **Insert Block** dialog box is displayed. The **Insert Block** dialog box provides the ability to select created symbols. Capacitor is selected by default.

Position X: Enter 70. Position Y: Enter 90.

Click OK.

: പ



The **Insert Block** dialog box is displayed. Capacitor is selected by default.

Position X: Enter 120.

Position Y: Enter 90. Click OK.

23 Save the Drawing. Click the **Save** icon **a** on the Standard toolbar.

Draw a Simple Schematic

24 Activate the Schematic Layer.

Click **Schematic** from the drop-down menu in the **Layers Manager**. Schematic is the active layer.

25 Apply the Line Command.

Draw two Horizontal Parallel Lines between the top symbols on the Schematic layer.

Click the Line icon 📉 on the Draw toolbar.

Click Snap from the Status bar. The command window displays: <Snap Off>.

Options: <u>S</u>egments, *Enter to continue from last point* or Specify start point>>

	🗕 🗢 🍗 🔵 Schematic	~
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5	📀 🖛 🖀 🔵 Schematic	
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DraftSight

Click a Point directly to the left as illustrated.

Specify next point>>

Click the top end point of the symbol to the right.

Options: Segments, Undo or Enter to exit or

Options: <u>S</u>egments, <u>U</u>ndo or *Enter to exit* or Specify next point>>,J

: പ

Options: Segments, Enter to continue from last point or Specify start point>>

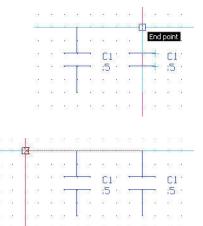
Click the bottom end point of the symbol to the right.

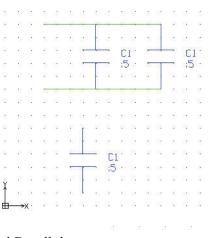
Options: <u>Segments</u>, <u>Undo or *Enter to exit* or Specify next point>></u>

Click a Point directly to the left as illustrated.

Options: <u>S</u>egments, <u>U</u>ndo or *Enter to exit* or Specify next point>>↓







26 Draw a Ground Symbol

Apply the **Line** command to draw three Horizontal and Parallel Lines: 30, 20, and 10mms in length and 5mm¹s apart. Draw a Vertical Line perpendicular at the midpoint to the top Horizontal Line 20mms in length.

Create a single entity (Ground symbol) with the MakeBlock

command. Use the **Define Block** dialog box. Apply the **Insert**, **Block** and to insert the Block (Ground symbol). Use the **Insert Block** dialog box.

- 27 Activate the Symbols Layer. Click Symbols from the drop-down menu in the Layers Manager. Symbols is the active layer.
- Image: Schematic
 Image: Schematic

 Image: Schematic
 Image: Schematic

28 Apply the Line Command.

Click the Line icon 💊 on the Draw toolbar.

Draw the first Horizontal Line.

Options: <u>S</u>egments, *Enter to continue from last point* or Specify start point>>160,50↓

Options: <u>S</u>egments, <u>U</u>ndo or Specify next point>>190,50↓ Right-click Enter. Draw the second Horizontal Line. : പ Options: Segments, Enter to continue from last point or Specify start point>>165,45, □ Options: <u>S</u>egments, <u>U</u>ndo or Specify next point>>185,45.↓ Right-click Enter. Draw the third Horizontal Line. : പ Options: Segments, Enter to continue from last point or Specify start point>>170,40.」 Options: Segments, Undo or Specify next point>>180,40, J Right-click **Enter**. Draw the Vertical Line. : പ Options: Segments, Enter to continue from last point or Specify start point>> Click the **midpoint** of the 30mm Horizontal Line. Options: <u>S</u>egments, <u>U</u>ndo or Specify next point>>@0,30, Right-click Enter. In the next section you will create a symbol called Ground.

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10	85		1	18
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29 Apply the MakeBlock Command.

Create a symbol named Ground on the Symbols layer.

Click the **MakeBlock** icon **Matheblock** on the Draw toolbar. The **Block definition** dialog box is displayed.

Name: Enter Ground.

Description: Enter Ground Symbol.

Units: Select Millimeters.

Block entities: Click the **Select in graphics area** icon.

Specify entities>>

Window-select the three horizontal and single Vertical Lines.

Specify entities>>↓

The **Define Symbol** dialog box is displayed. 4 entities specified is displayed.

Base Point: Click the **Select in graphics area (**icon.

Specify insertion base point>>

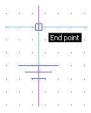
Click the top end point of the Vertical Line as illustrated.

The **Block Definition** dialog box is displayed.

Click OK.

Description: 4 entities selected Ground Symbol Settings Apply uniform scale Allow block to explode Attach Hyperlink Base Point Base Point Select in graphics area X: 175.00 Y: 80.00 Z: 0.00 Block definition Block Definition Name: Ground Preview: Ground Preview: Ground Preview: Convert to block Preview: Cancel Preview: Convert to block Preview: Convert to block Preview: P	Description: 4 entities selected Ground Symbol Settings Apply uniform scale Allow block to explode Allow block to explode Attach Hyperlink Base Point Block entities Select in graphics area X: 175.00 Convert to block Remove from drawing Convert to block Remove from drawing Convert to block Cancel Plep Block Definition Apply uniform scale Allow block to explode Allow block to explode Allow block to explode Allow block to explode Allow block to explode Attach Hyperlink Base Point Settings Allow block to explode Allow block to explode Allow block to explode Allow block to explode Attach Hyperlink Base Point Block entities Convert to block Settings Convert to block Settings Convert to block Settings Convert to block Settings Convert to block Convert to block	Block Det	finition		E E E E E E E E E E E E E E E E E E E
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30 Insert the Block.

Insert the Ground symbol.

Click **Insert**, **Block** from the Main Menu. The **Insert Block** dialog box is displayed.

The **Insert Block** dialog box provides the ability to select created symbols.

Ground is selected by default.

Position X: Enter 30.

Position Y: Enter 90.

Click OK.

lame:	Ground	Browse Preview:
ath:		
Positio	n	Scale
🔲 SI	pecify later	Specify later 🔲 Apply uniform scale
X:	0.00	X: 1.00
Y:	0.00	Y: 1.00
Z:	0.00	Z: 1.00
Rotate		Block UnitSystem
🔲 SI	pecify later	Units: Millimeters
Angle	: 0	Scale: 1.00

31 Save the Drawing.

Click the **Save** icon 🔳 on the Standard toolbar.

52	52	50	8	32		12	52	20	5	8	1	82	52	<u>20</u>	5	88	82	82	52
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Ring Command

The **Ring** command provides the ability to construct a ring of any thickness or size by specifying the diameters of two concentric Circles. The area between the Circles is filled. A filled Circle has an inner diameter of zero. Apply the Ring command in the next section. Create a **ring** with an inside diameter of **0**, and outside diameter of **2mms**.

32 Apply the Ring Command.

: Ring⊣ Default: 0.50 Specify inside diameter>>**0**,⊣

Default: 1.00 Specify outside diameter>>**2**↓ Defaul t: exi t Speci fy posi ti on>> Click the six endpoints as illustrated in the graphics area.

Default: exit Specify position>>↓

Explode a Block

DraftSight provides the ability to explode a Block into its component entities. A Block loses its

characteristics when exploded, and only a uniform scaled Block can be exploded.



33 Explode a Block.

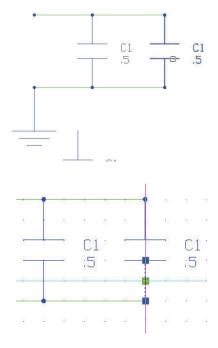
Click the **Explode** icon **a** on the Modify toolbar.

Specify entities>>

Click the **Block** as illustrated in the graphics area. Remember, a symbol is a single entity

1 found, 1 total

Specify entities>>,J



You can now select the individual Line entities and note.

Press Esc or Enter to exit.

34 Save the Drawing.

Click the **Save** icon 🗟 on the Standard toolbar.

DRAFTING APPLICATION

35 Edit the Note in the Exploded Symbol.

Double-click the **C1**.5 text in the exploded symbol.

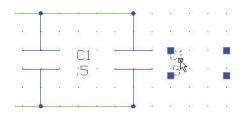
The **Edit Note** dialog box is displayed.

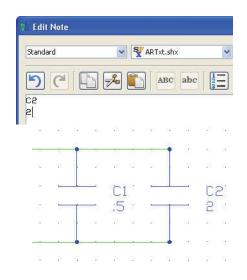
Delete C1 and .5 in the text box.

Enter C2 in the text box.

Press the Enter key.

Enter 2 in the text box.





Click OK.

View the results in the graphics area.

36 Save the Drawing.

Click the **Save** icon 🗟 on the Standard toolbar.



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Create a New Drawing

1 Create a New Drawing.

Click the **New** icon **(a)** on the Standard toolbar. The **Select Template** dialog box is displayed.

Set the Drawing Environment

Drawing Template

- 2 Select a Default Drawing Template. Select standard.dwt from the Specify Template dialog box.
- 3 Click Open.

Drawing Unit System

The default units system is the Decimal system and can be applied to either English (inches) or metric values (millimeters).

4 Set Linear Length Precision. Click Format, Unit System... from the Main Menu.

The Unit System section of the Options - Drawing Settings dialog box is displayed.

Select **0.00** for **Length Precision**.

5 Set Linear Units Scale.

The **Unit System** section of the **Options** - **Drawing Settings** dialog box allows you to set the linear and angular unit formats.

Click **Millimeters** for **Block units format** in the **Units scale** section of the dialog box. Click **OK**.

Drawing Boundary

Use a horizontal A4 default drawing sheet.

6 Set Drawing Boundary. Click Format, Drawing Boundary from the Main Menu. Defaul t: (0.00,0.00) Options: <u>OF</u>f, <u>ON</u> or Specify lower left corner»↓

Default: (12.00,9.00) Specify upper right corner» **297,210.**↓

Set Snap and Grid

Set the Snap to 5mms. Set the Grid to 10mms.

7 Set Snap and Grid.

Click **Tools**, **Options** from the Main Menu.

The **Options** dialog box is displayed.

Click the User Preferences tab.

Expand Drafting Options, Pointer Control and Snap Settings.

Click the **Enable Snap** <**F9**> box.

Enter the value of **5** in the **Horizontal Snap spacing** box.

Press the Tab key The Vertical Snap spacing box is updated.

Expand the Display, Grid Settings option

Click the **Enable Grid** <**F7**> box.

Enter **10** in the **Horizontal display spacing** box. Press the **Tab** key. The **Vertical display spacing** box is updated.

Click OK.



- 8 Display the Drawing with the New Boundary.
- 9 Click View, Zoom, Bounds 🔍 on the Main Menu.
- 10 Save the Drawing.

Click the **Save** icon 📕 on the Standard toolbar.

Enter File name: Lesson 4-2.

Click Save.

Create Six New Layers

In this section, create six new layers called: Dimensions, Object, Construction, Center, Hidden, and Section. Apply custom properties.

11 Create Six New Layers.

Click the Layers Manager icon 📓 on the Layers toolbar.

The **Layers Manager** dialog box is displayed. Create the layers as illustrated with custom properties.

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ctive layer:	Dimension. Total lay	ver(s) define	d: 8. Total l	ayer(s) c	lisplayed: 8.	Enter filter expre	ssion			
Status 🍸	Name	Show	Frozen	Lock	LineColor	LineStyle	LineWeight	PrintStyle	Print	Description
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<i>a</i>	Defpoints		0	-	O White	Continuolid line	Default	Color_7	1	
⇒	Dimension	۲	-	-	🕘 Red	Continuolid line	Default	Color_1	-	
0	Object	0	0	-	O White	Continuolid line	0.60 mm	Color_7		
9	Construction	0	0	-	🔵 Cyan	Continuolid line	Default	Color_4	8	
<i>a</i>	Center		0	1	Green	CENTER	Default	Color_3	8	
9	Hidden	•	0	-	O White	HIDDEN	0.30 mm	Color_7	8	
9	Section	•	0	-	O White	Continuolid line	0.30 mm	Color_7	0	

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NOTE: The LineWeight has been changed for the Object, Hidded and Section Layers.

12 Activate the Construction Layer.

Click inside the Status cell of the Construction layer.

Click the **Activate Layer** icon. A gold arrow is displayed in the Construction **Status** cell. Click **OK**.

13 Draw Six Horizontal InfiniteLines

Click the **Infinite Line** icon 📉 on the Draw toolbar.

Options: Angle, angle Bisect, Horizontal, Offset, Vertical, Enter to exit or

Specify position» $H \sqcup$

Options: *Enter to exit* or Specify next point»100,100,J

Options: *Enter to exit* or Specify next point»100,130,J

Options: *Enter to exit* or Specify next point»100,140,J

Options: *Enter to exit* or Specify next point»100,170,J

Options: *Enter to exit* or Specify next point»100,185.J

Options: *Enter to exit* or Specify next point»100,200.↓

Options: *Enter to exit* or Specify next point»,J

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14 Draw Seven Vertical InfiniteLines

Click the **Infinite Line** icon \searrow on the Draw toolbar. Options: <u>Angle</u>, angle <u>B</u>isect, <u>H</u>orizontal, <u>O</u>ffset, <u>V</u>ertical, <u>Enter to</u> *exit* or Specify position»V,J

Options: *Enter to exit* or Specify next point**»100,100**,J

Options: *Enter to exit* or Specify next point**»110,100**,J

Options: *Enter to exit* or Specify next point»**120,100**,J

Options: *Enter to exit* or Specify next point**»140,100**,J

Options: *Enter to exit* or Specify next point **»160,100.**↓

Options: *Enter to exit* or Specify next point»**190,100**,J

Options: *Enter to exit* or Specify next point **»220,100**.↓

Options: *Enter to exit* or Specify next point»→

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15 Activate the Object Layer and Display the LineWeight. Click **Object** from the drop-down menu in the **Layers Manager**.



Format Dimension Dra

۲

🚄 Layer...

Laver Tools

Line Style...

Object is the active layer.

Click Format, Line Weight from the Main Menu.

The **Options** dialog box is displayed.

Check the **Display weight in graphics** area box.

Click OK

16 Save the Drawing.

Click the **Save** icon **a** on the Standard toolbar.

€	Line Style: ByLayer			🔯 Load LineStyle
Đ	Line Color: ByLayer			
	Line Weight: ByLayer			
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	Default weight: —	—— 0.25 mm	💉 💽 Mil	limeters
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Draw Three Profiles. Utilize the Line command. 17 Draw the First Profile. Click the Line icon 🔨 on the Draw toolbar Options: Segments, Enter to continue from last point or Specify start point»100,100,1 Options: <u>Segments</u>, <u>Undo</u>, *Enter to exit* or Specify next point»160,100, □ Options: Segments, Undo, Enter to exit or Specify next point»160,130 → Options: <u>Segments</u>, <u>Undo</u>, *Enter to exit* or Specify next point»120,130, J Options: <u>Segments</u>, <u>Undo</u>, *Enter to exit* or Specify next point»120,140, J Options: <u>Segments</u>, <u>U</u>ndo, *Enter to exit* or Specify next point»100,140.」 Options: Segments, Undo, Enter to exit or

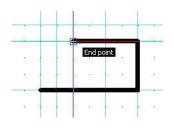
Specify next point»c, J

18 Draw the Second Profile. Click the **Line** icon 🔨 on the Draw toolbar

Options: <u>S</u>egments, *Enter to continue from last point* or Specify start point **>190,100**, J

Options: Segments, Undo, Enter to exit or Specify next point»220,100 \downarrow

Options: Segments, Undo, Enter to exit or Specify next point $220,140 \downarrow$



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Options: <u>Segments</u>, <u>Undo</u>, *Enter to exit* or Specify next point $\gg 190,140 \downarrow$

Options: <u>S</u>egments, <u>U</u>ndo, *Enter to exit* or Specify next point»c →

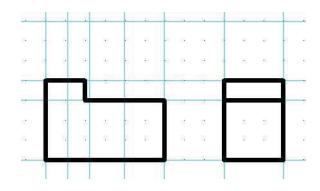
Click the Line icon 📉 on the Draw toolbar

Options: <u>S</u>egments, *Enter to continue from last point* or

Specify start point»190,130,

Options: <u>S</u>egments, <u>U</u>ndo, *Enter to exit* or Specify next point»**220,130**,

Right-click Enter.



19 Draw the Third Profile.

Click the **Line** icon 🔪 on the Draw toolbar

Options: <u>S</u>egments, *Enter to continue from last point* or Specify start point **>100,170**. J

Options: <u>S</u>egments, <u>U</u>ndo, *Enter to exit* or Specify next point»100,200 ↓

Options: <u>Segments</u>, <u>Undo</u>, *Enter to exit* or Specify next point»160,200 \downarrow

Options: Segments, Undo, Enter to exit or Specify next point» $160,170 \downarrow$

Options: <u>S</u>egments, <u>U</u>ndo, *Enter to exit* or Specify next point»c →

Click the Line icon 📐 from the Draw toolbar

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Options: <u>S</u>egments, *Enter to continue from last point* or Specify start point»120,170,J

Options: <u>S</u>egments, <u>U</u>ndo, *Enter to exit* or Specify next point»**120,200** ↓

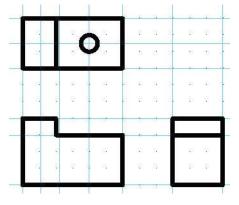
Right-click Enter.

Click the **Circle** icon **(**) on the Draw toolbar

Options: <u>3P</u>oint, <u>2P</u>oint, <u>T</u>tr, <u>TTT</u>, *Enter to exit* or

Specify Center point 140,185]

Options: <u>D</u>iameter or Specify Radius»**5**⊣



20 Save the Drawing.

Click the **Save** icon 🗟 on the Standard toolbar.

- 21 Activate the Hidden Layer. Click Hidden from the drop-down menu in the Layers Manager.
- **22 Draw the First Set of Hidden Lines View.** Click the **Line** icon **\scrimetrian** on the Draw toolbar

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Options: <u>S</u>egments, *Enter to continue from last point* or Specify start point»135,100.J

Options: <u>S</u>egments, <u>U</u>ndo, *Enter to exit* or Specify next point»135,130 ↓ Right-click **Enter**.

Click the Line icon 🔪 on the Draw toolbar

Options: <u>S</u>egments, *Enter to continue from last point* or

Specify start point»145,100.J

Options: <u>S</u>egments, *Enter to continue from last point* or

Specify start point $\$145,\!130 \mathrel{\lrcorner}$

Right-click **Enter**.

23 Draw the Second Set of Hidden Lines View.

Click the Line icon 📉 on the Draw toolbar

Options: <u>S</u>egments, *Enter to continue from last point* or Specify start point»**200,100**, J

Options: <u>S</u>egments, <u>U</u>ndo, *Enter to exit* or Specify next point»**200,130**, □

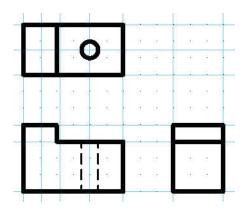
Right-click **Enter**.

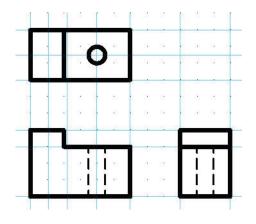
Click the Line icon 📉 on the Draw toolbar

Options: <u>Segments</u>, *Enter to continue from last point* or Specify start point»**210,100**,

Options: Segments, Undo, Enter to exit or Specify next point»210,130 \downarrow

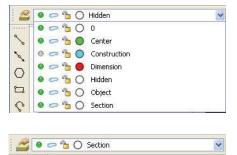
Right-click Enter.





24 Hide the Construction Layer.

Click the **Show** icon **•** for the Construction layer from the drop-down menu in the **Layers Manager**.



25 Activate the Section Layer.

Click **Section** from the drop-down menu in the **Layers Manager**.

Hatch Command

Use the **Hatch** is command to fill enclosed areas or selected entities with a hatch pattern. Hatching a drawing adds meaning and helps to differentiate the materials and areas. Some drawing applications such as construction drawings require hatch patterns which can increase the clarity and legibility of a drawing

🕦 Hatch / Fill		
Type Hatch Fill Pattern		Boundary settings
Type: Predefined Pattern: ANSI31 Add intersecting lines Pattern start point		Rebuild boundary Rebuild boundary Delete boundary entities Highlight boundary entities
Generate at: Current drawing origin User-defined location Specify coordinates Use boundary: Center Set as default	Angle: 0 V Scale: 25 V Spacing: 1.00 ISO pen width: V Scale based on sheet's units	Options Veep hatch and boundary related Create hatch for each boundary Use properties of selected hatch Placement: Send Behind Boundary Additional Options
	Previe	w V OK X Cancel Help

26 Hatch the Front View.

Click the **Hatch/Fill** icon **m** from the Draw toolbar.

Angle and scale: Scale: Enter 25.

Boundary settings:

Click the **Specify points** button.

Options: <u>D</u>eselect boundary, <u>E</u>ntities or

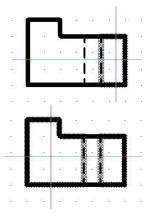
Specify internal point»

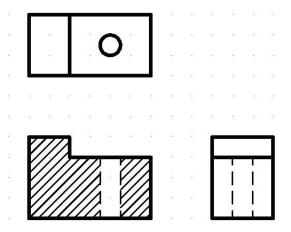
Click a Point to the right of the right Hidden Line as illustrated.

Click a Point to the left of the left Hidden Line. A boundary is created in both areas.

Options: <u>D</u>eselect boundary, <u>E</u>ntities or Specify internal point»↓

Click OK.

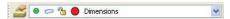




In normal drafting practice, the Hidden Lines need to be converted to an Object Line for this setion view. This is only an example of the command available.

Dimension the Views

NOTE:



1 Activate the Dimension Layer. Click Dimension from the drop-down menu in the Layers Manager.

The **DimensionStyle** command from the Dimension toolbar provides the ability to create and modify DimensionStyles.

2 Set DimensionStyle

Click the **DimensionStyle** icon 🗹 on the Dimension toolbar.

The **DimensionStyles Manager** dialog box is displayed.

Expand Fit, Dimension scale in the Navigation box.	🖨 - Fit
View the Fit options displayed on the right side of the dialog box.	 Geometry Dimension text Dimension scale
Scale factor: Enter 25.	 Scale factor: 25 Scale dimensions according to sheet Additional options
Modify the Linear dimension Precision. Expand Linear Dimensions in the Navigation box.	 Unear Dimension Une Radial/Diameter Dimensio
Precision: Select 0 from the drop-down menu.	æ Kadaquolameter Dimensio æ Text æ Tolerance
Modify the Center Mark Display. Expand Radial/Diameter Dimension, Center mark display in Navigation box.	e Line e Radial/Diameter Dimension e Text e Tolerance
Size: Enter 25.00 Click OK.	Radial/Diameter Dimension Center mark display None Size: As mark 25

Dimension Commands

Linear Command

3

4

The Linear 🔄 command creates a horizontal, vertical, or rotated linear dimension.

One way to use the Linear command is to define two Points of measurement which are the origins of the Extension Lines.

The measurement is placed as Dimension text based on the definitions of the current DimensionStyle.

Create Linear Dimensions

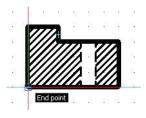
Dimension the rectangle in the graphics area. Create a linear horizontal and linear vertical dimension.

5 Dimension the Front View.

Click the **Linear** icon 🖃 on the Dimension toolbar.

Default: Entity Options: <u>E</u>ntity or Specify first extension line position» Click the lower left end point of the front view.

Specify second extension line position»



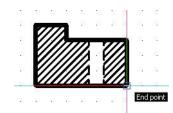
As centerline

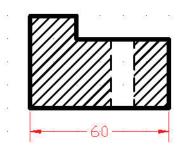
:

Click the lower right end point of the front view.

Options: <u>A</u>ngle, <u>H</u>orizontal, <u>N</u>ote, <u>R</u>otated, <u>T</u>ext, <u>V</u>ertical or Specify dimension Line position»

Click a position below the bottom Horizontal Line.





The **Note** option provides the ability to modify the Dimension text using the Note editor dialog box.

TIP:

NOTE:

Press the **Enter** (\downarrow) key to repeat the command.

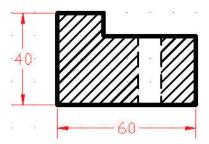
Default: Entity Options: <u>E</u>ntity or Specify first extension line position» Click the lower left end point of the front view.

Specify second extension line position» Click the upper left end point of the front view.

Options: <u>A</u>ngle, <u>H</u>orizontal, <u>N</u>ote, <u>R</u>otated, <u>T</u>ext, <u>V</u>ertical or

Specify dimension Line position»

Click a position to the left of the left Vertical Line. The front view has a vertical and horizontal linear dimension.



TIP:

The **Angle** option changes the angle of the Dimension text.

7 Add Two Additional Dimensions. Add two additional Dimensions as illustrated.

The Front view is complete.

- 8 Save the Drawing. Click the Save icon ☐ on the Standard toolbar.
- **9 Dimension the Top View.** Add **Linear** and **Diameter** Dimensions as illustrated.

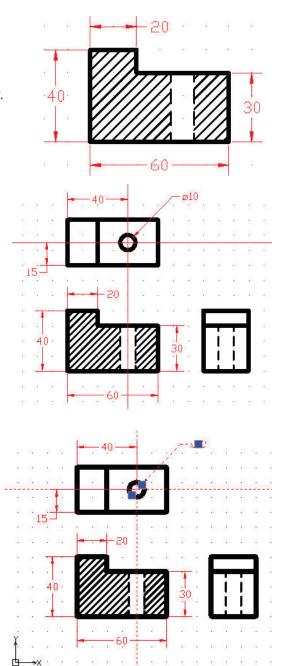
Click the 10mm diameter Dimension.

Tolerance section.

In the **Properties manager palette** scroll to the

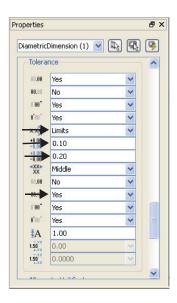
Apply the Tolerance

10 Modify the Tolerance precision.



DRAFTING APPLICATION

Change the Tolerance display to **Limits** Change the **Upper limit** to 0.10 Change the **Lower limit** to 0.20 Set **suppress tolerance trailing zeros** to **Yes** Press **Esc** or **Enter** to exit.



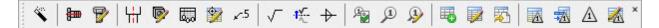
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You are finished with this section.

Lesson 5 TOOLBOX FUNCTIONALITY

When you complete this lesson, you will be able to:

- Choose a default **Standard** from a collection of data tables that define industry and national standards.
- Insert **Standard Hardware** into your drawing in a number of 2D views by choosing from a collection of Bolts and Screws, Nuts, Pins, and Washers.
- Edit Standard Hardware in the drawing.
- Add predefined Holes to a drawing by choosing from counter-bored or countersunk Holes with blind or through clearances. You can also specify geometric or textual Hole dimensions and create associative charts of hole locations, which you can then add as annotations to drawings.
- Detail drawings by creating, previewing, and adding **Surface Finishes** and **Welding Symbols.**
- Add **Balloons** to drawing entities to create a **Bill of Materials**.
- Use the item data associated with **Balloons** to add **Bill of Materials** tables to your drawings. You can export the contents of the tables as text files.
- Track changes in drawings by creating **Revision Tables** with links to **Revision Symbols**.



TOOLBOX OVERVIEW

The Toolbox add-in for DraftSight contains a set of standard 2D design, drafting, and detailing tools to help you create and annotate drawings.

You access the Toolbox features through the Toolbox menu or buttons on the Toolbox toolbar.

Toolbox is available through the Professional or Enterprise subscription of DraftSight®

CHOOSING A STANDARD

Choosing a Standard is based on your office standards and/or your client. DraftSight offers the most common of National and International Standards.

ACTIVE STANDARD

The **Active Standard** are from either the Basic or Custom Standards.

BASIC STANDARDS

The **Basic Standards** comprise of ANSI (American (Inch and Metric)), BSI (United Kingdom), DIN (Germany), ISO (International) and JIS (Japanese) standards. Each of these standards are fixed and cannot be edited, however, they can be copied and customized.

CUSTOM STANDARDS

The **Custom Standards** are modifications from the **Basic Standards**. Once the **Basic Standard** is copied, it is saved under the **Custom Standards**. At this point it can be edited. The **Custom Standards** can be either Exported or Imported to other users.

Basic standards	
ANSI Inch	ල් Copy
BSI DIN ISO JIS	Activat
Custom standards	
	С С С С С С Ору
	(2) Delete
	Elli Rename
	Edit
	Activati
	Too Too and
	Import.



INSERTING HARDWARE 🛛 🖶 🦻

The Active Standard will use the database to select the appropriate hardware to use. In Figure: 5-02, the Active Standard is "ANSI Metric."

The following Hardware is available.

- Bolts and Screws
- Nuts
- Pins
- Washers

Each has an option of Type, Size and Scale.

Active star	ndard: 🧮 Al	NSI Metric	Preview	
nd Screws Standard	ANSI Metric			
Type	Hex Bolt			
Size	M5x0.8			
Scale	1			
Name		Hex Bolt M5 x0.8		
-	method	Hex Bolt M5 x0.8 Side view 1		
shere Name	method			
	method			
shere Name	method			
Name				×

Figure 5-02

START A DRAFTSIGHT SESSION

- 1. Startup DraftSight
- 2. Start a new drawing and select "standard.dwt"
- 3. Set Drawing Unit System for a Metric drawing.
 - a. Decimal
 - b. Two decimal places
 - c. Millimeters
- 4. Set the Boundary to Landscape A4 (210 x 297)
- 5. Zoom Bounds
- 6. Set Snap and Grid to 10.
- 7. Create three new Layers with custom properties:
 - a. Object (White)
 - b. Centerline (Blue with Center LineStyle)
 - c. Construction (Red with Hidden LineStyle)
 - d. Dimension (Magenta)
 - e. Dash Line (Red with Dash LineStyle)

HINT: If you have been following along, you will see that we have repeated the same setting. Once you set the Units, Boundary, Snap, Grid and Layers save it as a Template for future use (i.e. ANSI_METRIC.DWT).

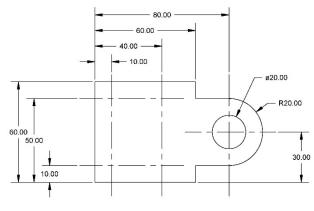
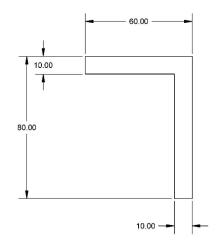


Figure 5-03





- Draw the two drawings in Figures 5-03 and 5-04
 Note that Figure: 5-03 is from Lesson 1, Quick Start
- 9. Project the Front View of Figure: 5-03
- 10. Join the two figures as shown in Figure: 5-05
- 11. From the Toolbox toolbar or pull-down menu, pick the Standards icon, set the Standard to ANSI Metric.
- 12. Select a M5 washer and create a side view, click "**OK**". *Optional: Place two washer on top.*

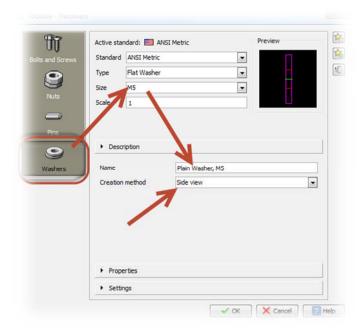


Figure 5-05

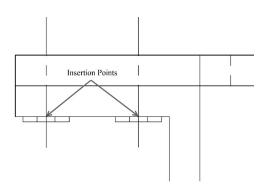
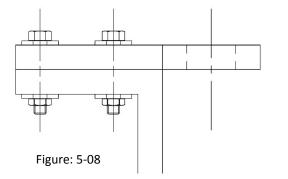


Figure 5-07

Note: Two clicks are required, one for insertion point and the other is for the Rotation.

Figure 5-06

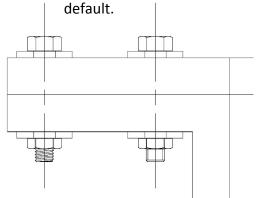
- 13. Select the M5 **Nut** and place under the washer. Remember you need one click for the insertion point and another point to select the rotation.
- 14. Select the M5 Bolt. Make sure Side View is still showing in the Creation Method.
 - a. Select Properties, in Material Thickness, pick at the top and then the bottom of the bolt. The distance is 26.575mm. An image will appear in the Preview.
 - b. Set the Fastener Length to 30. —
 - c. The Thread Display has options. This is optional for you to experience.



Bolts and Screws	10000000000	ANSI Metric			Ŕ
	Туре	Hex Bolt			10
	Size	M5x0.8			
Nuts	Scale	1			
Pins					
	▶ Desc	ription			
9		ortion			
Washers	- Frop	erues			
	Head op	tions	No bearing	•	
	Thread	lisolay	Simplified	•	
	Fastene	r length	30	•	
		angur			
	Material	thickness	26.575	°. `	
	 Setti 	ngs			

Figure 5-09

- 15. Edit the display by using the Edit Hardware icon on the Toolbox toolbar. 🕎
 - a. Once you pick the icon a dialog box will appear, see Figure: 5-10.
 - b. In this example, we will change the Thread Display to Detailed as shown in Figure 5-11.
 - c. You have an option to select All, Single or Multiple. Single is the



intity info					
Property	Value	*			
Quantity Found	1				
Description	Hex Bolt M5 x0.8				
Standard	ANSI Metric				
Size	M5x0.8				
Creation Method	Side view 3				
Thread Display	Simplified	ļ			
Thread Length	16				
intity edit					
Al O Single	 Multiple Edit. 				

Figure 5- 10

Figure 5-11

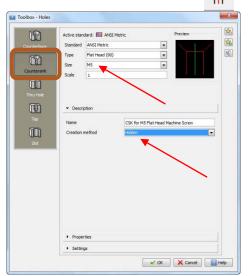
HOLES 💾 🖻 🕅 🕎 🖍

You can add 2D representations of Holes to drawings based on DraftSight's pre-set Standards or your custom standards. Once the **Holes** are created through this process they can be easily edited, create a **Hole Table**, edit the **Hole Table** and call out the **Hole** specs.

Let's take a look at the Plan View of the first drawing, see Figure 5-12.

We are going to change the design and go with Counter Sink Holes

Start by selecting the **Hole** icon.



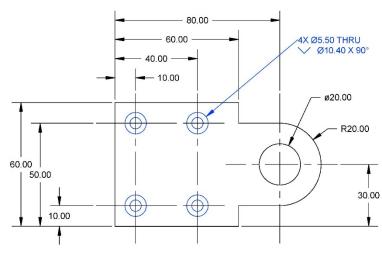


Figure 5-12

- 1. Select Countersink option.
 - a. Size = M5
 - b. Creation Method = Near Side
 - c. OK
- 2. Place 4 holes as shown in Figure 5-13
- 3. Select the **Hole Callout** _____5 Icon and place it as shown in Figure 5-13.

200

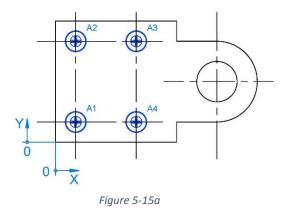
Figure 5-13

DRILL HOLE TABLE

- 1. Select the Hole Table Icon.
- 2. Select **OK** for the Default Table.
- 3. Select the lower left-hand corner of the object.
- 4. Select the Drill Holes.
- 5. Select the upper right hand corner of where the Hole Table is going to be generated, as shown in Figure 5-15b.

Toolbox - Hole T	able				×
Options					
Template: 🕅		Precision:	0.00	💌 🗹 Make uppercase	
Preview					
		DLE 1			
LABEL	X	Y		DESCRIPTION	
LABEL	XLOC	YLOC	DESC		
			~ (K X Cancel	7 Help





HOLE TABLE								
LABEL	Х	Y	DESCRIPTION					
A1	10.00	10.00	CSK FOR M5 FLAT HEAD MACHINE SCREW					
A2	10.00	50.00	CSK FOR M5 FLAT HEAD MACHINE SCREW					
A3	40.00	50.00	CSK FOR M5 FLAT HEAD MACHINE SCREW					
A4	40.00	10.00	CSK FOR M5 FLAT HEAD MACHINE SCREW					



Note: Scale factors may need to be added to tables and text.

SURFACE FINISH AND WELDING SYMBOLS

- Selecting the Surface Finish Finish Symbols dialog box.
 - a. Basic is the Default Style.
 - b. Specify Material Removal.
 - c. Specify Roughness min & max.
 - d. Under **Settings**, the options to change the TextStyle, Text Height, add a Leader and change Layer Properties.



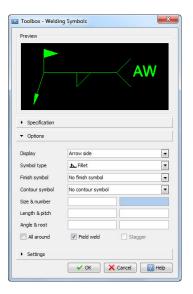
Figure 5-16

- 7. Selecting the Flip Surface Finish Icon 🥳 will flip the symbol to a mirror image.
- 8. Selecting the **Welding** Icon will activate the Toolbox Welding Symbols dialog box.
 - a. **Specification** options allows you to place the type of weld or your own text. (Figure: 5-17a)

-1×--

- b. **Options** allow you to change the display of the Weld Symbol. (Figure: 5-17b)
- c. **Settings** allow you to change the TextStyle, Text Size, and Layer Preferences. (Figure: 5-17c)
- 9. Place a Finish Symbol and Welding Symbol as shown in Figure 5-18.

Preview	AW
,	
 Specification 	
Abbreviation	AW (Arc welding)
Text	
Options	



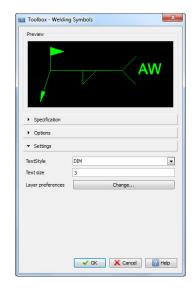


Figure 5-17a

Figure 5-17b

Figure 5-17c

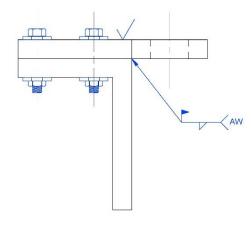


Figure 5-18

Note: Scale factors may need to be added to tables and text.

INSTERING BALLONS

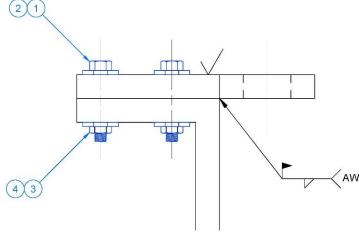


Toolbox lets you annotate drawings with **Balloons** and then use them to create a **Bill of Materials** (BOM) for the drawings.

Before you insert Toolbox **Balloons** and generate **BOM** from them, you can customize **Balloon Settings** to specify how the **Balloons** appear in your drawing.

9

- 1. Select the Balloon Settings
- 2. In the Style options:
 - Shape allows you to select a variety of geometric shapes that apply to most drafting standards, the Preview box will show the selected shape.
 - b. Size allows you to select the number of characters in the Balloon. The default is 1 character.
 - c. Leader can be on or off.
 - d. Arrowhead Size, it is scaled from the Dimension Style setting.
 - e. Stacked allows you to stack your Balloons to the Left, Right, Up, Down or No stacking.
- 3. In the Options:
 - a. Select the TextStyle
 - b. Select the Text Size (use factor scale to adjust the Text Size).
 - c. BOM Header Orientation for Top or the Bottom orientation.
 - d. Toggle the Auto Increment, BOM Make Uppercase, Verify Item Data, and Count Items.
- 4. Insert the Balloons as shown in Figure 5-21.



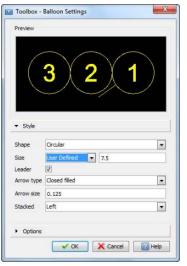


Figure 5-19

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✓ Options	
TextStyle	DIM
Text size	3
BOM header orientation	Тор
a des a succession de succession succession	BOM make uppercase
Auto increment	
	Count items

Figure 5-20

Figure 5-21

BILL OF MATERIALS (BOM) TABLE



You can create a **BOM** in a drawing after you insert **Balloons**. The **BOM** shows data from the blocks associated with **Balloons**.

The **Edit BOM** command lets you select a BOM and recreate it, delete it, or edit the rows.

You can export a **BOM** to a text file that shows data for the blocks associated with **Balloons**.

- 1. Select the Create BOM Icon
- 2. This will capture the data from the **Balloons** and create a **BOM**. Use factor scale (i.e. 25.4) to **Scale** up the **BOM**.
- 3. Edit BOM **BILL OF MATERIALS** will let you recreate it, ITEM PART NO DESCRIPTION QTY delete it, or edit the rows. 1 HEX BOLT M5 X0.8 4 2 PLAIN WASHER, M5 4. Export 4 HEX JAM NUT M5 X0.8 3 4 **BOM** will 4 PLAIN WASHER, M5 4 allow you create a

text file that can be imported into a spread sheet program.

INSERTING A REVISION TABLE

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Use **Revision Table** to track revisions in a drawing.

A **Revision Table** consists of a title and header row and individual revision rows. When you create a revision row, you can insert revision symbols that are linked to the row.

 Start the Revision Table, pick a point on the drawing, accept the defaults and it will activate dialog box to start a new Revision Table.

REVISION 1	DATE 15/08/2014	
	REVISION 1	REVISION 1 DATE 15/08/2014

- a. **ZONE**: Enter the area based on the title block grid system
- b. **REVISION**: Toolbox will keep track of the Revisions, however, it can be edited to your standards.
- c. **DATE:** Current date will show or recorded formatted date.
- d. **DESCRIPTION**: Description of revision done on drawing.
- e. **APPROVED**: Person name or initials approving the revision.

- 2. Once a **Revision Table** is established, **Revisions** can be added with the **Insert Revision** Icon to add a row to the **Revision Table**.
- 3. The Insert Revision Symbol Icon will add a Revision Symbol to the drawing sheet. The Revision Symbol is linked to the most recent revision row.
- 4. Edit Revision Table will allow the user to edit the table.

Note: Scale factors may need to be added to tables and text.

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