hsCADCreator Help

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Table of Contents

Index 314

1   Introduction

This help document includes information on how to Quickly get Started ⌐9⌐ in **hsCADCreator** as well as the more in-depth details of the hsCADCreator Handbook ⌐20⌐. Also available are Licensing ⌐296⌐, Support ⌐313⌐, and Version ⌐313⌐ Information, as well as Frequently Asked Questions ⌐310⌐.

1.1   Quick Start

Quick Start provides an overview of **hsCADCreator**. It is a primary goal of **Hachisoft Corporation** that **hsCADCreator** be a powerful tool in the hands of its user clientele. There are four fundamental ways to use **hsCADCreator**. Click on a links below to learn more about **hsCADCreator**.

1.   **Creating Drawings** ⌐11⌐
2.   **Editing Drawings** ⌐12⌐
3.   **Printing Drawings** ⌐14⌐
4.   **Viewing Drawings** ⌐13⌐

To insure your ability to utilize this software we recommend the following steps:

Become acquainted with the Tools
Become acquainted with all of the tools ⌐100⌐--one may be more effective for your job undertaking than another; knowing which tool to access and how to use it is critical to efficiently using **hsCADCreator**.

Become acquainted with User Interface

Our user interface is very dynamic and customizable ⌐275⌐. As you become familiar with **hsCADCreator** you will be able to modify the user interface and layout ⌐279⌐ to fit your personal needs. Each window (Help, Notification ⌐267⌐, Layers, Library,Drawing, and Viewport) within **hsCADCreator** has a purpose. Each will help you along on your way to utilizing this software to its full potential.

Use the Help Manual

Use the program Help manual (or the hard copy **Owners Manual**). Questions you may have can often be answered by reading the pertinent information there.

Check Frequently Asked Questions

Check the Frequently Asked Questions ⌐310⌐ (FAQ) page. Your inquiry may already be covered there.

Contact Us

Contact us ⌐313⌐ if steps (1), (2), or (3) have not alleviated your questions about how to use the program. We would be glad to assist you.

Note:

Evaluators of the free trial version of **hsCADCreator** may want to peruse the Evaluation Quick Start ⌐10⌐.

1.1.1   Open hsCADCreator

To open **hsCADCreator** click on **hsCADCreator** under your All Programs items. This will start the **hsCADCreator** program.

hsCADCreator is by default installed with an entry into your start menu. To start **hsCADCreator** go to you **Start Menu** ➡ **Hachisoft** ➡ **hsCADCreator** ➡ **hsCADCreator** as indicated in the image below.

1.1.2   Evaluation (Trial Version)

If you have acquired a free trial copy of **hsCADCreator**, then it will start in an "Evaluation" mode. The good news is that you now have everything you need to run a full version of **hsCADCreator** except a license key. No additional downloads are necessary should you wish to purchase and continue using **hsCADCreator** past the evaluation period.

For information on how to buy one or more licenses, please see the Upgrade Instructions ⌐296⌐.
For information on how to request trial extension, please see the Trial Extension Instructions ⌐302⌐.

When **hsCADCreator** starts in Evaluation mode, you will see this dialog:

Click the "Continue with **hsCADCreator** in Evaluation Mode" button to start Drawing and Designing.

When the trial period expires the prompt will continue to allow you enter a license key, but will prevent entrance to **hsCADCreator**. If more time is required to evaluate **hsCADCreator**, please see Trial Extension 302 for further information.

Return to the Quick Start 9 .

Also See:
    Licensing 296

1.1.3 Creating Drawings

hsCADCreator is the means by which the user can produce professional drawings with ease. It does not matter what level of CAD designer you perceive yourself to be, you can create sophisticated drawings by simply becoming familiarized with all of the tools and features of this program.

To create a new drawing, click on File 93 ➡ New 93 menu item after opening hsCADCreator 10 . This will create an empty drawing file with default drawing settings 76 derived from application settings 264 . Use variety of available create tools 103 to design using specialized drawing entities 21 .

The following entities can be drawn in **hsCADCreator**:
- **Points** 22
- **Lines** 23
- **Multi Lines** 24
- **Arcs** 28
- **Polylines** 26
- **Circles** 30
- **Ellipses** 32

Also see:

1.1.4   Editing Drawings

hsCADCreator editing tools allow the user to tweak or significantly alter portions or his/her drawing in progress. Several tools are at your disposal to improve your work to the level you desire even if it didn't seem that was possible at first:

Use modify tools 147 to modify already drawn entities.

Use <u>viewing tools</u> 173 to view current drawing from different angles.

Use <u>library tools</u> 217 to view and modify available library <u>objects</u> 50 for convenient reusability.

Also see:

       <u>Modify Tools</u> 147

       <u>Draw Plane (UCS) Tools</u> 193

       <u>View Tools</u> 173

       <u>Library Tools</u> 217

1.1.5   Viewing Drawings

hsCADCreator has variety of <u>view tools</u> 173 available to easily inspect the drawing from different angles and zoom levels. Alternatively, user can divide the default viewport into multiple viewports using the <u>Divide Viewport Tool</u> 184. This allows the user to see the model from different viewing angles at the same time. Each viewport is supported by individual <u>UCS system</u> 59 for easy editing of model.

Also see:
> View Tools |173|
> Divide Viewport Tool |184|

1.1.6   PrintingDrawings

hsCADCreator printing tools allow the user to print/plot newly created or existing drawings. Print Dialog |266| allows user to select paper size from more than 40 commercial paper sizes available, configure orientation and scale, and import plot styles |68|. **hsCADCreator** creates by default two paper space layouts |232| with newly created drawings. User can create/edit viewport entities |49| on these paperspace layouts for easy management of document printing.

Also see:

1.2   Terminology

Mouse-mode Terminology:

Left Click   : Left-button of mouse pressed and released. (also referred to as **click**)

Right Click   : Right-button of mouse pressed and released.

Middle Click/Wheel Click   : Middle/Wheel button of mouse pressed and released.

Wheel Scroll   : Middle/Wheel button of mouse rotated.

Left Mouse Drag   : Left-button of mouse pressed while moving mouse.

Right Mouse Drag   : Right-button of mouse pressed while moving mouse.

Middle Mouse Drag : Middle-button of mouse pressed while moving mouse.

Mouse Move   : Without pressing any mouse button, move the mouse.

Release Mouse: Release any button that is currently pressed on mouse.

Double Click   : Use left click operation in quick succession.

Mathematical Terminology:

Perpendicular: Perpendicular is a geometric term that may be used as a noun or adjective. The fundamental meaning pertains to the position of straight lines relative to one another. Two lines are said to be perpendicular if they meet at a right angle. Note that two line segments positioned at 90° to one another are perpendicular only if they meet.

***Orthographic Projection*:** Orthographic projection is a means of representing a three-dimensional object in two dimensions. It uses multiple views of the object, obtained by rotating camera about the object's center through increments of 90°. It produces two plan views (top, bottom) and four side views (front, back, left side, right side).

***Isometric Projection*:**Isometric projection is a form of orthographic projection, or more specifically, an axonometric projection. It is a method for the visual representation of three-dimensional objects in two dimensions in which the angles between the projection of the *x*, *y*, and *z* axes are all the same, or 120°. Isometric projection can be visualized by considering the view of a cubical room from an upper corner, looking towards the opposite lower corner. The term *isometric* comes from the Greek for "equal measure.", which reflects that the scale along each axis of the projection is the same (this is not

true of some other forms of graphical projections). Isometric projection is one of the projections used in drafting engineering drawings.

UCS: User Coordinate System/Space. The three-dimensional coordinate system where the drawing screen is located (as compared to the World Coordinate System [WCS] which may or may not coincide with the UCS).

WCS: World Coordinate System/Space. The three-dimensional coordinate system that is base for all UCS. Each UCS-axis is define in WCS coordinates. In a drawing there is only one WCS but may have one or more UCS.

2D space: 2D refers to the concept of a two-dimensional plane as in the Cartesian coordinate system with two axes, X and Y. Each axis is at right angles with the other axis. The point of intersection, where the axes meet is called the origin and normally labeled 0 (or 0,0). To specify a particular point in a two-dimensional coordinate system, the X unit (the horizontal distance from the origin, or abscissa), is indicated first, followed by the Y unit (the vertical distance from the origin, or ordinate) is second. So for example, the ordered pair (4,3) would represent that point four units to the **right**, and three units **above** the origin. Likewise, the ordered pair (-4,-3) would represent that point four units to the **left**, and three units **below** the origin.

3D space: 3D refers to the concept of a three-dimensional coordinate system that includes the Z-axis [altitude, or other third dimension of space measurement] in addition to the X-axis [horizontal] and Y-axis [vertical] of the two-dimensional Cartesian coordinate system. Each axis is at right angles with the other two. The point of intersection, where the axes meet is called the origin and normally labeled 0. To specify a particular point in a three-dimensional coordinate system, the X unit (the horizontal distance from the origin, or abscissa), is indicated first, followed by the Y unit (the vertical distance from the origin, or ordinate) is second, and the Z unit (the applicate) is third.

1.3   Feature List

Full CAD Editor functionality:

- Create new Drawings [11]
- Edit existing Drawings [12]
- View Drawings [13]
- Print/Plot Drawings [14]
- Export Drawings [286]

Uses industry-standard file formats for editing:

- **DWG** [287]
- **DXF** [287]
- **DWF** [288]

Design using specialized Drawing **Entities** [21]:

- **Points** [22]
- **Lines** [23]
- **Multi Lines** [24]
- **Arcs** [28]
- **Polylines** [26]
- **Circles** [30]
- **Ellipses** [32]
- **Images** [47]

- **Text** 43
- **Blocks** 61
- **Dimensions** 33
  - **Aligned** 35
  - **Angular** 42
  - **Radial** 39
  - **Diametric** 41
  - **Ordinate** 38
- **Hatches** 45
- **Poly Face Meshes** 46

Design in 2D and 3D:

- Powerful support of 2D editing
- Support of 3D editing through Draw Planes 59 and 3D Viewing 173
- No support for Creating or Editing 3D solids in this version, but coming soon!

Auto Save / Auto Recover ability:
- Never lose your drawing due to one of those power failures. **hsCADCreator** has Auto Save feature that saves your all open drawings at regular time interval. In case of abnormal/ unwanted shutdown, **hsCADCreator** automatically recovers the last Auto Saved files.

Design with the assistance of Intuitive Tools:

• Tool Interaction ➲ 100
- Point, click and drag *mouse-based mode*
- Context-sensitive tool properties allow for a precise *data-entry mode* (no cryptic "command line" necessary)
- "Notify window" 267 gives context-sensitive help, instructions, and feedback
- Visual aids 270 guide you through each tool process
- Unique cursors show you which tool you are using

• Entity Selection Tool ➲ 101
- Easily select one or more Entities
- Crossing rectangle selection
- Single click selection
- Filter current selection based on Color, Layer, Entity Type.
- View any Properties 77 in common across the entire selection in the Selection Property Tree 259
- Modify Properties 77 across the entire selection

• Entity Creation Tools for every Entity type ➲ 103
- Custom tool for every Entity type
- Tool to create repeated Blocks 61 of Entities
- Tool to insert Raster Images

• Entity Modification Tools ➲ 147
- Translate selected Entities
- Rotate selected Entities
- Explode selected Entities
- Delete selected Entities

- Scale selected Entities Uniformly
- Scale selected Entities Non-uniformly
- Trim selected Entities
- Extend selected Entities
- Clone selected Entities

- ## View Manipulation Tools ➲ 173
  - Pan
  - Zoom to Window
  - Zoom In
  - Zoom Out
  - Zoom Extent
  - Rotate
  - Snap to standard orthogonal and isometric views
  - Divide Viewport
  - Standard Viewports

- ## Rendering Tools ➲ 213
  - 2D Wireframe
  - 3D Wireframe
  - Hidden Lines
  - Flat Shaded
  - Gouraud Shaded
  - Flat Shaded with edges
  - Gouraud shaded with edges
  - Regenerate

- ## Measure Tools ➲ 244
  - Measure Distance
  - Measure Area
  - Measure Angle

- ## User Coordinate Tools ➲ 193
  - Preset UCS View Snaps
  - Rotate UCS Tools
  - Translate UCS Tools
  - Align UCS Tools

- ## Library Tools ➲ 217
  - Manage Blocks
  - Manage Colors
  - Manage Dimension Styles
  - Manage External References
  - Manage Hatch styles
  - Manage Image Definitions
  - Manage Layers
  - Manage Layouts
  - Manage Linetypes
  - Manage Multi Line Styles
  - Manage Named Views

- Manage Plot Settings
- Manage Text Styles
- Manage User Coordinate Systems
- Manage Viewports
- Quick Layer View

2   Handbook

Designing and drafting with the aid of a computer is equal parts art and craft. **hsCADCreator** seeks to facilitate the precision and practicality of CAD while still maintaining an enjoyable user experience. The following handbook details aspects of the art and craft of CAD.

- Concepts 20 - Background information and introductive CAD theory.
- User Interface 90 - The visual components of the **hsCADCreator** design interface, what they do, and why. Describes all Tools 100, Toolbars 250, Menus 90, and Dialogs 264 in detail.
- Files and Formats 286 - Details on what file types, formats, and versions that are supported in this application, as well as compatibility details.

2.1   Concepts

hsCADCreator employs specific concepts to facilitate drawing creation, editing and viewing:

Entities 21 : Various types of entities that can be drawn.

Objects 50 : Information holders for easy access to stored data.

Snap Point : Ability to direct cursor to various points on an entity.
Settings 71

Entity Grip : Each entity has one or more manipulation or grip points associated with it which allow
Points 74   the user to change visual properties of the entity.

Grid 75   : Display of equidistant points along X and Y axis for ease of drafting.

Drawing : Background color of drawing screen, Units, etc. settings for active drawing.
Settings 76

Data Types : Various types of information commonly used in drawings.
and
Properties
77

Property : Collection of properties 77 displayed in tree structure.
Trees 79

Tools 100   : **hsCADCreator** provides a number of ways to easily create/edit/delete/manage all
entities, objects, and their properties.

Toolbars : Collection of tools.
81

Dialogs 88 : Collection of various user interfaces to get done various complex tasks easily.

Undo/Redo : Process to remove or reinstate last change made in drawing.
 89

2.1.1   Entities

In **hsCADCreator** many distinct entities can be created in drawings.

1. *Point* 22
2. *Line* 23
3. *Multi Line* 24
4. *2D Polyline* 26
5. *3D Polyline* 27
6. *Arc* 28
7. *Circle* 30
8. *Ellipse* 32
9. *Dimension* 33
10. *Text* 43
11. *Hatch* 45
12. *Poly Face Mesh* 46
13. *Block Definition* 61
14. *Block Insertion* 46
15. *Image Definition* 63
16. *Image Insertion* 144
17. *Viewport* 49

All above entities(except Block Definition and Image Definition Entities) have some common properties that can be set at the time of creation or modified by selecting entity/ies in the drawing. These common properties are:

Field Name	Data Type	Description
Color	Color 220	Color of the pen with which this entity gets drawn.
Layer	Layer 64	Weight(Width) of the pen with which this entity gets drawn
Lineweight	Scientific Data 78	Imaginary transparent drawing sheet on which this entity gets drawn.
Linetype Scale	Scientific Data 78	Scaling factor for line type being used.
Linetype	Line Type 66	Type of the pen (dotted, dashed, dot-dash-dot, etc.) with which this entity gets drawn.

In **hsCADCreator** above properties are shown in <u>Selection Property Tree</u> 259 when any entity is selected using <u>Selection Tool</u> 101 .

2.1.1.1 Point

A *point* is an entity that has a location in space but no extent.

.

In **hsCADCreator**, a *Point Entity* consists of the following data in addition to the Common Data 21 :

Property Name	Data Type	Description
Position	3D Point 77 (X, Y, Z coordinates)	The point's location in space.

In **hsCADCreator** the above properties are shown in the <u>Selection Property Tree</u> 259 when a *point entity* is selected using <u>Selection Tool</u> 101 .

To draw a *point*:

 Entity toolbar 25⌐ **:**

 Tool Menu 97⌐ **:** *Create* ↳ *Point*

See also:
        Point Tool 105⌐

2.1.1.2   Line

A *line*, or *straight line*, **hsCADCreator** can be described as an (infinitely) thin, (infinitely) long, perfectly straight curve. The *line* provides the shortest connection between two points 22⌐. A *line segment* is a part of a line that is bounded by two end points 22⌐. The line entity represents a *line segment* in 3D space 17⌐.

In **hsCADCreator**, a *Line Entity* is a line segment that consists of the following data in addition to the Common Data 21⌐:

Property Name	Data Type	Description
Starting Point	3D Point 77⌐ (X, Y, Z coordinates)	The point locations in space.
Ending Point	3D Point 77⌐ (X, Y, Z coordinates)	The point locations in space.
Length	Scientific Data 78⌐	The distance of the line segment

		from one end point to the other

In **hsCADCreator** the above properties are shown in the Selection Property Tree |259| when a *line entity* is selected using the Selection Tool |101|.

To draw a **line**:

    Entity toolbar |251| **:**

    Tool Menu |97| **:** *Create* ✋ *Line*

See also:
    Line Tool |106|

2.1.1.3   Multi Line

A **Multi line** is a one or more parallel lines |23| drawn along a reference line. It allows easy editing/ creation of parallel lines as a single entity.

In **hsCADCreator**, a *Multi line Entity* consists of the following data in addition to the Common Data [21]:

Property Name	Data Type	Description
Multi line style	Mulit line style [67]	The point locations in space.
Justification	Multi-option [79] combo box. Available options are: -Align top with cursor -Align center with cursor -Align bottom with cursor	Justification of parallel lines with respect to the reference line defined by cursor inputs.
Number of vertices	Integer Number	Total number of vertices in this mulit-line.
Closed	Boolean [79]	When checked(true) joins start and end point of multi line with an extra multi line.
End Caps	Boolean [79]	Draws end caps at the end of multi-line if specified in multi line style.
Start Caps	Boolean [79]	Draws start caps at the start of multi-line if specified in multi line style.

In **hsCADCreator** the above properties are shown in the Selection Property Tree [259] when a *multi line entity* is selected using Selection Tool [101].

To draw a *Multi line*:

⬚ Entity toolbar [25] :

⬚ Tool Menu [97] : *Create* ⤳ *Multi Line*

See also:
> Multi Line Tool 108

2.1.1.4   Planar Polyline

A *Planar Polyline* is a continuous line 23 composed of one or more line 23 segments or arc 28 segments in a two dimensional space 17.

In **hsCADCreator**, the *Planar Polyline Entity* consists of the following data in addition to the Common Data 21:

Property Name	Data Type	Description
End Point(s)	3D Point 77 (X, Y, Z coordinates for each point) on a single plane.	The point locations in space.
Closed	Boolean 79	When checked(true) joins start and end point of polyline with an extra line.

In **hsCADCreator** the above properties are shown in Selection Property Tree 259 when a *planar polyline* is selected using Selection Tool 101. Currently the **End Point(s)** property is not available in property tree for editing, but **End Point(s)** can be edited using grip points 74 shown on the drawing screen.

To draw a **Planar Polyline**:

⬚ Entity toolbar [25] **:**

⬚ Tool Menu [97] **:** *Create* ⬐ *Polyline* ⬐ *Planar (2D)*

See also:
2D Polyline Tool [110]

2.1.1.5   3D Polyline

A *3D polyline* is a continuous line [23] composed of one or more line [23] segments in a three
dimensional space [17].

In **hsCADCreator**, the *3D Polyline Entity* consists of the following data in addition to the Common
Data [21]:

Property Name	Data Type	Description
End Point(s)	3D Point [77] (X, Y, Z coordinates for each point)	The point locations in space.
Closed	Boolean [79]	When checked(true) joins start and end point of polyline with an extra line.

In **hsCADCreator** the above properties are shown in the Selection Property Tree [259] when a **3D polyline** is selected using Selection Tool [101]. Currently the **End Point(s)** property is not available in the property tree for editing, but **End Point(s)** can be edited using grip points [74] shown on the drawing screen.

To draw a **3D Polyline**:

⬜ Entity toolbar [25] : 🔷

⬜ Tool Menu [97] : **Create** ⬧ **Polyline** ⬧ **Non-Planar (3D)**

See also:
3D Polyline Tool [112]

2.1.1.6 Arc

An **arc entity** is a continuous portion of a circle [30]; part of a circle's circumference (also called a circle segment).

In **hsCADCreator**, the *Arc Entity* consists of the following data in addition to the Common Data [21]:

Property Name	Data Type	Description
Center Point	3D Point [77] (X, Y, Z coordinates)	Center point of the circle that this arc represents.
Starting Angle	Scientific Data [78] (degrees, radians)	Angle of origin of the arc in relation to the Cartesian plane (360 degrees)
Ending Angle	Scientific Data [78] (degrees, radians)	Angle of completion of the arc in relation to the Cartesian plane (360 degrees)
Radius	Scientific Data [78]	The line segment distance from the arc's center to any point on the arc
Start Point	3D Point [77] (X, Y, Z coordinates)	Starting point of arc.
Through Point	3D Point [77] (X, Y, Z coordinates)	Any point on this arc. Initially, this point is set to be equal to Mid Point. When this property is changed, arc is recalculated such that it passes through Start Point, new Through Point and End Point. Mid Point is recalculated and Through Point is also updated to be equal to Mid Point.
Mid Point	3D Point [77] (X, Y, Z coordinates)	Mid point on arc between Start Point and End Point.
End Point	3D Point [77] (X, Y, Z coordinates)	Ending point of arc.

In **hsCADCreator** above properties are shown in the Selection Property Tree [259] when an *arc entity* is selected using the Selection Tool [101].

To draw an *Arc*:

Entity toolbar |25| :

      Arc(Start-Center-End) |115|

      Arc(Start-Center-Angle) |117|

      Arc(Start-Center-Chord Length) |119|

      Arc(Start-Middle-End) |114|

Tool Menu |97| : *Create* ⇗ *Arc* ⇗ *Start - Center - End*
                                ⇗ *Start - Center - Angle*
                                ⇗ *Start - Center - Chord Length*
                                ⇗ *Start - Middle - End*

See also:
      Start-Center-End Arc Tool |115|
      Start-Center-Angle Arc Tool |117|
      Start-Center-Chord Length Arc Tool |119|
      Start-Middle-End Arc Tool |114|

2.1.1.7   Circle

A *circle* is an entity that is the set of all points on a plane at a fixed distance, called the radius, from a fixed point, the center.

In **hsCADCreator**, a *Circle Entity* consists of the following data in addition to the Common Data 21 :

Property Name	Data Type	Description
Center Point	3D Point 77 (X, Y, Z coordinates)	The point's location in space
Radius	Scientific Data 78	The distance of the line segment from the circle's center to any point on the circle.

In **hsCADCreator** the above properties are shown in the Selection Property Tree 259 when a *circle entity* is selected using the Selection Tool 101 .

To draw a *Circle*:

Entity toolbar 251 :

Tool Menu 97 : *Create* ↳ *Circle*

See also:
Circle Tool 120

2.1.1.8 **Ellipse**

An *ellipse* is a plane algebraic curve (sometimes called an oval) where the sum of the distances from any point on the curve to two fixed points is constant. The two fixed points are called foci (plural of focus). An *ellipse* is a type of conic section: if a conical surface is cut with a plane which does not intersect the cone's base, the intersection of the cone and plane is an *ellipse*. The line segment which passes through the foci and terminates on the *ellipse* is called the major axis. The major axis is along the longest segment that passes through the *ellipse*. The line which passes through the center (halfway between the foci), at right angles to the major axis, is called the minor axis.

In **hsCADCreator**, an *Ellipse Entity* consists of the following data in addition to the Common Data [21]:

Property Name	Data Type	Description
Center Point	3D Point [77] (X, Y, Z coordinates)	The point's location in space.
Major Axis	3D Vector [78] (X, Y, Z coordinates)	The major axis of ellipse.
Start Angle	Scientific Data [78]	Start angle with reference to major axis.
End Angle	Scientific Data [78]	End angle with reference to major axis.
Radius Ratio	Real Number [78] (between 0 and 1)	Ratio of Minor Axis Length/Major Axis Length

In **hsCADCreator** the above properties are shown in the Selection Property Tree [259] when an *ellipse entity* is selected using the Selection Tool [101].

To draw an *Ellipse*:

Entity toolbar [25] **:**

Tool Menu [97] **:** *Create* ↳ *Ellipse*

See also:
Ellipse Tool [121]

2.1.1.9 Dimension

A *dimension* is a parameter or measurement required to define the characteristics of an object — ie. length, width, and height or size and shape. *Dimension entities* are used to show different type of measurements for various entities. In **hsCADCreator**, there are six type of *Dimension Entities*:

1. Aligned Dimension [35]

2. Linear Dimension [36]

3. Ordinate Dimension [38]

4. Radial Dimension [39]

5. Diametric Dimension [41]

6. Angular Dimension [42]

Each of these six *dimension entities* have some common properties in addition to their individual properties. These common properties are as follows:

Property Name	Data Type	Description
Common properties from Entities:		
Line Color	Color [220]	Color of the pen with which this entity gets drawn.

Line Weight	Real Number [78]	Weight(Width) of the pen with which this entity gets drawn
Layer	Layer [64]	Imaginary transparent drawing sheet on which this entity gets drawn.
Line Type	Line Type [66]	Type of the pen (dotted, dashed, dot-dash-dot, etc.) with which this entity gets drawn.
Line Type Scale	Real Number [78]	Scaling factor for line type being used.
Common properties from Dimension:		
Use Dimension Style only	Boolean [79]	When checked it uses Dimension Properties defined in dimension style and does not use local properties as override.
Dimension Style	Dimension Style [51]	Pre-created dimension style that defines various properties for dimension entities.
Dimension Properties	Dimension Style [51]	Local dimension style that defines various properties for dimension entities that can override properties defined by pre created dimension style.

In **hsCADCreator** the above properties are shown in the Selection Property Tree [259] when a *dimension entity* is selected using the Selection Tool [101].

To draw a *Dimension*:

Entity toolbar [251] :

Aligned Dimension [124]

Linear Dimension [126]

Ordinate Dimension [128]

Radial Dimension [130]

Diametric Dimension [132]

Angular Dimension [134]

Tool Menu [97] : *Create* ✎ *Dimension Indicator* ✎ *Aligned*
  ✎ *Linear*
  ✎ *Ordinate*
  ✎ *Radial*
  ✎ *Diametric*
  ✎ *Angular*

See also:
Aligned Dimension Tool [124]
Linear Dimension Tool [126]
Ordinate Dimension Tool [128]
Radial Dimension Tool [130]
Diametric Dimension Tool [132]
Angular Dimension Tool [134]

2.1.1.9.1 Aligned Dimension

Aligned Dimension is used for displaying and measuring length along any given linear entity. As the name suggests it can be aligned along any direction. Generally, these dimensions [33] are used to show absolute lengths along any direction.

In **hsCADCreator**, an *Aligned Dimension Entity* consists of the following data in addition to the Entity Common Data [21] and the Dimension Common Data [33]:

Property Name	Data Type	Description
Start Point	3D Point [77] (X, Y, Z coordinates)	Starting point for measuring length.
End Point	3D Point [77] (X, Y, Z coordinates)	Ending point for measuring length.

Offset Point	3D Point 77 (X, Y, Z coordinates)	Reference point where dimension entity should be drawn.

In **hsCADCreator** the above properties are shown in the Selection Property Tree 259 when an *aligned dimension entity* is selected using the Selection Tool 101.

To draw an *Aligned Dimension*:

Entity toolbar 251 :

Tool Menu 97 : *Create* ⬎ *Dimension Indicator* ⬎ *Aligned*

See also:
Aligned Dimension Tool 124

2.1.1.9.2 Linear Dimension

Linear Dimension is used for displaying and measuring length along X or Y axis. As the name suggests it can only be aligned along the X or Y axis. Generally, these *dimensions* are used to show absolute lengths along the X or Y axis.

In **hsCADCreator**, a *Linear Dimension Entity* consists of the following data in addition to the Entity Common Data [21] and the Dimension Common Data [33]:

Property Name	Data Type	Description
Start Point	3D Point [77] (X, Y, Z coordinates)	Starting point for measuring length.
End Point	3D Point [77] (X, Y, Z coordinates)	Ending point for measuring length.
Offset Point	3D Point [77] (X, Y, Z coordinates)	Reference point where dimension entity should be drawn.

In **hsCADCreator** the above properties are shown in the Selection Property Tree [259] when a *linear dimension entity* is selected using the Selection Tool [101].

To draw an ***Linear Dimension***:

    <u>Entity toolbar</u> [25] **:**

    <u>Tool Menu</u> [97] **:** ***Create*** ⤷ ***Dimension Indicator*** ⤷ ***Linear***

See also:
    <u>Linear Dimension Tool</u> [126]

2.1.1.9.3   Ordinate Dimension

Ordinate Dimensions are used for measuring the length along any X or Y axis and displaying length as a text with the use of a leader. Generally, these ***dimensions*** are used to show lengths of entities using leader lines.

In **hsCADCreator**, an ***Ordinate Dimension Entity*** consists of the following data in addition to the <u>Entity Common Data</u> [21] and the <u>Dimension Common Data</u> [33]:

Property Name	Data Type	Description
Origin Point	3D <u>Point</u> [77] (X, Y, Z coordinates)	Starting point for measuring length.
Defining Point	3D <u>Point</u> [77] (X, Y, Z coordinates)	Ending point for measuring length.
Leader End Point	3D <u>Point</u> [77] (X, Y, Z	Reference point where

	coordinates)	dimension entity should be drawn.
On X-axis	Boolean [79]	If checked(true) shows dimension text along X-axis, else shows it along Y-axis.

In **hsCADCreator** the above properties are shown in the Selection Property Tree [259] when an *ordinate dimension entity* is selected using the Selection Tool [101].

To draw an *Ordinate Dimension*:

    Entity toolbar [25] :

    Tool Menu [97] : *Create* ⭲ *Dimension Indicator* ⭲ *Ordinate*

See also:
    Ordinate Dimension Tool [128]

2.1.1.9.4 Radial Dimension

Radial Dimensions are used for measuring the radius of arcs [28], circles [30], and ellipses [32] and displaying it with a leader line.

In **hsCADCreator**, a *Radial Dimension Entity* consists of the following data in addition to the Entity Common Data 21 and the Dimension Common Data 33:

Property Name	Data Type	Description
Center Point	3D Point 77 (X, Y, Z coordinates)	Center point of arc, ellipse, circle.
Chord Point	3D Point 77 (X, Y, Z coordinates)	Point on circumference of arc, ellipse, circle.
Leader Length	Real Number 78	Length of leader line.

In **hsCADCreator** the above properties are shown in the Selection Property Tree 259 when a *radial dimension entity* is selected using the Selection Tool 101.

To draw an *Radial Dimension*:

⬚ Entity toolbar [25] **:** ✈

⬚ Tool Menu [97] **:** *Create* ✍ *Dimension Indicator* ✍ *Radial*

See also:
Radial Dimension Tool [130]

2.1.1.9.5 Diametric Dimension

Diametric Dimensions are used for measuring the diameter or chord of a circle [30] or ellipse [32] and showing it with a leader lines.

In **hsCADCreator**, a *Diametric Dimension Entity* consists of the following data in addition to the Entity Common Data [21] and the Dimension Common Data [33]:

Property Name	Data Type	Description
Near Chord Point	3D Point [77] (X, Y, Z coordinates)	Near chord point on circumference of arc, ellipse, circle.
Far Chord Point	3D Point [77] (X, Y, Z coordinates)	Far chord point on circumference of arc, ellipse, circle.
Leader Length	Real Number [77]	Length of leader line.

In **hsCADCreator** the above properties are shown in the Selection Property Tree [259] when a *diametric dimension entity* is selected using the Selection Tool [101].

To draw an *Diametric Dimension*:

⌨ Entity toolbar 25 **:** 🔍

⌨ Tool Menu 97 **:** *Create* ⬦ *Dimension Indicator* ⬦ *Diametric*

See also:
Diametric Dimension Tool 132

2.1.1.9.6 Angular Dimension

Angular Dimensions are used for measuring and displaying inside and outside angles.

In **hsCADCreator**, an *Angular Dimension Entity* consists of the following data in addition to the Entity Common Data 21 and the Dimension Common Data 33:

Property Name	Data Type	Description
Center Point	3D Point 77 (X, Y, Z coordinates)	Center point for angle measurement
Arc Point	3D Point 77 (X, Y, Z coordinates)	Offset point where angle text should be displayed
Line 1 Point	3D Point 77 (X, Y, Z coordinates)	Point on line measuring angle
Line 2 Point	3D Point 77 (X, Y, Z coordinates)	Point on line measuring angle

In **hsCADCreator** the above properties are shown in the Selection Property Tree 259 when an *angular dimension entity* is selected using the Selection Tool 101.

To draw an *Angular Dimension*:

Entity toolbar 25 :

Tool Menu 97 : *Create* ↳ *Dimension Indicator* ↳ *Angular*

See also:
Angular Dimension Tool 134

2.1.1.10 Text

A *Text Entity* is the body of printed or written matter on a drawing.

In **hsCADCreator**, a *Text Entity* consists of the following data in addition to the Common Data 21 :

Property Name	Data Type	Description
Text Style	Text Style 69	Pre-created text style that defines various properties for text entities.
Position	3D Point 77 (X, Y, Z coordinates)	Insertion point for this text.
Text	Text 79	Actual text
Text Height	Scientific Data 78	Text height

In **hsCADCreator** the above properties are shown in the Selection Property Tree 259 when a *text entity* is selected using the Selection Tool 101 .

To draw a *Text*:

    Entity toolbar 25 **:**

    Tool Menu 97 **:** *Create* ⬎ *Text*

See also:
    Text Tool 136

2.1.1.11 Hatch

Hatching and *cross-hatching* are artistic techniques used to create tonal or shading effects by drawing closely spaced parallel lines. When lines are placed at an angle to one another, it is called *cross-hatching*. Mostly *hatch entities* are used to show a cross-sectional view in a drawing.

In **hsCADCreator**, a *Hatch Entity* consists of the following data in addition to the Common Data [21]:

Property Name	Data Type	Description
Hatch Style	Hatch Style [60]	Hatch Style used to draw this hatch entity.

In **hsCADCreator** the above properties are shown in the Selection Property Tree [259] when a *hatch entity* is selected using the Selection Tool [101].

To draw a *Hatch*:

Entity toolbar [251] :

Tool Menu 97 : *Create* ✥ *Hatch*

See also:
Hatch Tool 138

2.1.1.12 Poly Face Mesh

Poly face mesh entity is a solid face entity that is created by meshing polylines 26 together. Viewing *poly face mesh entities* using Gouraud Shaded Tool 215, Flat Shaded Tool 214, or Hidden Tool 214, displays them as solid surfaces.

In **hsCADCreator**, a *Poly face mesh* does not have any special data different than Entity data 21.

To draw a *Poly face mesh*:

Entity toolbar 25 :

Tool Menu 97 : *Create* ✥ *Poly Face Mesh*

See also:
Poly Face Mesh Tool 139

2.1.1.13 Block Reference

Block Reference/Insertion entities represents copies of Block Definitions 61. In **hsCADCreator**, the Block Insertion Tool 142 is used to create *block insertion entities* from block definitions. For example, if the user frequently created drawings with a particular design of door, he/she could make a *block* (or stamp) of door and add it to Block Library 218 using Block Creation Tool 140, and insert it whenever his/her drawing called for it. Each door would look exactly like the others and would give the drawing a consistent appearance.

In **hsCADCreator**, a *Block Insertion Entity* consists of the following data in addition to the Common Data 21 :

Property Name	Data Type	Description
Block Name	Block Definition 61	Name of block definition used to create this block insertion.
Insertion Point	3D Point 77 (X, Y, Z coordinates)	The block's reference base point for insertions.
Scale Factors	3D Point 77 (X, Y, Z scale factors)	Scaling factors along X, Y and Z axis
Rotation	Scientific Data 78	Angle of rotation about Insertion Point

In **hsCADCreator** the above properties are shown in the Selection Property Tree 259 when a *block insertion entity* is selected using the Selection Tool 101 .

To draw a *Block* into drawing from Block Library:

⌨ Entity toolbar 25 **:** 🔳

⌨ Tool Menu 97 **:** *Create* ⬐ *Block Insertion*

See also:
　　Library Window 282
　　Block Entity 61
　　Manage Blocks 218
　　Block Insertion Tool 142

2.1.1.14 Image Insertion

Image Insertion Entity represents copies of Image Definitions 63 that are inserted in drawing.
Copies of *Image Entities* can be inserted into a drawing using the Image Insertion Tool 144 . To insert

an *Image Entity*, it is required that the image definition exists in Image Library²²⁸. Image definitions are added to the Image Library²²⁸ using the Manage Image Definitions Tool²²⁸.

In **hsCADCreator**, a *Image Insertion Entity* consists of the following data in addition to the Common Data²¹:

Property Name	Data Type	Description
Origin Point	3D Point⁷⁷ (X, Y, Z coordinates)	The Image's reference base point for insertions.
Dimension	2D Point⁷⁷ (X, Y size real numbers)	Size of Image.
Rotation Angle	Scientific Data⁷⁸	Angle of rotation about Insertion Point
File Name	Text⁷⁹	Full path including filename with extension for image.
Show Image	Boolean⁷⁹	If checked(true) shows actual image, else shows only border for the image.
Unaligned	Boolean⁷⁹	If checked(true) shows image unaligned.
Clip Image	Boolean⁷⁹	If checked(true) clips image to borders, else leaves image unclipped.
Transparent Image	Boolean⁷⁹	If checked(true) makes image transparent.

In **hsCADCreator** the above properties are shown in the Selection Property Tree²⁵⁹ when an *image insertion entity* is selected using the Selection Tool¹⁰¹.

To insert an *Image* into drawing from the Image Library 228:

⬜ Entity toolbar 25 **:** 🏠

⬜ Tool Menu 97 **:** *Create* ↳ *Image Insertion*

See also:
> Insert Image Tool 144
> Image Entity 63
> Manage Image Definitions 228

2.1.1.15 Paperspace Viewport

Paperspace Viewport Entity is a rectangular entity through which a 3D scene can be shown. In **hsCADCreator**, *Paperspace Viewport Entity* shows a view of the 3D Modelspace. *Paperspace Viewport Entity* can only be created on a paperspace layout. By default, **hsCADCreator** creates two paperspace layouts 64 with all new drawings. They are named "layout1" and "layout2".   To facilitate viewing of drawing through multiple viewports in Modelspace, **hsCADCreator** utilizes pre-set multiple viewports through Pre-configured Viewport Tool 186. Depending on whether the viewport exists on Modelspace or Paperspace they are defined as Modelspace viewport object 70 and *Paperspace viewport entity*.

In **hsCADCreator**, a *Paperspace Viewport Entity* consists of the following data:

Property Name	Data Type	Description
On	Boolean 79	Turns on/off this viewport entity.
Camera Settings	Multiple option 79 selection box.	Sets the camera settings for this viewport entity.
Center	3D Point 78	Center point for this viewport entity.
Width	Scientific Data 78	Width of viewport
Height	Scientific Data 78	Height of viewport
Transparency(%)	Real Number 78	Specifies transparency value for this viewport object.

In **hsCADCreator** the above properties are shown in the Selection Property Tree 259 when a *paperspace viewport entity* is selected using the Selection Tool 101.

To insert a ***paperspace viewport*** into drawing from the Viewport Library 242 :

Entity toolbar 251 **:**

Tool Menu 97 **:** ***Create*** 🖑 ***Viewport***

See also:
Create Viewport Entity 49
Divide Viewport Tool 184
Pre-configured Viewport Tool 186

2.1.2 Objects

Objects are information holders in **hsCADCreator**. ***Objects*** allows easy access to stored non-graphical data. There are total 11 types of ***objects*** in **hsCADCreator**. Visit the link on each ***object*** below for more information

1. Colors 51

2. Dimension Styles 51

3. Draw Plane (UCS) 59

4. External References 223

5. Hatch Style 60

6. Layers 64

7. Layouts 64

8. Linetypes 66

9. Plot Settings 238

10. Text Styles 69

11. Multiline Styles 67

2.1.2.1 Colors

Color Objects store color information in **hsCADCreator**. *Color Objects* are used to represent colors of background, layer, line color, etc. **hsCADCreator** creates nine colors on every drawing: Red, Green, Yellow, Cyan, Blue, Magenta, Foreground, Dark Background and Light Background.

In **hsCADCreator**, a *Color Object* consists of the following data:

Property Name	Data Type	Description
Color Method	Multi-option 79 selection box	Color method to be used for this color object
Red	Integer Number 78	Amount of red color.
Green	Integer Number 78	Amount of green color.
Blue	Integer Number 78	Amount of blue color.
Name	Text 79	Name of this color to save as.
Description	Text 79	Description for this color.
Explanation	Text 79	Explanation for this color.

To create *Color Objects* :

Library toolbar 217 **:**

Manage Menu 96 **:** *Colors...*

2.1.2.2 Dimension Styles

Dimension Style Objects represent formatting information for all dimension entities 33 in **hsCADCreator**. *Dimension Style* can be applied to dimension entities 33 to have similar look and feel across all dimension entities 33 in a drawing.

In **hsCADCreator**, a *Dimension Style Object* consists of the following data:

Property Name	Data Type	Description
Dimension Styles Properties:		
Arc Symbol Type	Multiple options 79 combo box.. Available options are: -Arc symbol precedes text. -Arc symbol is above text. -No arc symbol.	Sets the arc symbol type.
Text BG Color	Color 51	Sets the background color for text.
Extension line-1 Linetype	Linetype 66	Sets the first extension line linetype.

Extension line-2 Linetype	Linetype 66	Sets the second extension line linetype.
Dimension line Linetype	Linetype 66	Sets the dimension line linetype.
Extension Line Fix Length Enabled	Boolean 79	Controls if the extension line has a fixed length.
Extension Line Fix Length	Scientific Data 78	Sets the extension line fixed length.
Flip Arrow	Multiple options 79 combo box.. Available options are: -One arrow outside first extension line -One arrow outside second extension line -Arrows inside extension lines -Arrows outside extension lines	Sets the arrow flip option.
Jog Angle	Scientific Data 78	Sets the jog angle.
Dimension Line and Leader line properties:		
Hide first dimension line and arrow-head	Boolean 79	Show/hide first dimension line and arrow-head.
Hide second dimension line and arrow-head	Boolean 79	Show/hide second dimension line and arrow-head.
Dimension line extension	Scientific Data 78	The distance dimension lines extend beyond extension lines.
Dimension line distance	Scientific Data 78	The distance between dimension lines for baseline dimensions.
Dimension line Lineweight	Scientific Data 78	Lineweight for dimension lines
Hide dimension lines outside extension lines	Boolean 79	Show/Hide dimension lines outside extension lines.
Draw dimension lines when text is outside dimension lines	Boolean 79	Show/hide dimension lines between extension lines when text is outside dimension lines.
Extension line properties:		
Extension Lineweight	Scientific Data 78	Lineweight for extension lines
Hide first extension line	Boolean 79	Show/Hide first extension line.
Hide second extension line	Boolean 79	Show/Hide second extension line.
Extension line extension	Scientific Data 78	The distance extension lines extend beyond dimension lines.
Extension offset	Scientific Data 78	The distance extension lines are offset from their origin points.
Dimension text properties:		
Decimal separator	Text 79	Decimal separator for measurements. This value must be single character. If more than one characters are specified,

		application considers the first character as decimal separator.
Text style	Text Style 69	Text style for dimension text.
Text size	Scientific Data 78	Specify size of the dimension text. If text style being used has "Fix Size" variable set to TRUE then text size has no effect on dimension text.
Text horizontal position	Multiple options 79 combo box.. Available options are: -Centered above the dimension line. -By the first extension line. -By the second extension line. -Above the dimension line, parallel to the first extension line. -Above the dimension line, parallel to the second extension line.	Sets dimension text's horizontal position.
Text vertical position	Multiple options 79 combo box.. Available options are: -Centered about the dimension line. -Above the dimension line, unless DIMTIH=1 and the dimension line is not horizontal. -Side of the dimension line farthest from the defining points. -JIS standard.	Sets dimension text's vertical position.
Dimension type	Multiple options 79 combo box. Available options are: -Scientific -Decimal -Engineering -Architectural (stacked) -Fractional (stacked) -Microsoft Windows Desktop	Sets dimension formatting type for units in non-angular dimensions.
Decimal places	Integer Number 78	Number of decimal places
Zero suppression in linear dimensions	Multiple options 79 combo box. Available options are: -Suppress zero feet and exactly zero inches. -Include zero feet and exactly zero inches. -Include zero feet and suppress exactly zero inches. -Suppress zero feet and include exactly zero inches. -Suppress trailing decimal zeros. -Suppress leading decimal zeros.	Sets zero suppression options for linear dimension values.

Prefix and/or suffix	Text 79	Specify prefix and/or suffix for measurement text. The syntax is My Prefix<>MySuffix. Here "MyPrefix" is the prefix user wants to prepend before dimension value and "MySuffix" is the suffix user wants to append after dimension value. "<>" gets replaced by actual dimension value when dimension entity is created. e.g. <>in will be replaced by 10in when dimension entity is created. <>" will be replaced by 10" when dimension entity is created. @<> cm will be replaced by @10cm when dimension entity is created.
Text rounding	Real Number 78	Sets the rounding of measurements. If value is 0, no rounding is performed, else the measurement text is rounded to nearest multiple of rounding value specified. e.g. If measurement value is 8.81 and Number of Decimal places is set to 2, Rounding value = 0.000 will make measurement value 8.81. Rounding value = 1.000 will make measurement value 9.00 Rounding value = 2.000 will make measurement value 8.00 Rounding value = 5.000 will make measurement value 10.00 Rounding value = 1.100 will make measurement value 8.80 Rounding value = 5.200 will make measurement value 10.40
Make inside text X-aligned	Boolean 79	Controls the orientation of dimension text inside the extension lines. If checked makes text aligned with WCS X-axis otherwise aligns with dimension line.
Make outside text X-aligned	Boolean 79	Controls the orientation of dimension text outside the extension lines. If checked makes text aligned with WCS X-axis otherwise aligns with dimension line.
Force dimension text to put	Boolean 79	If checked it always puts

inside extension lines		dimension text inside extension lines, otherwise it puts inside only if there is room.
Text movement rules	Multiple options 79 combo box. Available options are: -Dimension line moves with dimension text. -Adds a leader for moved dimension text. -No leader, moves only dimension text.	Sets movement rules for the dimension text.
Dimension text gap(factor)	Real Number 78	The vertical distance from middle of the dimension text to the dimension line. This value is specified as factor of dimension text size.
Tolerance and fraction properties:		
Fraction format	Multiple options 79 combo box.. Available options are: -Horizontal Stacked. -Diagonal Stacked. -Not Stacked.	Sets the fraction format when dimension type is architectural or fractional. This value has no effect if "Dimension type" under "Dimension text properties" is not architectural or fractional.
Show tolerance	Boolean 79	Show/hide tolerance text.
Tolerance and fraction text size	Real Number 78	The size of tolerance and fraction text as a factor of Dimension text size.
Tolerance vertical position	Multiple options 79 combo box. Available options are: -Centered about the dimension line. -Above the dimension line, unless DIMTIH=1 and the dimension line is not horizontal. -Side of the dimension line farthest from the defining points. -JIS standard.	Controls vertical position of Tolerance with respect to dimension line.
Lower tolerance limit	Real Number 78	Sets lower tolerance limit.
Upper tolerance limit	Real Number 78	Sets upper tolerance limit.
Note: Since all decimal numbers are controlled by Application Settings variable "Decimal Places", if you want to specify tolerance limits with more decimal precision, change "Decimal Places" variable in Application settings. This can be accessed by following Menu: **Options ➡ Application settings..**.		
Zero suppression in tolerance values	Multiple options 79 combo box. Available options are: -Suppress zero feet and exactly zero inches. -Include zero feet and exactly zero inches. -Include zero feet and suppress exactly zero inches. -Suppress zero feet and include	Sets zero suppression in tolerance values.

	exactly zero inches. -Suppress trailing decimal zeros. -Suppress leading decimal zeros.	
Decimal places in tolerance	Integer Number [78]	Number of decimal place in tolerance values.
Dimension color properties:		
Dimension Color	Color [51]	Color of dimension line, arrow-head, leader lines.
Extension Color	Color [51]	Color of extension lines
Text Color	Color [51]	Color of dimension texts
Angular dimension properties:		
Precision(Angle)	Integer Number [78]	Decimal places in angular measurements
Suppress zero(Angular)	Multiple options [79] combo box. Available options are: -Suppress zero feet and exactly zero inches. -Include zero feet and exactly zero inches. -Include zero feet and suppress exactly zero inches. -Suppress zero feet and include exactly zero inches. -Suppress trailing decimal zeros. -Suppress leading decimal zeros.	Sets zero suppression in angular dimensions.
Arrow-head properties:		
Arrow-head size	Scientific Data [78]	Arrow-head size in dimension entities.
Arrow, Text Fit	Multiple options [79] combo box. Available options are: -Moves text and arrows outside extension lines. -Moves arrows, then text, outside extension lines. -Moves text, then arrows, outside extension lines. -Moves text or arrows for best fit.	Arrow and Text fit in dimensions. This property specifies how arrows and text are placed when they do not fit within the extension lines of dimensions.
Custom Arrow-head block	Block Definition [61]	Specify custom arrow-head block at ends of dimension and leader lines. If not specified it will use default arrow-head.
Separate arrow-heads	Boolean [79]	Use separate arrow-heads for dimension lines.
First Custom Arrow-head block	Block Definition [61]	Specify first custom arrow-head

		block of dimension lines. If not specified it will use default arrow-head. If "Separate arrow-heads" is not selected, this property has no effect on dimension entities.
Second Custom Arrow-head block	Block Definition 61	Specify second custom arrow-head block of dimension lines. If not specified it will use default arrow-head. If "Separate arrow-heads" is not selected, this property has no effect on dimension entities.
Leader line custom arrow block	Block Definition 61	Specify the arrowhead block at the end of leader lines. If not specified it will use default arrowhead.
Miscellaneous properties:		
Arc/Circle center mark	Real Number 78	The absolute value specifies the size of the center mark. -ve value: Center marks and center lines shown. 0 value: No center marks or center lines. +ve value: only Center marks are shown.
Move dimension line and the text	Boolean 79	If checked moves dimension line and the text, otherwise only the dimension line.
Dimension line and text gap	Scientific Data 78	The gap between dimension text and dimension lines. -ve value draws a box around text.
Linear scale factor	Scientific Data 78	Scale Factor for linear dimension.
Construct dimension limits	Boolean 79	Construct dimension limits
Overall scale factor	Real Number 78	Scale factor for whole dimension block.
Alternate dimension properties:		
Use alternate measurements	Boolean 79	Enables alternate measurements in dimensions.
Alternate dimension type	Multiple options 79 combo box. Available options are: -Scientific -Decimal -Engineering -Architectural (stacked) -Fractional (stacked) -Microsoft Windows Desktop	Sets alternate measurement type.
Zero suppression in alternate measurements	Multiple options 79 combo box. Available options are: -Suppress zero feet and exactly zero inches. -Include zero feet and exactly	Sets zero suppression in alternate measurements in dimensions.

	zero inches. -Include zero feet and suppress exactly zero inches. -Suppress zero feet and include exactly zero inches. -Suppress trailing decimal zeros. -Suppress leading decimal zeros.	
Prefix and/or suffix for alternate measurements text	Text 79	Specify prefix and/or suffix for alternate measurement text. The syntax is MyPrefix<>MySuffix. Here "MyPrefix" is the prefix user wants to prepend before dimension value and "MySuffix" is the suffix user wants to append after dimension value. "<>" gets replaced by actual dimension value when dimension entity is created. e.g. <>in will be replaced by 10in when dimension entity is created. <>" will be replaced by 10" when dimension entity is created. @<> cm will be replaced by @10cm when dimension entity is created.
Number of decimal places	Integer Number 78	Number of decimal places in alternate measurements in dimensions.
Distance multiplier	Real Number 78	Distance multiplier for alternate measurements in dimensions.
Rounding specifier	Real Number 78	Sets the rounding of alternate measurement. If value is 0, no rounding is performed, else the measurement text is rounded to nearest multiple of rounding value specified. e.g.If measurement value is 8.81 and Number of Decimal places is set to 2, Rounding value = 0.000 will make measurement value 8.81. Rounding value = 1.000 will make measurement value 9.00 Rounding value = 2.000 will make measurement value 8.00 Rounding value = 5.000 will make measurement value 10.00 Rounding value = 1.100 will make measurement value 8.80 Rounding value = 5.200 will make measurement value 10.40

Number of decimal places in tolerance	Integer Number [78]	Number of decimal places in tolerance values in alternate measurements in dimensions.
Zero suppression in tolerance	Multiple options [79] combo box. Available options are: -Suppress zero feet and exactly zero inches. -Include zero feet and exactly zero inches. -Include zero feet and suppress exactly zero inches. -Suppress zero feet and include exactly zero inches. -Suppress trailing decimal zeros. -Suppress leading decimal zeros.	Sets zero suppression in tolerance values in alternate measurements in dimensions.

To create *Dimension Style*:

    Library toolbar [217] **:**

    Manage Menu [96] **:** *Dimension Styles...*

2.1.2.3 User Coordinate System

User Coordinate System [17] (UCS) is a user defined coordinate space. User can define multiple coordinate systems and store them using Manage User Coordinate Systems [241]. UCS [17] is always defined with reference to WCS [17]. At any given time, there is only one UCS [17] active. The active UCS [17] can be modified using various UCS Tools [193].

In **hsCADCreator**, a *User Coordinate System* consists of the following data:

Property Name	Data Type	Description
Origin	3D Point [78]	(0, 0, 0) Origin Point for this User coordinate system
X-Axis Vector	3D Vector [78]	X-axis for this User Coordinate System
Y-Axis Vector	3D Vector [78]	Y-axis for this User Coordinate System

To create *User Coordinate System*:

    Library toolbar [217] **:**

    Manage Menu [96] **:** *User Coordinate System...*

See also:
    Draw Plane/UCS Tools [193]
    Manage User Coordinate Systems [241]

2.1.2.4 External References

External Reference Objects are references to external drawings. Using these objects, external drawing can be inserted into current drawing as a block.

In **hsCADCreator**, *External Reference Objects* consists of the following data:

Property Name	Data Type	Description
Name	Text [79]	Name of external reference
IsNested	Boolean [79]	Checked(true), if the external reference has more external references
IsResolved	Boolean [79]	Checked(true), if the external reference is loaded and resolved.

To create *External Reference Objects* :

Library toolbar [217] :

Manage Menu [96] : *External Reference...*

2.1.2.5 Hatch Style

Hatch Style objects represent the style used to draw a hatch entity [45]. *Hatch Style* object represents either a hatch gradient [60] or a hatch pattern [61].

In **hsCADCreator**, a *Hatch Style* consists of the following data:

Property Name	Data Type	Description
Preset	Multi-option [79] selection box	Name of pattern [61] or gradient [60] used for this hatch style.
Pattern or Gradient	Hatch Pattern [61] or Hatch Gradient [60]	Detailed properties of pattern or gradient used for this hatch style.

There are two major types of *Hatch Styles* within **hsCADCreator.**

Hatch Pattern Styles [227]

Hatch Gradient Styles [60]

2.1.2.5.1 Hatch Gradients

Hatch Gradient Objects define color gradient for filling in area inside boundaries. Gradient color is combination of two colors such that it starts off as first color on boundary and fades of to become second color on boundary. *Hatch Gradient Objects* are used for hatch [45] entities.

In **hsCADCreator**, a *Hatch Gradient Object* consists of the following data:

Field Name	Data Type	Description

Name	Text [79]	Name for this hatch gradient object
Description	Text [79]	Description for hatch gradient
Gradient Type	Multiple options [79] combo box..	Gradient direction type for hatch gradient object
Shift	Real Number [78]	Offset shift distance.
Luminance	Real Number [78]	Luminance value for hatch gradient.
Start Color	Color [51]	Starting color for gradient
End Color	Color [51]	Ending color for gradient
Angle	Real Number [78]	Angle for gradient

To create *Hatch Gradient Objet*:

    Library toolbar [217] :

    Manage Menu [96] : *Hatch Style...*

2.1.2.5.2 Hatch Patterns

Hatch Pattern Objects define patterns for filling in area inside boundaries. *Hatch Pattern Objects* are used for hatch [45] entities.

In **hsCADCreator**, a *Hatch Pattern Object* consists of the following data:

Field Name	Data Type	Description
Name	Text [79]	Name for this hatch gradient object
Description	Text [79]	Description for hatch gradient
Scale	Real Number [78]	Scale for hatch pattern
Angle	Real Number [78]	Angle for gradient

To create *Hatch Pattern Object*:

    Library toolbar [217] :

    Manage Menu [96] : *Hatch Pattern...*

2.1.2.6 Block Definition

Block Definition objects are collections of various drawn entities [21] so that they can be copied more than once. For example, if the user frequently created drawings with a particular design of door, he/she could make a *block* (or stamp) of door and add it to Block Library [218] using Block Creation Tool [140], and insert it whenever his/her drawing called for it. Each door would look exactly like the others and

would give the drawing a consistent appearance.

In **hsCADCreator**, a *Block Definition object* consists of the following data:

Property Name	Data Type	Description
Name	Text 79	Name of block definition
Xref Dependent	Boolean 79	Shows whether this block definition is dependent on external reference 60.
Xref Resolved	Boolean 79	Shows whether the external reference 60 is resolved or not.
Uniform Scaling	Boolean 79	If checked(true), allows only uniform scaling to be applied to block insertion entities 46 created from this block definition.
Explodable	Boolean 79	If checked(true), allows exploding of block insertion entities 46 created from this block definition using Explode Block Tool 152.
Has Preview Icon	Boolean 79	Shows whether this block definition has preview or not.
Is Anonymous	Boolean 79	Shows whether this block definition is anonymous.
From External Reference	Boolean 79	Shows whether this block definition was created from external reference 60.
From Overlay Reference	Boolean 79	Shows whether this block definition was created from overlay reference.
Is Layout	Boolean 79	Shows if this block definition is a layout 64.
Is Unloaded	Boolean 79	Shows if this block definition is unloaded.
Reference Count	Integer Number 78	Number of block insertion entities 46 existing in current drawing that are created from this block definition.
Origin	3D Point 78	Origin point for this block definition.
Path Name	Text 79	Path to external reference 60 file.
Block Insert Units	Multi-option 79 selection box	Scientific Units 78 associated with block insertion entities 46 created from this block definition.
Comments	Text 79	Any comments or description text for this block definition.
Layout	Text 79	Name of layout this block

		definition is associated with.

To add a new ***Block Definition*** to drawing's Block Library 218 :

    Entity toolbar 25 **:**

    Tool Menu 97 **:** ***Create*** 🖖 ***Block***

See also:
        Create Block Tool 140
        Block Insertion Entity 46
        Manage Blocks 218
        Block Insertion Tool 142

2.1.2.7 Image Definition

Image Definition object represents all objects that can be defined by standard bitmap types (.jpg, .bmp, .tiff, .gif, .png, .rle, .dib). ***Image Definitions*** are stored in Image Library 228 using Manage Image Definitions Tool 228 . Stored ***Image Definitions*** can be inserted in drawing using Image Insertion Tool 144 .

In **hsCADCreator**, an ***Image Definition object*** consists of the following data:

Property Name	Data Type	Description
Image Resolution	Real Number 78 | Real Number 78	Resolution of this image definition in millimeters per pixel. Size of single pixel Width | Height in millimeters
Is Loaded	Boolean 79	Shows whether the image file for this image definition is loaded in memory or not.
Size of Image	Real Number 78 | Real Number 78	Image size in pixels Width | Height.
Source File	Text 79	Path to image file from which this image definition was created.
Resolution Units	Multi-option 79 selection box	Resolution unit for this image definition.

To add an ***Image Definition*** to drawing's Image Library 228 :

    Library toolbar 257 **:**

    Tool Menu 97 **:** ***Manage*** 🖖 ***Image Definitions***

See also:
        Image Insertion Entity 47
        Manage Image Definitions 228

2.1.2.8 **Layers**

Layer Objects can be described as invisible drawing sheets that can be activated to draw, turn off/on to see entities on that *layer*, etc.

In **hsCADCreator**, a *Layer Object* consists of the following data:

Property Name	Data Type	Description
Name	Text 79	Name for this hatch gradient object
Description	Text 79	Description for hatch gradient
Color	Color 51	Default color for entities drawn on this layer.
Lineweight	Real Number 78	Default lineweight for entities drawn on this layer.
Linetype	Linetype 66	Default linetype for entities drawn on this layer.
Frozen	Boolean 79	If checked(true) freezes this layer else thaw this layer. Entities on frozen layers are not visible on screen. Entities on frozen layer are not passed to rendering engine.
Plottable	Boolean 79	If checked(true) makes this layer plottable.
Off	Boolean 79	If checked(true) turns this layer OFF. Entities on OFF layers are not visible on screen but are still passed to rendering engine.
Locked	Boolean 79	If checked(true) locks this layer. Entities on locked layers are visible on screen but can not be edited.

To create *Layer Object*:

⬚ Library toolbar 217 **:** ⬛

⬚ Manage Menu 96 **:** *Layers...*

2.1.2.9 **Layouts**

Layout Objects represents virtual drawing sheets. User can create snapshots of model using viewport 70 and create viewport tool 146 into *Layout Objects*.

In **hsCADCreator**, a *Layout Object* consists of the following data:

Property Name	Data Type	Description
Name	Text 79	Plot Settings Name

Paper Units	Multiple options [79] combo box.	Units the paper size will be based on.
Printer - Printer	Multiple options [79] combo box.	Selection of current printer/plotters available on system.
Printer - Paper Size	Multiple options [79] combo box.	Selection of common paper/potter sizes
Printer - Properties	Printer Properties Dialog (8.46 by 11 in Properties)	Launches printer configuration dialog.
Plot Rotation	Multiple options [79] combo box.Choices include -Inverted Landscape -Inverted Portrait -Landscape -Portrait	Modification of plot orientation.
Plot Type	Multiple options [79] combo box. Options include: -Display -Extents -Layout -Window	Selection of source type to be printed/ plotted
Paper Image Origin X	Real Number [78]	
Paper Image Origin Y	Real Number [78]	
Scale	Scale Ratio Selection (1 equals 1 1.000 : 1.000)	
Plot Arrangement		
Plot Centered	Boolean [79]	Center the printed area on the paper.
Origin X	Real Number [78]	Set the printed area's origin X component.
Origin Y	Real Number [78]	Set the printed area's origin Y component.
Plot Style Support		
Show Plot Styles	Boolean [79]	View with the selected plot style.
Plot Plot Styles	Boolean [79]	Print/Plot with the selected plot style.
Plot Style	File Selection [79]	Selection of a plot style file.
Advanced Plot Settings		
Scale Lineweights	Boolean [79]	Auto Scale the Lineweights.
Plot Lineweights	Boolean [79]	Print/Plot with the lineweights set within the drawing
Layout		
Layout Name	Text	User specified name

			for this layout.
Maximum Limits	2D Point 77		Maximum limits for this layout
Minimum Limits	2D Point 77		Minimum limits for this layout
Extent Maximum Point	3D Point 78		Maximum Extent point for this layout
Extent Minimum Point	3D Point 78		Minimum Extent point for this layout
Insertion Base	3D Point 78		Insertion point for this layout
Maximum Geometry Extents	3D Point 78		Maximum geometry extent point for this layout
Minimum Geometry Extents	3D Point 78		Minimum geometry extent point for this layout

To create **Layout Object**:

Library toolbar 217 **:**

Manage Menu 96 **: *Drawing Layout...***

2.1.2.10 Linetypes

Linetype Objects define different types of lines 23 to be used in drawing. Each **Linetype** defines a pattern that is repeated over the length of drawn line.

In **hsCADCreator**, a **Linetype Object** consists of the following data:

Property Name	Data Type	Description
Name	Text 79	Name for this linetype object
Comments	Text 79	Description for linetype
Scaled to fit	Boolean 79	Makes this linetype scale to fit the length
Dash count	Integer Number 78	Number of dashes in linetype pattern
Pattern length	Real Number 78	Total length of linetype pattern

To create **Linetype Object**:

Library toolbar 217 **:**

Manage Menu 96 **: *Linetype...***

2.1.2.11 Multi Line Style

Multi line style objects define different types of one or more parallel polylines to be used in a drawing. Each *Multi line style object* defines a pattern of parallel lines that is repeated over the length of drawn line. Multi line entities 24 use *multi line styles* to create uniform looking multi lines in a drawing.

In **hsCADCreator**, a *Multi line style object* consists of the following data:

Property Name	Data Type	Description
Name	Text 79	Name for this Multi Line Style object
Description	Text 79	Description for this Multi Line Style object
Fill Color	Boolean 79	If checked(true) fills color between cursor line and minimum offset line.
Fill color	Color 220	Color to be filled when Fill Color field is checked(true).
Start Angle	Scientific Data 78	Starting angle for all parallel lines.
End Angle	Scientific Data 78	Ending angle for all parallel lines.
Miters	Boolean 79	If checked(true), shows miter at each vertex.
Start Inner Arcs	Boolean 79	If checked(true), draws inner arcs at start point.
Start Round Caps	Boolean 79	If checked(true), draws round caps at start point.
Start Square Caps	Boolean 79	If checked(true), draws square caps at start point.
End Inner Arcs	Boolean 79	If checked(true), draws inner arcs at end point.
End Round Caps	Boolean 79	If checked(true), draws round caps at end point.
End Square Caps	Boolean 79	If checked(true), draws square caps at end point.
Linetypes: (More than one line types can be added by pressing "Add" button. All linetypes are drawn parallel to each other at specified offset distance.) Each Line Type has following properties:		
Line Type	Line Type 66	Type of the pen (dotted, dashed, dot-dash-dot, etc.) with which this entity gets drawn.
Offset	Scientific Data 78	Offset distance for this linetype.
Color	Color 220	Color of the pen with which this linetype gets drawn.

To create *Multi line style Object*:

Library toolbar 217 **:**

⊟ Manage Menu 96 : *Multi Line Styles...*

2.1.2.12 Plot Settings

Plot Settings Objects represents a group of settings for printing drawings.

In **hsCADCreator**, a *Plot Settings Objects* consists of the following data:

Property Name	Data Type	Description
Name	Text 79	Plot Settings Name
Paper Units	Multiple options 79 combo box.	Units the paper size will be based on.
Printer - Printer	Multiple options 79 combo box.	Selection of current printer/plotters available on system.
Printer - Paper Size	Multiple options 79 combo box.	Selection of common paper/potter sizes
Printer - Properties	Printer Properties Dialog (8.46 by 11 in [Properties])	Launches printer configuration dialog.
Plot Rotation	Multiple options 79 combo box.Choices include -Inverted Landscape -Inverted Portrait -Landscape -Portrait	Modification of plot orientation.
Plot Type	Multiple options 79 combo box. Options include: -Display -Extents -Layout -Window	Selection of source type to be printed/ plotted
Paper Image Origin X	Real Number 78	
Paper Image Origin Y	Real Number 78	
Scale	Scale Ratio Selection (1 equals 1 ▼ 1.000 : 1.000)	
Plot Arrangement		
Plot Centered	Boolean 79	Center the printed area on the paper.
Origin X	Real Number 78	Set the printed area's origin X component.
Origin Y	Real Number 78	Set the printed area's origin Y component.
Plot Style Support		
Show Plot Styles	Boolean 79	View with the selected plot style.
Plot Plot Styles	Boolean 79	Print/Plot with the selected plot style.

Plot Style	File Selection [79]	Selection of a plot style file.
Advanced Plot Settings		
Scale Lineweights	Boolean [79]	Auto Scale the Lineweights.
Plot Lineweights	Boolean [79]	Print/Plot with the lineweights set within the drawing

To create *Plot Settings Object*:

Library toolbar [217] :

Manage Menu [96] : *Drawing Layout...*

2.1.2.13 Text Styles

Text Styles Objects define detailed style for text [43] used in the drawings. *Text Style* can be applied to text entities [43] to have similar look and feel across all text entities [43] in a drawing. *Text Style* defines properties like text font, fixed size, type face, oblique angle, etc. for the text.

In **hsCADCreator**, a *Text Style Object* consists of the following data:

Property Name	Data Type	Description
Name	Text [79]	Name for this text style object
BigFont file name	File Selection [79]	Path name of BigFont file name from where font should be imported
Font file name	File Selection [79]	Path name of Font file from where font should be imported
Is Backwards	Boolean [79]	Specifies if this font is backwards
Is Shape File	Boolean [79]	Specifies if the font file is a shape file
Is Upside Down	Boolean [79]	Specifies if the font is defined as Upside down
Is Vertical	Boolean [79]	Specifies if the font is vertical
Obliquing Angle	Scientific Data [78]	Obliquing angle for font
Fix text size	Real Number [78]	Specifies Fix Text size. -If set to zero, allows to change text size using "text size" variable of Text Entity [43]. -If set to value other than zero, text size can not be changed using "text size" variable of Text Entity [43]. The text size can only be changed with Scale Entities Non-uniformly Tool [155].
Last Height	Scientific Data [78]	Specifies height of text inserted

		in last Text Entity 43
X Scale	Real Number 78	Scale along horizontal axis of text.

To create **Text Style Object**:

⬚ Library toolbar 217 **:** T

⬚ Manage Menu 96 **:** **Text Styles...**

2.1.2.14 Modelspace Viewport

Modelspace Viewport Object is the main drawing window through which most of entities are created and edited. It shows a view of 3D model space. To facilitate viewing of drawing through multiple viewports in Modelspace, **hsCADCreator** utilizes pre-set multiple viewports through Pre-configured Viewport Tool 186 and Divide Viewport Tool 184. Depending on whether the viewport exists on Modelspace or Paperspace they are defined as **Modelspace viewport object** and Paperspace viewport entity 49.. There can only one **Modelspace Viewport** exist in a drawing. **Modelspace Viewport** are always created by **hsCADCreator** on a layout 64 named "Model" when a new drawing is created.

In **hsCADCreator**, a **Modelspace Viewport Entity** consists of the following data:

Property Name	Data Type	Description
Name	Text 79	Name for this Viewport object
Camera Settings for this Viewport:		
Back Clip Distance	Real Number 78	Distance from the view plane to the back clipping plane
Back Clip Enabled	Boolean 79	Whether or not to draw entities beyond the back clipping plane.
Center Point	2D Point 77	Center point for this viewport object.
Elevation	Scientific Data 78	Distance from WCS origin to the UCS of this viewport
Clip at Eye	Boolean 79	Clip the scene from camera at eye location.
Front Clip Distance	Real Number 78	Distance from the view plane to the front clipping plane
Front Clip Enabled	Boolean 79	Whether or not to draw entities before the front clipping plane.
Height	Real Number 78	Height of scene in device coordinates
Lens Length	Real Number 78	The length of lens used in perspective mode
Is Perspective	Boolean 79	Make the view from camera as perspective view
Render Mode	Multiple options 79 combo box.	Specifies how to render entities.

Target	3D Point [78]	Target point for camera
UCS	User coordinate System [59]	User coordinate system associated with this viewport
Width	Real Number [78]	Width of this view in device coordinates
View Direction	Vector [78]	Vector representing direction the view faces relative to the WCS
Twist	Scientific Data [78]	Twisting angle in device coordinates
Modelspace viewport:		
UCS icon visible	Boolean [79]	Show UCS icon or hide it
UCS icon at origin	Boolean [79]	Always show UCS icon exactly at UCS origin or show it on left hand bottom corner when UCS origin is not on screen.
Follow UCS	Boolean [79]	Always display plan view [16] whenever the UCS for this viewport changes
Independent UCS	Boolean [79]	This viewport has its individual UCS and it becomes active whenever this viewport is activated
Grid visible	Boolean [79]	Show/hide grid [75] points in this viewport.
Grid snap	Boolean [79]	Enable/Disable grid snap points [74] in this viewport.
Grid Spacing along X-Axis	Scientific Data [78]	Spacing between grid [75] points along X-axis.
Grid Spacing along Y-Axis	Scientific Data [78]	Spacing between grid [75] points along X-axis.

See also:
Divide Viewport Tool [184]
Pre-configured Viewport Tool [186]

2.1.3   Snap Point Settings

Snap Settings define settings for Entity Snap Points [72] and Grid Snap Points [74]. These settings allow a user to specify a precise point on any entity without any difficulty. There are two major types of *Snap Points* in **hsCADCreator** Click on the links below for more information on the various types of *snap points*.

1. Entity Snap Points [72]

2. Grid Snap Points [74]

2.1.3.1 **Entity Snap Points**

Entity Snap Points are key points on any entity that allows a user to select a precise point without difficulty.

There are two settings for **Entity Snap Points** :

Permanent Entity Snaps:

This setting enables the *Entity Snap Points* till they are explicitly turned OFF using "Enabled" property under Snap Properties Tree 26. Or by the menu item "Enabled" found under the View Menu >> Entity Snaps Submenu 94.

Temporary Entity Snaps

This setting enables the *Entity Snap Points* only for one mouse click 15. This type of snap is found by right-clicking 15 and bringing up the Temporary Snap Context menu 90

There are 12 different types of *Entity Snap Points* in **hsCADCreator** :

Name of Snap Point	Description	Example
End Point	Snap to Entity end points (Line, Polylines)	
Mid Point	Snap to Entity mid points (Line, Polylines)	
Center Point	Snap to Entity center points (Arc, Circle, and Ellipse Entities)	
Insertion Point	Snap to insertion points (Block Insert Entities, Image Insert Entities, Text)	
Intersection Point	Snap to Intersection (between any two linear entities)	

Nearest Point	Snap to nearest point on adjacent Entity	
Node Point	Snap to Entity nodes (Point Entity or Dimension Entity reference points)	
Parallel Point	Snap to end point on a parallel line (When creating a Line Entity)	
Perpendicular	Snap to an end point on a perpendicular line to adjacent Entity (Line, Polyline Entity)	
Polar Point	Snap to Polar coordinates specified by Polar Angle	
Quadrant Point	Snap to quadrant (Circle, Arc, and Ellipse Entities)	
Tangent Point	Snap to an end point on a line tangent to adjacent curve Entity	

Polar Angle	Incremental angle for Polar Point snaps.	

See also:
Snap Properties Tree 261
View Menu 94
Snaps Toolbar 258

2.1.3.2 Grid Snap Points

Grid Snap Points allows easy access to snap the current cursor point to nearest grid 75 point available. This allows a user to easily jump from one point another point on regular intervals along UCS 17 X and Y axis. A user can also keep the Grid 75 invisible and still have access to its *Grid Snap Points*. The increments of *Grid Snap Points* can be changed by changing "Grid Spacing along X-Axis" and "Grid Spacing along Y-Axis" variables in Modelspace Viewport 70.

See Also:
Grid 75
Snaps Toolbar 258

2.1.4 Entity Grip Points

Entity Grip Points are key points on any entity that allows a user to modify location, scale, and rotation of the entity very easily by dragging 16 and releasing 16 it. Whenever a grip point is ready to edit, it is highlighted with a larger icon.

There are two types of grip points:

1. Move Grip Points: These *grip points* are represented by four arrows diagonally placed. By dragging these *grip points* user can move entity around in drawing.

2. Stretch Grip Points: These *grip points* are represented by square. By dragging these *grip points* user can rotate and scale entity in drawing.

2.1.5   Grid

Grid

The grid is a drawing aid that is a display of equidistant points oriented along the X and Y axis of the Drawing Plane 15 . The grid is located within the **drawing window** and is visible by default.

Basic Workspace Layout

Grid Properties

The grid increments can be modified by changing **"Grid Spacing along X-Axis"** and **"Grid Spacing along Y-Axis"** settings in Modelspace Viewport Settings 70 .

The Grid Snap 74 can be enabled/disabled from the following locations:
Snaps Toolbar 258
Snap Property Tree 261
View Menu 94 .

The size of the grid sometimes is either too small or too large to display on the drawing screen. When the user attempts to display the grid and a **small grid window** appears in the **lower right hand corner** of the drawing screen with a red ⌀ symbol, the grid cannot currently be displayed.

Grid is Too Sparse

If the grid is **too sparse** the user must **zoom out** to display the grid; this grid window will show a widely spaced grid like the following:

To zoom out, **roll the top mouse wheel toward you**, or find the **Zoom Out Tool** 177 and select a point on the drawing screen and Click.

Grid is Too Small

Likewise, if the grid is **too small** the user must **zoom in** to display the grid; this grid window will show a tightly spaced grid like the following:

To zoom in, **roll the top mouse wheel away from you**, or find the **Zoom In Tool** [176] and select a point on the drawing screen. Click.

2.1.6 Drawing Settings

Drawing Settings define settings for overall drawing. These settings allow user access to common properties for drawing.

In **hsCADCreator**, *Drawing Settings* consists of the following data:

Field Name	Data Type	Description
Active Layout	Multiple option [79] combo box.	Allows user to select from available Layout list and make it active.
Display Lineweight	Boolean [79]	When checked(true) draws all lines with actual lineweights specified as lineweight properties, otherwise it shows all lines with default lineweight.
Background Color	Color [51]	Background color for Layouts [64] .
Save preview	Boolean [79]	If checked(true) saves the preview along with drawing file. Saving preview allows user to see snap shot of file before actually opening it.
Units	Multiple option [79] combo box.	Allows user to select from available units and make it default unit for this drawing. All measurements are derived from drawing's unit.
Active Unit Family	Multiple option [79] combo box.	Allows user to select from available unit families and make it active unit family for this drawing. When using unit conversion feature for all Scientific data [78], it uses active unit family to show next conversion.
Filename	File Selection [79]	Filename to save the current drawing file.
Title	Text [79]	Title name for this drawing
Subject	Text [79]	Subject for this drawing

Comments	Text [79]	Any comments or description for this drawing
Author	Text [79]	Name of author for this drawing
Revision Number	Text [79]	Revision number for this drawing
Saved Via	Text [79]	Name of application that last saved this drawing

In **hsCADCreator** the above properties are shown in the Drawing Property Tree [262].

Click on image to see detail view.

2.1.7   Data Types and Properties

In **hsCADCreator** all entities [21] and objects [50] have different values(data) associated with it. These values are also know as properties data or simply *Properties.* Properties data are of various types (*Data Types*) as described below. **hsCADCreator** uses property trees [258] (located to the right hand side of the drawing screen) to edit individual *Properties* of entities [21], objects [50], and tools [80]. Each property has its own visual depiction and ways of editing data depending on its *Data Type.* **hsCADCreator** has following data types and properties associated with it:

Two-dimensional Point (2D Point):

Location of a point in an X-Y coordinate plane(2D space [17]).

Press Enter to update the information.
Press Escape to return the information to its previous value.
Press Tab to move to the next editable field.

Three-dimensional Point (3D Point):

Location of a point in an X-Y-Z coordinate space(3D space 17).

Starting Point	0.000	0.000	0.000

Press Enter to update the information.
Press Escape to return the information to its previous value.
Press Tab to move to the next editable field.

World Coordinate System Point (WCS Point):

Location of a point in the World (X-Y-Z) Coordinate Space.

Starting Point	0.000	0.000	0.000	W

Press Enter to update the information.
Press Escape to return the information to its previous value.
Press Tab to move to the next editable field.
Left click 15 **W** icon that appears to the right to convert the point property from the WCS to the active UCS.

User Coordinate System Point (UCS Point):

Location of a point in the User (X-Y-Z) Coordinate Space.

Starting Point	0.000	0.000	0.000	U

Press Enter to update the information.
Press Escape to return the information to its previous value.
Press Tab to move to the next editable field.
Left click 15 **U** icon that appears to the right to convert the point property from the active UCS to WCS.

Scientific Data:

A real number that has associated units (either metric or imperial) and can have minimum or maximum values. Scientific Data can also represent angles in either degrees or radians.

Length	0.000	mm
Included Angle	0.000	°

Left click 15 unit text (**mm**) to convert the scientific data property from the current units to next unit in the conversion set.

Integer Number:

An Integer is any number without a decimal portion (0, 1, 13, 155, 12345, -5, -12345 etc.). Integer values can be less than, equal to, or greater than zero. Integer Properties may have minimum and or maximum values.

Dash Count	0

Real Number:

A real number is any number that generally has a decimal portion, even if that decimal portion is zero(s). eg (2.45, 43.00, -6.5, 3.14159, 0.000 and even 35). This Real Number property can be set to have minimum and or maximum values.

Paper Image Origin X	0.000

Vector:

An object defining direction and length of a ray.

| View Direction | 0.00 | 0.00 | 1.00 |

Press Enter to update the information.
Press Escape to return the information to its previous value.
Press Tab to move to the next editable field.

Boolean (On/Off):

Check box specifying True/False or On/Off or Yes/No Value.

| UCS icon visible | ☑ |

Left click [15] on the check box to change the value.

Multiple option:

Left click here...

| Line Color | ☐ By Layer ▾ |

These properties are also known as drop-down list. They are accessed by left clicking [15] on the down arrow and then selecting required option from list using left click [15].

File Selection:

A File Selection property allows the user to select a file from the common file dialog by pressing the (...) button. The current or new selection will show up in the first portion of the property.

| C:\Documents and Settings\ce... | ... | * |

Text Field:

A Text Field property is for adding common text to a property. Simply give the property focus by clicking on it and type in the text desired.

| Name | **My Text Name** |

See also:

2.1.8   Property Trees

Property Tree is collection of properties [77] arranged in a tree fashion. Each property [77] in the *property tree* has associated brief description. A property in a *property tree* is selected by left clicking [15] on property name. Whenever a property is selected, its brief description is shown at the bottom of the property tree and the selected property name is highlighted. A *property tree* may have one or more sub-trees within it. Each of these trees can be expanded/collapsed by double-clicking [16] on the sub-tree name or the (+) or (-) icon in front of the sub-tree name.

For more information on **Property Trees** used in **hsCADCreator**, please see chapter on
hsCADCreator Property Trees 258.

See also:
hsCADCreator Property Trees 258

2.1.9 Tools

This application is built on the concept of tools. Loosely speaking, a tool for **hsCADCreator** is an implementation with the ability to perform a certain class of operation or procedure with regards to viewing and editing a drawing.

Tool Properties
Each tool may have a set of properties associated with it.

*Tool Modes :*Each tool may have one or more modes to work with. These modes are accessible from the *Tool Properties* window.
For example, Clone Tool 159 has more than one modes and they can be changed by selecting mode from drop-down list.

Active Tool
Only one tool may be active at a time. By choosing a tool with a toolbar, menu, or shortcut key, you are making it the active tool. In the case of Stackable Tools, only the "top most" tool is active. The Tool Properties 260
shows properties and options available for the active tool.

Stackable Tools and Tool Stack
In **hsCADCreator** some tools work in conjunction with each other and they are known as *stackable tools*. When more than one tools are active it creates a *tool stack.* In a tool stack only the topmost tool is active at a time. When the topmost tool is deactivated it is removed from tool stack and the "new" topmost tool becomes active and ready for use.
For example,
1. If you activate the Line Tool to create a line, and then hold down the mouse wheel while moving the mouse, you will make the screen (and line) pan or move. Releasing the mouse wheel will cease the

movement and other lines can now be created without having to select the Line Tool again.

2. If there is no entity selected when activating Clone Tool [159], both the Selection Tool [101] and Clone Tool will highlight until the selection of the entity is made. After the selection is done, the Selection Tool will be deactivated and only the Clone Tool will continue to be highlighted until the clone tool is finished or another tool becomes active.

2.1.10 Toolbars

Toolbar

Toolbar is a collection of **Tools** [80]. Tool buttons are grouped by their functionality into tool "bars". Similar tools are combined together in a toolbar for their easy placement and access. All of the toolbars have the ability to dock(be placed on the various edges of the drawing window) or place them anywhere on drawing screen. A toolbar can be moved by one of following methods depending on position of the toolbar:
1. Left mouse dragging [16] from its anchor line.

2. Left mouse dragging [16] from its title bar.

When a toolbar is moved near the edge of the **hsCADCreator** screen area, its title bar is removed and the toolbar tightly fits on the edge to save the screen space. Once you have managed to position the toolbars and tool windows to fit your needs you can manage their positions by using the following commands. Visit Workspace Layout [279] for more information on managing the layouts.
Save [279] **Layout** (Saves the current placement and sizes of all user interface elements within the workspace)
Revert [279] **Layout** (Returns the current placement and sizes of all user interface elements to the last Save [279] **Layout** state)
Save As... [279] **Layout** (Saves a layout file to a user defined location)
Load From... [279] **Layout** (Loads a layout file from a user defined location)

Each toolbar in **hsCADCreator** can be customized weather that be size, location, tools located on the bar, or visibility. To learn more about the modifications you can make to the toolbars visit the Customizing Toolbars [275] chapter.

Flyout Toolbar

Many toolbars have one or more than one toolbars hidden within them. These are known as *Flyout Toolbars*. *Flyout Toolbars* are provided to save the screen space and allow easy access to different Tool Modes [80]. *Flyout Toolbars* are sub-toolbars hidden withinthe normal toolbars and are designated by a small black arrow in the lower right hand corner of a toolbar icon (   ). By simply pointing at the icon with the mouse cursor and right clicking [15], this hidden submenu will appear. The user may now choose a tool from the sub toolbar by left

clicking 15. When a user selects a tool from the *Flyout toolbar* the selected tool will take the place of the tool in the parent menu.

1. Activate the Flyout Toolbar:

Left Click 15 on the ( ) down arrow located to the right of the icon. This will activate the flyout toolbar.

2. Choosing New Tool to Activate:

From the flyout toolbar select the tool you wish to use by left clicking 15 on it.

3. Chosen Tool Replaces Old Tool in Parent Toolbar:

After selection of the new tool notice that the chosen tool icon replaces the old icon in the parent toolbar.

Following *Flyout Toolbars* are present in **hsCADCreator**:

Entity Toolbar 251:
    Arc Tools 113 (4 additional tool choices available)
    Dimension Tools 123 (6 additional tool choices available)

View Toolbar: 253
    Rotate View Tools 179 (10 additional tool choices available)

Draw Plane/UCS Toolbar: 254
    Preset UCS Snap Tools 195 (6 additional tool choices available)
    Rotate Draw Plane Tools 198 (4 additional tool choices available)
    Translate Draw Plane Tools 205 (4 additional tool choices available)
    Clone Tools 159 (4 additional tool choices available)

See also:

2.1.11 Docking Window

Floating (UnDocked)

Docking windows have two major modes, **Docked** and **UnDocked**. The **UnDocked** mode is commonly referred to as **Floating**. A **Floating** window can be positioned anywhere on the screen where it will float above the various user interface elements. The image below shows the Notification Window 267 in **Floating mode**.

Docked

Another way to position windows is to **Dock** them to various other user elements. For instance the Notification Window 267 below has been **Docked** to the bottom of the Drawing space. When **docked** the window maintains a close relationship with the edges such that space is well managed.

Dockable Windows Include:
Notification Window 267
Layer Window 281
Library Window 282
Help Window 285
Selection Properties Window 259
Tool Properties Window 260
Drawing Properties Window
Viewport Properties Window
Snaps Window 71

Docking toolbars also have two visibility modes while *Docked*. They are **Pinned** and **UnPinned**.

Pinned Docking Window

In the upper right hand corner of a Docking Window is a small Pin Icon. When a **Docking window** is docked this icon becomes visible. When the pin icon looks like this (⬚) the **Docking Window** is in the **Pinned** mode. This means that the window will be constantly visible and in the position that is seen. The image below shows the Notification Window 267 **Docked** and **Pinned** to the bottom of the drawing space.

UnPinned Docking Window

When the Pin icon (📌) is selected the window becomes **UnPinned**. This **UnPinning** modifies the behavior of the window. The **Docking window** will now act like a drawer. When not active the window will slide away such that only the title tabs are visible on the screen. For more advanced users this is an excellent way to maximize the drawing space. When the window is either hovered over by the mouse or clicked on the window will become active. When active the drawer will slide out showing the contents of the window. Note that the Pin icon now is in the **UnPinned** state (📌).

The image below is the Notification Window 267 in its **Docked UnPinned** and **Inactive** State.

The image below is the same Notification Window 267 in its **Docked UnPinned** and **Active** State. Note that in its active state it appears very similar to the **Pinned** state.

2.1.11.1 Docking Procedure

Docking Window Management

1. To Dock a window the window must be in Pinned ⟨83⟩ **or** Floating ⟨83⟩ **mode.**

2. Select and hold the left mouse button down on the Title Bar of the window you want to dock.

3. Utilize the Docking aids to place the window in the desired location.

Drag the Docking toolbar over the desired docking aid ( ) and release to position. The images below depict the docking aids and the approximate resulting placement.

Main Window Docking Aids

Along with the above **Main Window Docking** aids there are **Current Window Docking** aids that will allow placement with relation to the window that is under the cursor. The image below depicts the docking aids approximate resulting placement if we were hovering over the Selection window.

The Docking aid ( ) is used for placing the docking window with relation to the window under the cursor.

The center of the docking aid above is also a Docking aid ( ). This will place the docking window as a tab within the window that is under the cursor. If that window is not a collection of tabs then this option will not be available.

See Also:

2.1.12 Dialogs

Dialog is a collection of various user interface elements (tools 80, property trees 79, and properties 77). *Dialogs* provides easy access and management of more complex objects and entities. *Dialogs* also provides easy way to do complex tasks. Most of dialogs when activated restricts access to the **hsCADCreator** main screen and other user interfaces outside the *dialog*. For more information on *dialogs* used in **hsCADCreator**, please see chapter on hsCADCreator Dialogs 264.

See also:
hsCADCreator Dialogs [264]

2.1.13 Undo/Redo

To Undo a recently made change in drawing:
Using the keyboard

: Press ***Ctrl+Z*** from the keyboard. Repeat as necessary to undo additional changes in reverse order.

Using the Mouse:

: Go to the Edit menu [94] and select ⬑ ***Undo***

or

: Go to the File Toolbar [256] and select the 🔄 icon button

To Redo a recently undone change in drawing:
Using the keyboard:

: Press ***Ctrl +Y*** from the keyboard. Repeat as necessary to redo additional undone changes in reverse order.

Using the mouse:

: Go to the Edit menu [94] and select ⬑ ***Redo***

or

🖱 : Go to the File Toolbar 256 and select the 🧲 icon button

2.2   User Interface

The *User Interface* of **hsCADCreator** is designed to be a simple, yet powerful tool for dealing with daily tasks in computer aided drafting and design. The following types of *user interfaces* are available in **hsCADCreator**.

- Menus 90
- Tools 100
- Toolbars 250
- Property Trees 258
- Dialogs 264
- Notify Window 267
- Status Bar 269
- Visual Aids 270
- Keyboard Shortcuts 271
- Customizing Toolbars 275
- Workspace Layout 279
- Layer Window 281
- Library Window 282
- Help Window 285

2.2.1   hsCADCreator Menus

There are eight main *Menus* available in **hsCADCreator**. They are located above the drawing screen in the upper left hand corner of the monitor screen.

Main Menu:

File   Edit   View   Manage   Tool   Options   Window   Help

1. **File Menu** 93
2. **Edit Menu** 94
3. **View Menu** 94

4. **Manage Menu** |96⌐

5. **Tools Menu** |97⌐

6. **Options Menu** |99⌐

7. **Window Menu** |99⌐

8. **Help Menu** |100⌐

Context Menus:

In addition, there are also context menus available. Depending upon the tool |100⌐ in which the user is working, these are accessed by ⌐right clicking |15⌐ the mouse or pressing the ⌨**Shift** key on the keyboard while ⌐right clicking |15⌐ the mouse.

1. *View Options* menu (available to most tools) Access by ⌐right clicking |15⌐ the mouse with the cursor in the drawing screen. Options available:

View Options
Pan
Rotate View
Zoom ▶

- Pan |178⌐
- Rotate View |183⌐
- Zoom
    - Zoom In |176⌐
    - Zoom Out |177⌐
    - Zoom Extents |178⌐
    - Zoom Window |175⌐

2. *Planar Polyline Tool* menu (available to the Planar Polyline Tool |110⌐ only) Access by ⌐right clicking |15⌐ the mouse with the cursor in the drawing screen. Options available:

Planar Polyline Tool
Line Mode
Bulge Mode   Tab
Toggle Closed   Ctrl+Enter

End Polyline   Shift+Enter

- Line Mode |110⌐
- Bulge Mode |110⌐
- Toggle Closed |110⌐
- End Polyline |110⌐

3. *3D Polyline Tool* menu (available to the 3D Polyline Tool only) Access by ⌐right clicking |15⌐ the mouse with the cursor in the drawing screen. Options available:

3D Polyline Tool
End Polyline   Shift+Enter
Toggle Closed   Ctrl+Enter

- End Polyline [112]
- Toggle Closed [112]

4. *Temporary Snaps* menu (available most tools) Access by holding ⌨**Shift** + 🖱right clicking [15] with the cursor in the drawing screen. The Temporary Snaps act much like the Permanent Entity Snaps [72] except that they last for only one mouse click. You can temporarily disable/enable the currently set Entity Snaps [72] and or Grid Snaps [74]. You may also select one or more Entity Snaps [72] you desire to use temporarily. This is useful for entity snapping that you do not commonly use but want to quickly switch too. Options available:

Temporary Snaps
Entity Snaps
Grid Snaps
End Point
Mid Point
Intersection
Center Point
Quadrant
Insertion
Nearest
Node
Perpendicular
Polar
Tangent
Parallel
Set Permanent Snaps...

- Entity Snaps [72]
- Grid Snaps [74]
- End Point [72]
- Mid Point [72]
- Intersection [72]
- Center Point [72]
- Quadrant [72]
- Insertion [72]
- Nearest [72]
- Node [72]
- Perpendicular [72]
- Polar [72]
- Tangent [72]
- Parallel [72]
- **Set Permanent Snaps** (This will translate your **Temporary Snap** choices into your permanent Entity Snaps [72].)

2.2.1.1   File Menu

The *File Menu* in **hsCADCreator** has the following options available:

1.   **New** (Start a new drawing)

2.   **Open...** (access a named drawing file previously saved)

3.   **Save** (store a drawing for future access)

4.   **Save As...**(store a drawing under a different file name)

5.   **Import...** (.dwf file format)

6.   **Export...** (save a copy of the drawing under a different file format)

7.   **Close** (terminate the current drawing file)

8.   **Print / Plot** 14 (print a drawing file)

9.   **Print / Plot Preview** 14 (see beforehand how a drawing file will be printed)

10.   **Recent Files** (displays files you have recently opened for quick access)

11.   **Exit** (terminate all documents as well as the program)

(Note: Each option above is equipped with a keyboard shortcut to improve drawing efficiency; to customize your shortcuts, see Key Accelerators 271 under Options Menu 99)

2.2.1.2 Edit Menu

The *Edit Menu* in **hsCADCreator** has the following options available:

1. **Undo** 89 (take back the most recent change made within the drawing)

2. **Redo** 89 (restore a change that has been removed using the **Undo** command)

3. **Select All** (selection of every entity)

4. **Copy** (duplicate one or more entities)

5. **Paste** (transfer selected entity to new location; used in conjunction with **Copy** above or **Cut** below)

6. **Paste as block** (transfer selected entities as a block 61 to a new location)

7. **Cut** (remove selected entity; used in conjunction with **Paste** or **Paste as block** above)

(Note: Each option above is equipped with a keyboard shortcut to improve drawing efficiency; to customize your shortcuts, see Key Accelerators 271 under Options Menu 99)

2.2.1.3 View Menu

The *View Menu* in **hsCADCreator** has the following options available:

1. **Grid** 75 (Toggles the framework of evenly spaced/parallel vertical and horizontal lines used as a drawing reference on/off)

2. **Grid Snap** 74 (Toggles the grid snap 74 ability on/off)

3. **Entity Snaps** 72 (Sub-menu)
   a. Enabled 72 (Enable/Disable Permanent Entity Snaps 72)
   b. End Point 72 (Snap to an end point of a line segment)
   c. Mid Point 72 (Snap to mid point of a line segment)
   d. Intersection 72 (Snap to the intersection point)
   e. Center Point 72 (Snap to center of a circle 30 , ellipse 32 or arc 28)
   f. Insertion 72 (Snap to the Insertion point of blocks 61 , images 47 , or text 43)
   g. Nearest 72 (Snap to the nearest point on the entity to the cursor)
   h. Node 72 (Snaps to the point entity 22 only)
   i. Parallel 72 (Snap parallel to nearest line to cursor)
   j. Perpendicular 72 (Snap to perpendicular point on line segment)
   k. Polar 72 (Snap to 0, 45deg, 90deg... etc)
   l. Quadrant 72 (Snap to the nearest quadrant on a circle 30 , ellipse 32 or arc 28)
   m. Tangent 72 (Snap to the point tangent to the circle 30 , ellipse 32 or arc 28)

4. **Toolbar** 250 (Sub-menu)
   a. File Toolbar 256 (Show/Hide this toolbar)
   b. Create Toolbar 103 (Show/Hide this toolbar)
   c. Modify Toolbar 253 (Show/Hide this toolbar)
   d. View Toolbar 253 (Show/Hide this toolbar)
   e. Draw Plane/UCS Toolbar 193 (Show/Hide this toolbar)
   f. Render Toolbar 256 (Show/Hide this toolbar)
   g. Library Toolbar 257 (Show/Hide this toolbar)
   e. Measure Toolbar 257 (Show/Hide this toolbar)

5. **Property Bar** 258 (Prop tree Tool Properties to the right of the drawing screen when a tool is activated)

6. **Status Bar** 269 (program information bar found at the bottom of drawing screen)

7. **Regen** 217 (redraw all entities with updated information)

(Note: Each option above is equipped with a keyboard shortcut to improve drawing efficiency; to customize your shortcuts, see Key Accelerators 271 under Options Menu 99)

Also See:
        Toolbars 250

2.2.1.4 Manage Menu

The *Manage Menu* in **hsCADCreator** has the following options available:

1. **Entire Library...** |217|(access to all aspects manageable in a single screen--the following below are separate screens for each aspect)

2. **Blocks...** |218|(entity/grouping of entities that can be placed in a drawing)

3. **Colors...** |220|(for background, lines, entities, gradients, etc.)

4. **Dimension Styles** |221| (add, remove, modify dimension styles)

5. **Drawing Layouts...** |232|(model, paperspace, etc. and **Layout**)

6. **External References...** |223|(attach, detach, load/reload, unload, and bind embedded drawings within a drawing)

7. **Hatch Styles...** |224|(add, remove, and import patterns/gradients)

8. **Image Definitions...** |228|(picture(s) that can be placed in a drawing)

9. **Layers...** |230|(set color, line weight, line style, visibility, printed or not, etc.)

10. **Line Types...** |233|(add, remove and import linestyles)

11. **Modelspace Viewports...** |242|(change properties)

12. **Multi Line Styles...** 235 (add, remove and edit multi-line styles)

13. **Named Views...** 236 (add, remove, set current, set new from active view)

14. **Text Styles...** 239 (way text is drawn...similar to fonts)

15. **User Coordinate Systems...** 241 (add, remove, set current, create new from active UCS)

16. **Quick Layers...** (set color, line weight, line style, visibility, printed or not, etc.)

(Note: Each option above is equipped with a keyboard shortcut to improve drawing efficiency; to customize your shortcuts, see Key Accelerators 271 under Options Menu 99)

Also See:
> Library Toolbar 257

2.2.1.5   Tool Menu

The *Manage Menu* in **hsCADCreator** is a second way to access all the tools that are found on the View 253 , Measure 257 , Modify 253 , Create 103 , UCS 193 , and Render 256 Toolbars 250 .

1. **View** 253
   - a.   Rotate 179
   - b.   Pan 178
   - c.   Zoom 175
   - d.   Snap 187
   - e.   Divide Viewport 184
   - f.   Preset Viewport 186

2. **Modify** 253
   - a.   Translate 148
   - b.   Rotate 150
   - c.   Explode 152 (Break down selected entity(ies) into their base entities)
   - d.   Delete 153 (Remove selected entity(ies) from drawing)
   - e.   Scale Uniform 154 (Scale entity(ies) by each axis consistently)
   - f.   Scale Non Uniform 155 (Scale entity(ies) by each axis independently)
   - g.   Trim 157 (Trim a line entity to a designated cutting edge)
   - h.   Extend 158 (Extend a line entity to a designated line)
   - I.   Clone 159 (Clone selected entity/ies)

3. **Create** 103
   - a.   Point 105

b. Line 106
c. Multi Line 108
d. Polyline 110
3. Arc 113
f. Circle 120
g. Ellipse 121
h. Dimension Indicator 123
i. Text 136
j. Block 140
k. Block Insertion 142
l. Image Insertion 144
m. Hatch 138
n. Viewport 146 (Only active while in a paperspace)
o. Poly Face Mesh 139

4. **UCS** 254
a. Rotate 198
b. Translate 205
c. Snap 195
d. Align UCS to View 210
e. Align View to UCS 211
f. UCS Origin to WCS origin 211
g. Align UCS Axis to WCS Axis 212
h. Select USC Via Entity(ies) 212

5. **Render** 256
a. 2D Wireframe 213
b. 3D Wireframe 214
c. Hidden 214
e. Flat Shaded 214
f. Gouraud Shaded 215
g. Flat Shaded w/Edges 216
h. Gouraud Shaded w/Edges 216

6. **Measure** 257
a. Measure Distance 245
b. Measure Area 247
c. Measure Angle 249

(Note: Many of the above options and sub menus are equipped with a keyboard shortcut to improve drawing efficiency; to customize your shortcuts, see Key Accelerators 271 under Options Menu 99)

Also See:
View Toolbar 253
Modify Toolbar 253
Create Toolbar 103
Draw Plane/UCS Toolbar 193
Render Toolbar 256
Measure Toolbar 257
Keyboard Shortcuts 271

2.2.1.6 Options Menu

The *Options Menu* in **hsCADCreator** is generally where you go to alter the default settings as well as manipulate the Keyboard shortcuts |271| and manage **hsCADCreator** plugins.

1.   **Settings...** |264|(Location of the Application settings)

2.   **Languages** (Language selection)
         a. English (Default)

3.   **Customize...** |275| (Customization of Toolbars |275| / Keyboard Shortcuts |271| / and more)

4.   **Manage Plugins...** (Manage additions to the **hsCADCreator** application.)

2.2.1.7 Window Menu

The *Window Menu* in **hsCADCreator** is where you can manage multiple documents as well as customize the Layout of the user interface.

1.   **Cascade** (Places all open documents in an overlapping stair stepping fashion within the **hsCADCreator** workspace)

2.   **Tile** (Places all open documents in a visible rectangle within the **hsCADCreator** workspace without any overlapping between documents)

3.   **Close All Windows** (Will close all open documents within the **hsCADCreator** workspace. This will not close the program.)

4.   **Layout** |279| (Workspace/User Interface Elements)
         a.   **Basic Layout** |279| (Changes the workspace layout to the basic window configuration)
         b.   **Advanced Layout** |279| (Changes the workspace to the advanced window configuration where more space is designated to the drawing area)
         c.   **Custom Layout** |279| (Allows the user to Save, Revert, Save As..., and Load From... layouts for custom user definted window configurations.)

5.   **Current Open Drawings** (List of open drawings within the workspace. Selection will activate that document and make it the current working document.)

Also See:
     Toolbars 250

2.2.1.8   Help Menu

The *Help Menu* in **hsCADCreator** provides an easy way to open the help documentation, manage the license 296 and discover the current program version 313 you are working with.

1.   **About hsCADCreator...** 264 (Displays the About dialog that will display the program version number and quick view at the license status.)

2.   **Tutorials** (This is a link to helpful tutorials on Hachisoft Corporation's website)

3.   **www.hachisoft.com** (This is a link to the designers of **hsCADCreator**. Check this site for update, version upgrades and plugins.)

4.   **Check for Updates** (This will link you to the **hsCADCreator** product site to ensure you have the latest version of the software.

4.   **Support** (Convenient ways to request support or a new feature to Hachisoft Corporation)

4.   **Purchase Upgrade to Full Version** (This is a web link to the online store where you can purchase one or more licenses of **hsCADCreator**.)

5.   **Remove License** 300 (Ability to remove a license from a computer for use on a different computer)

5.   **Activate License** 296 (Opens the License management dialog)

6.   **Help** (Activates this help documentation)

2.2.2   hsCADCreator Tools

This application is built on the concept of tools. Loosely speaking, a tool for **hsCADCreator** is an implementation of ability to perform a certain class of operation or procedure with regards to viewing and editing a drawing. All tool gives different type of notifications in notification window 267 as you advance to different steps.

Tool Modes

- Select, click and drag *mouse-based mode*
- Context-sensitive tool properties allow for a precise *data-entry mode*

Tool Properties

Each tool may have a set of properties associated with it. Tool properties can be changed to change the tool's behavior.

hsCADCreator has always one active tool ⌐80⌐ at any time. There may be more than one tools on Tool Stack ⌐80⌐ but only one tool is active at any given time.

hsCADCreator has six different tool categories available to the user in addition to the standard file tools:

    1 **Create Tools** ⌐103⌐ : Tools used to draw an entity

    2 **Modify Tools** ⌐147⌐ : Tools used to change an existing entity in the drawing.

    3 **View Tools** ⌐173⌐ : Tools used to alter viewing parameters (i.e. camera location, camera angle, camera zoom level, etc.)

    4 **Draw Plane(UCS) Tools** ⌐193⌐ : Tools used to create/edit User coordinate system.

    5. **Rendering Tools** ⌐213⌐ : Tools used to produce the different rendering view of the drawing.

    6. **Library Tools** ⌐217⌐ : Tools used to add, remove, or modify specialized object types

    7. **Measure Tools** ⌐244⌐ : Tools to measure lengths and area

2.2.2.1 Selection Tool

Entity toolbar ⌐251⌐ **:**

The *Selection Tool* is used to *choose/select* created entities for modification/deletion. After selection, an entity can be modified by utilizing the Modify Toolbar ⌐253⌐, changing the entity properties ⌐77⌐, or by changing its grip points ⌐74⌐.

Making a Selection:

Find the *Selection Tool* icon from the *Entity Toolbar* ⌐251⌐(color coded blue) highlight it. Click ⌐15⌐ on highlighted tool and it will become the active tool ready to perform a selection. Move the mouse close to the entity.

There are three methods to select any entity/entities:

Click : Simply *click* ⌐15⌐ on entity to select it. This is selection by a single selection point.

Fully enclosed : Drag left mouse ⌐16⌐ button over the entities from **top left** side of entities towards **bottom right** side of entities and release mouse ⌐16⌐. A temporary rectangle with a solid line be displayed. Any entities fully enclosed in the drawn rectangle will be selected.

Partially enclosed   : Drag left mouse ⌐16⌐ button over the entities from
bottom right side of entities towards **top left** side of
entities and release mouse ⌐16⌐. A temporary
rectangular box will be drawn with a dashed border.
Any entities that are crossing or fully enclosed in the
drawn rectangle boundary will be selected.

Selection Options:

There are multiple options within **hsCADCreator** to enable a more specific selection set.
Selections can be made based on Layer ⌐64⌐, Color ⌐51⌐, Entity Type ⌐21⌐, and UCS ⌐17⌐ relativity.
Below are the selection tool properties and their options.

To make a specific selection. Select the options from the **Selection Options property tree**.
Then perform the selection the same way as **Making a Selection** section above

Property Name	Data Type	Description
Layers	Multi-option ⌐79⌐ selection box	Selection of Entity(ies) on chosen Layer(s)
Color	Multi-option ⌐79⌐ selection box	Selection of Entity(ies) with chosen Color(s)
Selection Size	Integer Number ⌐78⌐	Maximum Size of Selection Set
Entity Type	Multi-option ⌐79⌐ selection box	Selection of Entity(ies) of chosen Type(s)
On UCS Plane	Real Number ⌐78⌐	Selection of Entity(ies) within a tolerance of current UCS Plane

Tool Options:

Escape (Esc): key clears the current selection set.

Space: key Resets this tool.

2.2.2.2 Create Tools

hsCADCreator has twenty-two different tools available to create drawing entities. The *Create Tools* can be accessed by using the blue colored Entity Toolbar 251, or by accessing the Create option from the Tool Menu 97. The Create tools primary function is to allow the creation of drawing entities through a user friendly process of steps.

1. **Selection Tool** 101 Facilitates the selection of entities.

2. **Point Tool** 105 Facilitates the creation of a Point Entity

3. **Line Tool** 106 Facilitates the creation of a Line Entity

4. **Multi Line Tool** 108 Facilitates the creation of a Multi Line Entity

5. **Planar Polyline Tool** 110 Facilitates the creation of a planar polyline.

6. **3D Polyline Tool** 112 Facilitates the creation of a polyline within 3D Space

7. **Arc Tools** 113 Facilitates the creation of Arc Entities

    Start-Middle-End Arc Tool 114 Facilitates the creation of an Arc Entity using Start, Middle and End points.

    Start-Center-End Arc Tool 115 Facilitates the creation of an Arc Entity using Start, Center and End points.

    Start-Center-Angle Arc Tool 117 Facilitates the creation of an Arc Entity using Start

and Center points and an Angle.

Start-Center-Chord Length Arc Tool|119| Facilitates the creation of an Arc Entity
using Start and Center points and a Chord length.

8.   **Circle Tool**|120| Facilitates the creation of a Circle Entity

9.   **Ellipse Tool**|121| Facilitates the creation of an Ellipse Entity

10.   **Dimension Tools**|123| Facilitates the creation of Dimension Entities

Aligned Dimension Tool|124| Facilitates the creation of a Dimension Entity that aligns
to two user directed points.

Linear Dimension Tool|126| Facilitates the creation of a Dimension Entity that depicts
the linear distance between to user defined points.

Ordinate Dimension Tool|128| Facilitates the creation of a Dimension Entity that
displays either the X or Y distance from the user specified origin

Radial Dimension Tool|130| Facilitates the creation of a Dimension Entity that depicts
the radius of an entity.

Diametric Dimension Tool|132| Facilitates the creation of a Dimension Entity that
depicts the diameter of an entity.

Angular Dimension Tool|134| Facilitates the creation of a Dimension Entity depicting
an angle specified from the user supplied center, start and end points.

11.   **Text Tool**|136| Facilitates the creation of a Text Entity.

12.   **Hatch Tool**|138| Facilitates the creation of a Hatch Entity within a specified area.

13.   **Face Mesh Tool**|139| Facilitates the creation of a Face Entity on the current UCS Draw
plane.

14.   **Block Definition Tool**|140| Facilitates the creation of a Block Definition or group of entities
combined together to create a single entity.

15.   **Block Insertion Tool**|142| Facilitates the creation and placement of a Block Entity from a
previously created Block Definition.

16.   **Image Insertion Tool**|144| Facilitates the creation of an Image Entity.

17.   **Viewport Creation Tool**|146| Facilitates the creation of a Viewport Entity (Used in Layouts
only)

2.2.2.2.1 Point Tool

Entity toolbar 25 **:**

The *Point Tool* is responsible for creating Point Entities 22 .

Creating a Point:
Activate the Point Tool:

Find the *Point Tool* icon from the *Entity Toolbar* 25 (color coded blue) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Creating a Point using the Mouse:

1.   Click 15 anywhere on the drawing screen to create a Point Entity 22 at that location.

Reset:
You have just added a Point Entity 22 to the drawing at the designated location. The tool has now **Reset** and you can either continue with other tools or add in another point. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating a Point using the Property Tree:

To add a Point Entity 22 from user supplied data do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260 .

☐ **Point Tool**				?
▷Point	650.000	250.000	0.000	Ⓤ

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

1.   To add a **Point** alter the numerical values for **X**, **Y**, and **Z**. on the 3D Point Property Item 77 **Point** and press **Enter** to accept.

Reset:
You have just added a Point Entity 22 to the drawing at the designated location. The tool has now **Reset** and you can either continue with other tools or add in another point. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool `101`).

Space: key Resets this tool.

Also See:
> Point Entity `22`
> Grid `75`
> Grid Snaps `74`
> Entity Snaps `72`
> Notification Bar `267`

2.2.2.2.2 Line Tool

Entity toolbar `251` **:**

The *Point Tool* is responsible for creating Line Entities `23`.

Creating a Line:
Activate the Line Tool:

Find the *Line Tool* icon from the *Entity Toolbar* `251` (color coded blue) and highlight it. While highlighted Click `15` on the tool. The tool is now the active tool and ready for use.

Creating a Line using the Mouse:

1. Click `15` anywhere on the drawing screen to designate the **Start Point** at that location.
2. Click `15` anywhere on the drawing screen to designate the **End Point** at that location.
3. If in **Specify Length Mode** move the mouse to the designated length of the Line Entity `23` and click `15`. (The length must be greater than zero and the new segment will always be created along the two specified points.)
4. A Line Entity `23` has been added. If in **Contiguous Mode** the tool will not **Reset**. The **End Point** of the newly created Line Entity `23` will automatically become the **Start Point** for a new Line Entity `23` and you will repeat steps 2,3 and 4 until finished with the tool.

Reset:
You have just added a Line Entity `23` to the drawing at the designated location. The tool has now **Reset** and you can either continue with other tools or add in another Line Entity `23`. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating a Line using the Property Tree:
To add a Line Entity `23` from user supplied data do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree `260`.

Line Tool Settings

Contiguous Mode	☐
Specify Length	☐

Line Tool   ?

▷Starting Point	**600.000**	**170.000**	**0.000**	Ⓤ
Ending Point	0.000	0.000	0.000	Ⓤ
Length			0.000 ◁ mm ▷

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

1. Type in the **Start Point** point data into the "**Start Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.
2. Type in the **End Point** point data into the "**End Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.
3. If in **Specify Length Mode** type in the **Length** data into the "**Length**" Scientific Data Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.
4. A Line Entity [23] has been added. If in **Contiguous Mode** the tool will not **Reset**. The **End Point** of the newly created Line Entity [23] will automatically become the **Start Point** for a new Line Entity [23] and you will repeat steps 2,3 and 4 until finished with the tool.

Reset:
You have just added a Line Entity [23] to the drawing at the designated location. The tool has now **Reset** and you can start adding another Line Entity [23] if desired. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

*Contiguous Mode***:** allows the user to create multiple Line Entities [23] that simulate a polyline's connectivity.
*Specify Length***:** allows the user to specify the **Length** of the line segment as part of the Line Entity [23] creation process.

⌨ *Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool [101]).

⌨ *Space:* key **Resets** this tool.

Also See:
Line Entity [23]
Grid [75]
Grid Snaps [74]
Entity Snaps [72]
Notification Bar [267]

2.2.2.2.3 Multi Line Tool

Entity toolbar 25 **:**

The *Mulit Line Tool* is responsible for creating Multi Line Entities 24.

Creating a Multi Line:
Activate the Multi Line Tool:

Find the *Mulit Line Tool* icon from the *Entity Toolbar* 25 (color coded blue) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Creating a Multi Line using the Mouse:

1. Click 15 anywhere on the drawing screen to designate the **Start Point** at that location.
2. Click 15 anywhere on the drawing screen to designate the **End Point** at that location.
3. If in **Specify Length Mode** move the mouse to the designated length of the Multi Line Entity 24 and click 15. (The length must be greater than zero and the new segment will always be created along the two specified points.)
4. A Line Entity 23 has been added. If in **Contiguous Mode** the tool will not **Reset**. The **End Point** of the newly created Multi Line Entity 24 will automatically become the **Start Point** for a new Multi Line Entity 24 and you will repeat steps 2,3 and 4 until finished with the tool.

Reset:
You have just added a Multi Line Entity 24 to the drawing at the designated location. The tool has now **Reset** and you can either continue with other tools or add in another Multi Line Entity 24. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating a Multi Line using the Property Tree:

To add a Multi Line Entity 24 from user supplied data do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

Multi Line Tool Settings

Multi line style	Standard
Multi line justification	Align Center with cursor
Closed	☐
End Caps	☐
Start Caps	☐
Contiguous Mode	☑
Specify Length	☐

Multi Line Tool　[?]

▷ Starting Point	**496.28**	**208.50**	**0.00**	[U]
Ending Point	0.00	0.00	0.00	[U]
Length			0.00 ◁ mm ▷	

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

1. Type in the **Start Point** point data into the "**Start Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.
2. Type in the **End Point** point data into the "**End Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.
3. If in **Specify Length Mode** type in the **Length** data into the "**Length**" Scientific Data Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.
4. A Multi Line Entity [24] has been added. If in **Contiguous Mode** the tool will not **Reset**. The **End Point** of the newly created Multi Line Entity [24] will automatically become the **Start Point** for a new Multi Line Entity [24] and you will repeat steps 2,3 and 4 until finished with the tool.

Reset:
You have just added a Multi Line Entity [24] to the drawing at the designated location. The tool has now **Reset** and you can start adding another Multi Line Entity [24] if desired. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

Contiguous Mode: allows the user to create continuous Multi Line Entity [24].
Specify Length: allows the user to specify the **Length** of the line segment as part of the Multi Line Entity [24] creation process.

⌨ **Escape (Esc):** key cancels current tool and activates the default tool (Selection Tool [101]).

⌨ **Space:** key **Resets** this tool.

Also See:
Multi Line Entity [24]

Grid [75]
Grid Snaps [74]
Entity Snaps [72]
Notification Bar [267]

2.2.2.2.4 Planar Polyline Tool

Entity toolbar [251] **:**

The *Planar Polyline Tool* is responsible for creating Planar Polyline Entities. [26]

Creating a Planar Polyline:
Activate the Planar Polyline Tool:

Find the *Planar Polyline Tool* icon from the *Entity Toolbar* [251] (color coded blue) and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Creating a Planar Polyline using the Mouse:

1.   Click [15] anywhere on the drawing screen to designate the first **Current Point** at that location. This will add a Polyline vertex at this location and set the **Previous Point** to this value.
2.   If there only one vertex currently associated with the Planar Polyline [26] then move your mouse and click [15] to add a second vertex, *otherwise skip this step*.
3.   If in **Specify Length Mode** move the mouse until it displays the desired length of the Planar Polyline [26] and then click [15].
4.   Repeat Step 1, 2 ,and 3 until all desired vertices have been added.
5.   Press **Shift+Enter** to finish and add the Planar Polyline [26] to the drawing.

Reset:
You have just added a Planar Polyline [26] to the drawing with vertices at the designated locations. The tool has now **Reset** and you can either continue with other tools or add in another Planar Polyline [26]. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating a Planar Polyline using the Property Tree:

To add a Planar Polyline [26] from user supplied data do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree [260].

Planar Polyline Tool Settings				
Draw Bulge	☐			
Draw Closed	☐			
Specify Length	☐			

Planar Polyline Tool				?
Previous Point	0.000	0.000	0.000	U
▷Current Point	**540.000**	**190.000**	**0.000**	U
Length			572.451	mm

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

1. Type in the **Current Point** point data into the "**Current Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. This will add a Polyline vertex at this location and set the **Previous Point** to this value.
2. If only one vertex has been added type in the **Current Point** point data into the "**Current Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.
3. If in **Specify Length Mode** type in the **Length** data into the "**Length**" Scientific Data Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.
4. Repeat Step 1, 2 ,and 3 until all desired vertices have been added.
5. Press [⌨] **Shift+Enter** to finish and add the polyline 26 to the drawing.

Reset:
You have just added a Planar Polyline 26 to the drawing with vertices at the designated locations. The tool has now **Reset** and you can either continue with other tools or add in another Planar Polyline 26. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

[⌨] ***Tab: Draw Bulge Mode*** allows the user to draw arcs as a polyline segment.

[⌨] ***Ctrl+Enter: Draw Closed Mode*** allows the user to create a Planar Polyline Entity 26 that will add a segment from the **Last Point** in the polyline back to the **First Point** in the polyline.
Specify Length Mode allows the user to specify the **Length** of a particular polyline segment as part of the Planar Polyline 26 creation process.

[🖱] ***Right Click: Options*** allows the user to view a Planar Polyline Context Menu 91 with options for use in creating the Planar Polyline Entities 26.

[⌨] ***Shift+Enter:*** this keystroke will complete and add the polyline to the drawing.

[⌨] ***Escape (Esc):*** key cancels current tool and activates the default tool (Selection Tool 101).

[⌨] ***Space:*** key **Resets** this tool.

Also See:
> Planar Polyline 26
> Grid 75
> Grid Snaps 74
> Entity Snaps 72
> Notification Bar 267

2.2.2.2.5 3D Polyline Tool

Entity toolbar 251 **:**

The *3D Polyline Tool* is responsible for creating 3D Polyline Entities 27

Creating a 3D Polyline:
Activate the 3D Polyline Tool:

> Find the *3D Polyline Tool* icon from the *Entity Toolbar* 251 (color coded blue) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Creating a 3D Polyline using the Mouse:
1. Click 15 anywhere on the drawing screen to designate the first **Current Point** at that location. This will add a Polyline vertex at this location and set the **Previous Point** to this value.
2. Repeat Step 1 until all desired vertices have been added.
3. Press **Shift+Enter** to finish and add the 3D Polyline 27 to the drawing.

Reset:
You have just added a 3D Polyline 27 to the drawing with vertices at the designated locations. The tool has now **Reset** and you can either continue with other tools or add in another 3D Polyline 27. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating a 3D Polyline using the Property Tree:
> To add a 3D Polyline 27 from user supplied data do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

3D Polyline Tool Settings				
Draw Closed	☐			
3D Polyline Tool				?
Previous Point	0.000	0.000	0.000	U
▷ Current Point	430.000	170.000	0.000	U

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

1. Type in the **Current Point** point data into the "**Current Point**" 3D Point Property Field ⌐77⌐ found on the Tool Property Tree ⌐260⌐ and press **Enter** to accept. This will add a Polyline vertex at this location and set the **Previous Point** to this value.
2. Repeat Step 1 until all desired vertices have been added.
3. Press ⌨ **Shift+Enter** to finish and add the 3D Polyline ⌐27⌐ to the drawing.

Reset:
You have just added a 3D Polyline ⌐27⌐ to the drawing with vertices at the designated locations. The tool has now **Reset** and you can either continue with other tools or add in another 3D Polyline ⌐27⌐. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

⌨ *Ctrl+Enter: Draw Closed Mode* allows the user to create a 3D Polyline Entity ⌐27⌐ that will add a polyline segment from the **Last Point** in the polyline back to the **First Point** in the polyline.

🖱 *Right Click: Options* allows the user to view a 3D Polyline Context Menu ⌐91⌐ with options for use in creating the 3D Polyline Entities ⌐27⌐.

⌨ *Shift+Enter:* this keystroke will complete and add the polyline to the drawing.

⌨ *Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool ⌐101⌐).

⌨ *Space:* key **Resets** this tool.

Also See:
3D Polyline ⌐27⌐
Grid ⌐75⌐
Grid Snaps ⌐74⌐
Entity Snaps ⌐72⌐
Notification Bar ⌐267⌐

2.2.2.2.6 Arc Tools

🖱 Entity toolbar ⌐251⌐ **:** 🔵 icons

The *Arc Tools* are responsible for creating Arc Entities ⌐28⌐.

Creating an Arc:

Activate an Arc Tool:

Find one of the *Arc Tool* icons 🔵 from the *Entity Toolbar* ⌐251⌐ (color coded blue) and highlight it. While highlighted Click ⌐15⌐ on the tool. The tool is now the active tool and ready for use. This set of tools is part of a Flyout toolbar ⌐81⌐.

Creating the Arc:
There are four different method that can be used to create an Arc Entity.

1. **Start-Middle-End Arc Tool** [114]: (default) Arcs that pass through three chosen points (a Start Point, Middle Point, and End Point)

2. **Start-Center-End Arc Tool** [115]: Arcs drawn from a Start Point to an End Point based on the location of a (circle's) Center Point

3. **Start-Center-Angle Arc Tool** [117]: Arcs drawn from a Start Point to a point on a circle with a specific included angle based on the location of the (circle's) Center Point

4. **Start-Center-Chord Length Arc Tool** [119]: Arcs drawn from a Start Point to a point of specified chord length based on the location of the (circle's) Center Point

Also See:
Arc Entity [28]
Flyout Toolbar [81]

2.2.2.2.6.1 Start-Middle-End Arc Tool

Entity toolbar [25] :

The *Arc Tools* is responsible for creating Arc Entities [28].

Creating an Arc (Start-Middle-End):
Activate the Arc (Start-Middle-End) Tool:

Find the *Arc (Start-Middle-End) Tool* icon from the *Entity Toolbar* [25] (color coded blue) or Arc Flyout Toolbar [81] and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Creating an Arc (Start-Middle-End) using the Mouse:
1. Click [15] anywhere on the drawing screen to designate the **Start Point** of the Arc at that location.
2. Move the mouse and Click [15] on a point you desire to be on the Arc but is not the same as the Start Point. This is the **Middle Point**.
3. Move the mouse and Click [15] on a point you wish to designate as the **End Point**. This is the location at which the arc will no longer be drawn.

Reset:
You have just added an Arc Entity [28] to the drawing using a Start Point, Middle Point (point along the arc), and End Point. The tool has now **Reset** and you can either continue with other tools or add in another Arc Entity [28]. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating an Arc (Start-Middle-End) using the Property Tree:
To add an Arc Entity [28] from user supplied data using the (Start-Middle-End) method do the following . Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree [260].

1. Type in the **Start Point** point data into the "**Start Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept. This will position the starting point for the arc.
2. Type in the **Middle Point** point data into the "**Middle Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.
3. Type in the **End Point** point data into the "**End Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept. This will finish the arc creation process.

Reset:
You have just added an Arc Entity [28] to the drawing using a Start Point, Middle Point (point along the arc), and End Point. The tool has now **Reset** and you can either continue with other tools or add in another Arc Entity [28]. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool [101]).

Space: key **Resets** this tool.

Also See:
Arc Entity [28]
Grid [75]
Grid Snaps [74]
Entity Snaps [72]
Notification Bar [267]
Flyout Toolbar [81]

2.2.2.2.6.2 Start-Center-End Arc Tool

Entity toolbar [251] :

The *Arc Tools* is responsible for creating Arc Entities [28] from a **Start Point** (where the arc starts) to an **End Point** (point where the arc ends) about the **Center Point** (point where the center would be located should the arc create a full circle).

Creating an Arc (Start-Center-End):
Activate the Arc (Start-Center-End) Tool:

Find the *Arc (Start-Center-End) Tool* icon ⬚ from the *Entity Toolbar* |251⌐(color coded blue) or Arc Flyout Toolbar |81⌐ and highlight it. While highlighted Click |15⌐ on the tool. The tool is now the active tool and ready for use.

Creating an Arc (Start-Center-End) using the Mouse:

1. Click |15⌐ anywhere on the drawing screen to designate the **Start Point** of the Arc at that location.
2. Move the mouse and Click |15⌐ on a point you desire to be the **Center Point** of an Arc that would create a full circle.
3. Move the mouse and Click |15⌐ on a point you wish to designate as the **End Point**. This is the location at which the arc will no longer be drawn. By default the Arc will be created in a Clockwise fashion. If in the **CCW Mode** the Arc will be drawn in the Counter-Clockwise fashion.

Reset:
You have just added an Arc Entity |28⌐ to the drawing using a **Start Point**, **Center Point**, and **End Point**. The tool has now **Reset** and you can either continue with other tools or add in another Arc Entity |28⌐. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating an Arc (Start-Center-End) using the Property Tree:

To add an Arc Entity |28⌐ from user supplied data using the (Start-Center-End) method do the following . Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree |260⌐.

⊟ **Arc Tool Settings**				?
CCW Arc	☑			
⊟ **Arc Tool (Start,Center,End)**				?
▷Start Point	470.000	180.000	0.000	Ⓤ
Center Point	0.000	0.000	0.000	Ⓤ
End Point	0.000	0.000	0.000	Ⓤ

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

1. Type in the **Start Point** point data into the "**Start Point**" 3D Point Property Field |77⌐ found on the Tool Property Tree |260⌐ and press **Enter** to accept. This will position the starting point for the arc.
2. Type in the **Center Point** point data into the "**Center Point**" 3D Point Property Field |77⌐ found on the Tool Property Tree |260⌐ and press **Enter** to accept.
3. Type in the **End Point** point data into the "**End Point**" 3D Point Property Field |77⌐ found on the Tool Property Tree |260⌐ and press **Enter** to accept. This will finish the arc creation process. By default the Arc will be created in a Clockwise fashion. If in the **CCW Mode** the Arc will be drawn in the Counter-Clockwise fashion.

Reset:
You have just added an Arc Entity |28⌐ to the drawing using a **Start Point**, **Center Point**, and **End Point**. The tool has now **Reset** and you can either continue with

other tools or add in another Arc Entity ⌐28⌐. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:
CCW Arc mode draws the Arc Entity ⌐28⌐ Counter-Clockwise from the **Start Point** to the **End Point** around the **Center Point**.

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool ⌐101⌐).

Space: key **Resets** this tool.

Also See:
Arc Entity ⌐28⌐
Grid ⌐75⌐
Grid Snaps ⌐74⌐
Entity Snaps ⌐72⌐
Notification Bar ⌐267⌐
Flyout Toolbar ⌐81⌐

2.2.2.2.6.3 Start-Center-Angle Arc Tool

Entity toolbar ⌐25⌐ **:**

The *Arc Tools* is responsible for creating Arc Entities ⌐28⌐ from a **Start Point** (where the arc starts) about the **Center Point** (point where the center would be located should the arc create a full circle) with a designated **Included Angle** (Angle at which the arc is drawn about the center point).

Creating an Arc (Start-Center-Angle):
Activate the Arc (Start-Center-Angle) Tool:

Find the *Arc (Start-Center-Angle) Tool* icon from the *Entity Toolbar* ⌐25⌐(color coded blue) or Arc Flyout Toolbar ⌐81⌐ and highlight it. While highlighted Click ⌐15⌐ on the tool. The tool is now the active tool and ready for use.

Creating an Arc (Start-Center-Angle) using the Mouse:
1. Click ⌐15⌐ anywhere on the drawing screen to designate the **Start Point** of the Arc at that location.
2. Move the mouse and Click ⌐15⌐ on a point you desire to be the **Center Point** of an Arc that would create a full circle.
3. Move the mouse and Click ⌐15⌐ on a point at which the arc creates the desired **Angle** from the Start Point about the Center Point. By default the Arc will be created in a Clockwise fashion. If in the **CCW Mode** the Arc will be drawn in Counter-Clockwise fashion.

Reset:
You have just added an Arc Entity ⌐28⌐ to the drawing using a **Start Point**, **Center Point**, and **Angle**. The tool has now **Reset** and you can either continue with other tools or add in another Arc Entity ⌐28⌐. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating an Arc (Start-Center-Angle) using the Property Tree:
To add an Arc Entity ⌐28⌐ from user supplied data using the (Start-Center-Angle)

method do the following . Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree[260].

Arc Tool Settings [?]

CCW Arc [✓]

Arc Tool (Start,Center,Angle) [?]

▷Start Point	**640.000**	**210.000**	**0.000**	[U]
Center Point	0.000	0.000	0.000	[U]
Included Angle			0.000 ◁ ° ▷	

Tool Properties

Details of the active Tool

Selection | Tool | Snaps | Drawing | Viewport

1. Type in the **Start Point** point data into the "**Start Point**" 3D Point Property Field [77] found on the Tool Property Tree[260] and press **Enter** to accept. This will position the starting point for the arc.
2. Type in the **Center Point** point data into the "**Center Point**" 3D Point Property Field [77] found on the Tool Property Tree[260] and press **Enter** to accept.
3. Type in the **Angle** data into the "**Included Angle**" Scientific Data Field [77] found on the Tool Property Tree[260] and press **Enter** to accept. This will finish the arc creation process. By default the Arc will be created in a Clockwise fashion. If in the **CCW Mode** the Arc will be drawn in a Counter-Clockwise fashion.

Reset:
You have just added an Arc Entity [28] to the drawing using a **Start Point**, **Center Point**, and **Angle**. The tool has now **Reset** and you can either continue with other tools or add in another Arc Entity [28]. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

CCW Arc mode draws the Arc Entity [28] Counter-Clockwise from the **Start Point** around the **Center Point** with a designated **Included Angle**.

⌨ *Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool [101]).

⌨ *Space:* key **Resets** this tool.

Also See:
Arc Entity [28]
Grid [75]
Grid Snaps [74]
Entity Snaps [72]
Notification Bar [267]
Flyout Toolbar [81]

2.2.2.2.6.4 Start-Center-Chord Length Arc Tool

⬚⬛ Entity toolbar ²⁵ **:** 𝒱

The ***Arc Start-Center-Chord Length Tool*** is responsible for creating Arc Entities ²⁸ from a **Start Point** (where the arc starts) about the **Center Point** (point where the center would be located should the arc create a full circle) with a designated **Chord Length** (shortest distance between the start point and ending point of the arc).

Creating an Arc (Start-Center-Chord Length):
Activate the Arc (Start-Center-Chord Length) Tool:

Find the ***Arc (Start-Center-Angle) Tool*** icon 𝒱 from the *Entity Toolbar* ²⁵ (color coded blue) or Arc Flyout Toolbar ⁸¹ and highlight it. While highlighted Click ¹⁵ on the tool. The tool is now the active tool and ready for use.

⬚⬛ Creating an Arc (Start-Center-Chord Length) using the Mouse:
1. Click ¹⁵ anywhere on the drawing screen to designate the **Start Point** of the Arc at that location.
2. Move the mouse and Click ¹⁵ on a point you desire to be the **Center Point** of an Arc that would create a full circle.
3. Move the mouse and Click ¹⁵ on a point at which **Chord Length** is the distance from the Start point that you desire. By default the Arc will be created in a Clockwise fashion. If in the **CCW Mode** the Arc will be drawn in Counter-Clockwise fashion.

Reset:
You have just added an Arc Entity ²⁸ to the drawing using a **Start Point**, **Center Point**, and **Chord Length**. The tool has now **Reset** and you can either continue with other tools or add in another Arc Entity ²⁸. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

⌨ Creating an Arc (Start-Center-Chord Length) using the Property Tree:
To add an Arc Entity ²⁸ from user supplied data using the (Start-Center-Chord Length) method do the following . Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree ²⁶⁰.

1. Type in the **Start Point** point data into the "**Start Point**" 3D Point Property Field ⁷⁷ found on the Tool Property Tree ²⁶⁰ and press **Enter** to accept. This will

position the starting point for the arc.

2.   Type in the **Center Point** point data into the "**Center Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.

3.   Type in the **Chord Length** data into the "**Chord Length**" Scientific Data Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. This will finish the arc creation process. By default the Arc will be created in a Clockwise fashion. If in the **CCW Mode** the Arc will be drawn in a Counter-Clockwise fashion. Note: The max cord length is the diameter of the circle created by the center point and the start point.

Reset:
You have just added an Arc Entity 28 to the drawing using a **Start Point**, **Center Point**, and **Chord Length**. The tool has now **Reset** and you can either continue with other tools or add in another Arc Entity 28. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

CCW Arc mode draws the Arc Entity 28 Counter-Clockwise from the **Start Point** around the **Center Point** with a designated **Chord Length**.

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
Arc Entity 28
Grid 75
Grid Snaps 74
Entity Snaps 72
Notification Bar 267
Flyout Toolbar 81

2.2.2.2.7 Circle Tool

Entity toolbar 251 **:**

The *Circle Tool* is responsible for creating Circle Entities 30.

Creating a Circle:
Activate the Circle Tool:

Find the *Circle Tool* icon from the Entity Toolbar 251 (color coded blue) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Creating a Circle using the Mouse:

1.   Click 15 anywhere on the drawing screen to designate the **Center Point** of the Circle Entity 30 at that location.

2.   Move the mouse anywhere on the drawing screen when the distance from the Center Point to the cursor is the desired **Radius** of the circle then click Click 15.

Reset:
You have just added a Circle Entity ⌐30⌐ to the drawing with the center of the circle being your designated center point. The tool has now **Reset** and you can either continue with other tools or add in another Circle Entity ⌐30⌐. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating a Circle using the Property Tree:

To add a circle from user supplied data do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree ⌐260⌐.

⊟	**Circle Tool**				?
▷	Center Point	430.000	170.000	0.000	U
	Radius			0.000 ◁ mm ▷	

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

1. Type in the **Center Point** point data into the "**Center Point**" 3D Point Property Field ⌐77⌐ found on the Tool Property Tree ⌐260⌐ and press **Enter** to accept.
2. Type in the **Radius** data into the "**Radius**" Scientific Data Field ⌐77⌐ found on the Tool Property Tree ⌐260⌐ and press **Enter** to accept.

Reset:
You have just added a Circle Entity ⌐30⌐ to the drawing with the center of the circle being your designated center point. The tool has now **Reset** and you can either continue with other tools or add in another Circle Entity ⌐30⌐. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool ⌐101⌐).

Space: key **Resets** this tool.

Also See:
Circle Entity ⌐30⌐
Grid ⌐75⌐
Grid Snaps ⌐74⌐
Entity Snaps ⌐72⌐
Notification Bar ⌐267⌐

2.2.2.2.8 Ellipse Tool

Entity toolbar ⌐251⌐ **:**

The *Ellipse Tool* is responsible for creating Ellipse Entities ⌐32⌐.

Creating an Ellipse:
Activate the Ellipse Tool:

Find the ***Ellipse Tool*** icon [icon] from the Entity Toolbar [251] (color coded blue) and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Creating an Ellipse using the Mouse:

1. Click [15] anywhere on the drawing screen to designate the **Center Point** of the Ellipse Entity [32] at that location.
2. Move the mouse anywhere on the drawing screen and designate the **Major Axis** for the ellipse. The Major axis is essentially the first radius of the two that made up the ellipse.
3. Move the mouse anywhere on the drawing screen and designate the **Minor Axis** for the ellipse. The Minor axis must be smaller or the same length as the distance between the center point and major axis point.

Reset:

You have just added a Ellipse Entity [32] to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another Ellipse Entity [32]. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating an Ellipse using the Property Tree:

To add an Ellipse Entity [32] from user supplied data do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree [260].

⊟ **Ellipse Tool**				[?]
▷ Center Point	420.000	170.000	0.000	[U]
Major Axis Point	0.000	0.000	0.000	[U]
Minor Axis Length			0.000 ◁ mm ▷	

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

1. Type in the **Center Point** point data into the "**Center Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.
2. Type in the **Major Axis Point** point data into the "**Major Axis Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept. The Major axis is essentially the first radius of the two that made up the ellipse.
3. Type in the **Minor Axis Length** data into the "**Minor Axis Length**" Scientific Data Field [77] found on the Tool Property Tree [260] and press **Enter** to accept. The Minor axis must be smaller or the same length as the distance between the center point and major axis point.

Reset:

You have just added a <u>Ellipse Entity</u> ⌐32⌐ to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another <u>Ellipse Entity</u> ⌐32⌐. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (<u>Selection Tool</u> ⌐10⌐).

Space: key **Resets** this tool.

Also See:
> <u>Ellipse Entity</u> ⌐32⌐
> <u>Grid</u> ⌐75⌐
> <u>Grid Snaps</u> ⌐74⌐
> <u>Entity Snaps</u> ⌐72⌐
> <u>Notification Bar</u> ⌐267⌐

2.2.2.2.9 Dimension Tool

<u>Entity toolbar</u> ⌐251⌐ **:**

The ***Dimension Tool*** is responsible for creating <u>Dimension Entities</u> ⌐33⌐.

Creating a Dimension:

Activate a Dimension Tool:

Find one of the ***Dimension Line Tool*** icons from the <u>Entity Toolbar</u> ⌐251⌐(color coded blue) and highlight it. While highlighted <u>Click</u> ⌐15⌐ on the tool. The tool is now the active tool and ready for use. This set of tools is part of a <u>Flyout toolbar</u> ⌐81⌐.

Creating the Dimension Line:

There are six different types of Dimension Line tools available to create various types of <u>Dimension Lines</u> ⌐33⌐.

1. **Aligned Dimension Tool** ⌐124⌐(default)   : A Linear dimension for objects with unusual shapes, or with edges at unusual angles; the dimension line will always parallel the imaginary line connecting the origins of the extension lines.

2. **Linear Dimension Tool** ⌐126⌐ : Linear dimension that will either be oriented along either the X-axis or the Y-axis (your choice).

3. **Ordinate Dimension Tool** ⌐128⌐ : Dimension of an X-axis or Y-axis point. (your choice)

4. **Radial Dimension Tool** ⌐130⌐ : Dimension for the radius of a circle.

5. ⊕ **Diametric Dimension Tool** |132⌐ : Dimension for the diameter of a circle.

6. ⊿ **Angular Dimension Tool** |134⌐ : Dimension for an angle (in radians or degrees).

Also See:
Dimension Entity |33⌐
Manage Dimension Styles |221⌐
Flyout Toolbar |81⌐

2.2.2.2.9.1 Aligned Dimension Tool

⊟ Entity toolbar |251⌐ **:** ⊿

The *Aligned Dimension Tool* is responsible for creating Dimension Entities |33⌐ that lay alongside the entity or spacing they display a dimension for.

Creating an Aligned Dimension:
Activate the Aligned Dimension Tool:

Find the *Aligned Dimension Tool* icon ⊿ from the Entity Toolbar |251⌐(color coded blue) or Dimension Flyout Toolbar |81⌐ and highlight it. While highlighted Click |15⌐ on the tool. The tool is now the active tool and ready for use.

⊟ Creating an Aligned Dimension using the Mouse:

1. Click |15⌐ anywhere on the drawing screen to designate the **Start Point** of the Dimension at that location.
2. Move the mouse and Click |15⌐ on a point you desire to be the **End Point.** The default dimension text will be the distance between this point and the Start Point.
3. Move the mouse and Click |15⌐ on a point you wish to designate as the **Offset Point**. This point is the basis for offsetting the notation of the dimension away from the dimensioned entity or space.

Reset:
You have just added an Aligned Dimension Entity |35⌐ to the drawing using a Start Point, End Point, and Offset Point. The tool has now **Reset** and you can either continue with other tools or add in another Aligned Dimension Entity |35⌐. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

⌨ Creating an Aligned Dimension using the Property Tree:
To add an Aligned Dimension |35⌐ from user supplied data, do the following . Move the mouse cursor beyond the right side of the drawing screen to the Tool Property

Tree 260.

Dimension Tool Settings ?

Dim Style	Empty ▼
Autosnap Enabled	✔
Show Unit	✔

Aligned dimension Tool ?

▶ Start Point	**430.000**	**170.000**	**0.000**	U
End Point	0.000	0.000	0.000	U
Offset Point	0.000	0.000	0.000	U

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

1. Type in the **Start Point** point data into the "**Start Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. This will position the starting point for the dimension.
2. Type in the **End Point** point data into the "**End Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.
3. Type in the **Offset Point** point data into the "**Offset Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. This will finish the Aligned Dimension Entity 33 creation process. If in **Show Unit Mode** the resultant dimension text will have the current unit displayed.

Reset:
You have just added an Aligned Dimension Entity 33 to the drawing using a Start Point, End Point, and Offset Point. The tool has now **Reset** and you can either continue with other tools or add in another Dimension Entity 33. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

Tab: **Auto snap Enabled** key enables/disables the auto snap feature. Auto snap automatically enables the end, mid, nearest and quadrant Entity Snap 72 features.

Show Unit will display the dimension's unit within the Dimension Entity's text.

Escape (Esc): key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:

2.2.2.2.9.2 Linear Dimension Tool

Entity toolbar 25 **:**

The *Linear Dimension Tool* is responsible for creating Linear Dimension Entities 36 that represent the distance between two points either along the X-Axis or Y-Axis.

Creating a Linear Dimension:
Activate the Aligned Dimension Tool:

Find the *Linear Dimension Tool* icon from the Entity Toolbar 25 (color coded blue) or Dimension Flyout Toolbar 81 and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Creating a Linear Dimension using the Mouse:

1.   Click 15 anywhere on the drawing screen to designate the **Start Point** of the Dimension at that location.
2.   Move the mouse and Click 15 on a point you desire to be the **End Point.** If in **On Y Axis Mode** the dimension will represent the distance between this point and the Start Point along the Y-Axis. If in **On X Axis Mode** the dimension will represent the distance between this point and the Start Point along the X-Axis.
3.   Move the mouse and Click 15 on a point you wish to designate as the **Offset Point**. This point is the basis for offsetting the notation of the dimension away from the dimensioned entity or space. The Dimension text will be positioned parallel to the axis of measurement

Reset:
You have just added a Linear Dimension Entity 36 to the drawing using a Start Point, End Point, and Offset Point. The tool has now **Reset** and you can either continue with other tools or add in another Linear Dimension Entity 36. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating an Linear Dimension using the Property Tree:

To add an Linear Dimension 36 from user supplied data, do the following . Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

□	**Dimension Tool Settings**				?
	Dim Style	Empty			⌄
	Autosnap Enabled	☑			
	Show Unit	☑			
□	**Linear Dimension Tool Settings**				
	On Y-Axis	☑			
	On X-Axis	☐			
□	**Linear dimension Tool**				?
	▷Start Point	**440.000**	**170.000**	**0.000**	Ⓤ
	End Point	0.000	0.000	0.000	Ⓤ
	Offset Point	0.000	0.000	0.000	Ⓤ

Tool Properties

Details of the active Tool

Selection | Tool | Snaps | Drawing | Viewport

1. Type in the **Start Point** point data into the "**Start Point**" 3D Point Property Field
   77 found on the Tool Property Tree 260 and press **Enter** to accept. This will
   position the starting point for the linear dimension.

2. Type in the **End Point** point data into the "**End Point**" 3D Point Property Field
   77 found on the Tool Property Tree 260 and press **Enter** to accept. If in **On Y
   Axis Mode** the dimension will represent the distance between this point and
   the Start Point along the Y-Axis. If in **On X Axis Mode** the dimension will
   represent the distance between this point and the Start Point along the X-Axis.

3. Type in the **Offset Point** point data into the "**Offset Point**" 3D Point Property
   Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. The
   Dimension text will be positioned parallel to the axis of measurement. On
   acceptance the Linear Dimension Entity 33 creation process will be complete.
   If in **Show Unit Mode** the resultant dimension text will have the current unit
   displayed.

   Reset:
   You have just added a Linear Dimension Entity 33 to the drawing using a Start Point,
   End Point, and Offset Point. The tool has now **Reset** and you can either continue
   with other tools or add in another Dimension Entity 33. Using **Reset** at any time while
   using this tool cancels all previous steps and restarts the tool.

Tool Options:

Tab: **Auto snap Enabled** key enables/disables the auto snap feature. Auto snap
automatically enables the end and mid Entity Snap 72 features.

Show Unit will display the dimension's unit within the Dimension Entity's 33 text.

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool
101).

Space: key **Resets** this tool.

Also See:
Dimension Entity | 33 |
Manage Dimension Styles | 221 |
Grid | 75 |
Grid Snaps | 74 |
Entity Snaps | 72 |
Notification Bar | 267 |
Flyout Toolbar | 81 |

2.2.2.2.9.3 Ordinate Dimension Tool

Entity toolbar | 25 | **:**

The *Ordinate Dimension Tool* is responsible for creating Ordinate Dimension Entities | 38 |.

Creating an Ordinate Dimension :
Activate Ordinate Dimension Tool:

Find the *Ordinate Dimension Tool* icon from the *Entity Toolbar* | 25 | (color coded blue) or Dimension Flyout Toolbar | 81 | and highlight it. While highlighted Click | 15 | on the tool. The tool is now the active tool and ready for use.

Creating an Ordinate Dimension using the Mouse:

1. Click | 15 | anywhere on the drawing screen to designate the **Start Point** of the Dimension at that location.
2. Move the mouse and Click | 15 | on a point you desire to be the **End Point.** If in **On Y Axis Mode** the dimension will represent the distance between this point and the Start Point along the Y-Axis. If in **On X Axis Mode** the dimension will represent the distance between this point and the Start Point along the X-Axis.
3. Move the mouse and Click | 15 | on a point you wish to designate as the **Insertion Point**. This point is the basis for positioning the dimension text.

Creating an Ordinate Dimension using the Property Tree:

To add an Ordinate Dimension | 38 | from user supplied data, do the following . Move the mouse cursor beyond the right side of the drawing screen to the Tool Property

Tree 260.

Dimension Tool Settings				?
Dim Style	ISO-25			∨
Autosnap Enabled	✔			
Show Unit	✔			
Ordinate Dimension Tool Settings				
On X-Axis	✔			
On Y-Axis	☐			
Ordinate dimension Tool				?
▷Start Point	500.000	190.000	0.000	U
End Point	0.000	0.000	0.000	U
Insertion Point	0.000	0.000	0.000	U

Tool Properties

Details of the active Tool

Selection | Tool | Snaps | Drawing | Viewport

1. Type in the **Start Point** point data into the "**Start Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. This will position the starting point for the dimension.
2. Type in the **End Point** point data into the "**End Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. If in **On Y Axis Mode** the dimension will represent the distance between this point and the Start Point along the Y-Axis. If in **On X Axis Mode** the dimension will represent the distance between this point and the Start Point along the X-Axis.

3. Type in the **Insertion Point** point data into the "**Insertion Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. The Dimension text will be positioned at this point. On acceptance the Ordinate Dimension Entity 38 creation process will be complete. If in **Show Unit Mode** the resultant dimension text will have the current unit displayed.

Reset:
You have just added an Ordinate Dimension Entity 38 to the drawing using a Start Point, End Point, and Offset Point. The tool has now **Reset** and you can either continue with other tools or add in another Ordinate Dimension Entity 38. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

*Tab***: Auto snap Enabled** key enables/disables the auto snap feature. Auto snap automatically enables the end and nearest Entity Snap 72 features.

Show Unit will display the dimension's unit within the Dimension Entity's 33 text.

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
>   Dimension Entity [33]
>   Manage Dimension Styles [221]
>   Grid [75]
>   Grid Snaps [74]
>   Entity Snaps [72]
>   Notification Bar [267]
>   Flyout Toolbar [81]

2.2.2.2.9.4 Radial Dimension Tool

Entity toolbar [25] :

The *Radial Dimension Tool* is responsible for creating Radial Dimension Entities [39].

Creating an Radial Dimension :
Activate Radial Dimension Tool:

>   Find the *Radial Dimension Tool* icon   from the *Entity Toolbar* [25] (color coded
>   blue) or Dimension Flyout Toolbar [81] and highlight it. While highlighted Click [15] on
>   the tool. The tool is now the active tool and ready for use.

Creating an Radial Dimension using the Mouse:

1.   Click [15] anywhere on the drawing screen to designate the **Center Point** of the
        Dimension at that location.
2.   Move the mouse and Click [15] on a point you desire to be the **Chord Point.**
        This is the *Chord Point* from which an extension line will reach. The distance
        between **Center Point** and **Chord Point** will be the radius measurement value.
3.   Move the mouse and Click [15] on at a distance you wish to designate as the
        Offset Distance. This distance is the basis for offsetting the notation of the
        dimension away from the dimensioned entity.

Creating an Radial Dimension using the Property Tree:

>   To add an Radial Dimension [39] from user supplied data, do the following . Move the
>   mouse cursor beyond the right side of the drawing screen to the Tool Property Tree
>   [260].

1. Type in the **Center Point** point data into the "**Center Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept. This will position the center point for the dimension.
2. Type in the **Chord Point** point data into the "**Chord Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.
3. Type in the **Offset Distance** data into the "**Offset Distance**" Scientific Data Field [77] found on the Tool Property Tree [260] and press **Enter** to accept. The Dimension text will be positioned at this much offset distance away from chord point. On acceptance the Radial Dimension Entity [39] creation process will be complete. If in **Show Unit Mode** the resultant dimension text will have the current unit displayed.

Reset:
You have just added an Radial Dimension Entity [39] to the drawing using a Chord Point, End Point, and Offset Point. The tool has now **Reset** and you can either continue with other tools or add in another Radial Dimension Entity [38]. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

Tab: **Auto snap Enabled** key enables/disables the auto snap feature. Auto snap automatically enables the end, mid, nearest and center Entity Snap [72] features.

Show Unit will display the dimension's unit within the Dimension Entity's [33] text.

Escape (Esc): key cancels current tool and activates the default tool (Selection Tool [101]).

Space: key **Resets** this tool.

Also See:
Dimension Entity [33]
Manage Dimension Styles [221]
Grid [75]
Grid Snaps [74]
Entity Snaps [72]
Notification Bar [267]
Flyout Toolbar [81]

2.2.2.2.9.5 Diametric Dimension Tool

Entity toolbar[251] :

The **Diametric Dimension Tool** is responsible for creating Diametric Dimension Entities[41].

Creating an Diametric Dimension :
Activate Diametric Dimension Tool:

Find the **Diametric Dimension Tool** icon from the *Entity Toolbar*[251](color coded blue) or Dimension Flyout Toolbar[81] and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Creating an Diametric Dimension using the Mouse:

1. Click[15] anywhere on the drawing screen to designate the **Far Chord Point** of the Dimension at that location.
2. Move the mouse and Click[15] on a point you desire to be the **Chord Point.** This is the **Chord Point** from which an extension line will reach. The distance between **Far Chord Point** and **Chord Point** will be the diameter measurement value.
3. Move the mouse and Click[15] on at a distance you wish to designate as the **Offset Distance**. This distance is the basis for offsetting the notation of the dimension away from the dimensioned entity.

Creating an Diametric Dimension using the Property Tree:

To add an Diametric Dimension Entities[41] from user supplied data, do the following . Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree[260].

1. Type in the **Far Chord Point** point data into the "**Far Chord Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. This will position the Far Chord point for the dimension.
2. Type in the **Chord Point** point data into the "**Chord Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.
3. Type in the **Offset Distance** data into the "**Offset Distance**" Scientific Data Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. The Dimension text will be positioned at this much offset distance away from chord point. On acceptance the Diametric Dimension Entities 41 creation process will be complete. If in **Show Unit Mode** the resultant dimension text will have the current unit displayed.

Reset:
You have just added an Diametric Dimension Entities 41 to the drawing using a Chord Point, End Point, and Offset Point. The tool has now **Reset** and you can either continue with other tools or add in another Diametric Dimension Entity 41. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

*Tab***: Auto snap Enabled** key enables/disables the auto snap feature. Auto snap automatically enables the end, mid, and quadrant Entity Snap 72 features.

Show Unit will display the dimension's unit within the Dimension Entity's 33 text.

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
    Dimension Entity 33
    Manage Dimension Styles 22
    Grid 75
    Grid Snaps 74
    Entity Snaps 72
    Notification Bar 267
    Flyout Toolbar 81

2.2.2.2.9.6 Angular Dimension Tool

Entity toolbar 25 :

The ***Angular Dimension Tool*** is responsible for creating Angular Dimension Entities 42 .

Creating an Angular Dimension :
Activate Angular Dimension Tool:

Find the ***Angular Dimension Tool*** icon from the *Entity Toolbar* 25 (color coded blue) or Dimension Flyout Toolbar 81 and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Creating an Angular Dimension using the Mouse:

1. Click 15 anywhere on the drawing screen to designate the **Center Point** of the Dimension at that location.
2. Move the mouse and Click 15 on a point you desire to be the **Start Point.** This is the point from which the angle will be measured with reference to **Center Point**.
3. Move the mouse and Click 15 on a point you desire to be the **End Point.** This is the point to which the angle will be measured with reference to **Center Point**.
4. Move the mouse and Click 15 on one side of **Center Point**. The angle between **Start Point** and **End Point** will measured on this side of the **Center Point.**

Creating an Angular Dimension using the Property Tree:

To add an Angular Dimension Entities 42 from user supplied data, do the following . Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260 .

Dimension Tool Settings				?
Dim Style	Empty			∨
Autosnap Enabled	☑			
Show Unit	☑			

Angular dimension Tool				?
▷ Center Point	**480.000**	**170.000**	**0.000**	Ⓤ
Start Point	0.000	0.000	0.000	Ⓤ
End Point	0.000	0.000	0.000	Ⓤ
Offset Side Point	0.000	0.000	0.000	Ⓤ

Tool Properties

Details of the active Tool

Selection | Tool | Snaps | Drawing | Viewport

1. Type in the **Center Point** point data into the "**Center Point**" 3D Point Property Field ⌐77⌐ found on the Tool Property Tree ⌐260⌐ and press **Enter** to accept. This will position the Center Point for the dimension.
2. Type in the **Start Point** point data into the "**Start Point**" 3D Point Property Field ⌐77⌐ found on the Tool Property Tree ⌐260⌐ and press **Enter** to accept.
3. Type in the **End Point** point data into the "**End Point**" 3D Point Property Field ⌐77⌐ found on the Tool Property Tree ⌐260⌐ and press **Enter** to accept.
4. Type in the **Offset Side Point** point data into the "**Offset Side Point**" 3D Point Property Field ⌐77⌐ found on the Tool Property Tree ⌐260⌐ and press **Enter** to accept.

Reset:
You have just added an Angular Dimension Entities ⌐42⌐ to the drawing using a Chord Point, End Point, and Offset Point. The tool has now **Reset** and you can either continue with other tools or add in another Angular Dimension Entity ⌐42⌐. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

⌨ *Tab*: **Auto snap Enabled** key enables/disables the auto snap feature. Auto snap automatically enables the end, mid, nearest and center Entity Snap ⌐72⌐ features.

⌨ *Show Unit* will display the dimension's unit within the Dimension Entity's ⌐33⌐ text.

⌨ *Escape (Esc)*: key cancels current tool and activates the default tool (Selection Tool ⌐101⌐).

⌨ *Space:* key **Resets** this tool.

Also See:
Dimension Entity ⌐33⌐
Manage Dimension Styles ⌐221⌐
Grid ⌐75⌐
Grid Snaps ⌐74⌐
Entity Snaps ⌐72⌐
Notification Bar ⌐267⌐
Flyout Toolbar ⌐81⌐

2.2.2.2.10 Text Tool

Entity toolbar 25 **:**

The *Text Tool* is responsible for creating Text Entities 43.

Creating a Text:
Activate the Text Tool:

Find the *Text Tool* icon from the Entity Toolbar 25 (color coded blue) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Creating an Text using the Mouse:

1. Click 15 anywhere on the drawing screen to designate the **Position** of the Text Entity 43 at that location.
2. Move the mouse cursor beyond right side of the drawing screen to the Tool Property Tree 260 and enter **Text** data in "**Text**" property field and press **Enter.**
    If "**Specify rotation**" is selected:
        Move the mouse cursor on drawing screen and Click 15 where required rotation is set.
    If "**Specify height**" is selected:
        Move the mouse cursor on drawing screen and Click 15 where required height is set.

    Reset:
    You have just added a Text Entity 43 to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another Text Entity 43. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating an Text using the Property Tree:
To add an Text Entity 43 from user supplied data do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

1. Type in the **Position Point** point data into the "**Position**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.
2. Type in the **Text** data into the "**Text**" property found on the Tool Property Tree 260 and press **Enter** to accept.
3. Type in the **Rotation angle** data into the "**Rotation**" Scientific Data Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.
4. Type in the **Text height** data into the "**Text Height**" Scientific Data Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.

Reset:
You have just added a Text Entity 43 to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another Text Entity 43. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
Text Entity 43
Text Styles 239
Grid 75
Grid Snaps 74
Entity Snaps 72
Notification Bar 267

2.2.2.2.11   Hatch Tool

Entity toolbar 25 **:**

The *Hatch Tool* is responsible for creating Hatch Entities 45 .

Creating a Hatch:
Activate the Hatch Tool:

Find the *Hatch Tool* icon from the Entity Toolbar 25 (color coded blue) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Creating an Hatch using the Mouse:

1. Click 15 anywhere inside the region to be hatched for the Hatch Entity 45 .
2. Press **Enter to create hatch.**

Reset:
You have just added a Hatch Entity 45 to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another Hatch Entity 45 . Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Create Hatch Tool	?
Hatch Style	
Preset	Choose a P... ▾
Type	Hatch Pattern ▾
Pattern	
Source	Defined in custom pat... ▾
Name	
Description	
Is Metric	☐
Scale	**1.000**
Angle	**0.000** ◂ ° ▸
Selection Mode	Region ▾
Island Mode	Normal ▾

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

Creating an Hatch using the Property Tree:
Create Hatch tool currently doesn't support input from keyboard to specify region for Hatch Entity 45 .

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
Hatch Entity 43
Hatch Styles 60
Grid 75
Grid Snaps 74
Entity Snaps 72
Notification Bar 267

2.2.2.2.12 Poly Face Mesh Tool

Entity toolbar 251 **:**

The *Poly Face Mesh Tool* is responsible for creating Poly Face Mesh Entities 46.

Creating a Face Mesh:
Activate the Face Mesh Tool:

Find the *Poly Face Mesh Tool* icon from the Entity Toolbar 251 (color coded blue) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Creating an Face Mesh using the Mouse:

1.   Click 15 anywhere inside the region to be Face Mesh for the Poly Face Mesh Entity 46.

2.   Press **Enter to create Poly Face Mesh.**

Reset:
You have just added a Poly Face Mesh Entity 46 to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another Poly Face Mesh Entity 46. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating an Face Mesh using the Property Tree:

Create Face Mesh tool currently doesn't support input from keyboard to specify region for Poly Face Mesh Entity 46 .

Tool Options:

Escape (Esc): key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
Poly Face Mesh Entities 46
Grid 75
Grid Snaps 74
Entity Snaps 72
Notification Bar 267

2.2.2.2.13 Block Definition Tool

Entity toolbar 251 :

The *Block Definition Tool* is responsible for creating Block Definition Entities 61 .

Creating a Block Definition:
Activate the Block Definition Tool:

Find the *Block Definition Tool* icon from the Entity Toolbar 251 (color coded blue) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Selecting the Entities to create Block Definition:

This tool is a Stacked tool 15 . If there is no selection prior to the activation of this tool the Selection Tool 101 will become active and allow the selection of the entities that will be used to create block.
1.   Select Entities to be to create Block Definition Entity 61 .
2.   Press **Enter** to finish the Selection Tool 101 .

Creating an Block Definition using the Mouse:

1. ⌨ Move the mouse cursor beyond right side of the drawing screen to the Tool Property Tree 260 and enter **Block Name** data in "**Block Name**" property field and press **Enter**.
2. Click 15 anywhere on the drawing screen to designate the **Block Origin** of the Block Definition Entity 61 at that location. This **Block Origin** is considered as base point for insertion whenever this block gets inserted into drawing.

Reset:
You have just added a Block Definition Entity 61 to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another Block Definition Entity 61. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

⌨ **Creating an Block Definition using the Property Tree:**

To add an Block Definition Entity 61 from user supplied data do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

⊟ **Create Block Tool Settings**		
Mode	Retain ...	∨
⊟ **Create Block Tool**		?
▷Block Name	**MyBlock**	
Block Origin	0 0 0	Ⓤ

Tool Properties
Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

1. If do not have any entities selected, Select entities 10 that defines new block. Press **Enter** when finished with selection.
2. Type in the **Block Name** data in "**Block Name**" property field and press **Enter**.
3. Type in the **Block Origin** point data into the "**Block Origin**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.

Reset:
You have just added a Block Definition Entity 61 to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another Block Definition Entity 61. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

⌨ *Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 10).

⌨ *Space:* key **Resets** this tool.

Also See:
Block Definition Entity 61

Grid 75
Grid Snaps 74
Entity Snaps 72
Notification Bar 267

2.2.2.2.14 Block Insertion Tool

Entity toolbar 25 **:**

The *Block Insertion Tool* is responsible for creating Block Insertion Entities 46.

Creating a Block Insertion:
Activate the Block Insertion Tool:

Find the *Block Insertion Tool* icon from the Entity Toolbar 25 (color coded blue) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Creating an Block Insertion using the Mouse:

1.   If do not have Block Definition 61 selected, move the mouse cursor beyond right side of the drawing screen to the Tool Property Tree 260 and select Block Definition 61 to be inserted.
2.   Click 15 anywhere on the drawing screen to designate the **Insertion Point** of the Block Insertion Entity 46 at that location.
   If **Scale Mode** is other than "Retain original Scale":
     Move the mouse cursor on drawing screen and Click 15 where required **Scale Factors** are set.
   If **Rotate Mode** is selected:
     Move the mouse cursor on drawing screen and Click 15 where required **Rotation Angle** is set.

   Reset:
   You have just added a Block Insertion Entity 46 to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another Block Insertion Entity 46. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating an Block Insertion using the Property Tree:
To add an Block Insertion Entity 61 from user supplied data do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

1. If do not have <u>Block Definition</u> [61] selected, select <u>Block Definition</u> [61] to be inserted.
2. Type in the **Insertion** point data into the "**Insertion Point**" <u>3D Point Property Field</u> [77] found on the <u>Tool Property Tree</u> [260] and press **Enter** to accept.

   If **Scale Mode** is other than "Retain original Scale":

   Type in the **Scaling** data in "**Scale Factors**" property field and press **Enter**.

   If **Rotate Mode** is selected:

   Type in the required rotation angle in "**Rotation Angle**" <u>scientific data</u> [78] property field and press **Enter**.

Reset:
You have just added a <u>Block Insertion Entity</u> [46] to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another <u>Block Insertion Entity</u> [46]. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (<u>Selection Tool</u> [101]).

Space: key **Resets** this tool.

Also See:
<u>Block Insertion Entity</u> [46]
<u>Block Management</u> [218]
<u>Grid</u> [75]
<u>Grid Snaps</u> [74]
<u>Entity Snaps</u> [72]
<u>Notification Bar</u> [267]

2.2.2.2.15 Image Insertion Tool

Entity toolbar 25 **:**

The *Image Insertion Tool* is responsible for creating Image Insertion Entities 47.

Creating a Image Insertion:
Activate the Image Insertion Tool:

Find the *Image Insertion Tool* icon from the Entity Toolbar 25 (color coded blue) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Creating an Image Insertion using the Mouse:

1. If do not have Image Definition 63 selected, move the mouse cursor beyond right side of the drawing screen to the Tool Property Tree 260 and select Image Definition 63 to be inserted.
2. Click 15 anywhere on the drawing screen to designate the **Insertion Point** of the Image Insertion Entity 47 at that location.

If **Specify Scale** is Selected:

Move the mouse cursor on drawing screen and Click 15 where required **Scale Factors** are set.

If **Specify Rotation** is selected:

Move the mouse cursor on drawing screen and Click 15 where required **Rotation Angle** is set.

Reset:

You have just added a Image Insertion Entity 47 to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another Image Insertion Entity 47. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating an Image Insertion using the Property Tree:

To add an Image Insertion Entity 61 from user supplied data do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

Image Insertion Tool Settings	
Image Definition	Empty ▾
Show Image while inserting	☐
Show Image after inserting	☑
Unaligned	☑
Clip Image	☐
Transparent Image	☐
Preserve Aspect Ratio	☑
Specify Scale	☑
Specify Rotation	☑

Image Insertion Tool				?
▷ Image Origin	**660.000**	**300.000**	**0.000**	Ⓤ
Image Scale		1.000	1.000	
Image Rotation			0.000 ◁ ° ▷	

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

1.   If do not have Image Definition [61] selected, select Image Definition [61] to be inserted.
2.   Type in the **Insertion** point data into the "**Insertion Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.
 If **Specify Scale** is Selected:
      Type in the **Scaling** data in "**Scale Factors**" property field and press **Enter**.
 If **Specify Rotation** is selected:
        Type in the required rotation angle in "**Rotation Angle**" scientific data [78] property field and press **Enter**.

Reset:
You have just added a Image Insertion Entity [47] to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another Image Insertion Entity [47]. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool [101]).

Space: key **Resets** this tool.

Also See:
    Image Insertion Entity [47]
    Image Management [228]
    Grid [75]
    Grid Snaps [74]

Entity Snaps 72
Notification Bar 267

2.2.2.2.16 Viewport Tool

Entity toolbar 251 **:**

The *Viewport Tool* is responsible for creating Viewport Entities 49.

Creating a Viewport:
Activate the Viewport Tool:

Activate paperspace layout (Layout1 or Layout2) by clicking small arrow button on bottom edge of drawing screen or select paperspace layout from Drawing Settings 76

. Find the *Viewport Tool* icon from the Entity Toolbar 251 (color coded blue) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Creating an Viewport using the Mouse:

1. Click 15 anywhere on the drawing screen to designate the **Start Point** of the Viewport Entity 49 at that location. **Start Point** left hand top/bottom corner for viewport.
2. Click 15 anywhere on the drawing screen to designate the **Width** of the Viewport Entity 49 at that location.
3. Click 15 anywhere on the drawing screen to designate the **Height** of the Viewport Entity 49 at that location.

Reset:
You have just added a Viewport Entity 49 to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another Viewport Entity 49. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Creating an Viewport using the Property Tree:
To add an Viewport Entity 61 from user supplied data do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

1. Type in the **Start** point data into the "**Start Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.
2. Type in the **Viewport width** data into the "**Width**" Scientific Data Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.
3. Type in the **Viewport height** data into the "**Height**" Scientific Data Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.

Reset:
You have just added a Viewport Entity 49 to the drawing. The tool has now **Reset** and you can either continue with other tools or add in another Viewport Entity 49. Using **Reset** at any time while using this tool cancels all previous steps and restarts the tool.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
Viewport Entity 49
Viewport Management 242
Grid 75
Grid Snaps 74
Entity Snaps 72
Notification Bar 267

2.2.2.3   Modify Tools

hsCADCreator has eight different tools available to modify/edit drawing entities. The *Modify Tools* can be accessed by using the green colored Modify Toolbar 253, or by accessing the Modify option from the Tool Menu 97. The Modify tools primary function is to allow the modification of drawing entities through a user friendly process of steps.

1. **Translate Tool** [148] Facilitates the linear movement of an object in 3D space

2. **Rotate Tool** [198] Facilitates the rotational movement of an object about a specified location

3. **Explode Tool** [152] Facilitates the breaking down of entities into their simple components

4. **Delete Tool** [153] Facilitates the removal of selected entities from the drawing.

5. **Scale Entities Uniformly Tool** [154]: Facilitates entire created entities to be resized proportionately

6. **Scale Entities Non-uniformly Tool** [155]: Facilitates portions of created entities to be resized disproportionately

7. **Trim Tool** [157]: Facilitates the removal of unnecessary portions of a created entity

8. **Extend Tool** [158]: Facilitates the elongation of selected portions of a created entity

9. **Clone Tools** [159] : Facilitates creation of multiple copies of selected entities

    a. **Linear Clone** [163]
    b. **Planar Clone** [165]
    c. **3-Axial Clone** [168]
    d. **Radial Clone** [160]

2.2.2.3.1 Translate Tool

Modify Toolbar [253] **:**

In **hsCADCreator** the *Translate Tool* facilitates movement of one or more entire entities from one location to another.

Translating Entities:
Activate the Translate Tool:

Find the *Translate Tool* icon from the Modify Toolbar [253] (color coded green) and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the Translate tool the Selection Steps will be skipped.

Selecting the Entities to Translate:

This tool is a Stacked tool [15]. If there is no selection prior to the activation of this tool the Selection Tool [101] will become active and allow the selection of the entities that will be translated.
1.   Select Entities to be Translated.
2.   Press **Enter** to finish the Selection Tool [101].

Translate Selected Entities using Mouse:

1.   Move the mouse and Click [15] anywhere on the screen. This will be the **Base Point** from which the selected entities will move in relation to.
2.   Move the mouse until the selected entities are placed as desired and Click to finish placement. If in either **Along X-Axis Mode**, **Along Y-Axis Mode**, or **Along Z-Axis Mode** the translation is restricted to that axis.

Reset:
You have just translated the selected Entity(ies). The tool has now **Reset** and you can either continue with other tools or translate the entity(ies) again.

Translate Selected Entities using Property Tree:

To **Translate** the selected entities in a more precise manner do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree [260].

1.   Type in the **Base Point** point data into the "**Base Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.
2.   Type in the **Move To Point** point data into the "**Move To Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept. If in either **Along X-Axis Mode**, **Along Y-Axis Mode**, or **Along Z-Axis Mode** the translation is restricted to that axis.

Reset:
You have just translated the selected Entity(ies). The tool has now **Reset** and you can either continue with other tools or translate the entity(ies) again.

Tool Options:

*R*elative Distance field displays the relative distance this translate will be for the selected entities.

*A*long X-Axis mode restricts translation along the X-Axis.
*A*long Y-Axis mode restricts translation along the Y-Axis.
*A*long Z-Axis mode restricts translation along the Z-Axis.

⌨ *Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool [101]).

⌨ *Space:* key **Resets** this tool.

Also See:
Selection Tool [101]
Grid [75]
Grid Snaps [74]
Entity Snaps [72]
Notification Bar [267]

2.2.2.3.2 Rotate Tool

🖱 Modify Toolbar [253] **:** ↻

In **hsCADCreator** the *Rotate Tool* facilitates movement of entire entities about a fixed point.

Rotating Entities:

Activate the Rotate Tool:

Find the *Rotate Tool* icon ↻ from the Modify Toolbar [253] (color coded green) and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the Translate tool the Selection Steps will be skipped.

🖱 Selecting the Entities to Rotate:

This tool is a Stacked tool [15]. If there is no selection prior to the activation of this tool the Selection Tool [101] will become active and allow the selection of the entities that will be translated.
1.   Select Entities to be Rotate.
2.   Press **Enter** to finish the Selection Tool [101].

🖱 Rotate Selected Entities using Mouse:

1.   Move the mouse and Click [15] anywhere on the screen. This will be the **Base Point** from which the selected entities will move in relation to.
2.   Click [15] and hold the left mouse button down. Move the mouse while holding the left button down until the selected entities are placed as desired and release the mouse button. If in either **X-Axis Mode**, **Y-Axis Mode**, or **Z-Axis Mode** the rotation is restricted to that axis.

Reset:
You have just **Rotated** the selected Entity(ies). The tool has now **Reset** and you can either continue with other tools or rotate the entity(ies) again.

Rotate Selected Entities using Property Tree:

To **Rotate** the selected entity(ies) in a more precise manner do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

Rotate Tool Settings	
X-Axis	☐
Y-Axis	☐
Z-Axis	☑

Rotate Tool	?
▷Base Point	660.000 280.000 0.000 U
Angle	0.000 °

Tool Properties
Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

1. Type in the **Base Point** point data into the "**Base Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.
2. Type in the **Angle** data into the "**Angle**" Scientific Data Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. If in either **X-Axis Mode**, **Y-Axis Mode**, or **Z-Axis Mode** the rotation is restricted to that axis.

Reset:
You have just **Rotated** the selected Entity(ies). The tool has now **Reset** and you can either continue with other tools or rotate the entity(ies) again.

Tool Options:

X-Axis mode restricts rotation along the X-Axis.
Y-Axis mode restricts rotation along the Y-Axis.
Z-Axis mode restricts rotation along the Z-Axis.

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:

2.2.2.3.3 Explode Tool

Modify Toolbar 253 :

In **hsCADCreator** the *Explode Tool* facilitates breaking down entire entities into component parts. *Note: **Not all entities can be exploded**. Points and lines cannot be broken into component parts.* However, polylines 26 , blocks 61 , dimensions 33 , text 43 , etc. can broken down into component parts. A complete list of non explodable entities are below.

Non Explodable Entities:
Points 22
Lines 23
Arcs 28
Circle 30
Ellipse 32
Text 43

Exploding Entities:
Activate the Explode Tool:

Find the *Explode Tool* icon from the Modify Toolbar 253 (color coded green) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the Translate tool the Selection Steps will be skipped.

Selecting the Entities to Explode:
This tool is a Stacked tool 15 . If there is no selection prior to the activation of this tool the Selection Tool 101 will become active and allow the selection of the entities that will be Exploded.
1.   Select Entities to be Explode.
2.   Press **Enter** to finish the Selection Tool 101 .

Explode Selected Entities using Mouse:
1.   On the Finish of the Selection Tool 101 the current selection set will be exploded. If there was a selection prior to the activation of the **Explode Tool** then a Dialog box asking "**Explode Current Selection Set**" will be shown.   Select "**Ok"** to **Explode** the selection.

Reset:
You have just **Exploded** the selected Entity(ies). The tool has now **Reset** and you can either continue with other tools or explode the entity(ies) again.

Note: To have correct behaviour on hide/show layer and selection for block insertion entities, always make blocks from entities those are on layer "0".

Also See:

Selection Tool |101|
Grid |75|
Grid Snaps |74|
Entity Snaps |72|
Notification Bar |267|

2.2.2.3.4 Delete Tool

⬜▭ Modify Toolbar |253| **:** ⊠

In **hsCADCreator** the *Delete Tool* facilitates erasing/removing entities from a drawing.

Deleting Entities:
Activate the Delete Tool:

Find the *Delete Tool* icon ⊠ from the Modify Toolbar |253| (color coded green) and highlight it. While highlighted Click |15| on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the Translate tool the Selection Steps will be skipped.

⬜▭ Selecting the Entities to Delete:

This tool is a Stacked tool |15|. If there is no selection prior to the activation of this tool the Selection Tool |101| will become active and allow the selection of the entities that will be Exploded.
1.   Select Entities to **Delete**.
2.   Press **Enter** to finish the Selection Tool |101|.

⬜▭ Delete Selected Entities using Mouse:

1.   On the Finish of the Selection Tool |101| the current selection set will be deleted. If there was a selection prior to the activation of the **Delete Tool** then a Dialog box asking "**Delete Current Selection Set**" will be shown.   Select "**Ok**" to **Delete** the selection.

Reset:
You have just **Deleted** the selected entity(ies). The tool has now **Reset** and you can either continue with other tools or delete more entity(ies).

⌨ Delete Selected Entities using DELETE key:

1.   Press the DELETE Key with some entity(ies) selected. A Dialog box asking "**Delete Current Selection Set**" will be shown.   Select "**Ok**" to **Delete** the selection.

Reset:
You have just **Deleted** the selected entity(ies). The tool has now **Reset** and you can either continue with other tools or delete more entity(ies).

Also See:

> Selection Tool 101
> Grid 75
> Grid Snaps 74
> Entity Snaps 72
> Notification Bar 267
> Keyboard Shortcuts 27

2.2.2.3.5 Scale Entities Uniformly Tool

Modify Toolbar 253 **:**

In **hsCADCreator,** the *Scale Entities Uniformly Tool* facilitates entire entities to be resized proportionately.

Scale Entities Uniformly:
Activate the Scale Entities Uniformly Tool:

Find the *Scale Entities Uniformly Tool* icon from the Modify Toolbar 253 (color coded green) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the Scale Uniformly tool the Selection Steps will be skipped.

Selecting the Entities to Rotate:

This tool is a Stacked tool 15. If there is no selection prior to the activation of this tool the Selection Tool 101 will become active and allow the selection of the entities that will be Scaled.
1.   Select Entities to be **Scale**.
2.   Press **Enter** to finish the Selection Tool 101.

Scale Selected Entities Uniformly using Mouse:
1.   Move the mouse and Click 15 anywhere on the screen. This will be the **Base Point** from which the selected entities will be scaled from.
2.   Move the mouse until the selected entities are scaled as desired and Click 15 mouse button.

Reset:
You have just **Scaled** the selected Entity(ies) in a uniform manner. The tool has now **Reset** and you can either continue with other tools or scale the entity(ies) again.

Scale Selected Entities using Property Tree:

To **Scale** the selected entity(ies) in a more precise manner do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

1. Type in the **Base Point** point data into the "**Base Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.
2. Type in the **Scale Factor** data into the "**Scale Factor**" Scalar Field [77] found on the Tool Property Tree [260] and press **Enter** to accept. A **Scale Factor** of 1.0 is the original size of the entity(ies). A **Scale Factor** < 1.0 will decrease the size and a **Scale Factor** > 1.0 will increase the size.

Reset:
You have just **Scaled** the selected Entity(ies) in a uniform manner. The tool has now **Reset** and you can either continue with other tools or scale the entity(ies) again.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool [101]).

Space: key **Resets** this tool.

Also See:
Selection Tool [101]
Grid [75]
Grid Snaps [74]
Entity Snaps [72]
Notification Bar [267]

2.2.2.3.6 Scale Entities Non-uniformly Tool

Modify Toolbar [253] **:**

In **hsCADCreator,** the *Scale Entities Non-Uniformly Tool* facilitates entire entities to be resized disproportionately.

Scale Entities Non-Uniformly:
Activate the Scale Entities Uniformly Tool:

Find the *Scale Entities Non-Uniformly Tool* icon from the Modify Toolbar [253] (color coded green) and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the Scale Non-Uniformly tool the Selection

Steps will be skipped.

Selecting the Entities to Rotate:

This tool is a Stacked tool 15 . If there is no selection prior to the activation of this tool the Selection Tool 101 will become active and allow the selection of the entities that will be Scaled.
1. Select Entities to be **Scale**.
2. Press **Enter** to finish the Selection Tool 101 .

Scale Selected Entities Non-Uniformly using Mouse:

1. Move the mouse and Click 15 anywhere on the screen. This will be the **Base Point** from which the selected entities will be scaled from.
2. Move the mouse until the selected entity(ies) are scaled as desired along the **X-Axis** and Click 15 mouse button.
3. Move the mouse until the selected entity(ies) are scaled as desired along the **Y-Axis** and Click 15 mouse button.
4. Move the mouse until the selected entity(ies) are scaled as desired along the **Z-Axis** and Click 15 mouse button.

Reset:

You have just **Scaled** the selected Entity(ies) in a non-uniform manner. The tool has now **Reset** and you can either continue with other tools or scale the entity(ies) again.

Scale Selected Entities using Property Tree:

To **Scale** the selected entity(ies) in a more precise manner do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260 .

Non-uniform Scale Tool			**?**
▷Base Point	**660.000**	**230.000**	**0.000** U
Scale on X-axis	1.000		
Scale on Y-axis	1.000		
Scale on Z-axis	1.000		

Tool Properties

Details of the active Tool

Selection | Tool | Snaps | Drawing | Viewport

1. Type in the **Base Point** point data into the "**Base Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.
2. Type in the **X Scale Factor** data into the "**Scale on X-axis**" Scalar Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. A **Scale Factor** of 1.0 is the original size along the x-axis of the entity(ies). A **Scale Factor** < 1.0 will decrease the size and a **Scale Factor** > 1.0 will increase the size.
3. Type in the **Y Scale Factor** data into the "**Scale on Y-axis**" Scalar Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. A **Scale Factor** of 1.0 is the original size along the y-axis of the entity(ies). A **Scale Factor** < 1.0 will decrease the size and a **Scale Factor** > 1.0 will increase the size.
4. Type in the **Z Scale Factor** data into the "**Scale on Z-axis**" Scalar Field 77

found on the Tool Property Tree |260| and press **Enter** to accept. A **Scale Factor** of 1.0 is the original size along the z-axis of the entity(ies). A **Scale Factor** < 1.0 will decrease the size and a **Scale Factor** > 1.0 will increase the size.

Reset:
You have just **Scaled** the selected Entity(ies) in a non-uniform manner. The tool has now **Reset** and you can either continue with other tools or scale the entity(ies) again.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool |101|).

Space: key **Resets** this tool.

Also See:
> Selection Tool |101|
> Grid |75|
> Grid Snaps |74|
> Entity Snaps |72|
> Notification Bar |267|

2.2.2.3.7 Trim Tool

Modify Toolbar |253| **:**

In **hsCADCreator***,* the *Trim Tool* facilitates the removal of select portions of certain entities.

Trim Entities:

Activate the Trim Entity Tool:

Find the *Trim Entity Tool* icon   from the Modify Toolbar |253| (color coded green) and highlight it. While highlighted Click |15| on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the Trim tool the Selection Steps will be skipped.

Selecting the Cutting Edge Entity(ies):

This tool is a Stacked tool |15|. If there is no selection prior to the activation of this tool the Selection Tool |101| will become active and allow the selection of the entities that will be used as the cutting edge in the Trim process.
1.   Select Entities to be used as a **Cutting Edge**.
2.   Press **Enter** to finish the Selection Tool |101|.

Select Entity(ies) to Trim:

1.   Move the mouse and Click |15| on an entity you wish to trim. *Note: If a cutting edge does not cross the entity then the entity will not be trimmed.*
2.   Repeat step 1 trimming as many entities as desired.

Reset:

You have just **Trimmed** one or more Entities. The tool has now **Reset** and you can either continue with other tools or use the trim tool again.

Note: Only line entities are available to be trimmed at this time.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
Selection Tool 101
Grid 75
Grid Snaps 74
Entity Snaps 72
Notification Bar 267

2.2.2.3.8 Extend Tool

Modify Toolbar 253 **:**

In **hsCADCreator,** the *Extend Tool* facilitates the lengthening of selected entities.

Extend Entities:
Activate the Extend Entity Tool:

Find the *Extend Entity Tool* icon from the Modify Toolbar 253 (color coded green) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the Extend tool the Selection Step will be skipped.

Select the Entity(ies) to Extend:

This tool is a Stacked tool 15 . If there is no selection prior to the activation of this tool the Selection Tool 101 will become active and allow the selection of the entities that will be extended to the **Extension Line**.
1.   Select Entities to be **Extended**.
2.   Press **Enter** to finish the Selection Tool 101 .

Specify Extension line for extension:

Extend Tool

| Extension Line Start Point | 640.000 | 230.000 | 0.000 |
| Extension Line End Point | 0.000 | 0.000 | 0.000 |

Tool Properties

Details of the active Tool

Selection | Tool | Snaps | Drawing | Viewport

Extension Line: All selected entities get extended to this line. For ease of point selection, it is always useful to draw **Extension Line** first before activating this tool.

1. Move the mouse and Click [15] on start point of Extension line.
   OR
   Move the mouse cursor beyond right side of the drawing screen to the Tool Property Tree [260] and type in the **Extension Line Start Point** point data into the "**Extension Line Start Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.

2. Move the mouse and Click [15] on end point of Extension line.
   OR
   Move the mouse cursor beyond right side of the drawing screen to the Tool Property Tree [260] and type in the **Extension Line End Point** point data into the "**Extension Line End Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.

Reset:
You have just **Extended** one or more Entities. The tool has now **Reset** and you can either continue with other tools or use the Extend tool again.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool [101]).

Space: key **Resets** this tool.

Also See:
      Selection Tool [101]
      Grid [75]
      Grid Snaps [74]
      Entity Snaps [72]
      Notification Bar [267]

2.2.2.3.9 Clone Tool

Modify toolbar [253] :

The **Clone Tool** is responsible for creating a complete duplicate of a selected entity and placing the designated number entities in various formations/spacings/locations.

Clone Entities:

Activate a Clone Tool:

Find one of the **Clone Tool** icons ⬚ ⬚ ⬚ ⬚ from the Modify toolbar |253| (color coded green) and highlight it. While highlighted Click |15| on the tool. The tool is now the active tool and ready for use. This set of tools is part of a Flyout toolbar |81|.

There are currently 4 different ways to clone entities.

1. ⬚ **Radial Clone Tool** |160|

2. ⬚ **Linear Clone Tool** |163|

3. ⬚ **Planar Clone Tool** |165|

4. ⬚ **3-Axial Clone Tool** |168|

Also See:
       Flyout Toolbar |81|

2.2.2.3.9.1 Radial Clone Tool

⬚ Modify Toolbar |253| **:** ⬚

In **hsCADCreator,** the **Radial Clone Tool** allows user to create multiple copies of an entity along radius of a circle.

Radial Clone:
Activate the Radial Clone Tool:

Find the **Radial Clone Tool** icon ⬚ from the Modify Toolbar |253| (color coded green) or **Clone Flyout Toolbar** |81| and highlight it. While highlighted Click |15| on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the Radial Clone Tool the Selection Steps will be skipped.

⬚ Selecting the Entities to Clone:

This tool is a Stacked tool |15|. If there is no selection prior to the activation of this tool the Selection Tool |101| will become active and allow the selection of the entities that will be cloned.
1.   Select Entities to Clone.
2.   Press **Enter** to finish the Selection Tool |101|.

Create Radial Clone from Selected Entities using Mouse:

1. Set Clone Tool Settings:
   - **Clone Mode** needs to be set to **Radial Clone(2D)** to create radial clone**.**
   - **Axis selection** can be set to **WCS axis**(WCS z-axis as radial axis), or **UCS axis**(UCS z-axis as radial axis), or **Arbitrary axis**(Define your own radial axis).
   - If **Insert as block** is checked, the created clones are inserted as a Block Insertion Entity 46.
   - If **Delete Entity** is checked, the original entity from which clones are created gets deleted after insertion.
   - **Number of entities** specifies total number of entities in this radial clone.
   - **Angle between items** specifies angle offset between clone entities.
2. If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Move the mouse and Click 15 anywhere on the screen. This will be the **Start Point** of **Radial Axis** direction. Move the mouse and Click 15 anywhere on the screen again. This will be the **End Point** of **Radial Axis** direction.
3. Move the mouse and Click 15 anywhere on the screen. This will be **Center Point(Base Point)** of the radial clone.
4. Move the mouse and Click 15 anywhere on the screen. The **Radial Clone** will be inserted at his **Insertion Point.**

Reset:
You have just **c**reated **Radial Clone** from selected entities. The tool has now **Reset** and you can either continue with other tools or select other entities to create **Radial Clone**.

Create Radial Clone from Selected Entities using Property Tree:

To create **Radial Clone from** the selected entity(ies) in a more precise manner do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

1. Set Clone Tool Settings:
    - **Clone Mode** needs to be set to **Radial Clone(2D)** to create radial Clone**.**
    - **Axis selection** can be set to **WCS axis**(WCS z-axis as radial axis), or **UCS axis**(UCS z-axis as radial axis), or **Arbitrary axis**(Define your own radial axis).
    - If **Insert as block** is checked, the created clone is inserted as a Block Insertion Entity 46 .
    - If **Delete Entity** is checked, the original entity from which clone is created gets deleted after insertion.
    - **Number of entities** specifies total number of entities in this radial clone.
    - If **Rotate about center** is checked, the original entity is rotated about the center as it is cloneed radially.
    - **Angle between items** specifies angle offset between clone entities.
2. If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Type in **Radial Axis direction** data into the "**Radial Axis**" 3D Vector 78 found on the Tool Property Tree 260 and press **Enter** to accept.
3. Type in the **Base Point** point data into the "**Base Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.
4. Type in the **Insertion Point** point data into the "**Insertion Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.

Reset:
You have just created a **Radial Clone** from selected entities. The tool has now **Reset** and you can either continue with other tools or select other entities to create **Radial Clone**.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
> Selection Tool 101
> Grid 75
> Grid Snaps 74
> Entity Snaps 72
> Notification Bar 267

2.2.2.3.9.2 Linear Clone Tool

Modify Toolbar 253 **:**

In **hsCADCreator***,* the *Linear Clone Tool* allows user to create multiple copies of an entity along one direction.

Linear Clone:
Activate the Linear Clone Tool:

Find the *Linear Clone Tool* icon from the Modify Toolbar 253 (color coded green) or Clone Flyout Toolbar 81 and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the Linear Clone Tool the Selection Steps will be skipped.

Selecting the Entities to Clone:

This tool is a Stacked tool 15 . If there is no selection prior to the activation of this tool the Selection Tool 101 will become active and allow the selection of the entities that will be cloned.
1.   Select Entities to create clone.
2.   Press **Enter** to finish the Selection Tool 101 .

Create Linear Clone from Selected Entities using Mouse:
1.   Set Clone Tool Settings:
   - **Clone Mode** needs to be set to **Linear Clone(1D)** to create Linear clones**.**
   - **Axis selection** can be set to **WCS axis**(WCS x-axis as **Column offset direction**), or **UCS axis**(UCS x-axis as **Column offset direction**), or **Arbitrary axis**(Define your own **Column offset direction**).
   - If **Insert as block** is checked, the created clone is inserted as a Block Insertion Entity 46 .
   - If **Delete Entity** is checked, the original entity from which clone is created gets deleted after inserting clone.
   - **Row Offset** specifies unit offset distance between cloneed entities.
   - **Number of columns** specifies number of entities to create along the

direction.

2. If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Move the mouse and Click 15 anywhere on the screen. This will be the **Start Point** of **Column offset direction**. Move the mouse and Click 15 anywhere on the screen again. This will be the **End Point** of **Column offset direction**.

3. Move the mouse and Click 15 anywhere on the screen. This will be **Base Point** of the Linear clone.

4. Move the mouse and Click 15 anywhere on the screen. The **Linear Clone** will be inserted at his **Insertion Point.**

Reset:
You have just **c**reated **Linear Clone** from selected entities. The tool has now **Reset** and you can either continue with other tools or select other entities to create **Linear Clone**.

Create Linear Clone from Selected Entities using Property Tree:

To create **Linear Clone from** the selected entity(ies) in a more precise manner do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

Array Tool Settings			
Array Mode	Linear Array(1D)		
Axis selection	UCS axis		
Insert as block	☑		
Delete Entity	☐		
Row Offset	10.00 mm		
Number of columns	2		
Array Tool			?
Column offset direction	1.00	0.00	0.00	U
▷Base Point	635.65	208.50	0.00	U
Insertion Point	0.00	0.00	0.00	U
Start Point	0.00	0.00	0.00	U
End Point	0.00	0.00	0.00	U

Array Mode

Type of array to create.

Selection | Tool | Snaps | Drawing | Viewport

1. Set Clone Tool Settings:
    - **Clone Mode** needs to be set to **Linear Clone(1D)** to create Linear clone**.**
    - **Axis selection** can be set to **WCS axis**(WCS x-axis as **Column offset direction**), or **UCS axis**(UCS x-axis as **Column offset direction**), or **Arbitrary axis**(Define your own **Column offset direction**).
    - If **Insert as block** is checked, the created clone is inserted as a Block Insertion Entity 46.
    - If **Delete Entity** is checked, the original entity from which clone is created gets deleted after inserting clone.
    - **Row Offset** specifies unit offset distance between cloneed entities.

- **Number of columns** specifies number of entities to create along the direction.

2. If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Type in **Column offset direction** data into the "**Column Offset direction**" 3D Vector [78] found on the Tool Property Tree [260] and press **Enter** to accept.

3. Type in the **Base Point** point data into the "**Base Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.

4. Type in the **Insertion Point** point data into the "**Insertion Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.

Reset:

You have just **c**reated **Linear Clone** from selected entities. The tool has now **Reset** and you can either continue with other tools or select other entities to create **Linear Clone**.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool [101]).

Space: key **Resets** this tool.

Also See:

Selection Tool [101]
Grid [75]
Grid Snaps [74]
Entity Snaps [72]
Notification Bar [267]

2.2.2.3.9.3 Planar Clone Tool

Modify Toolbar [253] **:**

In **hsCADCreator,** the *Planar Clone Tool* allows user to create multiple copies of an entity along two direction.

Planar Clone:
Activate the Planar Clone Tool:

Find the *Planar Clone Tool* icon from the Modify Toolbar [253] (color coded green) or Clone Flyout Toolbar [81] and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the Planar Clone Tool the Selection Steps will be skipped.

Selecting the Entities to Clone:

This tool is a Stacked tool 15. If there is no selection prior to the activation of this tool the Selection Tool 101 will become active and allow the selection of the entities that will be cloneed.

1.   Select Entities to create clone.
2.   Press **Enter** to finish the Selection Tool 101.

Create Planar Clone from Selected Entities using Mouse:

1.   Set Clone Tool Settings:
     · **Clone Mode** needs to be set to **Planar Clone(2D)** to create Planar clone.
     · **Axis selection** can be set to **WCS axis**(WCS x-axis is **column offset direction** and y-axis is **row offset direction**), or **UCS axis**(UCS x-axis is **column offset direction** and y-axis is **row offset direction**), or **Arbitrary axis**(Define your own **column** and **row offset directions**).
     · If **Insert as block** is checked, the created clone is inserted as a Block Insertion Entity 46.
     · If **Delete Entity** is checked, the original entity from which clone is created gets deleted after inserting clone.
     · **Row Offset** specifies unit offset distance between rows.
     · **Column Offset** specifies unit offset distance between columns.
     · **Number of columns** specifies number of columns.
     · **Number of rows** specifies number of rows.
2.   If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Move the mouse and Click 15 anywhere on the screen. This will be the **Start Point** of **Column offset direction**. Move the mouse and Click 15 anywhere on the screen again. This will be the **End Point** of **Column offset direction**.
3.   If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Move the mouse and Click 15 anywhere on the screen. This will be the **Start Point** of **Row offset direction**. Move the mouse and Click 15 anywhere on the screen again. This will be the **End Point** of **Row offset direction**.
4.   Move the mouse and Click 15 anywhere on the screen. This will be **Base Point** of the Planar clone.
5.   Move the mouse and Click 15 anywhere on the screen. The **Planar Clone** will be inserted at his **Insertion Point.**

Reset:
You have just **c**reated **Planar Clone** from selected entities. The tool has now **Reset** and you can either continue with other tools or select other entities to create **Planar Clone**.

Create Planar Clone from Selected Entities using Property Tree:

To create **Planar Clone from** the selected entity(ies) in a more precise manner do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

1. Set Clone Tool Settings:
   - **Clone Mode** needs to be set to **Planar Clone(2D)** to create Planar clone**.**
   - **Axis selection** can be set to **WCS axis**(WCS z-axis as Planar axis), or **UCS axis**(UCS z-axis as Planar axis), or **Arbitrary axis**(Define your own Planar axis).
   - If **Insert as block** is checked, the created clone is inserted as a <u>Block Insertion Entity</u> 46 .
   - If **Delete Entity** is checked, the original entity from which clone is created gets deleted after inserting clone.
   - **Row Offset** specifies unit offset distance between rows.
   - **Column Offset** specifies unit offset distance between columns.
   - **Number of columns** specifies number of columns.
   - **Number of rows** specifies number of rows.
2. If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Type in **Column offset direction** data into the "**Column Offset direction**" <u>3D Vector</u> 78 found on the <u>Tool Property Tree</u> 260 and press **Enter** to accept.
3. If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Type in **Row offset direction** data into the "**Row Offset direction**" <u>3D Vector</u> 78 found on the <u>Tool Property Tree</u> 260 and press **Enter** to accept.
4. Type in the **Base Point** point data into the "**Base Point**" <u>3D Point Property Field</u> 77 found on the <u>Tool Property Tree</u> 260 and press **Enter** to accept.
5. Type in the **Insertion Point** point data into the "**Insertion Point**" <u>3D Point Property Field</u> 77 found on the <u>Tool Property Tree</u> 260 and press **Enter** to accept.

Reset:

You have just **c**reated **Planar Clone** from selected entities. The tool has now **Reset** and you can either continue with other tools or select other entities to create **Planar Clone**.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool [101]).

Space: key **Resets** this tool.

Also See:
Selection Tool [101]
Grid [75]
Grid Snaps [74]
Entity Snaps [72]
Notification Bar [267]

2.2.2.3.9.4 3-Axial CloneTool

Modify Toolbar [253] **:**

In **hsCADCreator,** the *3-Axial Clone Tool* allows user to create multiple copies of an entity along three direction.

3-Axial Clone:
Activate the 3-Axial Clone Tool:

Find the *3-Axial Clone Tool* icon   from the Modify Toolbar [253] (color coded green) or Clone Flyout Toolbar [81] and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the 3-Axial Clone Tool the Selection Steps will be skipped.

Selecting the Entities to Clone:
This tool is a Stacked tool [15]. If there is no selection prior to the activation of this tool the Selection Tool [101] will become active and allow the selection of the entities that will be cloneed.
1.   Select Entities to create clone.
2.   Press **Enter** to finish the Selection Tool [101].

Create 3-Axial Clone from Selected Entities using Mouse:
1.   Set Clone Tool Settings:
   · **Clone Mode** needs to be set to **3-Axial Clone(2D)** to create 3-Axial clone**.**
   · **Axis selection** can be set to **WCS axis**(WCS x-axis is **column offset direction** and y-axis is **row offset direction**), or **UCS axis**(UCS x-axis is

column offset direction and y-axis is **row offset direction**), or **Arbitrary axis**(Define your own **column** and **row offset directions**).

- If **Insert as block** is checked, the created clone is inserted as a <u>Block Insertion Entity</u> ⌐46⌐.
- If **Delete Entity** is checked, the original entity from which clone is created gets deleted after inserting clone.
- **Row Offset** specifies unit offset distance between rows.
- **Column Offset** specifies unit offset distance between columns.
- **Tier Offset** specifies unit offset distance between tiers.
- **Number of columns** specifies number of columns.
- **Number of rows** specifies number of rows.
- **Number of tiers** specifies number of tiers.

2. If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Move the mouse and <u>Click</u> ⌐15⌐ anywhere on the screen. This will be the **Start Point** of **Column offset direction**. Move the mouse and <u>Click</u> ⌐15⌐ anywhere on the screen again. This will be the **End Point** of **Column offset direction**.

3. If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Move the mouse and <u>Click</u> ⌐15⌐ anywhere on the screen. This will be the **Start Point** of **Row offset direction**. Move the mouse and <u>Click</u> ⌐15⌐ anywhere on the screen again. This will be the **End Point** of **Row offset direction**.

4. If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Move the mouse and <u>Click</u> ⌐15⌐ anywhere on the screen. This will be the **Start Point** of **Tier offset direction**. Move the mouse and <u>Click</u> ⌐15⌐ anywhere on the screen again. This will be the **End Point** of **Tier offset direction**.

5. Move the mouse and <u>Click</u> ⌐15⌐ anywhere on the screen. This will be **Base Point** of the 3-Axial clone.

6. Move the mouse and <u>Click</u> ⌐15⌐ anywhere on the screen. The **3-Axial Clone** will be inserted at his **Insertion Point.**

Reset:
You have just **c**reated **3-Axial Clone** from selected entities. The tool has now **Reset** and you can either continue with other tools or select other entities to create **3-Axial Clone**.

Create 3-Axial Clone from Selected Entities using Property Tree:

To create **3-Axial Clone from** the selected entity(ies) in a more precise manner do the following. Move the mouse cursor beyond the right side of the drawing screen to the <u>Tool Property Tree</u> ⌐260⌐.

1. Set Clone Tool Settings:
   - **Clone Mode** needs to be set to **3-Axial Clone(2D)** to create 3-Axial clone**.**
   - **Axis selection** can be set to **WCS axis**(WCS z-axis as 3-Axial axis), or **UCS axis**(UCS z-axis as 3-Axial axis), or **Arbitrary axis**(Define your own 3-Axial axis).
   - If **Insert as block** is checked, the created clone is inserted as a Block Insertion Entity 46 .
   - If **Delete Entity** is checked, the original entity from which clone is created gets deleted after inserting clone.
   - **Row Offset** specifies unit offset distance between rows.
   - **Column Offset** specifies unit offset distance between columns.
   - **Tier Offset** specifies unit offset distance between tiers.
   - **Number of columns** specifies number of columns.
   - **Number of rows** specifies number of rows.
   - **Number of tiers** specifies number of tiers.
2. If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Type in **Column offset direction** data into the "**Column Offset direction**" 3D Vector 78 found on the Tool Property Tree 260 and press **Enter** to accept.
3. If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Type in **Row offset direction** data into the "**Row Offset direction**" 3D Vector 78 found on the Tool Property Tree 260 and press **Enter** to accept.

4. If **Arbitrary axis** is not selected as **Axis selection** this step is skipped. Type in **Tier offset direction** data into the "**Tier Offset direction**" 3D Vector 78 found on the Tool Property Tree 260 and press **Enter** to accept.
5. Type in the **Base Point** point data into the "**Base Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.
6. Type in the **Insertion Point** point data into the "**Insertion Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.

Reset:
You have just **c**reated **3-Axial Clone** from selected entities. The tool has now **Reset** and you can either continue with other tools or select other entities to create **3-Axial Clone**.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
Selection Tool 101
Grid 75
Grid Snaps 74
Entity Snaps 72
Notification Bar 267

2.2.2.3.10 Mirror Tool

Modify Toolbar 253 **:**

In **hsCADCreator,** the *Mirror Tool* facilitates the mirroring of selected entities.

Mirror Entities:
Activate the Mirror Entity Tool:

Find the *Mirror Entity Tool* icon from the Modify Toolbar 253 (color coded green) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the Mirror tool the Selection Step will be skipped.

Select the Entity(ies) to Mirror:
This tool is a Stacked tool 15. If there is no selection prior to the activation of this tool

the Selection Tool [101] will become active and allow the selection of the entities that
will be mirrored.
1.   Select Entities to be **Mirrored**.
2.   Press **Enter** to finish the Selection Tool [101].

Specify Mirroring axis for extension:

Mirroring Axis: All selected entities get mirrored about this line. For ease of point
    selection, it is always useful to draw **Mirror Line** first before activating this tool.

1.   Move the mouse and Click [15] on first point of Mirroring Axis.
            OR

        Move the mouse cursor beyond right side of the drawing screen to the
        Tool Property Tree [260] and type in the **First Point** point data into the "**First
        Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and
        press **Enter** to accept.

2.   Move the mouse and Click [15] on end point of Extension line.
            OR

        Move the mouse cursor beyond right side of the drawing screen to the
        Tool Property Tree [260] and type in the **Second Point** point data into the "
        Second Point" 3D Point Property Field [77] found on the Tool Property Tree [260]
        and press **Enter** to accept.

    Reset:
    You have just **Mirrored** one or more Entities. The tool has now **Reset** and you can
    either continue with other tools or use the Mirror tool again.

Tool Options:

    Delete Original mode erases selected entities after they are mirrored.

    *Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool
[101]).

    Space: key **Resets** this tool.

Also See:

2.2.2.4 View Tools

Following tools are available in **hsCADCreator** for manipulating the view of created entities within model or paperspace:

1. **Zoom to Window Tool** [175]: Allows to draw a rectangle for zoom level

2. **Zoom In Tool** [176] : Allows entity (or region of entity) to be enlarged to fill the drawing screen

3. **Zoom Out Tool** [177]: Allows entity (or region of entity) to be shrunk to fill the drawing screen

4. **Zoom Extent Tool** [178]: Allows user to easily change view to cover full extents of drawing using single click

5. **Pan View Tool** [178] : Allows entire drawing screen to be translated

6. **Rotate View Tools** [179] : Tools Allowing entity movement about one or three axes

   Rotate About Eye Vector Tool [180]: Allows rotation of drawing screen about Z-axis (axis coming to/going away from user)

   Rotate About Vertical Vector Tool [181]: Allows rotation of drawing screen about Y-axis (up/down axis)

   Rotate About Horizontal Vector Tool [182]: Allows rotation of drawing screen about X-axis (left/right axis)

   Rotate View 3D Tool [183]: Allows simultaneous rotation of drawing screen about all three axes (X,Y, and Z)

7. **Preset View Snap Tools** [187] : Tools representing three-dimensional objects in two dimensions (w/ two axes)

   Top View Tool [188]: Tool representing overhead view of an object (traditional representation)

   Bottom View Tool [189] : Tool representing underside of an object

   Front (South) View Tool [189]: Tool representing front side of an object (traditional

representation)

Back (North) View Tool 190 : Tool representing back side of an object

Left Side (West) View Tool 190 : Tool representing left side of an object

Right Side (East) View Tool 191 : Tool representing the right side of an object (traditional representation)

Southwest View Tool 191 : Tool representing front-left view of an object

Southeast View Tool 192 : Tool representing front-right view of an object

Northeast View Tool 192 : Tool representing back-right view of an object

Northwest View Tool 193 : Tool representing back-left view of an object

8. **Divide Viewport Tool** 184 : Allows user to divide active Modelspace viewport vertically or horizontally.

9. **Pre-configured Viewport Tools** 186 : Various (10) screen division formats to accommodate simultaneous multiple viewing

Single Viewport 186

Two Vertical Viewports 186

Two Horizontal Viewports 186

Three - Right Viewports 186

Three - Left Viewports 186

Three - Bottom Viewports 186

Three - Top Viewports 186

Four Even Viewports 186

Four - Right Viewports 186

Four - Left Viewports 186

2.2.2.4.1 Zoom Tools

View Toolbar [253] :

Following **Zoom Tools** are available to change zoom level of current view in **hsCADCreator**:

Using the Zoom View Tools:

Activate a Zoom View Tool:

Find one of the *Zoom View Tool* icons from the View Toolbar [253] (color coded red) and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Zoom the View:
There are four different tools that will allow modification of the zoom level.

1. **Zoom to Window Tool** [175]: Allows to draw a rectangle for zoom level

2. **Zoom In Tool** [176] : Allows entity (or region of entity) to be enlarged to fill the drawing screen

3. **Zoom Out Tool** [177]: Allows entity (or region of entity) to be shrunk to fill the drawing screen

4. **Zoom Extent Tool** [178]: Allows user to easily change view to cover full extents of drawing using single click

Also See:

2.2.2.4.1.1 Zoom to Window Tool

View Toolbar [253] :

The *Zoom to Window Tool* allows for a user to designate a rectangle and the view will zoom to that user specified window.

Using the Zoom to Window Tool:

Activate the Zoom to Window Tool:

Find one of the *Zoom to Window Tool* icon from the View Toolbar [253] (color coded red) and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Use the Mouse to Zoom to Window:

1.   Click ⌐15⌐ and hold the left mouse button to start drawing the **Zoom** rectangle.
2.   Move the mouse while still holding the left button down. This will display a temporary zoom rectangle. Release the left button when the rectangle is positioned as desired. The drawing will now update with the proper zoom for the designated rectangle to fit within the view screen.

Reset:

You have just zoomed to a custom zoom window. The Tool has now **Reset** and you can choose a different tool or select another **Zoom Window**.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool ⌐101⌐)

2.2.2.4.1.2 Zoom In Tool

View Toolbar ⌐253⌐ **:**

In **hsCADCreator** the *Zoom In Tool* is used to enlarge the drawing screen.

Using the Zoom in Tool:

Activate the Zoom in Tool:

Find one of the *Zoom in Tool* icon from the View Toolbar ⌐253⌐ (color coded red) and highlight it. While highlighted Click ⌐15⌐ on the tool. The tool is now the active tool and ready for use.

Note: This tool can be utilized via the middle mouse scroll wheel ⌐15⌐ without activating this tool and without interrupting other tool processes. This usage is noted below.

Use the Mouse to Zoom In:

1.   Click ⌐15⌐ on a point within the drawing screen to zoom to that location. This point of focus will be placed at the center of the screen for easier viewing.

Reset:

You have just zoomed in. The Tool has now **Reset** and you can choose a different tool or select another point to **Zoom In** on.

Note: Zooming in ⌐176⌐ or Zooming out ⌐176⌐ too far may cause the loss of sight of the drawing grid ⌐75⌐. See the topic Grid ⌐75⌐ for more information.

Use the Mouse Scroll Wheel ⌐15⌐ to Zoom In:

1.   Use the mouse Scroll Wheel ⌐15⌐ to zoom in. Move the scroll wheel ⌐15⌐ in a counter-clockwise motion. The mouse must contain a middle mouse scroll wheel ⌐15⌐ to utilize this feature.

No Reset:

Using the Scroll Wheel to zoom does not Reset the tool. You can utilize other tools and within a tool process you may **zoom in**, zoom out [177] and pan [178] using the mouse without interrupting that tool's procedure.
Note: Zooming in [176] or Zooming out [176] too far may cause the loss of sight of the drawing grid [75]. See the topic Grid [75] for more information.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool [101])

2.2.2.4.1.3 Zoom Out Tool

View Toolbar [253] **:**

In **hsCADCreator** the ***Zoom Out Tool*** is used to shrink the view size within the drawing screen.

Using the Zoom Out Tool:

Activate the Zoom Out Tool:

Find the ***Zoom Out Tool*** icon from the View Toolbar [253] (color coded red) and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Note: This tool can be utilized via the middle mouse scroll wheel [15] without activating this tool and without interrupting other tool processes. This usage is noted below.

Use the Mouse to Zoom Out:
1.   Click [15] on a point within the drawing screen to zoom out from that location. This point of focus will be placed at the center of the screen for easier viewing.

Reset:
You have just zoomed out. The Tool has now **Reset** and you can choose a different tool or select another point to **Zoom Out** from.
Note: Zooming in [176] or Zooming out [176] too far may cause the loss of sight of the drawing grid [75]. See the topic Grid [75] for more information.

Use the Mouse **Scroll Wheel** [15] **to Zoom Out:**
1.   Use the mouse Scroll Wheel [15] to zoom out. Move the scroll wheel [15] in a clockwise motion. The mouse must contain a middle mouse scroll wheel [15] to utilize this feature.

No Reset:
Using the Scroll Wheel to zoom does not Reset the tool. You can utilize other tools and within a tool process you may zoom in [176], **zoom out** and pan [178] using the mouse without interrupting that tool's procedure.
Note: Zooming in [176] or Zooming out [176] too far may cause the loss of sight of the drawing grid [75]. See the topic Grid [75] for more information.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101)

2.2.2.4.1.4 Zoom Extent Tool

View Toolbar 253 **:**

In **hsCADCreator** the *Zoom To Extents* is used zoom in or out such that the entire drawing is visible within the drawing screen.

Using the Zoom To Extents Tool:

Using the Zoom to Extents Tool:

Find the *Zoom To Extents Tool* icon from the View Toolbar 253 (color coded red) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will display all of the drawing such that it will fit to the limits of the drawing screen.

Note: This tool will not Activate. It is a one time tool. This means you can use this tool at any time and without interrupting other tool processes.

Also See:
View Toolbar 253

2.2.2.4.2 Pan View Tool

View Toolbar 253 **:**

In **hsCADCreator** the *Pan View Tool* is used to translate the entire drawing within the drawing screen.

Using the Pan View Tool:

Activate the Pan View Tool:

Find one of the *Pan View Tool* icon from the View Toolbar 253 (color coded red) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Note: This tool can be utilized via the middle mouse button click 15 without activating this tool and without the interruption other tool processes. This usage is noted below.

Use the Mouse to Pan the View:
1. Click 15 and hold the left mouse button 15 on a point within the drawing screen. While holding the button down move the mouse and translate the entire drawing within the drawing screen to the desired location.

Reset:

You have just performed a Pan of the View. The Tool has now **Reset** and you can choose a different tool or perform another **Pan View**.
Note: It is possible to pan the drawing to where you can no longer see any of the drawing on the screen. If you cannot find the drawing use the Zoom to Extents [178] *to reset the view on the drawing.*

Use the **Middle Mouse Button** [15] **to Pan the View:**
1.   Click [15] and hold the middle mouse button [15]. While holding the button down move the mouse and translate the entire drawing within the drawing screen to the desired location. *Note: you must have a middle mouse button to utilize this feature.*

No Reset:
Using the Middle Mouse button [15] to **Pan the View** does not Reset the tool. You can utilize other tools and within a tool process you may **zoom in**, zoom out [177] and pan [178] using the mouse without interrupting that tool's procedure.
Note: It is possible to pan the drawing to where you can no longer see any of the drawing on the screen. If you cannot find the drawing use the Zoom to Extents [178] *to reset the view on the drawing.*

Tool Options:
*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool [101])

2.2.2.4.3 Rotate View Tools

View Toolbar [253] **:**

In **hsCADCreator**, entities can be rotated around each axis in a 2D or 3D plane separately, or all three simultaneously. The following tools facilitate viewing from different perspectives:

Using the Rotate View Tools:

Activate a Rotate View Tool:
Find one of the *Rotate View Tool* icons from the View Toolbar [253] (color coded red) and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Rotating the View:
There are four different tools that will allow rotation of the view.

1.   **Rotate About Eye Vector Tool** [180] : The eye vector is that which approaches (or goes away from) the user's eye. This tool rotates entities about an axis in a circular motion (similar to a pinwheel twirling on a wall).

2.   **Rotate About Vertical Vector Tool** [181] : This tool rotates entities around an up and down axis (similar to a tetherball circling an upright pole).

3.   **Rotate About Horizontal Vector Tool** [182] : This tool rotates entities around a

side to side axis (similar to a gymnast rotating about a high bar).

4. **Rotate View 3D Tool** |183| : This tool rotates entities about all three axes simultaneously (like a skydiver rolling and tumbling in a free fall--any rotational movement is possible).

Also See:
> View Toolbar |253|

2.2.2.4.3.1 Rotate About Eye Vector Tool

 View Toolbar |253| **:**

In **hsCADCreator**, the *Rotate About Eye Vector Tool* facilitates rotational movement about an axis coming to and going away from the eye of the user. (Note: depending upon the arbitrary assignment of axes this axis can either be X,Y, or Z or some other user designated vector.)

Using the Rotate About Eye Vector Tool:

Activate the Rotate About Eye Vector Tool:

Find the *Rotate About Eye Vector* |15| icon from the View Toolbar |253| (color coded red) and highlight it. While highlighted Click |15| on the tool. The tool is now the active tool and ready for use.

Use the Mouse to Rotate the View about the Eye Vector:

1. Click |15| and hold the left mouse button down. While holding the left button down move the mouse around the center of the drawing screen. As you move the mouse you will see the drawing rotate about the vector created from the center of the screen to the eye of the user or Eye Vector |15|. Release the left mouse button when the rotation is as desired.

Reset:
You have just **Rotated about the Eye Vector** |15|. The Tool has now **Reset** and you can choose a different tool or rotate about the Eye Vector |15| again.

Note: It may be helpful to watch the UCS |15| icon in the bottom left corner of the drawing screen as you rotate the entity. Any axis that is moving toward you is represented with a solid line while an axis moving away is dotted.

Tool Options:

Horizontal vector mode changes the active tool to <u>Rotate about Horizontal Vector Tool</u>|182̅|
.

Up vector mode changes the active tool to <u>Rotate about Vertical Vector Tool</u>|18ᵀ|
Eye vector mode is the mode where the view is **Rotated about Eye Vector**
Set target mode changes the active tool to <u>Rotate about 3D Tool</u>|18З̅|.
Target point depicts the point about which the rotation will occur.

*Escape (Esc)***:** key cancels current tool and activates the default tool (<u>Selection Tool</u>
|10ᵀ|)

2.2.2.4.3.2 Rotate About Vertical Vector Tool

<u>View Toolbar</u>|25З̅| **:**

In **hsCADCreator**, the *Rotate About Vertical Vector Tool* facilitates rotational movement around
an up/down axis or <u>vertical vector</u>|15̅|. (Note: depending upon the arbitrary assignment of axes this axis
can either be X,Y, Z, or some other user defined vector.)

Using the Rotate About Vertical Vector Tool:

Activate the Rotate About Vertical Vector Tool:

Find the *Rotate About Vertical Vector* icon from the <u>View Toolbar</u>|25З̅| (color
coded red) and highlight it. While highlighted <u>Click</u>|15̅| on the tool. The tool is now the
active tool and ready for use.

Use the Mouse to Rotate the View about the Vertical Vector:

1.   <u>Click</u>|15̅| and hold the left mouse button down. While holding the left button
down move the mouse from right to left for a clockwise rotation about the
<u>vertical vector</u>|15̅| or left to right for counter-clockwise rotation about the <u>vertical</u>
<u>vector</u>|15̅|. As you move the mouse you will see the drawing rotate about the
<u>vertical vector</u>|15̅|. Release the left mouse button when the rotation is as
desired.

Reset:
You have just **Rotated about the <u>Vertical Vector</u>**|15̅|. The Tool has now **Reset** and
you can choose a different tool or rotate
about the <u>vertical vector</u>|15̅| again.

Note: It may be helpful to watch the UCS 15 *icon in the bottom left corner of the drawing screen as you rotate the entity. Any axis that is moving toward you is represented with a solid line while an axis moving away is dotted.*

⊟ **Rotate View Tool**	[?]
Horizontal vector	☐
Up vector	☑
Eye vector	☐
Set target	☐
Target	344.187 148.500 0.000 [U]

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

Tool Options:

Horizontal vector mode changes the active tool to Rotate about Horizontal Vector Tool 182.

Up vector mode is the current mode for rotating the view about the vertical vector 15.
Eye vector mode changes the active tool to Rotate about Eye Vector Tool 180.
Set target mode changes the active tool to Rotate about 3D Tool 183.
Target point depicts the point about which the rotation will occur.

⌨ *Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101)

2.2.2.4.3.3 Rotate About Horizontal Vector Tool

⬚ View Toolbar 253 **:** 🔄

In **hsCADCreator**, the *Rotate About Horizontal Vector Tool* facilitates the rotational movement of an object about a side to side axis or Horizontal Vector 15. (Note: depending upon the arbitrary assignment of axes this axis can either be X,Y, or Z.)

Using the Rotate About Horizontal Vector Tool:

Activate the Rotate About Horizontal Vector Tool:

Find the *Rotate About Horizontal Vector* 15 icon 🔄 from the View Toolbar 253 (color coded red) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

⬚ Use the Mouse to Rotate the View about the Horizontal Vector:

1.   Click 15 and hold the left mouse button down. While holding the left button down move the mouse from top to bottom or bottom to top for rotation about the Horizontal Vector 15. As you move the mouse you will see the drawing rotate about the Horizontal Vector 15. Release the left mouse button when the rotation is as desired.

Reset:
You have just **Rotated about the Horizontal Vector** 15 . The Tool has now **Reset** and you can choose a different tool or rotate about the Horizontal Vector 15 again.

Note: It may be helpful to watch the UCS 15 *icon in the bottom left corner of the drawing screen as you rotate the entity. Any axis that is moving toward you is represented with a solid line while an axis moving away is dotted.*

Rotate View Tool		?
Horizontal vector	☑	
Up vector	☐	
Eye vector	☐	
Set target	☐	
Target	344.187 148.500 0.000	U

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

Tool Options:

Horizontal vector mode is the current mode for rotating the view about the Horizontal Vector 15 .
*U*p **vector** mode changes the active tool to Rotate about Vertical Vector Tool 181 .
Eye vector mode changes the active tool to Rotate about Eye Vector Tool 180 .
Set target mode changes the active tool to Rotate about 3D Tool 183 .
Target point depicts the point about which the rotation will occur.

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101)

2.2.2.4.3.4 Rotate View 3D Tool

View Toolbar 253 **:**

In **hsCADCreator**, the *Rotate View 3D Tool* facilitates the simultaneous rotational movement of an entity about all three axes with the origin at a user defined **Target**. (Note: depending upon the arbitrary assignment of axes a particular axis can be X,Y, or Z.)

Using the Rotate 3D Tool:

Activate the Rotate 3D Tool:

Find the *Rotate 3D Tool* icon from the View Toolbar 253 (color coded red) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Use the Mouse to Rotate the View in 3D about the Target:
1. Click 15 and hold the left mouse button down. While holding the left button

down move the mouse in any direction. As you move the mouse you will see the drawing rotate about the **Target**. Release the left mouse button when the rotation is as desired.

Reset:

You have just **Rotated about the Target Point**. The Tool has now **Reset** and you can choose a different tool or rotate about the Target Point again.

Note: It may be helpful to watch the UCS [15] *icon in the bottom left corner of the drawing screen as you rotate the entity. Any axis that is moving toward you is represented with a solid line while an axis moving away is dotted.*

Note: It may be necessary to reset the rotation. To reset the rotation to a known position use the Preset View Snaps [187] *or to find the draw plane* [15] *use the Align View to UCS tool* [211].

Rotate View Tool				?
Horizontal vector	☐			
Up vector	☐			
Eye vector	☐			
Set target	☐			
Target	344.187	148.500	0.000	U

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

Tool Options:

Horizontal vector mode changes the active tool to Rotate about Horizontal Vector Tool [182].

Up vector mode changes the active tool to Rotate about Vertical Vector Tool [181].

Eye vector mode changes the active tool to Rotate about Eye Vector Tool [180].

Set target mode is the current mode for rotating the view about the **Target** in 3D.

Target point depicts the point about which the rotation will occur.

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool [101])

2.2.2.4.4 Divide Viewport Tool

View Toolbar [253] **:**

In **hsCADCreator** the *Divide Viewport Tool* will custom divide the drawing screen into multiple viewports allowing you to create you own personal workspace with the views you need.

Using the Viewport Divide Tool:

Activate the Viewport Divide Tool:

Find the *Viewport Divide Tool* icon from the View Toolbar [253] (color coded red)

and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Use the Mouse Divide the Active Viewport:

1.   Move the mouse around and you will see a temporary cutting edge. When this cutting edge is positioned as desired Click 15 to divide the viewport at that location. The tool will divide the active viewport either horizontally or vertically based on the **Split Type** found in the **Viewport Divide Settings**.
Note: The Viewport Divide tool will only divide the Active Viewport. To divide a different viewport you will need to select it before using this tool.

Reset:
You have just divided this viewport. The Tool has now **Reset** and you can choose a different tool or divide the active viewport.

Divide the Active Viewport using the Property Tree:

To divide the viewport with more precise data do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

⊟ **Viewport Divide Settings**		
Split Type	Vertical	⌄
⊟ **Viewport Divide Tool**		?
▷Percentage	**90.000**	
Tool Properties		
Details of the active Tool		

Selection | Tool | Snaps | Drawing | Viewport

1.   Select the **Split Type** from the "**Split Type**" Drop down Combo. The two choices are Vertical and Horizontal.
2.   Type in the **Percentage** data into the "**Percentage**" Scalar Data Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. In **Horizontal Mode** 0% is at the bottom of the viewport and 100% is the top of the viewport. If in **Vertical Mode** 0% is located at the left and 100% is located at the right of the viewport.

Reset:
You have just divided this viewport. The Tool has now **Reset** and you can choose a different tool or divide the active viewport.

Tool Options:

Split Type is the type of cut the divide tool will make. The choices are Vertical or Horizontal.
Percentage allows the user to specify splitting percentage of the active viewport.

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool

$\boxed{101})$

2.2.2.4.5   Pre-configured Viewport Tools

View Toolbar $\boxed{253}$ **:**

In **hsCADCreator** there are ten *Pre-configured Viewport Tools* from which to choose. Each offers a different drawing screen format to aid the user in viewing multiple projections of an entity simultaneously. The following descriptions will aid you in choosing the best format for your drawing project:

Using the Viewport Divide Tool:

Activate a Pre-Configured Viewport Tool:

Find one of the *Pre-Configured Viewport Tool* icons from the View Toolbar $\boxed{253}$ (color coded red) and highlight it. While highlighted Click $\boxed{15}$ on the tool. The tool has now changed the viewport settings for your drawing.

Note: These tools will not Activate. They are one time tools. This means you can use this tool at any time and without interrupting other tool processes to change the way you work within the drawing screen.

There are ten different Pre-Configured ways to display the drawing screen broken into one or more viewports.

1. **Single Viewport Tool** : Drawing screen remains whole. Find the **Single Viewport Tool** icon <insert image> from the View Toolbar (**color coded red**) and maneuver the mouse to it. The cursor will highlight the icon and a small box around it. Right click $\boxed{15}$. Ten pre-configured viewport tools will appear in a flyout toolbar. The **Single Viewport Tool** is the **first** (default). Click $\boxed{15}$. The drawing screen viewport will remain whole.

2. **2 Vertical Viewports Tool** : The drawing screen will be divided into **two equal** viewports with a vertical divider. Select the **second** tool from the flyout toolbar. Click.

3. **2 Horizontal Viewports Tool** : The drawing screen will be divided into **two equal** viewports with a horizontal divider. Select the **third** tool from the flyout toolbar. Click.

4. **3 Right Viewports Tool** : The drawing screen will be divided into **two equal** viewports vertically. The portion on the **left** will be further divided horizontally into **two** viewports (for a total of three). Select the **fourth** tool from the flyout toolbar. Click.

5. **3 Left Viewports Tool** : The drawing screen will be divided into **two equal** viewports vertically. The portion on the **right** will be further divided horizontally into **two** viewports (for a total of three). Select the **fifth** tool from the flyout toolbar. Click.

6. **3 Bottom Viewports Tool** : The drawing screen will be divided into **two equal** viewports horizontally. The **upper** portion will be further divided vertically into **two** viewports (for a total of three). Select the **sixth** tool from the flyout toolbar. Click.

7. **3 Top Viewports Tool** : The drawing screen will be divided into **two equal** viewports horizontally. The **lower** portion will be further divided vertically into **two** viewports (for a total of three). Select the **seventh** tool from the flyout toolbar. Click.

8. **4 Even Viewports Tool** : The drawing screen will be divided into **four equal** viewports (horizontally and vertically). Select the **eighth** tool from the flyout toolbar. Click.

9. **4 Right Viewports Tool** : The drawing screen will be divided into **two equal** viewports vertically. The portion on the **left** will be further divided horizontally into **three** equal viewports (for a total of four). Select the **ninth** tool from the flyout toolbar. Click.

10. **4 Left Viewports Tool** : The drawing screen will be divided into **two equal** viewports vertically. The portion on the **right** will be further divided horizontally into **three** equal viewports (for a total of four). Select the **tenth** (or last tool) from the flyout toolbar. Click.

Also See:

2.2.2.5   Preset View Snap Tools

View Toolbar |253| **:**

In **hsCADCreator**, Viewing the drawing in the best manner possible as well as snapping to common views within 3D space make working in this drawing environment much easier. The View Snap Tools provide a quick view manipulation toolbox for this very reason.

Using the Preset View Snap Tools:

Activate a Preset View Snap Tool:

Find one of the *Preset View Snap Tool* icons from the View Toolbar |253| (color coded red) and highlight it. While highlighted Click |15| on the tool. The tool is now the active tool and ready for use. This set of tools is part of a Flyout toolbar |81|. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar |81|.

Use a Preset View Tool:

There are six different Orthographic View Tools to use for quick view manipulation

1. **Top View Tool** [188]

2. **Bottom View Tool** [189]

3. **Front (South) View Tool** [189]

4. **Back (North) View Tool** [190]

5. **Left Side (West) View Tool** [190]

6. **Right Side (East) View Tool** [191]

There are four different Isometric View Tools to use for quick view manipulation.

1. **Southwest View Tool** [191]

2. **Southeast View Tool** [192]

3. **Northeast View Tool** [192]

4. **Northwest View Tool** [193]

Also See:

2.2.2.5.1 Top View Tool

View Toolbar [253] :

In **hsCADCreator** the Orthographic Projection [15] *Top View Tool* facilitates viewing of an object from above. This traditional plan view used in architecture and building engineering can be thought of as a diagram of a room, a building, or a level (floor) of a building as if seen from a "bird's eye" view.

Using the Top View Tool:

Using the Top View Tool:

Find the *Top View Tool* icon from the View Toolbar [253] (color coded red) and highlight it. While highlighted Click [15] on the tool. The drawing screen will now refresh and will display all of the drawing such that it is looking down in a "bird's eye" view of the 3D drawing.

Note: This tool will not Activate. It is a one time tool. This means you can use this

tool at any time and without interrupting other tool processes.
Note: This tools is part of a Flyout toolbar 81. *To learn how to change tool selection on a flyout toolbar see Flyout Toolbar* 81.

Also See:
        View Toolbar 253
        Flyout toolbar 81

2.2.2.5.2 Bottom View Tool

View Toolbar 253 **:**

In **hsCADCreator** the Orthographic Projection 15 **Bottom View Tool** facilitates viewing of an object from underneath. This nontraditional plan view is used primarily in instances where additional object detail (to a top view) is necessary.

Using the Bottom View Tool:

Using the Bottom View Tool:

Find the **Bottom View Tool** icon from the View Toolbar 253 (color coded red) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will display all of the drawing such that it is looking up at the underside of 3D drawing.

Note: This tool will not Activate. It is a one time tool. This means you can use this tool at any time and without interrupting other tool processes.
Note: This tools is part of a Flyout toolbar 81. *To learn how to change tool selection on a flyout toolbar see Flyout Toolbar* 81.

Also See:
        View Toolbar 253
        Flyout toolbar 81

2.2.2.5.3 Front (South) View Tool

View Toolbar 253 **:**

In **hsCADCreator** the Orthographic Projection 15 **Front (South) View Tool** facilitates viewing of an object from its front side. This traditional orthographic view is used in architecture and building engineering and is the most critical elevation (i.e the above-ground-level front view of a building)

Using the Front (South) View Tool:

Using the Front (South) View Tool:

Find the **Front (South) View Tool** icon from the View Toolbar 253 (color coded red) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will display all of the drawing such that it is looking at the front side of a 3D drawing.

Note: This tool will not Activate. It is a one time tool. This means you can use this tool at any time and without interrupting other tool processes.

Note: This tools is part of a Flyout toolbar [81]. *To learn how to change tool selection on a flyout toolbar see Flyout Toolbar* [81].

Also See:
> View Toolbar [253]
> Flyout toolbar [81]

2.2.2.5.4 Back (North) View Tool

View Toolbar [253] :

In **hsCADCreator** the Orthographic Projection [15] *Back (North) View Tool* facilitates viewing of an object from behind. This traditional elevation (i.e. above ground level rear view of a building) used in architecture and building engineering is second only to the front elevation.

Using the Back (North) View Tool:

Using the Back (North) View Tool:

Find the *Back (North) View Tool* icon from the View Toolbar [253] (color coded red) and highlight it. While highlighted Click [15] on the tool. The drawing screen will now refresh and will display all of the drawing such that it is looking at the back side of a 3D drawing.

Note: This tool will not Activate. It is a one time tool. This means you can use this tool at any time and without interrupting other tool processes.
Note: This tools is part of a Flyout toolbar [81]. *To learn how to change tool selection on a flyout toolbar see Flyout Toolbar* [81].

Also See:
> View Toolbar [253]
> Flyout toolbar [81]

2.2.2.5.5 Left Side (West) View Tool

View Toolbar [253] :

In **hsCADCreator** the Orthographic Projection [15] *Left Side (West) View Tool* facilitates viewing of an object from its left side. This nontraditional orthographic view, however, is important in architecture and building engineering (i.e the above-ground-level left side view of a building)

Using the Left Side (West) View Tool:

Using the Left Side (West) View Tool:

Find the *Left Side (West) View Tool* icon from the View Toolbar [253] (color coded red) and highlight it. While highlighted Click [15] on the tool. The drawing screen will now refresh and will display all of the drawing such that it is looking at the Left or West side of a 3D drawing.

Note: This tool will not Activate. It is a one time tool. This means you can use this tool at any time and without interrupting other tool processes.
Note: This tools is part of a Flyout toolbar 81. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar 81.

Also See:
   View Toolbar 253
   Flyout toolbar 81

2.2.2.5.6 Right Side (East) View Tool

   View Toolbar 253 **:**

In **hsCADCreator** the Orthographic Projection 15 *Right Side (East) View Tool* facilitates viewing of an object from its right side. This traditional orthographic view is also used in architecture and building engineering (i.e the above-ground-level right side view of a building)

Using the Right Side (East) View Tool:

Using the Right Side (East) View Tool:

Find the *Right Side (East) View Tool* icon from the View Toolbar 253 (color coded red) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will display all of the drawing such that it is looking at the Right or East side of a 3D drawing.

Note: This tool will not Activate. It is a one time tool. This means you can use this tool at any time and without interrupting other tool processes.
Note: This tools is part of a Flyout toolbar 81. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar 81.

Also See:
   View Toolbar 253
   Flyout toolbar 81

2.2.2.5.7 Southwest View Tool

   View Toolbar 253 **:**

In **hsCADCreator** the Isometric Projection 15 *Southwest View Tool* facilitates viewing an object from above, as well as from the front (south) and left (west) sides. *Note: After a front view is arbitrarily assigned, four different isometric projections can be viewed as the object is rotated clockwise.*

Using the Southwest View Tool:

Using the Southwest View Tool:

Find the *Southwest View Tool* icon from the View Toolbar 253 (color coded red) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will display all of the drawing such that it is looking down at the **Front Left** or **South West** corner of a 3D drawing.

Note: This tool will not Activate. It is a one time tool. This means you can use this tool at any time and without interrupting other tool processes.
Note: This tools is part of a Flyout toolbar 81. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar 81.

Also See:
    View Toolbar 253
    Flyout toolbar 81

2.2.2.5.8 Southeast View Tool

View Toolbar 253 :

In **hsCADCreator** the Isometric Projection 15 *Southeast View Tool* facilitates viewing an object from above, as well as from the front (south) and right (east) sides. *Note: After a front view is arbitrarily assigned, four different isometric projections can be viewed as the object is rotated clockwise.*

Using the Southeast View Tool:

Using the Southeast View Tool:

Find the *Southeast View Tool* icon from the View Toolbar 253 (color coded red) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will display all of the drawing such that it is looking down at the **Front Right** or **South East** corner of a 3D drawing.

Note: This tool will not Activate. It is a one time tool. This means you can use this tool at any time and without interrupting other tool processes.
Note: This tools is part of a Flyout toolbar 81. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar 81.

Also See:
    View Toolbar 253
    Flyout toolbar 81

2.2.2.5.9 Northeast View Tool

View Toolbar 253 :

In **hsCADCreator** the Isometric Projection 15 *Northeast View Tool* facilitates viewing an object from above, as well as from the back (north) and right (east) sides. *Note: After a front view is arbitrarily assigned, four different isometric projections can be viewed as the object is rotated clockwise.*

Using the Northeast View Tool:

Using the Northeast View Tool:

Find the *Northeast View Tool* icon ⬢ from the View Toolbar [253] (color coded red) and highlight it. While highlighted Click [15] on the tool. The drawing screen will now refresh and will display all of the drawing such that it is looking down at the **Back Right** or **North East** corner of a 3D drawing.

Note: This tool will not Activate. It is a one time tool. This means you can use this tool at any time and without interrupting other tool processes.
Note: This tools is part of a Flyout toolbar [81]. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar [81].

Also See:
>	View Toolbar [253]
>	Flyout toolbar [81]

2.2.2.5.10 Northwest View Tool

View Toolbar [253] **:** ⬢

In **hsCADCreator** the Isometric Projection [15] *Northwest View Tool* facilitates viewing an object from above, as well as from the back (north) and left (west) sides. *Note: After a front view is arbitrarily assigned, four different isometric projections can be viewed as the object is rotated clockwise.*

Using the Northwest View Tool:

Using the Northwest View Tool:

Find the *Northwest View Tool* icon ⬢ from the View Toolbar [253] (color coded red) and highlight it. While highlighted Click [15] on the tool. The drawing screen will now refresh and will display all of the drawing such that it is looking down at the **Back Left** or **North West** corner of a 3D drawing.

Note: This tool will not Activate. It is a one time tool. This means you can use this tool at any time and without interrupting other tool processes.
Note: This tools is part of a Flyout toolbar [81]. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar [81].

Also See:
>	View Toolbar [253]
>	Flyout toolbar [81]

2.2.2.6 **Draw Plane Tools**

In **hsCADCreator**, following *UCS/Draw Plane Tools* facilitate the creation of entities in three-dimensional space:

1. ⊞ **Preset View Snaps** 195 : Allow access to six different planes to facilitate creation of three-dimensional drawings. The planes correspond to the six orthographic views: top, bottom, front, back, left side, and right side.

    ⊞ Snap UCS to Top View 195

    ⊞ Snap UCS to Bottom View 196

    ⊞ Snap UCS to Front (South) View 197

    ⊞ Snap UCS to Back (North) View 197

    ⊞ Snap UCS to Left (West) View 197

    ⊞ Snap UCS to Right (East) View 198

2. **Rotate UCS/UCS/Draw Plane Tools** 198 :

    Rotate UCS About X-axis Tool 199

    Rotate UCS About Y-axis Tool 201

    Rotate UCS About Z-axis Tool 202

    Rotate UCS About Line Tool 203

3. **Translate UCS/Draw Plane Tools** 205 :

    Translate UCS Origin Along X-axis Tool 206

    Translate UCS Origin Along Y-axis Tool 207

    Translate UCS Origin Along Z-axis Tool 208

    Translate UCS Origin To Point Tool 209

4. **Align UCS to Current View Tool** 210 Aligns the User Coordinate System with whatever current view the user has activated.

5. **Align Current View to UCS Tool** 211 Aligns whatever current view the user has activated with the User Coordinate System.

6. **Move UCS Origin to WCS Origin Tool** 211 Moves the User Coordinate System origin to coincide with the World Coordinate System origin.

7. **Align UCS Axis to WCS Axis Tool** 212 Aligns the User Coordinate System axis to the World Coordinate System axis.

8. **Align UCS to Selected Entities Tool** 212 Aligns the User Coordinate System such that if all selected entities are on same plane, the selected entities plane becomes X-Y plane of UCS.

2.2.2.6.1 Preset UCS View Snaps

Drawplane Toolbar|254| **:**

In **hsCADCreator** six *Preset View Snaps* are available to aid the user in creating three-dimensional drawing entities:

Using the Preset USC View Snap Tools:

Activate a Preset View Snap Tool:

Find one of the *Preset USC view Snap Tool* icons from the Drawplane Toolbar|254| (color coded purple) and highlight it. While highlighted Click|15| on the tool. The tool is now the active tool and ready for use. This set of tools is part of a Flyout toolbar|81|.

Use a Preset View Tool:

There are six different USC View snap tools to use for quick UCS manipulation

1. *Snap UCS to Top View*|195| : Traditional overhead plane

2. **Snap UCS to Bottom View**|196| *:* Underneath plane

3. **Snap UCS to Front (South) View**|197| *:* Traditional front (side) plane

4. **Snap UCS to Back (North) View**|197| *:* Back (side) plane

5. **Snap UCS to Left (West) View**|197| *:* Left (side) plane

6. **Snap UCS to Right (East) View**|198| : Traditional right (side) plane

Also See:
Draw Plane Toolbar|254|
Flyout Toolbar|81|

2.2.2.6.1.1 Snap UCS to Top View Tool

UCS Draw Plane Toolbar|254| **:**

The *Snap UCS to Top View Tool* facilitates **drawing** on an overhead plane (i.e. floor plan, map, etc.) by changing the current UCS|59| to align with overhead plane.

Using the Snap UCS to Top View Tool:

Using the Snap UCS to Top View Tool:

Find one of the ***Snap UCS to Top View Tool*** icon from the UCS Draw Plane Toolbar|254⌐ (color coded purple) and highlight it. While highlighted Click|15⌐ on the tool. The UCS|59⌐ has changed to align with overhead plane. The drawing screen will now refresh and will show the changed UCS|59⌐.

Note*: This tool will not Activate. It is a one click tool and when finished it will activate selection tool.*
Note*: This tools is part of a Flyout toolbar|81⌐. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar|81⌐.*

Also See:
  UCS Draw Plane Toolbar|254⌐

2.2.2.6.1.2 Snap UCS to Bottom View Tool

UCS Draw Plane Toolbar|254⌐ **:**

The ***Snap UCS to Bottom View Tool*** facilitates **drawing** on an underneath plane (i.e. floor plan, map, etc.) by changing the current UCS|59⌐ to align with underneath plane.

Using the Snap UCS to Bottom View Tool:

Using the Snap UCS to Bottom View Tool:

Find one of the ***Snap UCS to Bottom View Tool*** icon from the UCS Draw Plane Toolbar|254⌐ (color coded purple) and highlight it. While highlighted Click|15⌐ on the tool. The UCS|59⌐ has changed to align with underneath plane. The drawing screen will now refresh and will show the changed UCS|59⌐.

Note*: This tool will not Activate. It is a one click tool and when finished it will activate select tool.*
Note*: This tools is part of a Flyout toolbar|81⌐. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar|81⌐.*

Also See:
  UCS Draw Plane Toolbar|254⌐

2.2.2.6.1.3 Snap UCS to Front (South) View Tool

⬜️ UCS Draw Plane Toolbar 254 **:** 🔲

The *Snap UCS to Front (South) View Tool* facilitates **drawing** on an front side plane by changing the current UCS 59 to align with front side plane.

Using the Snap UCS to Front (South) View Tool:

⬜️ Using the Snap UCS to Front (South) View Tool:

Find one of the *Snap UCS to Front (South) View Tool* icon 🔲 from the UCS Draw Plane Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The UCS 59 has changed to align with front side plane. The drawing screen will now refresh and will show the changed UCS 59.

Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.
Note: This tools is part of a Flyout toolbar 81. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar 81.

Also See:
UCS Draw Plane Toolbar 254

2.2.2.6.1.4 Snap UCS to Back (North) View Tool

⬜️ UCS Draw Plane Toolbar 254 **:** 🔲

The *Snap UCS to Back (North) View Tool* facilitates **drawing** on an back side plane by changing the current UCS 59 to align with back side plane.

Using the Snap UCS to Back (North) View Tool:

⬜️ Using the Snap UCS to Back (North) View Tool:

Find one of the *Snap UCS to Back (North) View Tool* icon 🔲 from the UCS Draw Plane Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The UCS 59 has changed to align with back side plane. The drawing screen will now refresh and will show the changed UCS 59.

Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.
Note: This tools is part of a Flyout toolbar 81. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar 81.

Also See:
UCS Draw Plane Toolbar 254

2.2.2.6.1.5 Snap UCS to Left (West) View Tool

⬜️ UCS Draw Plane Toolbar 254 **:** 🔲

The *Snap UCS to Left (West) View Tool* facilitates **drawing** on an left side plane by changing the current UCS 59 to align with left side plane.

Using the Snap UCS to Left (West) View Tool:

Using the Snap UCS to Left (West) View Tool:

Find one of the *Snap UCS to Left (West) View Tool* icon from the UCS Draw Plane Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The UCS 59 has changed to align with left side plane. The drawing screen will now refresh and will show the changed UCS 59.

Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.
Note: This tools is part of a Flyout toolbar 81. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar 81.

Also See:
      UCS Draw Plane Toolbar 254

2.2.2.6.1.6   Snap UCS to Right (East) View Tool

UCS Draw Plane Toolbar 254 **:**

The *Snap UCS to Right (East) View Tool* facilitates **drawing** on an right side plane by changing the current UCS 59 to align with right side plane.

Using the Snap UCS to Right (East) View Tool:

Using the Snap UCS to Right (East) View Tool:

Find one of the *Snap UCS to Right (East) View Tool* icon from the UCS Draw Plane Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The UCS 59 has changed to align with right side plane. The drawing screen will now refresh and will show the changed UCS 59.

Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.
Note: This tools is part of a Flyout toolbar 81. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar 81.

Also See:
      UCS Draw Plane Toolbar 254

2.2.2.6.2   Rotate UCS Tools

Drawplane Toolbar 254 **:**

In **hsCADCreator** the *Rotate UCS Tools* facilitate the rotation of active UCS 59 about co-ordinate axis or about an arbitrary line. There are four tools available from which the user can choose:

Using the Rotate USC Tools:

Activate a Rotate UCS Tool:

Find one of the **Rotate UCS Tool** icons from the Drawplane Toolbar[254] (color coded purple) and highlight it. While highlighted Click[15] on the tool. The tool is now the active tool and ready for use. This set of tools is part of a Flyout toolbar[81].

Use a Rotate UCS Tool:

There are four different USC Rotation tools to for fast UCS manipulation. The tools are listed below.

1. **Rotate UCS About X-axis Tool**[199]

2. **Rotate UCS About Y-axis Tool**[201]

3. **Rotate UCS About Z-axis Tool**[202]

4. **Rotate UCS About Line Tool**[203]

Also See:
Draw Plane Toolbar[254]
Flyout Toolbar[81]

2.2.2.6.2.1 Rotate UCS About X-axis Tool

UCS Toolbar[254] :

The **Rotate UCS About X-axis Tool** facilitates rotation of active UCS[59] about X-axis by specified angle.

Using the Rotate UCS About X-axis Tool:

Activate the Rotate UCS About X-axis Tool:

Find the **Rotate UCS About X-axis Tool** icon from the UCS Toolbar[254] (color coded purple) and highlight it. While highlighted Click[15] on the tool. The tool is now the active tool and ready for use.

Note: *This tools is part of a Flyout toolbar[81]. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar[81].*

Use the Mouse to Rotate UCS About X-axis:

1. Click 15 and hold the left mouse button down. While holding the left button down move the mouse around the center of the drawing screen. As you move the mouse you will see the UCS 59 rotate about the X-axis. Release the left mouse button when the rotation is as desired.

Reset:
You have just **Rotated UCS about X-axis**. The Tool has now **Reset** and you can choose a different tool or rotate UCS 59 about X-axis again.

UCS Rotate Tool Settings				
about X-Axis	☑			
about Y-Axis	☐			
about Z-Axis	☐			
UCS Rotate Tool				?
Point on Axis	0.000	0.000	0.000	U
Point on Axis	1.000	0.000	0.000	U
Angle		**0.000** ◄ ° ►		

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

Use the keyboard to Rotate UCS About X-axis:

1. Type in the **Angle** data into the "**Included Angle**" Scientific Data Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.

Reset:
You have just **Rotated UCS about X-axis**. The Tool has now **Reset** and you can choose a different tool or rotate UCS 59 about X-axis again.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
UCS Toolbar 254
UCS 59

2.2.2.6.2.2 Rotate UCS About Y-axis Tool

UCS Toolbar |254| **:**

The *Rotate UCS About Y-axis Tool* facilitates rotation of active UCS |59| about Y-axis by specified angle.

Using the Rotate UCS About Y-axis Tool:

Activate the Rotate UCS About Y-axis Tool:

Find the **Rotate UCS About Y-axis Tool** icon from the UCS Toolbar |254| (color coded purple) and highlight it. While highlighted Click |15| on the tool. The tool is now the active tool and ready for use.

Note: *This tools is part of a Flyout toolbar |81|. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar |81|.*

Use the Mouse to Rotate UCS About Y-axis:

1.   Click |15| and hold the left mouse button down. While holding the left button down move the mouse around the center of the drawing screen. As you move the mouse you will see the UCS |59| rotate about the Y-axis. Release the left mouse button when the rotation is as desired.

Reset:
You have just **Rotated UCS about Y-axis**. The Tool has now **Reset** and you can choose a different tool or rotate UCS |59| about Y-axis again.

UCS Rotate Tool Settings			
about X-Axis	☐		
about Y-Axis	☑		
about Z-Axis	☐		
UCS Rotate Tool			?
Point on Axis	0.000	0.000	0.000	U
Point on Axis	0.000	1.000	0.000	U
Angle	**0.000** ◄ ° ►		

Tool Properties
Details of the active Tool

Selection | Tool | Snaps | Drawing | Viewport

Use the keyboard to Rotate UCS About Y-axis:

1.   Type in the **Angle** data into the "**Included Angle**" Scientific Data Field |77| found on the Tool Property Tree |260| and press **Enter** to accept.

Reset:
You have just **Rotated UCS about Y-axis**. The Tool has now **Reset** and you can choose a different tool or rotate UCS |59| about Y-axis again.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
       UCS Toolbar 254
       UCS 59

2.2.2.6.2.3   Rotate UCS About Z-axis Tool

UCS Toolbar 254 **:**

The ***Rotate UCS About Z-axis Tool*** facilitates rotation of active UCS 59 about Z-axis by specified angle.

Using the Rotate UCS About Z-axis Tool:

Activate the Rotate UCS About Z-axis Tool:

Find the **Rotate UCS About Z-axis Tool** icon from the UCS Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Note*: This tools is part of a Flyout toolbar 81. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar 81.*

Use the Mouse to Rotate UCS About Z-axis:

1.   Click 15 and hold the left mouse button down. While holding the left button down move the mouse around the center of the drawing screen. As you move the mouse you will see the UCS 59 rotate about the Z-axis. Release the left mouse button when the rotation is as desired.

Reset:
You have just **Rotated UCS about Z-axis**. The Tool has now **Reset** and you can choose a different tool or rotate UCS 59 about Z-axis again.

UCS Rotate Tool Settings		
about X-Axis	☐	
about Y-Axis	☐	
about Z-Axis	☑	

UCS Rotate Tool				?
Point on Axis	0.000	0.000	0.000	U
Point on Axis	0.000	0.000	1.000	U
Angle		**0.000** ◄	°	►

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

⌨ Use the keyboard to Rotate UCS About Z-axis:

1. Type in the **Angle** data into the "**Included Angle**" Scientific Data Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.

Reset:
You have just **Rotated UCS about Z-axis**. The Tool has now **Reset** and you can choose a different tool or rotate UCS 59 about Z-axis again.

Tool Options:

⌨ *Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

⌨ *Space:* key **Resets** this tool.

Also See:
UCS Toolbar 254
UCS 59

2.2.2.6.2.4 Rotate UCS About Line Tool

⌨ UCS Toolbar 254 **:** 🗘

The *Rotate UCS About Line Tool* facilitates rotation of active UCS 59 about a given Line by specified angle.

Using the Rotate UCS About Line Tool:

Activate the Rotate UCS About Line Tool:

Find the **Rotate UCS About Line Tool** icon 🗘 from the UCS Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The tool is now

the active tool and ready for use.

Note: This tools is part of a _Flyout toolbar_ |81|. To learn how to change tool selection on a flyout toolbar see _Flyout Toolbar_ |81|.

Use the Mouse to Rotate UCS About Line:

1. _Click_ |15| anywhere on screen to specify the first point on rotate about line.
2. _Click_ |15| anywhere on screen to specify the second point on rotate about line.
3. _Click_ |15| and hold the left mouse button down. While holding the left button down move the mouse around the center of the drawing screen. As you move the mouse you will see the _UCS_ |59| rotate about the Z-axis. Release the left mouse button when the rotation is as desired.

Reset:
You have just **Rotated UCS about Line**. The Tool has now **Reset** and you can choose a different tool or rotate _UCS_ |59| about Line again.

	UCS Rotate Tool Settings				
	about X-Axis	☐			
	about Y-Axis	☐			
	about Z-Axis	☐			
	UCS Rotate Tool				❓
▷	Point on Axis	0.000	0.000	0.000	Ⓤ
▷	Point on Axis	0.000	0.000	1.000	Ⓤ
	Angle			0.000 ◁ ° ▷	

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

Use the keyboard to Rotate UCS About Line:

1. Type in the **Start Point** point data into the "**Point on Axis**" 3D Point Property Field |77| found on the Tool Property Tree |260| and press **Enter** to accept.
2. Type in the **End Point** point data into the "**Point on Axis**" 3D Point Property Field |77| found on the Tool Property Tree |260| and press **Enter** to accept.
3. Type in the **Angle** data into the "**Included Angle**" Scientific Data Field |77| found on the Tool Property Tree |260| and press **Enter** to accept.

Reset:
You have just **Rotated UCS about Line**. The Tool has now **Reset** and you can choose a different tool or rotate _UCS_ |59| about Line again.

Tool Options:

Escape (Esc): key cancels current tool and activates the default tool (Selection Tool |101|).

Space: key **Resets** this tool.

Also See:

UCS Toolbar 254

UCS 59

2.2.2.6.3 Translate UCS Tools

Drawplane Toolbar 254 :

In **hsCADCreator** the *Translate UCS Tools* facilitate translation (movement) of all active UCS along co-ordinate axis or to an arbitrary point. There are four tools available from which the user can choose:

Using the Translate USC Tools:

Activate a Rotate UCS Tool:

Find one of the *Rotate UCS Tool* icons from the Drawplane Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use. This set of tools is part of a Flyout toolbar 81.

Use a Translate UCS Tool:

There are four different USC Translation tools to for fast UCS manipulation. The tools are listed below.

1. **Translate UCS Origin Along X-axis Tool** 206

2. **Translate UCS Origin Along Y-axis Tool** 207

3. **Translate UCS Origin Along Z-axis Tool** 208

4. **Translate UCS Origin To Point Tool** 209

Also See:

Draw Plane Toolbar 254

Flyout Toolbar 81

2.2.2.6.3.1 Translate UCS Origin Along X-axis Tool

UCS Toolbar 254 **:**

The *Translate UCS Origin Along X-axis Tool* facilitates translation/movement of active UCS 59 origin along X-axis by specified distance.

Using the Translate UCS Origin Along X-axis Tool:

Activate the Translate UCS Origin Along X-axis Tool:

Find the **Translate UCS Origin Along X-axis Tool** icon from the UCS Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Note: This tools is part of a Flyout toolbar 81. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar 81.

Use the Mouse to Translate UCS Origin Along X-axis:

1. Click 15 anywhere on screen when desired distance is achieved for translation.

Reset:

You have just **Translated UCS along X-axis**. The Tool has now **Reset** and you can choose a different tool or translate UCS 59 along X-axis again.

Use the keyboard to Translate UCS Origin Along X-axis:

1. Type in the **Distance** data into the "**Distance**" Scientific Data Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.

Reset:

You have just **Translated UCS along X-axis**. The Tool has now **Reset** and you can choose a different tool or translate UCS 59 along X-axis again.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
> UCS Toolbar 254
> UCS 59

2.2.2.6.3.2 Translate UCS Origin Along Y-axis Tool

UCS Toolbar 254 **:**

The *Translate UCS Origin Along Y-axis Tool* facilitates translation/movement of active UCS 59 origin along Y-axis by specified distance.

Using the Translate UCS Origin Along Y-axis Tool:

Activate the Translate UCS Origin Along Y-axis Tool:

Find the **Translate UCS Origin Along Y-axis Tool** icon from the UCS Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Note: *This tools is part of a Flyout toolbar 81. To learn how to change tool selection on a flyout toolbar see Flyout Toolbar 81.*

Use the Mouse to Translate UCS Origin Along Y-axis:
1.   Click 15 anywhere on screen when desired distance is achieved for translation.

 Reset:
You have just **Translated UCS along Y-axis**. The Tool has now **Reset** and you can choose a different tool or translate UCS 59 along Y-axis again.

UCS Translate Tool Settings	
on X-Axis	☐
on Y-Axis	☑
on Z-Axis	☐

UCS Translate Tool				?
▷Distance		**260.000** ◀ mm ▶		
New Origin	0.000	0.000	0.000	U

Tool Properties

Details of the active Tool

Selection	Tool	Snaps	Drawing	Viewport

Use the keyboard to Translate UCS Origin Along Y-axis:

1.   Type in the **Distance** data into the "**Distance**" Scientific Data Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.

Reset:
You have just **Translated UCS along Y-axis**. The Tool has now **Reset** and you can choose a different tool or translate UCS 59 along Y-axis again.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
UCS Toolbar 254
UCS 59

2.2.2.6.3.3 Translate UCS Origin Along Z-axis Tool

UCS Toolbar 254 **:**

The *Translate UCS Origin Along Z-axis Tool* facilitates translation/movement of active UCS 59 origin along Z-axis by specified distance.

Using the Translate UCS Origin Along Z-axis Tool:

Activate the Translate UCS Origin Along Z-axis Tool:

Find the **Translate UCS Origin Along Z-axis Tool** icon from the UCS Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Note: This tools is part of a Flyout toolbar 81 . To learn how to change tool selection on a flyout toolbar see Flyout Toolbar 81 .

Use the Mouse to Translate UCS Origin Along Z-axis:

1.   Click 15 anywhere on screen when desired distance is achieved for translation.

Reset:
You have just **Translated UCS along Z-axis**. The Tool has now **Reset** and you can choose a different tool or translate UCS 59 along Z-axis again.

UCS Translate Tool Settings

on X-Axis	☐
on Y-Axis	☐
on Z-Axis	☑

UCS Translate Tool [?]

| ▷Distance | **0.000** ◀ mm ▶ |
| New Origin | 0.000 | 0.000 | 0.000 | Ⓤ |

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

Use the keyboard to Translate UCS Origin Along Z-axis:

1. Type in the **Distance** data into the "Distance" Scientific Data Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.

Reset:

You have just **Translated UCS along Z-axis**. The Tool has now **Reset** and you can choose a different tool or translate UCS [59] along Z-axis again.

Tool Options:

Escape (Esc): key cancels current tool and activates the default tool (Selection Tool [101]).

Space: key **Resets** this tool.

Also See:
UCS Toolbar [254]
UCS [59]

2.2.2.6.3.4 Translate UCS Origin to Point Tool

UCS Toolbar [254] **:** ⊕

The *Translate UCS Origin to Point Tool* facilitates translation/movement of active UCS [59] origin to a specified point.

Using the Translate UCS Origin to Point Tool:

Activate the Translate UCS Origin to Point Tool:

Find the **Translate UCS Origin to Point Tool** icon ⊕ from the UCS Toolbar [254] (color coded purple) and highlight it. While highlighted Click [15] on the tool. The tool is now the active tool and ready for use.

Note: This tools is part of a Flyout toolbar [81]. To learn how to change tool selection

on a flyout toolbar see Flyout Toolbar 81 .

Use the Mouse to Translate UCS Origin to Point:

1.   Click 15 anywhere on screen where user wants to move active UCS 59 origin.

   Reset:

You have just **Translated UCS to Point**. The Tool has now **Reset** and you can choose a different tool or translate UCS 59 to Point again.

UCS Translate Tool Settings	
on X-Axis	☐
on Y-Axis	☐
on Z-Axis	☐
UCS Translate Tool	?
Distance	0.000 mm
▷New Origin	660.000 290.000 0.000 Ⓤ

Tool Properties

Details of the active Tool

| Selection | Tool | Snaps | Drawing | Viewport |

Use the keyboard to Translate UCS Origin to Point:

1.   Type in the **Origin Point** point data into the "**New Origin**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.

   Reset:

You have just **Translated UCS to Point**. The Tool has now **Reset** and you can choose a different tool or translate UCS 59 to Point again.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:

     UCS Toolbar 254
     UCS 59

2.2.2.6.4   Align UCS to Current View Tool

UCS Draw Plane Toolbar 254 **:**

The *Align UCS to Current View Tool* allows user to align active UCS 59 to Current View such that UCS 59 X-Y plane is aligned with view's Horizontal and Vertical axis respectively.

Using the Align UCS to Current View Tool:

Using the Align UCS to Current View Tool:

Find one of the **Align UCS to Current View Tool** icon from the UCS Draw Plane Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The UCS 59 has changed to align with view plane. The drawing screen will now refresh and will show the changed UCS 59.

Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.

Also See:
UCS Draw Plane Toolbar 254

2.2.2.6.5 Align Current View to UCS Tool

UCS Draw Plane Toolbar 254 :

The **Align Current View to UCS Tool** allows user to change Current View such that view's Horizontal and Vertical axis are aligned with UCS 59 X-Y plane respectively.

Using the Align Current View to UCS Tool:

Using the Align Current View to UCS Tool:

Find one of the **Align Current View to UCS Tool** icon from the UCS Draw Plane Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The Current View has changed to align with active UCS 59. The drawing screen will now refresh and will show the changed view of the modelspace.

Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.

Also See:
UCS Draw Plane Toolbar 254

2.2.2.6.6 Move UCS Origin to WCS Origin Tool

UCS Draw Plane Toolbar 254 :

The **Move UCS Origin to WCS Origin Tool** allows user to move active UCS 59 origin point to World Coordinate System(WCS) Origin point (0, 0, 0).

Using the Move UCS Origin to WCS Origin Tool:

Using the Move UCS Origin to WCS Origin Tool:

Find one of the **Move UCS Origin to WCS Origin Tool** icon from the UCS Draw Plane Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will show the changed UCS 59.

Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.

Also See:
UCS Draw Plane Toolbar 254

2.2.2.6.7 Align UCS Axis to WCS Axis Tool

UCS Draw Plane Toolbar 254 **:**

The *Align UCS Axis to WCS Axis Tool* allows user to align all axis of active UCS 59 to corresponding axis of World Coordinate System(WCS).

Using the Align UCS Axis to WCS Axis Tool:

Using the Align UCS Axis to WCS Axis Tool:

Find one of the *Align UCS Axis to WCS Axis Tool* icon from the UCS Draw Plane Toolbar 254 (color coded purple) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will show the changed UCS 59.

Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.

Also See:
UCS Draw Plane Toolbar 254

2.2.2.6.8 Align UCS to Selected Entities

UCS Draw Plane Toolbar 254 **:**

The *Align UCS to Selected Entities Tool* facilitates aligning of the User Coordinate System such that if all selected entities are on same plane, the selected entities plane becomes X-Y plane of UCS 59.

Align UCS to Selected Entities:

Activate the Align UCS to Selected Entities Tool:

Find the *Extend Entity Tool* icon from the UCS Draw Plane Toolbar 254 (color coded green) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Note: If there is a selection prior to activating the Align UCS to Selected Entities Tool, the Selection Step will be skipped.

Select the Entity(ies) to define UCS Plane:
This tool is a Stacked tool 15. If there is no selection prior to the activation of this tool the Selection Tool 101 will become active and allow the selection of the entities that will be used to define plane.
1.   Select Entities to define plane.
2.   Press **Enter** to finish the Selection Tool 101. The tool tries to find the plane

defined by selected entities. If all selected entities are on same plane then the tool aligns active UCS 59 to this plane.

Reset:
You have just changed active UCS 59. The tool has now **Reset** and you can either continue with other tools or use the Align UCS to Selected Entities Tool again.

Tool Options:

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101).

Space: key **Resets** this tool.

Also See:
UCS Draw Plane Toolbar 254

2.2.2.7   Rendering Tools

In **hsCADCreator,** Eight *Rendering Tools* are available to the user to render drawings in different ways:

1. ▢ **2D Wireframe Tool** 213

⊞ 3D Wireframe Tool 214

◨ Hidden Tool 214

◪ Flat Shaded Tool 214

◪ Gouraud Shaded Tool 215

⊡ Flat Shaded With Edges Tool 216

◪ Gouraud Shaded With Edges Tool 216

8. ↻ **Regenerate Tool** 217

2.2.2.7.1 2D Wireframe Tool

Rendering Toolbar 256 **:** ▢

Using the 2D Wireframe Tool:

Using the 2D Wireframe Tool:

Find the **2D Wireframe Tool** icon ☐ from the Rendering Toolbar 256 (color coded teal) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will display 2D wireframe model for all solid entities.

Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.

Also See:
Rendering Toolbar 256

2.2.2.7.2 3D Wireframe Tool

☐ Rendering Toolbar 256 **:** 田

Using the 3D Wireframe Tool:

Using the 3D Wireframe Tool:

Find the **3D Wireframe Tool** icon 田 from the Rendering Toolbar 256 (color coded teal) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will display 3D wireframe model for all solid entities.

Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.

Also See:
Rendering Toolbar 256

2.2.2.7.3 Hidden Tool

☐ Rendering Toolbar 256 **:** ▢

Using the Hidden Tool:

Using the Hidden Tool:

Find the **Hidden Tool** icon ▢ from the Rendering Toolbar 256 (color coded teal) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will display model such that all lines that are behind any solid face are hidden from view.
Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.

Also See:
Rendering Toolbar 256

2.2.2.7.4 Flat ShadedTool

In **hsCADCreator** the **Flat Shaded Tool** alters the 3D rendering mode. Flat shading is lighting technique used in 3D computer graphics. It shades each polygon of an object based on the angle

between the polygon's surface normal and the direction of the light source, their respective colors and the intensity of the light source. It is usually used for high speed rendering where more advanced shading techniques are too computationally expensive.

The disadvantage of flat shading is that it gives low-polygon models a faceted look. Sometimes this look can be advantageous though, such as in modeling boxy objects. Artists sometimes use flat shading to look at the polygons of a solid model they are creating. More advanced and realistic lighting and shading techniques include Gouraud shading and Phong shading.

This tool incorporates the flat shading technique but **without** visible object edge lines.

Rendering Toolbar 256 :

Using the Flat Shaded Tool:

Using the Flat Shaded Tool:

Find the *Flat Shaded Tool* icon from the Rendering Toolbar 256 (color coded teal) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will display modelspace entities with Flat Shaded mode.
Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.

Also See:
Rendering Toolbar 256

2.2.2.7.5 Gouraud ShadedTool

In **hsCADCreator** the *Gouraud Shaded Tool* alters the 3D rendering mode. Gouraud shading is a method used in computer graphics to simulate the differing effects of light and color across the surface of an object. In practice, Gouraud shading is used to achieve smooth lighting on low-polygon surfaces without the heavy computational requirements of calculating lighting for each pixel.

This tool incorporates the Gouraud shading technique but **without** visible object edge lines.

Rendering Toolbar 256 :

Using the Gouraud Shaded Tool:

Using the Gouraud Shaded Tool:

Find the *Gouraud Shaded Tool* icon from the Rendering Toolbar 256 (color coded teal) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will display modelspace entities with Gouraud Shaded mode.
Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.

Also See:
Rendering Toolbar 256

2.2.2.7.6 Flat Shaded With EdgesTool

In **hsCADCreator** the *Flat Shaded With Edges Tool* alters the 3D rendering mode. Flat shading is lighting technique used in 3D computer graphics. It shades each polygon of an object based on the angle between the polygon's surface normal and the direction of the light source, their respective colors and the intensity of the light source. It is usually used for high speed rendering where more advanced shading techniques are too computationally expensive.

The disadvantage of flat shading is that it gives low-polygon models a faceted look. Sometimes this look can be advantageous though, such as in modeling boxy objects. Artists sometimes use flat shading to look at the polygons of a solid model they are creating. More advanced and realistic lighting and shading techniques include Gouraud shading and Phong shading.

This tool incorporates the flat shading technique **with** visible object edge lines.

Rendering Toolbar 256 :

Using the Flat Shaded With Edges Tool:

Using the Flat Shaded With Edges Tool:

Find the *Flat Shaded With Edges Tool* icon from the Rendering Toolbar 256 (color coded teal) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will display modelspace entities with Flat Shaded With Edges mode.
Note: This tool will not Activate. It is a one click tool and when finished it will activate select tool.

Also See:
Rendering Toolbar 256

2.2.2.7.7 Gouraud Shaded With EdgesTool

In **hsCADCreator** the *Gouraud Shaded With Edges Tool* alters the 3D rendering mode. Gouraud shading is a method used in computer graphics to simulate the differing effects of light and color across the surface of an object. In practice, Gouraud shading is used to achieve smooth lighting on low-polygon surfaces without the heavy computational requirements of calculating lighting for each pixel.

This tool incorporates the Gouraud shading technique **with** visible object edge lines.

Rendering Toolbar 256 :

Using the Gouraud Shaded With Edges Tool:

Using the Gouraud Shaded With Edges Tool:

Find the *Gouraud Shaded With Edges Tool* icon from the Rendering Toolbar 256 (color coded teal) and highlight it. While highlighted Click 15 on the tool. The drawing screen will now refresh and will display modelspace entities with Gouraud Shaded With Edges mode.
Note: This tool will not Activate. It is a one click tool and when finished it will activate

select tool.

Also See:

2.2.2.7.8 Regenerate Tool

In **hsCADCreator** the *Regenerate Tool* initiates a refreshment of the drawing in progress.

: ⟳

Using the Regenerate Tool:

Using the Regenerate Tool:

Find the *Regenerate Tool* icon ⟳ from the Rendering Toolbar|256| (color coded
teal) and highlight it. While highlighted Click |15| on the tool. The drawing screen will
now redraw all the entities visible on the screen.
*Note: This tool will not Activate. It is a one click tool and when finished it will activate
select tool.*

Also See:

2.2.2.8 Library Tools

In **hsCADCreator,** *Library Tools* allow easy creation/edition/deletion of stored Objects|50|. There
are tools available to edit each type of Objects|50|.

1. 🗂 *Manage Layers Tool*|230|

2. 🗂 *Quick Layer View Tool*|243|

3. 🅱 *Manage Blocks Tool*|218|

4. 🏠 *Manage Image Definitions Tool*|228|

5. ⊕ *Manage Colors Tool*|220|

6. T *Manage Text Styles Tool*|239|

7. ☰ *Manage Linetypes Tool*|233|

8. ⫽ *Manage Multi Line Styles Tool*|235|

9. ▨ *Manage Hatch Styles Tool*|224|

10. ⤡ *Manage Dimension Styles Tool*|221|

11. ***Manage User Coordinate Systems Tool*** 241

12. ***Manage External References Tool*** 223

13. ***Manage Layouts Tool*** 232

  Manage Plot Settings Tool 238

14. ***Manage Viewports Tool*** 242

15. ***Manage Named Views Tool*** 236

2.2.2.8.1 Manage Blocks Tool

Library Toolbar 257 **:**

Manage Blocks Tool is used to edit/delete Block Definition Objects 61 . See detail description for different properties of Block Definition Objects 61 **.**

Using the Manage Blocks Tool:

Find one of the ***Manage Blocks Tool*** icon from the Library Toolbar 257 (color coded peach) and highlight it. While highlighted Click 15 on the tool. Manage Blocks Dialog will show up and allows editing of Block Definition Objects 61 . After performing required operation in manage dialog, left click 15 on OK button to save it to drawing. If Cancel button is left clicked 15 , all the changes performed since opening of this manage dialog are undone 89 .

Select Block Definition:

To select a block definition, Left click [15] on one of the block names from the list displayed on left hand side of manage dialog.

View/Edit Block Definitions:

To view properties of block definition [61], select a block definition as described in **Select Block Definition**. The **Preview** section will show small view of selected block definition. All properties for selected block definition are displayed on right hand side of manage dialog. **Action Buttons** available for selected block definition will be displayed under **Actions** section below the block name list.

Add Block Definitions:

To add block definitions [61] to drawing, Block Definition Tool [140] is used. The Manage Block dialog does not allow to add new block definitions. Manage block dialog allows editing of block definition's properties and deletion of block definitions.

Delete Block Definitions:

To delete block definitions [61] from drawing, select a block definition as described in **Select Block Definition** section. Left click [15] on delete button(⊟) under **Actions** section. The selected block definition will be deleted. Delete button(⊟) becomes unavailable when there is no block definition selected or selected block definition is **hsCADCreator**'s default created block definition.

Also See:

2.2.2.8.2 Manage Colors Tool

Library Toolbar 257 **:**

Manage Colors Tool is used to create/edit/delete Color Objects 51. See detail description for different properties of Color Objects 51**.**

Using the Manage Colors Tool:

Find one of the *Manage Colors Tool* icon from the Library Toolbar 257 (color coded peach) and highlight it. While highlighted Click 15 on the tool. Manage Colors Dialog will show up and allows editing of Color Objects 51. After performing required operation in manage dialog, left click 15 on OK button to save it to drawing. If Cancel button is left clicked 15, all the changes performed since opening of this manage dialog are undone 89.

Select Color:
To select a Color 51, Left click 15 on one of the color names from the list displayed on left hand side of manage dialog.

View/Edit Colors:
To view properties of Color 51, select a color as described in **Select Color**. The **Preview**

section will show the selected color as it will appear in drawing. All properties for selected color are displayed on right hand side of manage dialog. **Action Buttons** available for selected color will be displayed under **Actions** section below the color names list.

⊞ Add Colors:

To add Color⌐51⌐ to drawing, Left click⌐15⌐ on add button (⊞) under **Actions**. This opens a standard dialog to choose colors. After selecting required color click on OK and a new color is added to drawing's color library. By default the Name property of color object is set to R,G,B value for newly created color.

⊟ Delete Colors:

To delete Color⌐51⌐ from drawing, select a color as described in **Select Color** section. Left click⌐15⌐ on delete button(⊟) under **Actions** section. The selected color will be deleted. Delete button(⊟) becomes unavailable when there is no color selected or selected color is **hsCADCreator**'s default created color.

▤ Import Colors from a drawing:

To import Color⌐51⌐ from a drawing, Left click⌐15⌐ on import button(▤) under **Actions** section. This opens a standard file select dialog. After selecting required drawing file click on OK and colors existing in selected drawing file will be added to current drawings color library.

Also See:
> Color Objects⌐51⌐
> Library Toolbar⌐257⌐

2.2.2.8.3 Manage Dimension Styles Tool

▭ Library Toolbar⌐257⌐ **:** ◺

Manage Dimension Styles Tool is used to create/edit/delete Dimension Style Objects⌐51⌐. See detail description for different properties of Dimension Style Objects⌐51⌐.

▭ Using the Manage Dimension Styles Tool:

Find one of the **Manage Dimension Styles Tool** icon ◺ from the Library Toolbar⌐257⌐ (color coded peach) and highlight it. While highlighted Click⌐15⌐ on the tool. Manage Dimension Styles Dialog will show up and allows editing of Dimension Style Objects⌐51⌐. After performing required operation in manage dialog, left click⌐15⌐ on OK button to save it to drawing. If Cancel button is left clicked⌐15⌐, all the changes performed since opening of this manage dialog are undone⌐89⌐.

Select Dimension Style:

To select a Dimension Style 51, Left click 15 on one of the dimension style names from the list displayed on left hand side of manage dialog.

View/Edit Dimension Styles:

To view properties of Dimension Style 51, select a dimension style as described in **Select Dimension Style**. The **Preview** section will show the selected dimension style as it will appear in drawing. All properties for selected dimension style are displayed on right hand side of manage dialog. **Action Buttons** available for selected dimension style will be displayed under **Actions** section below the dimension style names list.

Add Dimension Styles:

To add Dimension Style 51 to drawing, Left click 15 on add button (⊞) under **Actions**. This creates a new dimension style with default parameters and adds it to the dimension style library.

Delete Dimension Styles:

To delete Dimension Style 51 from drawing, select a dimension style as described in **Select Dimension Style** section. Left click 15 on delete button(⊟) under **Actions** section. The selected dimension style will be deleted. Delete button(⊟) becomes unavailable when there is no dimension style selected.

Also See:

Dimension Style Objects 51

Library Toolbar 257

2.2.2.8.4 Manage External References Tool

Library Toolbar 257 :

Manage External References Tool is used to create/edit/delete External Reference Objects 60 . See detail description for different properties of External Reference Objects 60 .

Using the Manage External References Tool:

Find one of the *Manage External References Tool* icon from the Library Toolbar 257 (color coded peach) and highlight it. While highlighted Click 15 on the tool. Manage External References Dialog will show up and allows editing of External Reference Objects 60 . After performing required operation in manage dialog, left click 15 on OK button to save it to drawing. If Cancel button is left clicked 15 , all the changes performed since opening of this manage dialog are undone 89 .

Manage

T External References

Preview

No Preview

doughnut

External Reference ?

Is Nested	✓
Status	Resolved
Name	**doughnut**
Block	doughnut

Object Properties

Choose one or more items from the list

Actions:

OK   Cancel   Apply   Help

Select External Reference:
To select an External Reference 60 , Left click 15 on one of the external reference names from the list displayed on left hand side of manage dialog.

View External References:
To view properties of External Reference 60 , select an external reference as described in **Select External Reference**. The **Preview** area will show the selected external reference

as it will appear in drawing. All properties for selected external reference are displayed on right hand side of manage dialog. **Action Buttons** available for selected external reference will be displayed under **Actions** section below the external reference names list.

Attach/Add External References:

To attach/add Underline External Reference 51 to drawing, Left click 15 on attach button ( ) under **Actions**. This opens a standard file select dialog. After selecting required drawing file, click on OK. A new external reference with selected file name will be added drawing's library.

Detach/Delete External References:

To detach/delete External Reference 51 from drawing, select an external reference as described in **Select External Reference**. Left click 15 on detach button( ) under

Actions section. The selected external reference will be deleted. Detach button( ) becomes unavailable when there is no external reference selected or selected external reference is **hsCADCreator**'s default created external reference.

Load External References:

To load/reload an existing External Reference 51, select a external reference as described in **Select External Reference**. Left click 15 on load button( ) under **Actions** section. The selected external reference will be loaded into drawing's memory.

Unload External References:

To unload an existing External Reference 51 from drawing's memory, select an external reference as described in **Select External Reference**. Left click 15 on unload button( ) under **Actions** section. The selected external reference will be unloaded from drawing's memory.

Bind External Reference as Block Definition 61:

To bind an existing External Reference 51 as Block Definition 61, select an external reference as described in **Select External Reference**. Left click 15 on bind button( ) under **Actions** section. The selected external reference will be removed from external references library and binded into current drawing as a block definition. Binding an external reference as block definition creates a new block definition with external reference's name. Any block definition's within the external reference are also added as new block definitions to current drawing with this naming convention: *ExternalReferenceName$ BlockNameInExternalReference*.

Also See:
External Reference Objects 60
Library Toolbar 257

2.2.2.8.5 Manage Hatch Styles Tool

Library Toolbar 257 **:**

Manage Hatch Pattern Tool is used to create/edit/delete Hatch Pattern Objects 61. See detail description for different properties of Hatch Pattern Objects 61.

Manage Hatch Styles:

Activate a Manage Hatch Style Dialog:

Find the ***Manage Hatch Style Tool*** icons ▨ from the Library Toolbar 257 (color coded peach) and highlight it. While highlighted Click 15 on the tool. The tool will activate the **Manage Hatch Style** dialog..

There are two different types of **Hatch Styles.** The two types are listed below. Visit the links to get a more in depth look at the hatch styles.

1. ▨ ***Manage Hatch Gradient*** 225

2. ▨ ***Manage Hatch Pattern*** 227

2.2.2.8.5.1 Manage Hatch Gradient

▭ Library Toolbar 257 **:** ▨

Manage Hatch Gradient Tool is used to create/edit/delete Hatch Gradient Objects 60 **.** See detail description for different properties of Hatch Gradient Objects 60 **.**

▭ Using the Manage Hatch Gradient Tool:

Find one of the ***Manage Hatch Gradient Tool*** icon ▨ from the Library Toolbar 257 (color coded peach) and highlight it. While highlighted Click 15 on the tool. Manage dialog will be displayed. Click 15 on the Hatch Gradients tab and it allows editing of Hatch Gradient Objects 60. After performing required operation in manage dialog, left click 15 on OK button to save it to drawing. If Cancel button is left clicked 15, all the changes performed since opening of this manage dialog are undone 89.

Select Hatch Gradient:

To select a Hatch Gradient 51, Left click 15 on one of the hatch gradient names from the list displayed on left hand side of manage dialog.

View/Edit Hatch Gradients:

To view properties of Hatch Gradient 51, select a hatch gradient as described in **Select Hatch Gradient**. The **Preview** section will show the selected hatch gradient as it will appear in drawing. All properties for selected hatch gradient are displayed on right hand side of manage dialog. **Action Buttons** available for selected hatch gradient will be displayed under **Actions** section below the hatch gradient names list.

Add Hatch Gradients:

To add Hatch Gradient 51 to drawing, Left click 15 on add button (⊞) under **Actions**. This adds a new hatch gradient to drawing's hatch gradient library. By default, the Name property of hatch gradient object is set to "Gradient*Number"*.

Delete Hatch Gradients:

To delete Hatch Gradient 51 from drawing, select a hatch gradient as described in **Select Hatch Gradient** section. Left click 15 on delete button(⊟) under **Actions** section. The selected hatch gradient will be deleted. Delete button(⊟) becomes unavailable when there is no hatch gradient selected or selected hatch gradient is **hsCADCreator**'s default created hatch gradient.

Import Hatch Gradients from a gradient file (.GRA file):

To import Hatch Gradient 51 from a file, Left click 15 on import button( ) under **Actions** section. This opens a standard file select dialog. After selecting required gradient file click on OK and gradients existing in selected file will be added to current drawings hatch gradient library.

Export Hatch Gradients to a gradient file (.GRA file):

To export Hatch Gradient 51 to a file, Left click 15 on export button( ) under **Actions** section. This opens a standard file save dialog. After entering required gradient file name, click on OK and selected gradients information will be saved in .GRA file.

Also See:
Hatch Gradient Objects 60
Library Toolbar 257

2.2.2.8.5.2   Manage Hatch Pattern

Library Toolbar 257 **:**

Manage Hatch Pattern Tool is used to create/edit/delete Hatch Pattern Objects 61. See detail description for different properties of Hatch Pattern Objects 61.

Using the Manage Hatch Pattern Tool:

Find one of the *Manage Hatch Pattern Tool* icon from the Library Toolbar 257 (color coded peach) and highlight it. While highlighted Click 15 on the tool. Manage Hatch Pattern Dialog will show up and allows editing of Hatch Pattern Objects 61. After performing required operation in manage dialog, left click 15 on OK button to save it to drawing. If Cancel button is left clicked 15, all the changes performed since opening of this manage dialog are undone 89.

Select Hatch Pattern:

To select a Hatch Pattern 51 , Left click 15 on one of the hatch pattern names from the list displayed on left hand side of manage dialog.

View/Edit Hatch Patterns:

To view properties of Hatch Pattern 51 , select a hatch pattern as described in **Select Hatch Pattern**. The **Preview** section will show the selected hatch pattern as it will appear in drawing. All properties for selected hatch pattern are displayed on right hand side of manage dialog. **Action Buttons** available for selected hatch pattern will be displayed under **Actions** section below the hatch pattern names list.

Import Hatch Patterns from a pattern file (.PAT file):

To import Hatch Pattern 51 from a pattern file, Left click 15 on import button( ) under **Actions** section. This opens a standard file select dialog. After selecting required pattern file click on OK and pattern will be added to current drawings hatch pattern library.

Also See:

Hatch Pattern Objects 61
Library Toolbar 257

2.2.2.8.6 Manage Image Definitions Tool

Library Toolbar 257 **:**

Manage Image Definitions Tool is used to create/edit/delete Hatch Pattern Objects 61 . See detail

description for different properties of Hatch Pattern Objects 61 .

Using the Manage Image Definitions Tool:

Find one of the ***Manage Image Definitions Tool*** icon 🏠 from the Library Toolbar 257 (color coded peach) and highlight it. While highlighted Click 15 on the tool. Manage Image Definitions Dialog will show up and allows editing of Hatch Pattern Objects 61 . After performing required operation in manage dialog, left click 15 on OK button to save it to drawing. If Cancel button is left clicked 15 , all the changes performed since opening of this manage dialog are undone 89 .

Manage	✕

🏠 Image Definitions

Preview

Hachisoft Corporation

CIMG2048.JPG
graph.jpg
hsLetterHead.jpg

⊟ **Image Definition**	?	
Image Resolution	0.001	0.001
Is Loaded	☑	
Size of image	1545.000	306.000
Source File	C:\Documents and Settings\Developer\...	... *
Resolution Units	Undefined	⌄

Object Properties

Choose one or more items from the list

Actions: ➕ ➖

OK	Cancel	Apply	Help

Select Image Definition:

To select an Image Definition 51 , Left click 15 on one of the image definition names from the list displayed on left hand side of manage dialog.

View/Edit Image Definitions:

To view properties of Image Definition 51 , select an image definition as described in **Select Image Definition**. The **Preview** section will show the selected image definition as it will appear in drawing. All properties for selected image definition are displayed on right hand side of manage dialog. **Action Buttons** available for selected image definition will be displayed under **Actions** section below the image definition names list.

➕ Add Image Definitions:

To add new Image Definition 51 to drawing's library, Left click 15 on add button (➕) under **Actions**. This opens a standard dialog to choose image files. After selecting

required file, click on OK and a new image definition is added to drawing's image definition library. By default the Name property of image definition object is set to name of the source file.

Delete Image Definitions:

To delete Image Definition 51 from drawing's library, select an image definition as described in **Select Image Definition**. Left click 15 on delete button( ) under **Actions** section. The selected image definition will be deleted. Delete button( ) becomes unavailable when there is no image definition selected.

Also See:
Hatch 61 Pattern Objects 61
Library Toolbar 257

2.2.2.8.7 Manage Layers Tool

Library Toolbar 257 :

Manage Layers Tool is used to create/edit/delete Layer Objects 64. See detail description for different properties of Layer Objects 64.

Using the Manage Layers Tool:

Find one of the *Manage Layers Tool* icon from the Library Toolbar 257 (color coded peach) and highlight it. While highlighted Click 15 on the tool. Manage Layers Dialog will show up and allows editing of Layer Objects 64. After performing required operation in manage dialog, left click 15 on OK button to save it to drawing. If Cancel button is left clicked 15, all the changes performed since opening of this manage dialog are undone 89.

Select Layer:

To select a Layer⌐51¬, Left click⌐15¬ on one of the layer names from the list displayed on left hand side of manage dialog.

View/Edit Layers:

To view properties of Layer⌐51¬, select a layer as described in **Select Layer**. All properties for selected layer are displayed on right hand side of manage dialog. **Action Buttons** available for selected layer will be displayed under **Actions** section below the layer names list.

⊞ Add Layers:

To add new Layer⌐51¬ to drawing, Left click⌐15¬ on add button (⊞) under **Actions**. This adds a new layer to drawing's layer library. By default the Name property of layer object is set to Layer*Number*.

⊟ Delete Layers:

To delete Layer⌐51¬ from drawing, select a layer as described in **Select Layer**. Left click⌐15¬ on delete button(⊟) under **Actions** section. The selected layer will be deleted. Delete button(⊟) becomes unavailable when there is no layer selected or selected layer is **hsCADCreator**'s default created layer.

Also See:

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2.2.2.8.8 Manage Layouts Tool

Library Toolbar 257 **:**

Manage Layouts Tool is used to create/edit/delete Layout Objects 64. See detail description for different properties of Layout Objects 64.

Using the Manage Layouts Tool:

Find one of the *Manage Layouts Tool* icon from the Library Toolbar 257 (color coded peach) and highlight it. While highlighted Click 15 on the tool. Manage Layouts Dialog will show up and allows editing of Layout Objects 64. After performing required operation in manage dialog, left click 15 on OK button to save it to drawing. If Cancel button is left clicked 15, all the changes performed since opening of this manage dialog are undone 89.

Select Layout:

To select a Layout 51, Left click 15 on one of the layout names from the list displayed on left hand side of manage dialog.

View/Edit Layouts:

To view properties of Layout 51, select a layout as described in **Select Layout**. The **Preview** section will show the selected layout as it will appear in drawing. All properties for selected layout are displayed on right hand side of manage dialog. **Action Buttons** available for selected layout will be displayed under **Actions** section below the layout

names list.

⊞ Add Layouts:

To add new <u>Layout</u> 51 to drawing, <u>Left click</u> 15 on add button (⊞) under **Actions**. This adds new layout to drawing's layout library. By default the Name property of layout object is set to Layout*Number*.

⊟ Delete Layouts:

To delete <u>Layout</u> 51 from drawing, select a layout as described in **Select Layout**. <u>Left click</u> 15 on delete button(⊟) under **Actions** section. The selected layout will be deleted. Delete button(⊟) becomes unavailable when there is no layout selected or selected layout is **hsCADCreator**'s default created layout.

▢ Import Layouts from print settings:

To import <u>Layout</u> 51 from a print settings, <u>Left click</u> 15 on import button(▢) under **Actions** section. This opens an object selection dialog. After selecting required print settings click on OK and new layout will be imported into drawings layout library.

Also See:
<u>Layout Objects</u> 64
<u>Library Toolbar</u> 257

2.2.2.8.9 Manage Linetypes Tool

<u>Library Toolbar</u> 257 **:** ⌨

Manage Linetypes Tool is used to create/edit/delete <u>Linetypes Objects</u> 66. See detail description for different properties of <u>Linetypes Objects</u> 66.

Using the Manage Linetypes Tool:

Find one of the *Manage Linetypes Tool* icon ⌨ from the <u>Library Toolbar</u> 257 (color coded peach) and highlight it. While highlighted <u>Click</u> 15 on the tool. Manage Linetypes Dialog will show up and allows editing of <u>Linetypes Objects</u> 66. After performing required operation in manage dialog, <u>left click</u> 15 on OK button to save it to drawing. If Cancel button is <u>left clicked</u> 15, all the changes performed since opening of this manage dialog are <u>undone</u> 89.

Select Linetype:

To select a <u>Linetype</u> 51 , <u>Left click</u> 15 on one of the linetype names from the list displayed on left hand side of manage dialog.

View/Edit Linetypes:

To view properties of <u>Linetype</u> 51 , select a linetype as described in **Select Linetype**. The **Preview** section will show the selected linetype as it will appear in drawing. All properties for selected linetype are displayed on right hand side of manage dialog. **Action Buttons** available for selected linetype will be displayed under **Actions** section below the linetype names list.

Add Linetypes:

To add <u>Linetype</u> 51 to drawing, <u>Left click</u> 15 on add button (⊞) under **Actions**. This adds a new linetype to drawing's linetype library. By default the Name property of linetype object is set to Line Type*Number.*

Delete Linetypes:

To delete <u>Linetype</u> 51 from drawing, select a linetype as described in **Select Linetype** section. <u>Left click</u> 15 on delete button(⊟) under **Actions** section. The selected linetype will be deleted. Delete button(⊟) becomes unavailable when there is no linetype selected or selected linetype is **hsCADCreator**'s default created linetype.

Import Linetypes from a linetype file (.LIN file):

To import <u>Linetype</u> 51 from a linetype file, <u>Left click</u> 15 on import button( ) under

Actions section. This opens a standard file select dialog. After selecting required linetype file click on OK and linetypes existing in selected file will be added to current drawings linetype library.

Also See:
Linetypes Objects 66
Library Toolbar 257

2.2.2.8.10 Manage Multi Line Styles

Library Toolbar 257 :

Manage Multi Line Styles Tool is used to create/edit/delete Multi Line Styles Objects 67. See detail description for different properties of Multi Line Styles Objects 67.

Using the Manage Multi Line Styles Tool:

Find one of the ***Manage Multi Line Styles Tool*** icon from the Library Toolbar 257 (color coded peach) and highlight it. While highlighted Click 15 on the tool. Manage Multi Line Styles Dialog will show up and allows editing of Multi Line Styles Objects 67. After performing required operation in manage dialog, left click 15 on OK button to save it to drawing. If Cancel button is left clicked 15, all the changes performed since opening of this manage dialog are undone 89.

Manage

Multi-line Styles

Preview		Multi-Line Style	?
		Name	**Standard**
No Preview		Description	
		Fill Color	☑
Standard		Fill color	☐ By Layer ⌄
Multi-Line_Style1		Start Angle	90.000 °
Multi-Line_Style2		End Angle	90.000 °
Multi-Line_Style3		Miters	☐
Multi-Line_Style4		Start Inner Arcs	☐
Multi-Line_Style5		Start Round Caps	☐
		Start Square Caps	☐
		End Inner Arcs	☐
		End Round Caps	☐

Object Properties

Choose one or more items from the list

Actions: ⊞ ⊟

| OK | Cancel | Apply | Help |

Select Multi Line:
To select a Multi Line 51, Left click 15 on one of the multi line names from the list

displayed on left hand side of manage dialog.

View Multi Lines:

To view properties of Multi Line [51], select a multi line as described in **Select Multi Line**. The **Preview** section will show the selected multi line as it will appear in drawing. All properties for selected multi line are displayed on right hand side of manage dialog. **Action Buttons** available for selected multi line will be displayed under **Actions** section below the multi line names list.

Add Multi Lines:

To add Multi Line [51] to drawing, Left click [15] on add button (⊞) under **Actions**. This adds a new multi line to drawing's multi line library. By default the Name property of multi line object is set to Multi-Line_Style*Number*.

Delete Multi Lines:

To delete Multi Line [51] from drawing, select a multi line as described in **Select Multi Line** section. Left click [15] on delete button(⊟) under **Actions** section. The selected multi line will be deleted. Delete button(⊟) becomes unavailable when there is no multi line selected or selected multi line is **hsCADCreator**'s default created multi line.

Also See:
> Multi Line Styles Objects [67]
> Library Toolbar [257]

2.2.2.8.11 Manage Named Views Tool

Library Toolbar [257] **:**

Manage Named Views Tool is used to create/edit/delete View Objects [70]. See detail description for different properties of View Objects [70].

Using the Manage Named Views Tool:

Find one of the ***Manage Named Views Tool*** icon from the Library Toolbar [257] (color coded peach) and highlight it. While highlighted Click [15] on the tool. Manage Named Views Dialog will show up and allows editing of View Objects [70]. After performing required operation in manage dialog, left click [15] on OK button to save it to drawing. If Cancel button is left clicked [15], all the changes performed since opening of this manage dialog are undone [89].

Select View:

To select a View⌐51⌐, Left click⌐15⌐ on one of the view names from the list displayed on left hand side of manage dialog.

View Views:

To view properties of View⌐51⌐, select a view as described in **Select View** section. The **Preview** section will show the selected view as it will appear in drawing. All properties for selected view are displayed on right hand side of manage dialog. **Action Buttons** available for selected view will be displayed under **Actions** section below the view names list.

⊞ Add Views:

To add new View⌐51⌐ to drawing, Left click⌐15⌐ on add button (⊞) under **Actions**. This adds a new view to drawing's view library. By default the Name property of view object is set to View*Number*.

⊟ Delete Views:

To delete View⌐51⌐ from drawing, select a view as described in **Select View** section. Left click⌐15⌐ on delete button(⊟) under **Actions** section. The selected view will be deleted. Delete button(⊟) becomes unavailable when there is no view selected.

Import/Add active view from   drawing:

To add/import View⌐51⌐ from current drawing, Left click⌐15⌐ on add current button(⊞) under **Actions** section. This creates a new view from currently active view in working drawing and adds it to drawing's library.

Set view active in current drawing:

To set active View⌐51⌐ in current drawing, select a view as described in **Select View.** Left click⌐15⌐ on set active button(   ) under **Actions** section. This will make currently selected view active in working drawing.

Also See:
> View Objects⌐70⌐
> Library Toolbar⌐257⌐

2.2.2.8.12 Manage Plot Settings Tool

Library Toolbar⌐257⌐ **:**

Manage Plot Settings Tool is used to create/edit/delete Plot Setting Objects⌐68⌐. See detail description for different properties of Plot Setting Objects⌐68⌐.

Using the Manage Plot Settings Tool:

Find one of the ***Manage Plot Settings Tool*** icon   from the Library Toolbar⌐257⌐ (color coded peach) and highlight it. While highlighted Click⌐15⌐ on the tool. Manage Plot Settings Dialog will show up and allows editing of Plot Setting Objects⌐68⌐. After performing required operation in manage dialog, left click⌐15⌐ on OK button to save it to drawing. If Cancel button is left clicked⌐15⌐, all the changes performed since opening of this manage dialog are undone⌐89⌐.

Select Plot Settings:

To select a Plot Settings [51], Left click [15] on one of the plot settings names from the list displayed on left hand side of manage dialog.

View Plot Settings:

To view properties of Plot Settings [51], select a plot settings as described in **Select Plot Settings** section. The **Preview** section will show the selected plot settings as it will appear in drawing. All properties for selected plot settings are displayed on right hand side of manage dialog. **Action Buttons** available for selected plot settings will be displayed under **Actions** section below the plot settings names list.

Add Plot Settings:

To add Plot Settings [51] to drawing, Left click [15] on add button ( ) under **Actions**. This adds a new plot settings to drawing's plot settings library. By default the Name property of plot settings object is set to Plot Settings*Number.*

Delete Plot Settings:

To delete Plot Settings [51] from drawing, select a plot settings as described in **Select Plot Settings** section. Left click [15] on delete button( ) under **Actions** section. The selected plot settings will be deleted. Delete button( ) becomes unavailable when there is no plot settings selected or selected plot settings is **hsCADCreator**'s default created plot settings.

Import Plot Settings from a drawing:

To import Plot Settings [51] from an existing layout [64], Left click [15] on import button( ) under **Actions** section. This opens an object selection dialog. After selecting required layout click on OK and plot settings from selected layout will be added to current drawings plot settings library.

Also See:
Plot Setting Objects [68]
Library Toolbar [257]

2.2.2.8.13 Manage Text Styles Tool

Library Toolbar [257] : T

Manage Text Styles Tool is used to create/edit/delete Text Style Objects [69]. See detail description for different properties of Text Style Objects [69].

Using the Manage Text Styles Tool:

Find one of the ***Manage Text Styles Tool*** icon T from the Library Toolbar [257] (color coded peach) and highlight it. While highlighted Click [15] on the tool. Manage Text Styles Dialog will show up and allows editing of Text Style Objects [69]. After performing required operation in manage dialog, left click [15] on OK button to save it to drawing. If Cancel button is left clicked [15], all the changes performed since opening of this manage dialog are undone [89].

Select Text Style:

To select a Text Style 51, Left click 15 on one of the text style names from the list displayed on left hand side of manage dialog.

View Text Styles:

To view properties of Text Style 51, select a text style as described in **Select Text Style** section. The **Preview** section will show the selected text style as it will appear in drawing. All properties for selected text style are displayed on right hand side of manage dialog. **Action Buttons** available for selected text style will be displayed under **Actions** section below the text style names list.

Add Text Styles:

To add Text Style 51 to drawing, Left click 15 on add button (＋) under **Actions**. This adds a new text style to drawing's text style library. By default the Name property of text style object is set to TextStyle*Number.*

Delete Text Styles:

To delete Text Style 51 from drawing, select a text style as described in **Select Text Style** section. Left click 15 on delete button(▬) under **Actions** section. The selected text style will be deleted. Delete button(▬) becomes unavailable when there is no text style selected or selected text style is **hsCADCreator**'s default created text style.

Import Text Styles from a font file (.TTF file):

To import Text Style 51 from a font file, Left click 15 on import button(TTF) under **Actions**

section. This opens a standard file select dialog. After selecting required font file click on OK and text styles existing in selected font file will be displayed in object selection dialog. Select required text styles and click on Import. Selected text styles will be added to current drawings text style library.

Also See:
> Text Style Objects 69
> Library Toolbar 257

2.2.2.8.14 Manage User Coordinate Systems Tool

Library Toolbar 257 **:**

Manage User Coordinate Systems Tool is used to create/edit/delete User Coordinate System Objects 59 . See detail description for different properties of User Coordinate System Objects 59 .

Using the Manage User Coordinate Systems Tool:

Find one of the *Manage User Coordinate Systems Tool* icon   from the Library Toolbar 257 (color coded peach) and highlight it. While highlighted Click 15 on the tool. Manage User Coordinate Systems Dialog will show up and allows editing of User Coordinate System Objects 59 . After performing required operation in manage dialog, left click 15 on OK button to save it to drawing. If Cancel button is left clicked 15 , all the changes performed since opening of this manage dialog are undone 89 .

Select User Coordinate System:
To select a User Coordinate System 51 , Left click 15 on one of the user coordinate

systems names from the list displayed on left hand side of manage dialog.

View/Edit User Coordinate Systems:

To user coordinate systems properties of User Coordinate System 51, select a user coordinate systems as described in **Select User Coordinate System** section. The **Preview** section will show the selected user coordinate systems as it will appear in drawing. All properties for selected user coordinate systems are displayed on right hand side of manage dialog. **Action Buttons** available for selected user coordinate systems will be displayed under **Actions** section below the user coordinate systems names list.

Add User Coordinate Systems:

To add new User Coordinate System 51 to drawing, Left click 15 on add button ( ) under **Actions**. This adds a new user coordinate systems to drawing's user coordinate systems library. By default the Name property of user coordinate systems object is set to User Coordinate System*Number*.

Delete User Coordinate Systems:

To delete User Coordinate System 51 from drawing, select a user coordinate systems as described in **Select User Coordinate System** section. Left click 15 on delete button( ) under **Actions** section. The selected user coordinate systems will be deleted. Delete button( ) becomes unavailable when there is no user coordinate systems selected.

Import/Add active user coordinate systems from   drawing:

To add/import User Coordinate System 51 from current drawing, Left click 15 on add current button( ) under **Actions** section. This creates a new user coordinate systems from currently active user coordinate systems in working drawing and adds it to drawing's library.

Set user coordinate systems active in current drawing:

To set active User Coordinate System 51 in current drawing, select a user coordinate systems as described in **Select User Coordinate System.** Left click 15 on set active button( ) under **Actions** section. This will make currently selected user coordinate systems active in working drawing.

Also See:
    User Coordinate System Objects 59
    Library Toolbar 257

2.2.2.8.15 Manage Viewports Tool

Library Toolbar 257 **:**

Manage Viewports Tool is used to create/edit/delete Viewport Objects 70. See detail description for different properties of Viewport Objects 70.

Using the Manage Viewports Tool:

Find one of the ***Manage Viewports Tool*** icon from the Library Toolbar 257 (color coded peach) and highlight it. While highlighted Click 15 on the tool. Manage Viewports Dialog will show up and allows editing of Viewport Objects 70. After performing required operation in manage dialog, left click 15 on OK button to save it to drawing. If Cancel button is left clicked 15, all the changes performed since opening of this manage dialog are undone 89.

Manage

Modelspace Viewports

Preview

No Preview

*Active

Named Object	?
Name	*Active
Xref Dependent	☐
Xref Resolved	☐

Camera Settings	
Back Clip Distance	0.000
Back Clip Enabled	☐
Center Point	0.000	0.000
Elevation	0.000
Clip at Eye	☑
Front Clip Distance	0.000
Front Clip Enabled	☐
Height	730.915

Object Properties

Choose one or more items from the list

Actions:

OK   Cancel   Apply   Help

Select Model Viewport:

To select a Model Viewport 51, Left click 15 on one of the model viewport names from the list displayed on left hand side of manage dialog.

View Model Viewports:

To view properties of Model Viewport 51, select a model viewport as described in **Select Model Viewport** section. The **Preview** section will show the selected model viewport as it will appear in drawing. All properties for selected model viewport are displayed on right hand side of manage dialog.

Also See:

Viewport Objects 70
Library Toolbar 257

2.2.2.8.16 Quick Layer View Tool

Library Toolbar 257 :

Quick Layer View Tool is used to create/edit/delete Layer Objects 64. See detail description for different properties of Layer Objects 64.

Using the Quick Layer View Tool:

Find one of the **Quick Layer View Tool** icon from the Library Toolbar 257 (color coded

peach) and highlight it. While highlighted <u>Click</u> ¹⁵ on the tool. Quick Layer View Dialog will show up and allows quick turn on/off, freeze/thaw <u>Layer Objects</u> ⁶⁴.

To turn ON/OFF any layer:
> Click on the "light-bulb" icon infront of the layer name. Depending on drawing's size, it may take longer to reflect changes in drawing.

To FREEZE/THAW any layer:
> Click on the "Sun/Snow" icon infront of the layer name. Depending on drawing's size, it may take longer to reflect changes in drawing.

To Lock/Unlock any layer:
> Click on the "Lock" icon infront of the layer name. Depending on drawing's size, it may take longer to reflect changes in drawing.

To include/exclude a layer in printing:
> Click on the "Printer" icon infront of the layer name. Depending on drawing's size, it may take longer to reflect changes in drawing.

To edit any other properties, click on "Manage Layers..." button to access detailed layer properties dialog.

Also See:
> <u>Layer Objects</u> ⁶⁴
> <u>Library Toolbar</u> ²⁵⁷

2.2.2.9 Measure Tools

In **hsCADCreator**, *Measure Tools* allow easy measurement of length and area. There are two different tools for measuring length and area.

1. <u>Measure Distance Tool</u> ²⁴⁵

2. Measure Area Tool 247

3. Measure Angle Tool 249

2.2.2.9.1 Measure Distance Tool

Measure Toolbar 257 :

In **hsCADCreator** the *Measure Distance Tool* facilitates measurement of length along polyline or arc.

Measuring Lengths:
Activate the Measure Distance Tool:

Find the *Measure Distance Tool* icon from the Measure Toolbar 257 (color coded lavender) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Measure Distance using Mouse:
1. Move the mouse and Click 15 anywhere on the screen. This will be the **Start Point** from which the current length will be measured.
2. Move the mouse and Click 15 anywhere on the screen. This will be the **Tape Point** to which the current length will be set. And current length will be added to **Total Length**.
3. Successively select next **Tape Points** to keep on adding length to **Total Length.**
4. Alternatively, to change **Start Point** of tape in-between adding **Total Length,** use SHIFT+ENTER shortcut.
5. To measure length along an arc, change **Measure Mode** in Tool Property Tree 260 to "*Measure along arc*" and select **Start Point, Point on Arc,** and **Tape Point**.

Reset:
You have just finished measuring lengths. The tool has now **Reset** and you can either continue with other tools or measure other lengths again.

Measure Distance using Property Tree:

To **Measure Distance** in a more precise manner do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

1. Type in the **Start Point** point data into the "**Start Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept.
2. Type in the **Tape Point** point data into the "**Tape Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept. The tool measures current length from **Start Point** to **Tape Point** and adds it to Total Length.
3. Successively, type in the **Tape Point** point data into the "**Tape Point**" 3D Point Property Field 77 found on the Tool Property Tree 260 and press **Enter** to accept to keep on adding length to **Total Length**.
4. Alternatively, to change **Start Point** of tape in-between adding **Total Length,** use SHIFT+ENTER shortcut.
5. To measure length along an arc, change **Measure Mode** in Tool Property Tree 260 to "**Measure along arc**" and type in **Start Point,** press **Enter,** type **Point on Arc,** press **Enter,** type **Tape Point,** and press **Enter.**

Reset:

You have just finished measuring lengths. The tool has now **Reset** and you can either continue with other tools or measure other lengths again.

Measure Distance Options Menu :

Options Menu for **Measure Distance Tool** can be accessed by right click 15 on drawing screen while **Measure Distance Tool** is active.

Tool Options:

Continuous Measure keeps on adding the current length to Total length.
Auto Snap turns on common entity snaps for easy selection of points on entities.
Measure along polyline mode allows measuring length along polylines.
Measure along **arc** mode allows measuring length along circular arcs.

Change Start Point allows user to change current start point of tape.

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool 101)

2.2.2.9.2 Measure Area Tool

Measure Toolbar 257 **:**

In **hsCADCreator** the *Measure Area Tool* facilitates measurement of area of polygons.

Measuring Areas:
Activate the Measure Area Tool:

Find the *Measure Area Tool* icon from the Measure Toolbar 257 (color coded lavender) and highlight it. While highlighted Click 15 on the tool. The tool is now the active tool and ready for use.

Measure Area using Mouse:

1. Move the mouse and Click 15 anywhere on the screen. This will be the first vertex for polygon, from which the current area will be measured.
2. Move the mouse and Click 15 anywhere on the screen. This will be the second vertex for polygon.
3. Successively select next vertext points to measure **Current Area.**
4. Alternatively, use SHIFT+ENTER to start new polygon. This will add **Current Area** to **Total Area** and reset **Current Area** to zero to measure new polygon area.

Reset:
You have just finished measuring area. The tool has now **Reset** and you can either continue with other tools or measure other polygon area again.

Measure Area using Property Tree:

To **Measure Area** in a more precise manner do the following. Move the mouse cursor beyond the right side of the drawing screen to the Tool Property Tree 260.

1. Type in the **Current Point** point data into the "**Current Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept.
2. Successively, type in the **Current Point** point data into the "**Current Point**" 3D Point Property Field [77] found on the Tool Property Tree [260] and press **Enter** to accept to keep on adding area to **Current Area**.
4. Alternatively, use SHIFT+ENTER to start new polygon. This will add **Current Area** to **Total Area** and reset **Current Area** to zero to measure new polygon area.
5. To measure area along an arc, change **Measure Mode** in Tool Property Tree [260] to "*Measure along arc*" and type in **Start Point,** press **Enter,** type **Point on Arc,** press **Enter,** type **Tape Point,** and press **Enter.**

Reset:
You have just finished measuring area. The tool has now **Reset** and you can either continue with other tools or measure other polygon area again.

Measure Area Options Menu :
Options Menu for **Measure Area Tool** can be accessed by right click [15] on drawing screen while **Measure Area Tool** is active.

NOTE: Measure Area tool can not calculate area of self intersecting polygons or non-planar polygons. In order to calculate these, use "**Continuous Mode**" and dis-integrate self-intersecting/non-planar polygons into simple polygons.

Tool Options:

Continuous Measure keeps on adding the current area to Total area.
Auto Snap turns on common entity snaps for easy selection of points on entities.

Measure along polyline mode allows measuring area along polylines.
Measure along **arc** mode allows measuring area along circular arcs.
Change Start Point allows user to start a new polygon to measure area.

*Escape (Esc)***:** key cancels current tool and activates the default tool (Selection Tool
|101|)

2.2.2.9.3 Measure Angle Tool

Measure Toolbar|257| **:**

In **hsCADCreator** the *Measure Angle Tool* facilitates measurement of angles.

Measuring Areas:
Activate the Measure Angle Tool:

Find the *Measure Angle Tool* icon from the Measure Toolbar|257| (color coded
lavender) and highlight it. While highlighted Click|15| on the tool. The tool is now the active
tool and ready for use.

Measure Angle using Mouse:
1. Move the mouse and Click|15| anywhere on the screen. This will be the start
point for the angle being measured.
2. Move the mouse and Click|15| anywhere on the screen. This will be the center
point for the angle being measured.
3. Move the mouse and Click|15| anywhere on the screen. This will be the end
point for the angle being measured.

Reset:
You have just finished measuring angle. The tool has now **Reset** and you can either
continue with other tools or measure angles again.

Measure Angle using Property Tree:

To **Measure Angle** in a more precise manner do the following. Move the mouse
cursor beyond the right side of the drawing screen to the Tool Property Tree|260|.

1. Type in the **Start Point** point data into the "**Start Point**" 3D Point Property Field |77⌐| found on the Tool Property Tree |260⌐| and press **Enter** to accept.
2. Type in the **Center Point** point data into the "**Center Point**" 3D Point Property Field |77⌐| found on the Tool Property Tree |260⌐| and press **Enter** to accept.
3. Type in the **End Point** point data into the "**End Point**" 3D Point Property Field |77⌐| found on the Tool Property Tree |260⌐| and press **Enter** to accept.

Reset:
You have just finished measuring angles. The tool has now **Reset** and you can either continue with other tools or measure angles again.

Measure Angle Options Menu :
Options Menu for **Measure Area Tool** can be accessed by right click |15⌐| on drawing screen while **Measure Area Tool** is active.

Tool Options:

Continuous Measure keeps on adding the current area to Total area.
Auto Snap turns on common entity snaps for easy selection of points on entities.

Escape (Esc): key cancels current tool and activates the default tool (Selection Tool |101⌐|)

2.2.3   hsCADCreator Toolbars

hsCADCreator makes its Tools|100⌐| available primarily through the use of *Toolbars*|81⌐| and Menus |90⌐|. Each tool has a representational icon and a tool button. By pressing the tool button, you activate that tool.

Toolbar Visibility:

hsCADCreator has the ability to show/hide each of toolbars and tool windows. To show/ hide a toolbar/tool window open the **View Menu** 94 ➡**Toolbar** and select the name of the toolbar to show/hide.

Toolbars:

File Toolbar 256 :

Create Toolbar 251 :

Modify Toolbar 253 :

View Toolbar 253 :

Rendering Toolbar 256 :

Draw Plane/UCS Toolbar 254 :

Library Toolbar 257 :

Measure Toolbar 257 :

Also See:

2.2.3.1 Entity Toolbar

Description:
The tools associated with the *Entity toolbar* are the basis for the creation and placement of entities onto a drawing.

1. 2. 3. 4. 5. 6. 7. 8. 9. 10. 11. 12. 13. 14. 15. 16. 17.

1.   **Selection Tool** 101

2. **Point Tool** 105
3. **Line Tool** 106
4. **Multi Line Tool** 108
5. **2D Polyline Tool** 110
6. **3D Polyline Tool** 112
7. **Arc Tools** 113 has a Flyout Toolbar 81

    a. b. c. d.
    a. Arc (Start, Middle, End) 114
    b. Arc (Start, Center, End) 115
    c. Arc (Start, Center, Angle) 117
    d. Arc (Start, Center, Chord Length) 119

8. **Circle Tool** 120
9. **Ellipse Tool** 121
10. **Dimension Tools** 123 has a Flyout Toolbar 81

    a. b. c. d. e. f.
    a. Aligned Dimension Tool 124
    b. Linear Dimension Tool 126
    c. Ordinate Dimension Tool 128
    d. Radial Dimension Tool 130
    e. Diametric Dimension Tool 132
    f. Angular Dimension Tool 134

11. **Text Tool** 136
12. **Hatch Tool** 138
13. **Face Mesh Tool** 139
14. **Block Tool** 140
15. **Block Insertion Tool** 142
16. **Image Insertion Tool** 144
17. **Viewport Tool** 146 *Note: This tool is only active if working in a paper space. It will be disabled within model space.*

Also See:
    Tool Menu 97
    Flyout Toolbar 81

2.2.3.2 Modify Toolbar

Description:
The tools associated with the *Modify Toolbar* have the purpose of manipulating, altering, or removing entities that have already been created and placed on a drawing.

1.   2.   3.   4.   5.   6.   7.   8.   9.

1. **Translate Tool** 148
2. **Rotate Tool** 150
3. **Explode Tool** 152
4. **Delete Tool** 153
5. **Scale Entity Uniformly Tool** 154
6. **Scale Entity Non-uniformly Tool** 155
7. **Trim Tool** 157
8. **Extend Tool** 158
9. **Clone Tools** 159

      a.   b.   c.   d.
      a.   **Linear Clone** 163
      b.   **Planar Clone** 165
      c.   **3-Axial Clone** 168
      d.   **Radial Clone** 160

Also See:
      Tool Menu 97

2.2.3.3 View Toolbar

Description:
The tools associated with the *View Toolbar* are used to alter the 2D or 3D view of the drawing.

1.   2.   3.   4.   5.   6.   7.   8.   9.   10. 11. 12.

1. *Zoom To Window Tool* 175
2. *Zoom In Tool* 176
3. *Zoom Out Tool* 177
4. *Zoom Extent Tool* 178
5. *Pan View Tool* 178

6. ***Rotate About Eye Vector Tool*** 180

7. ***Rotate About Vertical Vector Tool*** 181

8. ***Rotate About Horizontal Vector Tool*** 182

9. ***Rotate View 3D Tool*** 183

10. **View Snap Tools** 173 has a Flyout Toolbar 81

    a. b. c. d. e. f. g. h. i. j.

a. *Top View Tool* 188
b. *Bottom View Tool* 189
c. *Front (South) View Tool* 189
d. *Back (North) View Tool* 190
e. *Left Side (West) View Tool* 190
f. *Right Side (East) View Tool* 191
g. *Southwest View Tool* 191
h. *Southeast View Tool* 192
i. *Northeast View Tool* 192
j. *Northwest View Tool* 193

11. **Divide Viewport** 184 (Splits the current viewport into two parts either horizontally or vertically based on the tool setting)

12. **Viewport Presets** 186 has a Flyout Toolbar 81

    a. b. c. d. e. f. g. h. i. j.

a. Single Viewport Tool 186
b. 2 Vertical Viewports Tool 186
c. 2 Horizontal Viewports Tool 186
d. 3 Right Viewports Tool 186
e. 3 Left Viewports Tool 186
f. 3 Bottom Viewports Tool 186
g. 3 Top Viewports Tool 186
h. 4 Even Viewports Tool 186
i. 4 Right Viewports Tool 186
j. 4 Left Viewports Tool 186

Also See:
    Tool Menu 97
    Flyout Toolbar 81

2.2.3.4 Draw Plane/UCS Toolbar

Description:

The tools associated with the ***Draw Plane/UCS Toolbar*** are used to alter the working draw plane. "UCS" stands for User Coordinate System. We use "Draw plane" to help users understand the purpose of the UCS 15. The UCS 15 is a moveable, manipulatable coordinate system to help our users more easily work in 2D and 3D drawing space.

1. 2. 3. 4. 5. 6. 7. 8.

1. **Preset UCS Snap Tools** [195] has a Flyout Toolbar [81]

    a. b. c. d. e. f.
a.   Snap UCS to Top Tool [195]
b.   Snap UCS to Bottom Tool [196]
c.   Snap UCS to Front (South) Tool [197]
d.   Snap UCS to Back (North) Tool [197]
e.   Snap UCS to Left Side (West) Tool [197]
f.   Snap UCS to Right Side (East) Tool [198]

2. **Rotate Draw Plane Tools** [198] has a Flyout Toolbar [81]

    a. b. c. d.
a.   Rotate UCS about X axis Tool [199]
b.   Rotate UCS about Y axis Tool [201]
c.   Rotate UCS about Z axis Tool [202]
d.   Rotate UCS about X,Y and or Z axis Tool [203]

3. **Translate Draw Plane Tools** [205] has a Flyout Toolbar [81]

    a. b. c. d.
a.   Translate UCS along X axis Tool [206]
b.   Translate UCS along Y axis Tool [207]
c.   Translate UCS along Z axis Tool [208]
d.   Translate UCS origin to a Point Tool [209]

4. **Align UCS to View** [210] (aligns the Draw plane [15] with the current view)

5. **Align View to UCS** [211] (aligns the View with the current UCS [15] /Draw plane [15])

6. **UCS Origin to WCS Origin** [211] (moves the UCS [15] Origin to the WCS [15] Origin)

7. **Align UCS Axis to WCS Axis** [212] (aligns and orients the UCS [15] axis to the WCS [15] axis)

8. **Align UCS to Selected Entities Tool** [212] (moves and orients the UCS [15] to align with plane of selected entities)

Also See:
    Tool Menu [97]
    Flyout Toolbar [81]
    UCS [15]
    WCS [15]

2.2.3.5 **Rendering Toolbar**

Description:
The tools associated with the *Rendering Toolbar* are used to alter the rendering of the drawing.

1.   2.   3.   4.   5.   6.   7.   8.

1.   **2D Wireframe Tool** 213 (Default Render mode)
2.   **3D Wireframe Tool** 214
3.   **Hidden Tool** 214
4.   **Flat shaded Tool** 214
5.   **Gouraud Shaded Tool** 215
6.   **Flat Shaded With Edges Tool** 216
7.   **Gouraud Shaded With Edges Tool** 216
8.   **Regenerate Tool** 217 (Refreshes all entities and rendering within a drawing.)

Also See:
        Tool Menu 97

2.2.3.6 **File Toolbar**

Description:
The tools associated with the *File Toolbar* are used to for the basic document operations of open, save, print, and help.

1.   2.   3.   4.   5.   6.

1.   **New** (Create a new document)
2.   **Save** (Save the active document)
3.   **Open** (Open an existing named document)
4.   **Print/Plot** 14 (Print/Plot the active document)
5.   **Print/Plot Preview** 14 (Preview how the active document will print)
6.   **Help** (Opens the help documentation)

Also See:
        File Menu 93
        File Formats 286
        Print/Plot 14

2.2.3.7 **Library Toolbar**

Description:

The tools associated with the *Library Toolbar* are used for the quick management of useful entities and objects used within the drawing or for help in creating a drawing.

1. 2. 3. 4. 5. 6. 7. 8. 9. 10. 11. 12. 13. 14. 15.

1. **Manage Layers** [230]
2. **Quick Layer View** [243]
3. **Manage Blocks** [218]
4. **Manage Image Definitions** [228]
5. **Manage Colors** [220]
6. **Manage Text Styles** [239]
7. **Manage Linetypes** [233]
8. **Manage Multi Line Styles** [235]
9. **Manage Hatch Styles** [224]
10. **Manage Dimension Styles** [221]
11. **Manage User Coordinate Systems** [241]
12. **Manage External References** [223]
13. **Manage Layouts** [232]
14. **Manage Viewports** [242]
15. **Manage Named Views** [236]

Also See:
> Manage Menu [96]

2.2.3.8 **Measure Toolbar**

Description:

The tools associated with the *Measure Toolbar* are used to for measuring length, area and angle.

1. 2. 3.

1. **Measure Distance** [245] **:** Measure Distance along lines and arc.
2. **Measure Area** [247] : Measure area of polygons.
3. **Measure Angle** [249] : Measure angles

Also See:

2.2.3.9 Snaps Toolbar

Description:

The tools associated with the *Snap Toolbar* have the purpose of quickly enabling or disabling various snapping abilities within the drawing space.

Enabled	End Point	Mid Point	Intersection	Center Point	Insertion	Nearest	Node	Parallel	Perpendicular	Polar	Quadrant	Tangent	Grid Snap	Grid
1.	**2.**	**3.**	**4.**	**5.**	**6.**	**7.**	**8.**	**9.**	**10.**	**11.**	**12.**	**13.**	**14.**	**15.**

1. **Enable** 72 /Disable (quickly turn on/off all snapping features)

2. **End Point** 72

3. **Mid Point** 72

4. **Intersection** 72

5. **Center Point** 72

6. **Insertion** 72

7. **Nearest** 72

8. **Node** 72

9. **Parallel** 72

10. **Perpendicular** 72

11. **Polar** 72

12. **Quadrant** 72

13. **Tangent** 72

14. **Grid Snap** 74 (toggle the Grid Snapping feature)

15. **Grid** 74 (toggle the grid visibility)

2.2.4 hsCADCreator Property Trees

hsCADCreator uses property trees 79 to edit individual properties 77. Following property trees are accessible in **hsCADCreator** :

- Selection Property Tree 259
- Tool Property Tree 260
- Snap Property Tree 261
- Drawing Property Tree 262
- Viewport Property Tree 263

2.2.4.1   Selection Property Tree

2.2.4.2 Tool Property Tree

2.2.4.3   Snap Property Tree

2.2.4.4 Drawing Property Tree

Click on image to see detail view.

2.2.4.5 Viewport Property Tree

Named Object		
Name	*Active	
Xref Dependent	☐	
Xref Resolved	☐	

Camera Settings		
Back Clip Distance	0.000	
Back Clip Enabled	☐	
Center Point	0.000	0.000
Elevation	0.000	
Clip at Eye	☑	
Front Clip Distance	0.000	
Front Clip Enabled	☐	
Height	422.183	
Lens Length	50.000	
Is Perspective	☐	
Render Mode	2D Optimized	
Target	344.187	148.500	0.000
UCS	Empty	
Width	635.754	
View Direction	0.000	0.000	1.000
Twist	0.000	

Modelspace Viewport		
UCS icon visible	☑	
UCS icon at origin	☑	
Follow UCS	☐	
Independent UCS	☑	
Grid visible	☑	
Grid snap	☑	
Grid Spacing along X-Axis	10.000	
Grid Spacing along Y-Axis	10.000	

Viewport Properties

Details of the active Modelspace Viewport

2.2.5 hsCADCreator Dialogs

Following dialogs are used in **hsCADCreator**:

- About Dialog [264]
- Application Settings Dialog [264]
- Print Dialog [266]

2.2.5.1 About hsCADCreator Dialog

The About Dialog displays information about **hsCADCreator** and the current status of the License.

- **Current Version Number for hsCADCreator**
- **Licensee Name**: Name of License Owner or Company License Contact
- **Licensee Company**: Name of Company
- **Licensee Email** Email of License Owner or Company Contact
- **Licensee Phone Number** : Phone number of License Owner or Company Contact
- **License Site Code** : Generated
- **Machine Code** : Generated
- **License Group Code** : Code from online purchase or media case.
- **Activation Code** : Generated

2.2.5.2 Application Settings Dialog

Application Settings are program level settings. These settings are not saved with drawing, but saved with **hsCADCreator** program. Whenever a new drawing or an existing drawing is opened previously set Application Settings are applied to it. Application Settings are accessible through Application Settings Dialog. To access this dialog use **Options** [99] ➡ **Settings...** Menu item.

Field Name	Data Type	Description
Use Auto Save	Boolean [79]	Enable/disable auto save feature
Auto Save Interval	Integer Number	Time interval between each auto save in minutes.
Auto Save Directory	Folder Path	Path to folder for temporary auto save files.
Drawings Directory	Folder Path	Path to current drawing's directory.
Raster Directory	Folder Path	Path to current drawing's raster files.
Plot Style Directory	Folder Path	Path to current drawing's plot style directory.
Tesselation Deviation	Real Number	Value that controls the resolution for drawing non-linear curves. Lower the value, higher the resolution.
Region Selection UCS	Real Number	Tolerance for selecting region

Tolerance		that is on UCS plane.
Region Selection Point Tolerance	Real Number	Tolerance for selecting point that is on UCS plane.
Decimal Places	Integer Number	Number of decimal places to display for all numbers used in drawing.
Default Drawing Units	Multi-option [79] selection box	Default drawing units to be used for newly created drawings.
Active Unit Family	Multi-option [79] selection box	Currently active unit family
Notification: Show Tool Options	Boolean [79]	Enable/disable tool options notifications.
Notification: Show Tool Activations	Boolean [79]	Enable/disable tool activations notifications.
Trim Trailing Zeros	Boolean [79]	Enable/disable trimming of trailing zeros.
Default Viewer Settings: Permanent Entity Snap Settings	Entity Snap Settings [72]	Default entity snap settings to be used for newly created drawings.
Default Drawing Settings	Drawing Settings [76]	Default drawing settings to be used for newly created drawings.

2.2.5.3   Print Dialog

Print dialog gives access to print/plot settings [238]. This dialog is accessible from **File** ➡ **Print/Plot** menu item.

Print

Plot Settings

| Model (layout) | ▼ |

⊟	**Print Settings**		^
	Copies	**1**	
⊟	**Plot Settings**		?
	Name		
	Paper Units	Inches	⌄
	Printer		⌄
			⌄
		0 by 0 in	Properties
	Plot Rotation	Portrait	⌄
	Plot Type	Display	⌄
	Scale	Scale to fit ⌄ 0.000 : 0.000	
⊞	Plot Arrangement		
⊞	Plot Style Support		
⊞	Advanced Plot Settings		

No Item Selected

Select an item to see its description

| Apply To Layout | Add As New Plot Settings | | OK | Cancel | Help |

2.2.6   Notify Window

The ***Notify Window*** tracks and displays a variety of information regarding the current state of the design process. Messages are displayed in order of arrival, with the latest message first. Messages have one of 6 color-coded types.

The ***Notification Window*** is a Docking window 83. Docking windows 83 are easy to position and size. Docking windows 83 may also be pinned 83 or unpinned 83 depending on preference. In the Basic Workspace Layout 279 (default layout) the ***Notification window*** is docked 83 to the bottom of the screen and is pinned 83. In the image below the Notification window in floating 83 mode.

Basic Workspace Layout

Type	Format	Purpose
Option	*OPTION:ACTION*	Displays information on how to activate special options in the currently **active tool** 80. OPTION is the name of the choice, and ACTION is the key or mouse action used to activate the

		choice. These messages can be turned on or off in the Settings Dialog [264].
Instruction	[INSTRUCTION]	When a tool is expecting user input, it uses these messages to explain precisely what action is necessary in order to continue.
Action	ENTITY=(0,1,2)	When a tool completes an action of interest, it relates this information back to the user.
Activate / Deactivate	Tool Name Activated Tool Name Deactivated	These messages occur when a tool is activated or deactivated. These also show when Stackable Tools [80] are being used. These messages can be turned on or off in the Settings Dialog [264].
Warning	Warning! "I" before "E" EXCEPT after "C".	A tool uses these messages to notify the user that input and actions may not have the desired result, or if there exists a potential for error.
Error	ERROR! "Ain't" ain't a word.	This mechanism notifies the user when a problem has occurred in the system, either through incorrect input or an internal error.

2.2.7   Status Bar

Status Bar is located at the bottom of **hsCADCreator's** window. It shows the current status of the **hsCADCreator**. It also shows brief description of a tool when user moves over mouse on any tool.

2.2.8 Visual Aids

Beside the usual visual feedback of the mouse cursor, different tools will employ visual aids to enhance the use of that tool. Different tools will use different visual aids as appropriate. Entity Creation Tools 103, typically use all of the visual aids.

Preview Entities

Entity Creation tools incorporate a preview entity and one or more entity design widgets. The preview entity shows what an entity would look like if you were to complete the current process with the current choices. Thus, a line preview entity would show you what the line would look like if you were to click it. The preview entities are shown with dotted lines.

Cursor Icons

There are various **cursor icons** to give user a visual feedback of currently active tool 80. The cursor icon changes whenever active tool is changed. Apart from active tool cursor icons, there is also invalid input cursor icon ⊘. Whenever the input under cursor becomes invalid the cursor icon changes to invalid input cursor ⊘.

Cursor Targets

This is a very common visual aid. There are two types of cursor target representation.

- Round cursor target (⊕)
- Square cursor target (⊞)

Round target cursor(⊕) is shown whenever user moves the mouse around before selecting any point. Square target cursor (⊞) is drawn whenever user selects some point during any tool step. In example below drawing arc with *Start-Middle-End Arc Tool* [114], start and middle points are already selected and are shown with square target ⊞ . The end point is still not selected and hence is shown as round target ⊕ :

See Also:
Grid [75]

2.2.9 Keyboard Shortcuts

hsCADCreator is installed with color coded shortcuts from the keyboard for very easy access to

most frequently used tools. These shortcut keys may be modified by selecting the down arrow (⊽) located to the right side any toolbar. Choose **Add or Remove Buttons** ➡ **Customize...** This will activate the **Customize** window. Select the **Keyboard** tab. Within this tab it is possible assign, remove, or reset keyboard shortcuts for one or many commands. Note that the keyboard shortcuts will not be saved unless you save your current layout. Visit Saving My Layout for more information.

Command Key Mapping dialog can be opened from **Options ➡ Key Accelerators...** menu item. In the following picture, each default shortcut key is shown with its toolbars color.

In **hsCADCreator**, default shortcut keys are as follows:

Shortcut Key	Function
F1	Opens Help for hsCADCreator

Create Toolbar 251 **:**

Shortcut Key	Function
ESC	Selection Tool 101
O	Point Tool 105
L	Line Tool 106
M	Multiline Tool 108
P	Planar Polyline Tool 110

SHIFT + P	3D Polyline Tool [112]
A	Arc (Start, Middle, End) Tool [114]
SHIFT + A	Arc (Start, Center, End) Tool [115]
C	Circle Tool [120]
E	Ellipse Tool [121]
D	Aligned Dimension Tool [124]
SHIFT + D	Linear Dimension Tool [126]
T	Text Tool [136]
H	Hatch Tool [138]
F	Poly Face Mesh Tool [139]
SHIFT + B	Block Definition Tool [140]
B	Block Insertion Tool [142]
I	Image Insertion Tool [144]

Modify Toolbar [253] **:**

Shortcut Key	Function
V	Translate Tool [148]
R	Rotate Tool [150]
X	Explode Tool [152]
DELETE	Delete Tool [153]
S	Scale Uniformly Tool [154]
SHIFT + S	Scale Non-Uniformly Tool [155]
[	Trim Tool [157]
]	Extend Tool [158]

View Toolbar [253] **:**

Shortcut Key	Function
W	Zoom to Window Tool [175]
NUMPAD +	Zoom In Tool [176]
NUMPAD -	Zoom Out Tool [177]
SHIFT + E	Zoom Extents Tool [178]
Z	Pan View Tool [178]
NUMPAD 5	Top View Tool [188]
NUMPAD 0	Bottom View Tool [189]
NUMPAD 8	Front (South) View Tool [189]

NUMPAD 2	Back (North) View Tool [190]
NUMPAD 4	Left Side (West) View Tool [190]
NUMPAD 6	Right Side (East) View Tool [191]
NUMPAD 1	Southwest View Tool [191]
NUMPAD 3	Southeast View Tool [192]
NUMPAD 7	Northwest View Tool [193]
NUMPAD 9	Northeast View Tool [192]

File Toolbar [256] **:**

Shortcut Key	Function
CTRL + Z	Undo
CTRL + Y	Redo
CTRL + A	Select All
CTRL + C	Copy
CTRL + V	Paste
CTRL + SHIFT + V	Paste as Block
CTRL + X	Cut Selection

Draw Plane/UCS Toolbar [254] **:**

Shortcut Key	Function
ALT + NUMPAD 5	Snap UCS to Top View [195]
ALT + NUMPAD 0	Snap UCS to Bottom View [196]
ALT + NUMPAD 8	Snap UCS to Front View [197]
ALT + NUMPAD 2	Snap UCS to Back View [197]
SHIFT + RIGHT ARROW	Rotate UCS about X-axis Tool [199]
SHIFT + UP ARROW	Rotate UCS about Y-axis Tool [201]
SHIFT + DOWN ARROW	Rotate UCS about Z-axis Tool [202]
SHIFT + LEFT ARROW	Rotate UCS [203]
RIGHT ARROW	Translate UCS along X-axis Tool [206]
UP ARROW	Translate UCS along Y-axis Tool [207]
DOWN ARROW	Translate UCS along Z-axis Tool [208]
,	Align UCS to Current View Tool [210]
.	Align Current View to UCS Tool [211]
SHIFT+ >	Align UCS Origin to WCS Origin [212]
/	Align UCS to Selected Entities [212]

Library Toolbar 257 :

Shortcut Key	Function
ALT + E	Manage Entire Library Tool
ALT + K	Manage Layers Tools 230
ALT + L	Quick Layer View Tool 243
ALT + B	Manage Blocks Tool 218
ALT + I	Manage Image Definitions Tool 228
ALT + C	Manage Colors Tool 220
ALT + T	Manage Text Styles Tool 239
ALT + Y	Manage Linetypes Tool 233
ALT + M	Manage Multiline Styles Tool 235
ALT + H	Manage Hatch Styles Tool 224
ALT + D	Manage Dimension Styles Tool 221
ALT + U	Manage UCS Tool 241
ALT + R	Manage External References Tool 223
ALT + W	Manage Layouts Tool 232
ALT + V	Manage Viewports Tool 242
ALT + N	Manage Named Views Tool 236

2.2.10 Customizing Toolbars

hsCADCreator has the ability customize each toolbar. This customization includes the ability to show hide individual toolbar buttons; creation/management of custom toolbars; activation and deactivation of a toolbar's visibility; and quick key or key accelerator management.

Toolbar Show/Hide:

You may show/hide a toolbar/tool window by opening the ➡**Toolbar** and select the name of

the toolbar you wish to show/hide. You may also select the down arrow (▼) located to the right side of the toolbars and choose **Add or Remove Buttons** ➡**Customize...** This will activate a **Customize** window. To show/hide a toolbar simply check/uncheck the appropriate toolbar in the list.

Creating a Custom Toolbar:

It is possible to create a custom toolbar with the tools that you use most often. To create a

new toolbar select the down arrow (▼) located to the right side of the toolbars and choose
Add or Remove Buttons ➡ Customize... This wll will activate a **Customize** window. By
default the active tab should be **Toolbars**. Select the **New...** button found on the right hand
side of the **Toolbars** tab window. This will activate a small window where you can enter the
name of the new toolbar. In the screenshot below we name our new toolbar "Custom 1".
When you press **OK** a new empty toolbar will become visible in the upper left hand corner of
hsCADCreator.

Once you have created the toolbar you will need to add commands to the currently empty toolbar. To do this go to the **Commands** tab in the **Customize** window. Select a command using the mouse. Click and hold the left mouse button down and drag the selected command to the empty toolbar. Release the left mouse button when you are over the top of the toolbar. If the command has a toolbar icon you will now see that icon in the toolbar otherwise you will see the command's name.

Keyboard Accelerators:

hsCADCreator has key accelerators that are setup by default on the installation of the program. To view the default key configurations visit . To modify a tool's default keyboard accelerator select the down arrow (🔽) located to the right side any toolbar and choose **Add or Remove Buttons** ➡ **Customize...** This will activate the **Customize** window. Select the **Keyboard** tab. Within this tab it is possible assign, remove, or reset keyboard shortcuts for one or many commands. Note that the keyboard shortcuts will not be saved unless you save your current layout. Visit Saving My Layout for more information.

Toolbar Options:

Select the down arrow (🔽) located to the right side any toolbar and choose **Add or Remove Buttons** ➡ **Customize...** This will activate the **Customize** window. Select the **Options** tab. In this tab it is possible to manage the personalized menu data as well as change the toolbar icons size. You may also modify if the tool tips and their shortcut keys are shown when your mouse hovers over the top of a tool for a certain amount of time. These options are only saved when you save your layout. Visit Saving My Layout

2.2.11 Workspace Layout

The User Interface of **hsCADCreator** is designed to be a simple, yet powerful tool for dealing with daily tasks in computer aided drafting and design. We know that you may wish to modify your layout to more fully utilize the drafting space / tools / etc. We have provided the ability to manage how exactly user interface elements are displayed. This includes the easy movement and docking of all default and custom toolbars, Drawer windows, Flyout toolbars for like minded tools, and much more. Layouts also contain the key accelerator configurations. Visit this link for more on changing the Keyboard Shortcuts 27 .

Basic and Advanced Layouts:

Installed by default is two different layouts designed for two groups of our users **Basic** and **Advanced**. The **Basic** layout is designed to get the unfamiliar user more comfortable with the use of hsCADCreator. This will typically have a visible Library Bar, Notification Window, and Help Window open and easily seen. The Advanced layout is designed more for our advanced user and every day usage of hsCADCreator. This layout will have most windows set as unpinned such that the drawing space is maximized. Also in the Advanced layout all toolbars and tools are visible.

Basic Workspace Layout

Advanced Workspace Layout

Custom Layouts:

The layout of **hsCADCreator** is fully customizable. This include placement and visibility of almost every user interface element including toolbars, windows, menus, etc. Below we will cover the steps to **Save**, **Revert**, **Save As...**, and **Load From...**.

Save my current layout:
1. Modify the layout of the windows/toolbars as desired.
2. Go to **Window** ➡ **Layout** ➡ **Custom Layout** ➡ **Save** from the main menu.
3. After accepting to perform this operation a file will be created in the
hsCADCreator install directory under **Layouts**. The file's name is **Current.xml**.

Revert my current layout:
1. Go to **Window** ➡ **Layout** ➡ **Custom Layout** ➡ **Revert** from the main menu
2. After accepting that this will modify the current layout the layout will be modified to the previously saved layout configuration.

Save to File:
You may save your current configuration and saving to a file will save the active configuration with which you could then share with friends and colleagues.
1. Go to **Window** ➡ **Layout** ➡ **Custom Layout** ➡ **Save As...** from the main menu
2. Select name and location and save the layout.

Load from File:
This will allow you to specify which layout you wish to utilize. You may load a previously saved layout at any time. Note that if you do not **Save** the current layout after loading from a file the layout will not be the same as the loaded file the next time **hsCADCreator** is started.

1. Go to **Window** ➡ **Layout** ➡ **Custom Layout** ➡ **Load From** from the main menu
2. Select the Layout file you wish to Load.
3. Layout will be updated from the file
4. If you wish this layout to be the active layout the next time **hsCADCreator** is opened you must go to **Window** ➡ **Layout** ➡ **Custom Layout** ➡ **Save** to save the layout as the current active layout.

*Note: Every time you **Save** you will overwrite the **Current.xml** file with the active configuration. Every time you **Revert** it will load the **Current.xml** file if one exists.*

Changing Layouts:

You may change your layout at any time. You may change to the **Basic** layout by going to **Window** ➡ **Layout** ➡ **Basic Layout** from the main menu. You may change to the **Advanced** layout by going to the **Window** ➡ **Layout** ➡ **Advanced Layout** from the main menu.

2.2.12 Layer Window

The Layer Window is a quick and easy way to manipulate the layers of a drawing. For more information on managing the layers and what symbols mean visit <u>Quick Layer View Tool</u> 243 and/or <u>Manage Layers Tool</u> 230.

2.2.13 Library Window

The *Library Window* within **hsCADCreator** is designed to be a simple and powerful tool. Installed by default is a robust library of common architectural drawing items, furniture, appliances, general shapes, and much more. You may simply click and drag a desired item from this window directly onto your drawing.

By default within the Basic Layout 279 the *Library Window* is docked on the left side of the screen and

---

has the "All" category Selected.

Using the Library Window/Bar

To use the window bar move the mouse over the top of the preview item you wish to add to the drawing. Click and hold down the left mouse button on the selection. In the image below we are selecting "Fan Ceiling 3". As you can see when you drag the item you will have a cursor representation of that item.

Once you reach the drawing space the cursor representation will turn into the actual representation of the item being added to the drawing.

Simply release the <u>Left mouse</u> [15] when it is in the desired location and a block item will have been quickly added to your drawing.

Finding Library Items

The ***Library Window*** has a generic text search capabilities. To quickly find an item type in the what you are looking for within the category combo box. For instance type in "win" and all the "Window" items will be within the preview portion as seen in the image below.

It is simple to change the category of items displayed within the *Library Window*. Click on the category drop down arrow and choose a category from the list. After the category is chosen the preview list will be updated with that category's items.   Seen below is changing the category from "Fans" to "Showers".

2.2.14 Help Window

By default **hsCADCreator** uses the Basic Workspace Layout [279]. In this layout the *Help Window* is located to the right hand side of the main window. With it easily visible, browsing through the help will support a better understanding of **hsCADCreator** quicker. The *Help Window* is also a Docking Window [83]. This window is easily UnPinned [83] to save valuable drawing space. For more information on managing the window space visit Workspace Layout [279].

Basic Workspace Layout

Navigating Through the Help Documentation

To move throughout the help documentation use the red links to navigate. Also, found at the top

of the window are three icons ( ). These icons are used to move around throughout the help.

1. ( ) will move back to the home or start of the help documentation.

2. ( ) will move you forward to a page where you had recently gone back from.

3. ( ) will move you back to the previous page you had looked at.

2.3 Export

hsCADCreator can export the current drawing file to following formats.

- DWG 287
- DWF : Multi Layout 288
- DWF : Single Layout 289
- DXB 290
- DXF 287
- Email Attachment 291
- Image 292
- PDF 293
- SVG 294

2.3.1 DWG Format

DWG(Drawing) format is standard format for most of the CAD(Computer Aided Drafting) drawing files.

hsCADCreator supports DWG formats for open/edit/create drawings.

Following release formats of DWG are supported:
        DWG Release 9
        DWG Release 10
        DWG Release 12
        DWG Release 13
        DWG Release 14
        DWG Release 15
        DWG Release 18
        DWG Release 21

Exporter Dialog can be used to export existing drawing to any of above format.
Exporter Dialog is accessed from <u>File Menu</u> 93 :: Exporter... menu option.

Pressing **Export...** button creates new DWG file with given File Name.

2.3.2 DXF Format

DXF is a **Drawing Interchange Format.**

hsCADCreator supports DXF formats for open/edit/create drawings.

Following release formats of DXF are supported:

DXF Release 9
DXF Release 10
DXF Release 12
DXF Release 13
DXF Release 14
DXF Release 15
DXF Release 18
DXF Release 21

Exporter Dialog can be used to export existing drawing to any of above format.

Exporter Dialog is accessed from File Menu 93 :: Exporter... menu option.

Pressing **Export...** button creates new DXF file with given File Name.

2.3.3   DWF : Multi Layout Format

DWF is a **Drawing Web Format.** This format is commonly used to upload drawings on website for easy viewing and access.

hsCADCreator supports DWF: Multi Layout formats for open/edit/create drawings.

Following release formats of DWF : Multi Layout are supported:
Binary DWF v6.0
Zipped ASCII Encoded 2D Stream DWF v6.0

Exporter Dialog can be used to export existing drawing to any of above format.

Exporter Dialog is accessed from File Menu 93 :: Exporter... menu option.

Pressing **Export...** button creates new DWF file with given File Name.

2.3.4 DWF : Single Layout Format

DWF is a **Drawing Web Format.** This format is commonly used to upload drawings on website for easy viewing and access.

hsCADCreator supports DWF : Single Layout formats for open/edit/create drawings.

Following release formats of DWF : Single Layout are supported:
      3-D DWF v6.01
      Binary DWF v6.0
      Zipped ASCII Encoded 2D Stream DWF v6.0
      Compressed Binary DWF v5.5
      Uncompressed Binary DWF v5.5
      ASCII DWF v5.5
      Compressed Binary DWF v4.2
      Uncompressed Binary DWF v4.2
      ASCII DWF v4.2

Exporter Dialog can be used to export existing drawing to any of above format.

Exporter Dialog is accessed from File Menu 93 :: Exporter... menu option.

Pressing **Export...** button creates new DWF file with given File Name.

2.3.5   DXB Format

DXB is a **Drawing Interchange Binary** Format**.**

hsCADCreator can convert any opened drawing to **DXB** format:

Following release formats of DXB are supported:
    DXB Release 9
    DXB Release 10
    DXB Release 12
    DXB Release 13
    DXB Release 14
    DXB Release 15
    DXB Release 18
    DXB Release 21

Exporter Dialog can be used to export existing drawing to any of above format.

Exporter Dialog is accessed from File Menu 93 :: Exporter... menu option.

Pressing **Export...** button creates new DXB file with given File Name.

2.3.6   Email Attachment

hsCADCreator can be used to attach current drawing in an email easily with *Exporter Dialog*.

Exporter Dialog is accessed from File Menu ⌐93⌐ :: Exporter... menu option.

Pressing **Export...** button opens a new email to compose with current drawing already attached with email.

2.3.7 Image Format

Currently **hsCADCreator** supports BMP (Bitmap) Image format. Using *Exporter Dialog,* **hsCADCreator** can convert current drawing to BMP format.

Exporter Dialog is accessed from File Menu 93 :: Exporter... menu option.

Pressing **Export...** button creates new BMP image file with given File Name.

2.3.8   PDF Format

PDF is a **Portable Document Format.**

hsCADCreator can convert any opened drawing to **PDF** format using *Exporter Dialog*.

Exporter Dialog is accessed from File Menu 93 :: Exporter... menu option.

Pressing **Export...** button creates new PDF file with given File Name.
If there is any plot style(stylesheet) associated with exported layout |232|, it gets applied to pdf document.

2.3.9 SVG Format

SVG is a **Scalable Vector Graphics** format**.**

hsCADCreator can convert any opened drawing to **SVG** format using *Exporter Dialog*.

Exporter Dialog is accessed from File Menu |93| :: Exporter... menu option.

Pressing **Export...** button creates new SVG file with given File Name.

2.4   Supported Files and Formats

hsCADCreator can open a variety of files and formats and convert between variety of formats.

Input Formats that can be opened/viewed/edited by **hsCADCreator** are :

DWG 287
DXF 287
DWF : Multi Layout 288
DWF : Single Layout 289

Output Formats can be created/converted to from opened drawing in **hsCADCreator** are :

DWG 287
DWF : Multi Layout 288
DWF : Single Layout 289
DXB 290
DXF 287
Email Attachment 291
Image 292
PDF 293
SVG 294

3 Licensing

- This software is controlled by a licensing system. A license key is a unique identifier used to activate your copy of the software for use under the End-User License Agreement 303.
- This software requires a valid license key and running this software causes a License Key Entry Dialog 296 to appear until one is entered. By default, **hsCADCreator** is installed with an Evaluation license key which allows use for a trial period.
- Full version licenses may be purchased by opening the "Help" menu and clicking on the "Purchase Upgrade to Full Version" menu item, or by visiting hsCADCreator website.

This software is controlled by a seat enforcing licensing system. The evaluation version of **hsCADCreator** allows a user a trial period of 5 days and/or 5 uses. After the evaluation term has expired the software is locked and cannot be used without an activation code. Hachisoft Corporation sells its software as license groups. A license group can contain one or more license seats. You may purchase a license group by visiting (http blah).

3.1 Licensing Application

Licensing hsCADCreator:

1. Press the "**Buy Full Version hsCADCreator**" or use hsCADCreator website to buy **Full Version License Key**.

2. Press the "**Enter Full Version License Key**" or use the **Help ➡ Activate License** Menu to activate **hsCADCreator**.

or

3. The Manage **hsCADCreator** Licensing dialog will popup and will allow you to enter the "**License Group Code**" you purchased.

4. Press the "**Activate**" button to activate your copy of **hsCADCreator**.

Also See:

3.2 Managing License Groups

Managing License Groups:

Hachisoft Corporation sells its software as license groups. A license group can contain one or more license seats. On activation the licensing system ties a seat of **hsCADCreator** to a given computer. The licensing system supports adding more seats to a license group, activating seats, removing seats and transferring seats of **hsCADCreator** from machine to machine.

The hsCADCreator setup can be passed command line parameters that allow remote administration for volume and site license management. These parameters are:

/VERYSILENT -- runs the installation without showing any of the GUI configurations and forms. /hsKEY -- uses the passed parameter as the license group code for each installation of hsCADCreator. On initialization, hsCADCreator will attempt to automatically activate a seat from the license group pool. This simplifies deployment of hsCADCreator across large groups of users.

Also See:

3.2.1 Activating a Seat

Web Activation:

Enter the "**License Group Code**" you received in your confirmation email (online purchases) or the "**License Group Code**" on the cd case (media purchases), a unique "**Seat Name**" and press "**Activate**". Hachisoft Corporation will automatically check the validity of the "**License Group Code**" and generate an "**Activation Code**" that will unlock **hsCADCreator**. Press "**Done**" to use **hsCADCreator...** licensing is now complete.

Phone Activation:

Call Hachisoft Corporation at 1 877 685 0760. Tell the operator the "**License Group Code**" you received in your confirmation email (online purchases) or the "**License Group Code**" on the cd case (media purchases), the "**Site Code**", "**Machine Code**", and a unique "**Seat Name**". The operator will check the validity of the "**License Group Code**" and generate an "**Activation Code**" that will unlock **hsCADCreator**. Enter the "**Activation Code**" that they told you and press "**Activate**". Press "**Done**" to use **hsCADCreator...** licensing is now complete.

3.2.2 Remove a Seat

Web Removal:

Press "**Remove**" to remove this license from current machine and restore into the "**License Group**" pool. Please note that the application will close on successful removal and you will be required to activate a seat to use **hsCADCreator** on this computer. A dialog will ask you if you really want to remove the seat. Press "**Yes**" to remove this seat of **hsCADCreator** and return it to the "**License Group**" pool. **hsCADCreator** will return to its evaluation mode after a successful seat removal.

Phone Removal:

Call Hachisoft Corporation at 1 877 685 0760. Tell the operator the "**License Group Code**", "**Site Code**", and "**Machine Code**". Please note that the application will close on successful removal and you will be required to activate a seat to use **hsCADCreator** on this computer. A dialog will ask you if you really want to remove the seat. Press "**Yes**" to remove this seat of **hsCADCreator**. Tell the operator the "**Removal Code**" that was generated. The operator will return it to the "**License Group**" pool. **hsCADCreator** will return to its evaluation mode after a successful seat removal.

3.3 Trial Extension

Thanks for trying our software!   We want you to feel comfortable with our products and appreciate you taking the time to do a thorough evaluation!

If you need more time simply click 15 on **Request Trial Extension** button on the **hsCADCreator** startup screen.

This will open **hsCADCreator**'s **New Trial Extension** website in your browser. Fill in required information on the form and click on the **Submit** button. This will redirect you to a page listing your trial extension as well as send you a **Trial Extension** code at your specified email address.

Note that the **Product and Version**, **Site Code**, and **Machine Code** will automatically be filled with your information.

3.4   License Agreements

- hsCADCreator End-User License Agreement

hsCADCreator utilizes the following open source libraries:

- Apache Xerces (XML processing library): Apache License Agreement
- OpenCascade (Modeling library): OpenCascade License Agreement

3.4.1   hsCADCreator End-User License Agreement

hsCADCreator **End-User License Agreement**

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3.4.2   Apache License Agreement

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4   Frequently Asked Questions

If your question is not answered here? Please contact our Support 313.

FAQs

Q: I bought a license key for hsCADCreator but whenever I run the program it still says "Evaluation Version". How can I unlock and enable my licensed copy of software?

▼ Answer

hsCADCreator has two type of downloads available for install. You have "Evaluation Version" download installed on computer. Please download and install the "Full Version" of **hsCADCreator**. When you run the "Full Version" **hsCADCreator** first time, it will ask you to enter your licensing information and unlock the software.

Q: After every 5 minutes, whichever tool I am using gets Reset or Restarted. What is wrong?

▼ Answer

hsCADCreator has Auto Save feature. The default time settings for Auto Save is 5 minutes. Whenever Auto Save saves any file, it resets currently active tool. You can disable Auto Save feature or increase the time interval from **Options** 99 ➡ **Settings...** Menu item.

Q: I added few objects 50 (Layer, Linetype, Textstyle, Dimension Style, etc.) using Library Tools/Manage Diaglogs 217. When I close the dialog, those objects 50 are not in drawing for use, as if they were never created. What is wrong?

▼ Answer

You might have pressed "**Cancel**" button or "**Esc**" key on your keyboard after you were done adding your objects 50 through Library Tools/Manage Diaglogs 217. **hsCADCreator** assumes that when you "**Cancel**" or "Es**c**" any Library Tools/Manage Diaglogs 217, you want to **cancel** all your changes that has been made through that dialog. After adding desired objects 50 through Library Tools/Manage Diaglogs 217, use "**OK**" button in order to save your changes to drawing.

Q: I recently changed my linestlye(linetype) 66 in drawing to Dashed (or any other linetype) and saved, when I reopened my drawing linestyle was lost. It shows solid line instead of my selected linestyle(linetype). What is wrong?

▼ Answer

Nothing is wrong. Just try refresh your view with "Regen" tool and your saved linestyle(linetype) will be shown properly.

Q: I recently drew few entities but they are not showing on drawing. What is wrong?

▼ Answer

Check and make sure that your entities are not being drawn on a hidden layer. The current layer is shown in tool properties 260 under "Entity Defaults".

Q: My circle looks more like a Hexagon. What is wrong?

▼ Answer

Just try refreshing your view with "Regen" tool.

Q: My middle mouse doesn't work for "Pan |178〕**". What is wrong?**
▼ Answer
You might have mouse software installed that allows you to configure actions on different mouse buttons. Change Middle Button's action to "Middle Click" to be able to "Pan" using middle mouse button. These settings are accessible in Mouse Properties of your Operating System. Mouse Properties are located under Control Panel.

Q: Entity snap settings |72〕 **are not saved when i close hsCADCreator. Every time I start hsCADCreator I have to reselect my entity snaps through Snaps Tab** |261〕**. Is there any way to save my entity snaps?**
▼ Answer
The snap settings shown on Snaps Tab |261〕 are only valid for current drawing. If a user wants to set **hsCADCreator's** Entity Snaps they can be accessed from **Options** |99〕 ➡**Settings...** Menu item. Every time a new drawing is opened, it sets default entity snaps from **hsCADCreator's** Entity Snaps. Snap settings changed through **Options** |99〕 ➡**Settings...** Menu item will be saved with **hsCADCreator** and will be activated everytime a new drawing is opened.

Q: I imported different linetypes from a .LIN file and changed linetype for one of my entities. But I still see the entity being drawn with solid line and not with the linetype that I have set. What is wrong?
▼ Answer
May be your linetype scale is not set properly. Select the targeted entity and increase Linetype Scale under selection properties |259〕.

Q: Why am I not able to hide a block insertion entity |46〕 **even though its layer** |28〕 **is hidden?**
▼ Answer
If the block definition |46〕 is made up of entities that are on layers |28〕 other than "0", those entities retain their original layer |28〕 behaviour. To have correct behaviour on hide/show layer |28〕 and selection for block insertion entities, always make blocks from entities those are on layer |28〕 "0".

Q: Why am I not able to select a block insertion entity |46〕 **in my drawing?**
▼ Answer
Make sure that layer |28〕 of block insertion entity |46〕 is not hidden/frozen |28〕. If the block definition is made up of entities that are on layers other than "0", those entities retain their original layer behaviour. To have correct behaviour on hide/show layer and selection for block insertion entities, always make blocks from entities those are on layer "0".

Q: I want to delete Defpoints layer. But when I try to access it through Layer Manager Dialog, the "delete" button is not visible. How can I delete this layer?
▼ Answer
Defpoints layer is created by **hsCADCreator.** This layer defines points used by dimension entitiies |33〕. Since Defpoints layer is created and used by system, this layer should not be deleted. To restrict access to system created layers, the "delete" button is disabled. You can not delete this layer.

Q: Why I am not able to see the Grip Points |74〕 **for the selected entities?**
▼ Answer
Grip Points |74〕 for entities are only shown when:
    1. Only Selection Tool |10〕 is active. i.e. You are not using Selection Tool |10〕 as part of Move Tool |148〕, Rotate Tool |150〕, etc.
    2. The selected entities are not on OFF/FROZEN/LOCKED layer |64〕.

Q: I have a Block Insertion Entity |46〕 **on "Blocks" layer** |64〕**. When I turn OFF** |243〕 **"Blocks" layer** |64〕**, my Block Insertion Entity** |46〕 **is still visible. How can I hide Block Insertion Entity** |46〕 **on**

"Blocks" layer 64 **?**

▼ Answer

Block Insertion Entity 46 is always created from Block Definition Object 61. Block Definition Object 61 preserves original layer 64 of its individual entities 21 except system default layer ("0"). Due to this reason, eventhough Block Insertion Entity 46 is on "Blocks" layer 64, its' individual entities are on different layers 64 (original layers of entities). You can hide entities by either turning OFF the original layer, OR by FREEZING 243 "Blocks" layer.

Q: The Angular Dimension Tool 134 **rounds to the closest whole number. How can I change that to nearest tenths, or nearest hundredth. For instance the NE direction is 22.5 degrees (compass).**

▼ Answer

This can be achieved by creating a Dimension Style Object 51 for required format. By default, the Angular Dimension Tool 134 uses "ISO-25" Dimension Style Object 51. ISO-25 dimension style specifies decimal places in angular dimension as 0.

1. Use Manage Dimension Styles 221 through menu 96 or Library Toolbar 257 to access Dimension Style Dialog 221. Create New dimension and select it. Change name to required name(eg. myAnglularDimension). Scroll down on right hand pane to find "Dimension Text" properties. Click on "+" sign in front of "Dimension Text" to expand it. Set "Text Size" property to an appropriate value(eg. 2.0). Scroll down on right hand pane to find "Angular Dimensions" properties. Click on "+" sign in front of "Angular Dimensions" to expand it. Change "Precision (Angle)" property to required number of decimal places in angular measurments. Now activate Angular Dimension Tool 134. Specify Dim Style to be required dimension style(eg. myAngularDimension). Create new dimension.

2. Alternately, to change property on already created "Angular Dimension Entity", select the entity and edit its "Angular Dimensions" property to set "Precision (Angle) to required decimal places.

Q: While creating Dimension Entities 33 **using Dimension Tools** 123 **, I select "Show Unit" but the created dimension only shows number. It doesn't show unit along with it. How can I display unit?**

▼ Answer

There can be 2 resolutions for this:

1. Check Drawing Settings 76. Dimension Entities 33 gets drawing's dimension unit from "Units" property of Drawing Settings 76. If "Units" is undefined/unspecified/empty Dimension Entities 33 can not display unit text along with dimension. Change "Units" to appropriate unit to be used and create dimensions again.

2. Use Manage Dimension Styles 221 to create Dimension Style Object 51. Set all properties as required. Change "Prefix and/or suffix" property in "Dimension Text" to set the Unit for measurment. For detailed explanation see description of Dimension Style Object 51. Select this newly created Dimenstion Style Object when creating any dimension entity 33 and unselect "Show Unit" property.

Q: I've changed my units in the property tree, and then drag an entity grip. Why did my units change back?

▼ Answer

The property tree gets rebuilt when a grip is used. The units will return to the drawing default for that unit class.

Q: I am not able to use Nearest Entity Snap 71 **on an entity not on UCS-plane** 17 **, Why?**

▼ Answer

The **Nearest Entity Snap** 71 only works on current/active **UCS-plane** 17. Change your **UCS** 17 to align with entity in question using modify UCS tools 193 and then try to use **Nearest Entity Snap** 71.

Q: I used <u>Trim Tool</u> 157 to trim more than one <u>entities</u> 21 at a time(using box selection), but the tool trimmed some entities on wrong side of trimming line. Why?

▼ Answer

<u>Trim Tool</u> 157 trims entities on the side that is closest to the point where mouse button is released (in order to finish the box selection). Try trimming single entity or only few entities at at time such that mouse release point is close to the side that needs to be trimmed.

Q: I used <u>Measure Area Tool</u> 247 to measure area of a non-planar/self intersecting polygon, but it shows me the area as Zero or Negative value. What is correct area?

▼ Answer

<u>Measure Area Tool</u> 247 can not measure area of non-planar/self intersecting polygons correctly. The best way to calculate correct area is to dis-integrate the required polygon into simple polygons conceptually and then use <u>Measure Area Tool</u> 247 with Continuous Mode ON to find total area of polygons.

5   Support and Feedback

The most current support details can be found on the web at <u>Hachisoft Support Website</u>.

Email Support:

The primary form of support contact is via email. Our support staff can be reached at: <u>support@hachisoft.com</u>

Forums:

At <u>Hachisoft Support Website</u> we have user forums for our products. Please use these to check for known bugs, view upcoming update details and progress, ask questions and give feedback.

Phone Support:

If the other support options are insufficient, then phone support is also available:

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6   Version History

Visit <u>hsCADCreator online version history</u> to view the latest **hsCADCreator** changes.

Index

- . -

- 2 -

- 3 -

- A -

- B -

- C -

- W -

- Z -