Lesson 9
Auxiliary Views and Editing with GRIPS

Learning Objectives

- Use 2D Projection Method to Draw Auxiliary Views
- Create Rectangles
- Use the Basic GRIPS Editing Commands
- Create and Edit the Plot Style Table
- Set Up and Use the Polar Tracking Option
- Create Multiple Viewports in Paper Space
AutoCAD Certified User Examination Objectives Coverage

This table shows the pages on which the objectives of the Certified User Examination are covered in Chapter 9.

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>Creating Basic Drawings</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Polar Tracking</td>
<td>9-12</td>
</tr>
<tr>
<td>3</td>
<td>Manipulating Objects</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Grips Editing</td>
<td>9-6</td>
</tr>
<tr>
<td></td>
<td>Copy Selections, Grips</td>
<td>9-6</td>
</tr>
<tr>
<td></td>
<td>Rotate, Grips</td>
<td>9-7</td>
</tr>
<tr>
<td>4</td>
<td>Drawing Organization and Inquiry Commands</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Layers Controls</td>
<td>9-5</td>
</tr>
<tr>
<td>5</td>
<td>Altering Objects</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Trim Command</td>
<td>9-18</td>
</tr>
<tr>
<td>12</td>
<td>Plotting Your Drawings</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Plot Style Table Settings</td>
<td>9-25</td>
</tr>
</tbody>
</table>
Introduction

An important rule concerning multiview drawings is to draw enough views to accurately describe the design. This usually requires two or three of the regular views, such as a front view, a top view and/or a side view. Many designs have features located on inclined surfaces that are not parallel to the regular planes of projection. To truly describe the feature, the true shape of the feature must be shown using an auxiliary view. An auxiliary view has a line of sight that is perpendicular to the inclined surface, as viewed looking directly at the inclined surface. An auxiliary view is a supplementary view that can be constructed from any of the regular views. This lesson will demonstrate the construction of an auxiliary view using various CAD techniques.

In this lesson, we will examine the use of the very powerful AutoCAD GRIPS feature. In AutoCAD 2015, a GRIP is a small square displayed on a pre-selected object. Grips are key control locations such as the endpoints and midpoints of lines and arcs. Different types of objects display different numbers of grips. Using grips, we can stretch, move, mirror, scale, rotate, and copy objects without entering commands or clicking toolbars. Grips reduce the keystrokes and object selection required in performing common editing commands. To edit with grips, we select the objects before issuing any commands. To remove a specific object from a selection set that displays grips, we hold down the [SHIFT] key as we select the object. To exit the grips mode and return to the command prompt, press the [ESC] key.

The V-Block Design
The V-Block Example

Before going through the tutorial, make a rough sketch of a multiview drawing of the part. How many 2D views will be necessary to fully describe the part? Based on your knowledge of AutoCAD 2015 so far, how would you arrange and construct these 2D views? Take a few minutes to consider these questions and do preliminary planning by sketching on a piece of paper. You are also encouraged to construct the orthographic views on your own prior to going through the tutorial.

Starting Up AutoCAD 2015

1. Select the AutoCAD 2015 option on the Program menu or select the AutoCAD 2015 icon on the Desktop.

2. In the AutoCAD 2015 Startup dialog box, select the Template option.

3. Select the Acad-A-H-Title template file from the list of template files. If the template file is not listed, click on the Browse button to locate and proceed to open a new drawing file.
Setting up the Principal Views

1. In the Layer Control box, set layer Construction as the Current Layer, if it is not the default layer.

2. In the Status Bar area, switch ON the following options in the status toolbar: GRID Display, Object Snap, Object Snap Tracking, Dynamic Input and Line Weight Display.

- We will first create construction geometry for the front view.

3. Select the Rectangle icon in the Draw toolbar. In the command prompt area, the message “Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:” is displayed.

4. Place the first corner point of the rectangle near the lower left corner of the screen. Do not be overly concerned about the actual coordinates of the location; the CAD drawing space is a very flexible virtual space.

5. Create a 3" × 2.25" rectangle. Using the Dynamic Input option, enter @3,2.25 [ENTER].

- The Rectangle command creates rectangles as polyline features, which means all segments of a rectangle, are created as a single object.
6. Next, we will use the GRIPS editing tools to make a copy of the rectangle. Pick any edge of the rectangle we just created. Notice that small squares appear at different locations on the rectangle.

7. Inside the Drawing Area, right-mouse-click to bring up the popup option menu.
   - In the center section of the popup menu, the set of GRIPS editing commands includes Erase, Move, Copy Selection, Scale, and Rotate.

8. In the popup menu, select the Copy Selection option.

9. In the command prompt area, the message “Specify base point or displacement, or [Multiple]:” is displayed. Pick the lower right corner as the base point. A copy of the rectangle is attached to the cursor at the base point.

10. In the command prompt area, the message “Specify second point of displacement, or <use first point as displacement>:” is displayed. Enter: @0,0.75 [ENTER].
   - This will position the second rectangle at the location for the vertical 30-degree angle.

11. Inside the Drawing Area, right-mouse-click once to bring up the option menu and choose Enter to exit the Copy option.
12. **Pre-select** the **copy** by picking the top horizontal line on the screen. The second rectangle, the copy we just created, is selected.

13. Inside the Drawing Area, **right-mouse-click** to bring up the popup option menu and select the **Rotate** option.

14. In the command prompt area, the message “Specify base point:” is displayed. Pick the **lower right corner** of the **selected rectangle** as the base point.

15. In the command prompt area, the message “Specify the rotation angle or [Reference]:” is displayed. Enter: **30 [ENTER]**.
Setting up the Top View

1. Select the **Rectangle** icon in the *Draw* toolbar. In the command prompt area, the message “Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:” is displayed.

2. Move the cursor over the top left corner of the first rectangle we created. This will activate the **object tracking** alignment feature to the corner.

3. Move the cursor upward to a location that is about 1.5” away from the reference point. (Read the OTRACK display on the screen.) Left-click once to place the first corner-point of the rectangle.

4. We will create a 3” x 2” rectangle. Enter: @3,2 [ENTER].

- We have created the outline of the top view of the V-block design.

5. Pre-select the **rectangle** we just created by left-clicking any edge of the rectangle.

6. Select the **Explode** icon in the *Modify* toolbar.

- The top rectangle now consists of four separate line segments.
Using the OFFSET Command

1. Click the Offset icon in the Modify toolbar. In the command prompt area, the message “Specify offset distance or [Through]:” is displayed.

2. In the command prompt area, enter: 0.2 [ENTER].

3. In the command prompt area, the message “Select object to offset or <exit>:” is displayed. Pick the top horizontal line of the top view on the screen.

4. AutoCAD next asks us to identify the direction of the offset. Pick a location that is below the selected line.

5. In the command prompt area, the message “Select object to offset or <exit>:” is displayed. Pick the bottom horizontal line of the top view on the screen.

6. AutoCAD next asks us to identify the direction of the offset. Pick a location that is above the selected line.

7. Inside the Drawing Area, right-mouse-click and select Enter to end the Offset command.

8. Inside the Drawing Area, right-mouse-click to bring up the popup option menu and select the Repeat Offset option.

   - Notice in the popup menu, none of the GRIPS editing commands are displayed; the GRIPS editing commands are displayed only if objects are pre-selected.
9. In the command prompt area, the message “Specify offset distance or [Through]:” is displayed. Enter: 0.75 [ENTER].

10. In the command prompt area, the message “Select object to offset or <exit>:” is displayed. Pick the top horizontal line of the top view on the screen.

11. AutoCAD next asks us to identify the direction of the offset. Pick a location that is below the selected line.

12. In the command prompt area, the message “Select object to offset or <exit>:” is displayed. Pick the bottom horizontal line of the top view on the screen.

13. AutoCAD next asks us to identify the direction of the offset. Pick a location that is above the selected line.

14. Inside the Drawing Area, right-mouse-click and choose Enter to end the Offset command.

- The four parallel lines will be used to construct the top v-cut feature and the two 0.75" x 30° cut features at the base of the V-block in the top view.
Creating Object Lines in the Front View

1. On the Object Properties toolbar, choose the Layer Control box with the left-mouse-button.

2. Move the cursor over the name of layer Object_Lines; the tool tip “Object_Lines” appears.

3. Left-mouse-click once and layer Object_Lines is set as the Current Layer.

4. Select the Line command icon in the Draw toolbar. In the command prompt area, the message “_line Specify first point:” is displayed.

5. Pick the lower left corner of the bottom horizontal line in the front view as the starting point of the line segments.

6. Pick the lower right corner of the bottom horizontal line in the front view as the second point.

7. Select the third and fourth points as shown in the figure below.
Setting the **Polar Tracking** Option

1. In the Status Bar area, turn **ON** the Polar Tracking option.

- Note that the Polar Tracking option is one of the AutoCAD AutoTrack™ features. The AutoTrack features include two tracking options: polar tracking and object snap tracking. When the Polar Tracking option is turned on, alignment markers are displayed to help us create objects at precise positions and angles. A quick way to change the settings of the AutoTrack feature is to use the option menu.

2. Move the cursor on top of the Polar Tracking option in the Status Bar area.

3. Click once with the right-mouse-button to bring up the option menu.

4. Select **Settings** in the option menu as shown in the figure.

5. In the Drafting Settings dialog box, set the Increment angle to **30** as shown in the figure below.

6. Under the **Object Snap Tracking Settings** option, turn **ON** the **Track using all polar angle settings** option as shown in the figure.

7. Click **OK** to accept the modified settings.
8. Move the cursor near the left vertical line as shown and notice that *AutoCAD AutoTrack* automatically snaps the cursor to the intersection point and displays the alignment marker as shown.

➢ In the following steps, we will illustrate the use of different *POLAR* settings to achieve the same result.

9. Click once with the right-mouse-button on the *POLAR* option in the *Status Bar* area to bring up the option menu.

10. Select *Settings* in the option menu as shown in the figure.

11. In the *Drafting Settings* dialog box, set the *Increment angle* to 90 as shown in the figure below.

12. Under the *Object Snap Tracking Settings* option, turn ON the *Track orthogonally only* option as shown in the figure.

13. Under the *Object Snap Tracking Settings* option, turn ON the *Relative to last segment* option as shown in the figure above.

14. Click OK to accept the modified settings.
15. Move the cursor near the left vertical line and notice the AutoTrack feature automatically snaps the cursor to the intersection point and displays the alignment marker as shown.

- On your own, experiment with changing the settings to achieve the same SNAP/POLAR results.

16. Left-click at the intersection point as shown.

17. Click the Close option in the command prompt area. AutoCAD will create a line connecting the last point to the first point of the line sequence.
Setting up an Auxiliary View

1. **Pre-select** all objects in the top view by enclosing the objects inside a selection window.

2. Inside the *Drawing Area*, right-mouse-click to bring up the popup option menu and select the **Copy Selection** option.

3. In the command prompt area, the message “Specify base point or displacement, or [Multiple]:” is displayed. Pick the **lower left corner** of the top view as the base point.

4. Using the *AutoTrack* feature, place the copy of the top-view by aligning it to the inclined object line we just created. **Left-click** once to position the copy about 2” away from the top corner of the front-view.
Aligning the Auxiliary View to the Front View

1. **Pre-select** all objects in the auxiliary view by enclosing the objects inside a selection window.

2. Inside the Drawing Area, right-mouse-click to bring up the popup option menu and select the Rotate option.

3. In the command prompt area, the message "Specify base point:" is displayed. Pick the **bottom left corner** of the auxiliary view as the base point.

4. In the command prompt area, the message "Specify the rotation angle or [Reference]:" is displayed. Enter: -60 [ENTER].

Creating the V-cut in the Auxiliary View

1. Select the **Line** icon in the Draw toolbar. In the command prompt area, the message "_Line Specify first point:" is displayed.

2. Pick the **top corner** of the inclined object line in the front view as the starting point of the line segments.

3. Pick the **second top end point** in the auxiliary view as the second point.
4. Inside the *Drawing Area*, right-mouse-click and select **Enter** to end the *Line* command.

5. Pre-select the line we just created.

6. Inside the Drawing Area, right-mouse-click to bring up the popup option menu and select the **Rotate** option.

7. In the command prompt area, the message "**Specify base point:**" is displayed. Pick the top right endpoint of the line as the base point.

8. In the command prompt area, the message "**Specify the rotation angle or [Reference]:**" is displayed. Enter: 45 [ENTER].

9. On your own, repeat the above steps and create the other line as shown.
10. Select the **Trim** command icon in the **Modify** toolbar. In the command prompt area, the message “Select boundary edges... Select objects:” is displayed.

11. Pick the two inclined lines we just created in the auxiliary view as the boundary edges.

12. Inside the **Drawing Area**, **right-mouse-click** to proceed with the Trim command. The message “Select object to trim or [Project/Edge/Undo]:” is displayed in the command prompt area.

13. Pick the two lower endpoints of the two inclined lines to remove the unwanted portions.

14. Inside the Drawing Area, right-mouse-click to activate the option menu and select **Enter** with the left-mouse-button to end the **Trim** command.

The V-cut is shown at its true size and shape only in the auxiliary view. It is therefore necessary to create the V-cut in the auxiliary view. Now that we have constructed the feature in the auxiliary view, we can use projection lines to transfer the feature to the front view and top view.
Creating the V-cut in the Front View and Top View

1. On the Object Properties toolbar, choose the Layer Control box with the left-mouse-button.

2. Move the cursor over the name of layer Hidden_Lines; the tool tip “Hidden_Lines” appears.

3. Left-mouse-click once and layer Hidden_Lines is set as the Current Layer.

4. Select the Line command icon in the Draw toolbar. In the command prompt area, the message “_line Specify first point:” is displayed.

5. Pick the vertex of the V-cut in the auxiliary view as the first point of the line.

6. Inside the Drawing Area, hold down the [SHIFT] key and right-mouse-click once to bring up the Object Snap shortcut menu.

7. Select the Perpendicular option in popup window.

8. Move the cursor to the front view on the inclined edge and notice the perpendicular symbol appears at different locations. Select the intersection point on the inclined line as shown.
9. Inside the *Drawing Area*, right-mouse-click to activate the option menu and select *Enter* with the left-mouse-button to end the *Line* command.

10. On your own, use the **Trim** and **Extend** commands to adjust the hidden lines in the front view.

> On your own, first create the three construction lines and then construct the V-cut feature in the top-view. Use the **Trim** and **Extend** commands to assist the construction.
Setting the **POLAR TRACKING** Option

1. Move the cursor to the *Status Bar* area, over the *Polar Tracking* option button.

2. **Left-mouse-click once** on the triangle icon to bring up a popup option menu.

3. Select the **30, 60, 90, 120...** option by clicking once with the left-mouse-button. This is the shortcut to set the angles for the *Polar Tracking Angle Setting*.

   - Notice the other settings that are available. We will use the absolute polar angle measurement for this example.

Completing the Top View

1. Click on the **Zoom Realtime** icon in the *View display* toolbar area.

2. Move the cursor near the center of the *Drawing Area*.

3. **Push and hold down the left-mouse-button**, then move upward to enlarge the current display scale factor. (Press the [Esc] key to exit the command.)

4. On your own, use the **Pan Realtime** option to reposition the display so that we can work on the top view of the V-block.

5. Select the **Line** icon in the *Draw* toolbar.
6. In the command prompt area, the message "line Specify first point:" is displayed. Pick the right endpoint of the third horizontal line in the top view as the starting point of the line segments.

7. Move the cursor toward the top horizontal line and observe the AutoTracking markers over different locations.

8. Left-click at the intersection of the polar tracking and the top horizontal line as shown. Do not select the intersection between the top horizontal line and the vertical line. (Use the Zoom Realtime command to zoom-in further, if necessary.)

9. Inside the Drawing Area, right-mouse-click and select Enter to end the Line command.

10. Repeat the above steps and create the other inclined line in the top view.

➢ On your own, complete the top view by adding all the necessary object lines in the top view. Use the Trim and Extend commands to assist the construction.

- Notice that the two 30° cut features are shown as true size and shape only in the top view and therefore it is necessary for us to construct the features in the top view.
On your own, create the **vertical construction line** through the corner of the 30° cut as shown.

11. Complete the front view by adding the object line along the construction line as shown.
On your own, complete the views by adding all the necessary object lines in the views.

Complete the drawing by adding the proper dimensions.
Edit the Plot Style Table

1. Inside the Drawing Area, right-mouse-click and select Options in the popup menu.

2. In the Options dialog box, select the Plotting tab.

3. Click on the Plot Style Table Settings button as shown.

4. In the Plot Style Table Settings dialog box, switch ON the Use named plot styles option as shown.

5. Choose the acad.stb as the default plot style table as shown.

6. In the Options dialog box, click on the Add or Edit Plot Style Tables button.

7. The Plot Styles folder appears on the screen.

8. Double-click the acad.stb icon with the left-mouse-button to open the plot style file.
9. In the Plot Style Table Editor, select the Table View tab.

![Plot Style Table Editor - acad-styles.png](image)

- The Plot Style Table Editor displays the plot styles that are in the current plot style table. The Table View and Form View tabs provide two methods to modify the existing plot style settings. Both tabs list all of the plot styles in the plot style table and their settings. In general, the Table View tab is more convenient if there are only a small number of plot styles. We can modify plot style color, screening, linetype, linewidth, and other settings. The first plot style in a named plot style table is Normal and represents an object's default properties (no plot style applied). We cannot modify or delete the Normal style.

10. Change the Color setting for Style 1 to Black, so that all layers using this plot style will print using black.

11. Pick the Save & Close button to accept the settings and exit the Plot Style Table Editor.

> On your own, print out a copy of the V-block drawing using the modified plot style table.
Review Questions: (Time: 25 minutes)

1. What is an auxiliary view and why would it be important?

2. What is a GRIP? What are the advantages of using the GRIPS?

3. List three GRIPS editing commands you have used in the tutorial.

4. What does the Polar Tracking option allow us to do?

5. Find the area A defined by the two arcs, as described in the figure below.

6. Find the area A defined by the three arcs, as described in the figure below.
Exercises: (Time: 150 minutes)

1. Angle Base (Dimensions are in inches.)

2. Indexing Guide (Dimensions are in inches.)
3. **Spindle Base** (Dimensions are in millimeters.)

4. **Transition Support** (Dimensions are in inches.)