Lesson 8
Parametric Drawing Tools

Learning Objectives

- Understand the Basics of Parametric Modeling
- Apply the Geometric Constraints Manually and Automatically
- Use the Dimensional Constraints command
- Control Geometry through the use of Constraints
- Show and Hide the Applied Constraints
AutoCAD Certified User Examination Objectives Coverage

This table shows the pages on which the objectives of the Certified User Examination are covered in Chapter 8.

**Section 1: Introduction to AutoCAD**
- Show All Constraints ................................................................. 8-33
- Show Dynamic Constraints ....................................................... 8-33

**Section 2: Creating Drawings**
- Geometric Constraints ............................................................... 8-9
- Dimensional constraints ............................................................ 8-12
- Infer Constraints ........................................................................ 8-34

**Section 3: Manipulating Objects**
- Geometric Constraints ............................................................... 8-10
- Dimensional constraints ............................................................ 8-12
- Mirror command ....................................................................... 8-25
- Auto Constrain command ......................................................... 8-28

**Section 4: Drawing Organization and Inquiry Commands**
- Layers Controls ......................................................................... 8-5

**Section 5: Altering Objects**
- Mirror command ....................................................................... 8-25
- Trim command ........................................................................... 8-26
Introduction

One of the most important advancements in the CAD/CAE technology was the invention of parametric modeling tools in the late 1980s. The introduction of the parametric technology revolutionized the CAD industry by allowing users to use CAD software as true design tools. The parametric modeling approach has elevated the traditional CAD technology to the level of a very powerful design tool. Parametric modeling techniques can be used to automate the design and revision procedures; this is done through the use of parametric features. Parametric features control the model geometry by the use of design variables. In 2009, a new set of parametric drawing tools was first introduced in AutoCAD. The word parametric means that the geometric definitions of the design, such as dimensions, can be varied at any time during the design process. The concept of parametric modeling makes the way CAD works more closely match the actual design-manufacturing process than the mathematics of a CAD program. By using the parametric drawing tools available in AutoCAD 2015, CAD drawings can now be updated more easily when the design is refined.

The main characteristic of parametric modeling involves the use of Constraints. Constraints are geometric rules and restrictions applied to 2D geometry.

There are two general types of constraints: Geometric constraints and Dimensional constraints.

Geometric constraints are used to control the geometric relationships of objects with respect to each other; for example, a line that is tangent to an arc, a line that is horizontal, or two lines that are collinear.

Dimensional constraints are used to control the size and location of geometric entities; for example, the distance between two parallel lines, the length of a line, the angle of two lines, or radius values of arcs.

When a design is created or changed, a drawing will be in one of three states:
- **Unconstrained**: No constraints are applied to the constructed geometry.
- **Under constrained**: Some constraints, but not all, are applied to the constructed geometry.
- **Fully constrained**: The necessary definitions of the design have been properly applied to the constructed geometry, which means all relevant geometric and dimensional constraints are present.

Note that AutoCAD will prevent the user from applying any constraints that result in an over-constrained condition (having duplicating or conflicting definitions.)

Generally speaking, during the initial stages of a design process, creating an under constrained, or even unconstrained, drawing can be very beneficial in helping a designer to determine the forms and shapes of the design. But as the design begins to reach the final stages, a fully constrained drawing is necessary eventually, as this will assure the manufacturability of the finalized design.
In this lesson, we will examine the use of the very powerful *parametric drawing tools* that are available in **AutoCAD 2015**. The parametric drawing tools can be used to (1) assist the construction of designs, especially when more complex geometric relations are present, and (2) help maintain the design intents and thus ease the tedious tasks involved in design modifications.

The concepts and procedures described in this chapter represent the basic techniques involved in the use of 3D parametric modeling software, such as the *Autodesk Inventor 2015* software.

*Note the parametric drawing tools allow users to concentrate on the design itself. With the parametric tools, the users now have full control of the specific geometric properties, as well as the size and location definitions. This approach can be quite effective, as it can be used to supplement traditional geometric construction techniques.*
Starting Up AutoCAD 2015

1. Select the AutoCAD 2015 option on the Program menu or select the AutoCAD 2015 icon on the Desktop. Once the program is loaded into the memory, the AutoCAD 2015 drawing window will appear on the screen.

2. In the Startup window, select Start from Scratch, as shown in the figure below.

3. In the Default Settings section, pick Metric as the drawing units.

4. On your own, set the display to no digits after the decimal point and also both the Snap and Grid options to 10 for both X and Y directions.

Layers Setup

1. Pick Layer Properties Manager in the Layers toolbar.

2. Click on the New icon to create new layers.

3. Create two new layers with the following settings:

<table>
<thead>
<tr>
<th>Layer</th>
<th>Color</th>
<th>Linetype</th>
<th>Lineweight</th>
</tr>
</thead>
<tbody>
<tr>
<td>Construction</td>
<td>White</td>
<td>Continuous</td>
<td>Default</td>
</tr>
<tr>
<td>Center</td>
<td>Red</td>
<td>CENTER</td>
<td>Default</td>
</tr>
<tr>
<td>Object</td>
<td>Blue</td>
<td>Continuous</td>
<td>0.3mm</td>
</tr>
</tbody>
</table>

4. On your own, set layer Construction as the Current Layer.

5. In the Status Bar area, reset the option buttons so that only GRID Display is switched ON.
Creating Rough Sketches

Quite often during the early design stage, the shape of a design may not have any precise dimensions. Most conventional CAD systems require the user to input the precise lengths and locations of all geometric entities defining the design, which are not available during the early design stage. With parametric modeling, we can use the computer to elaborate and formulate the design idea further during the initial design stage. With Autodesk parametric drawing tools, we can use the computer as an electronic sketchpad to help us concentrate on the formulation of forms and shapes for the design. This approach is the main advantage of parametric drawing over conventional CAD drawing techniques.

As the name implies, a rough sketch is not precise at all. When sketching, we simply sketch the geometry so that it closely resembles the desired shape. Precise scale or lengths are not needed. AutoCAD provides us with many tools to assist us in finalizing sketches. For example, geometric entities such as horizontal and vertical lines can be set at anytime. Here are some general guidelines for creating sketches in AutoCAD 2015:

- **Create a sketch that is proportional to the desired shape.** Concentrate on the shapes and forms of the design.

- **Keep the sketches simple.** Initially, leave out the small geometry features such as fillets, rounds and chamfers. Add those in after the major parts of the sketch have been established.

- **Exaggerate the geometric features of the desired shape.** For example, if the desired angle is 85 degrees, create an angle that is 50 or 60 degrees. It is not necessary to construct everything precisely.

- **Confirm all necessary constraints are maintained correctly throughout the construction/editing.** In parametric modeling, editing geometry may cause the removal of certain applied constraints by the system. For example, trimming a line may cause the removal of a tangent constraint which was applied earlier.

Note that in AutoCAD 2015, it is also feasible to use the parametric drawing tools on precisely constructed geometry. In this tutorial, to illustrate the concepts of geometric constraints, a set of randomly created geometry will be used.

1. Click on and switch to the **Home** tab in the **Ribbon** tabs and panels area.

2. Select the **Circle – Center, Radius** command icon in the **Draw** toolbar. In the command prompt area, the message "_circle Specify center point for the circle or [3P/2P/Ttr (tan tan radius)]:" is displayed.
3. On your own, create two circles of arbitrary sizes near the center of the screen as shown.

4. Select the **Line** command icon in the *Draw* toolbar. In the command prompt area, near the bottom of the AutoCAD drawing screen, the message "*line Specify first point:*" is displayed. AutoCAD expects us to identify the starting location of a straight line.

5. On your own, create a line of arbitrary length just below the two circles as shown in the figure below.
Parametric Drawing Tools

1. In the Ribbon tabs area, left-mouse-click once on the Parametric Tools tab as shown.

- Three tool panels are available under the Parametric Tools tab: Geometric Constraints, Dimensional Constraints and the Manage panel.

- Geometric Constraints panel contains the tools to apply geometric constraints either manually or automatically.

- Dimensional Constraints panel contains the tools to applied dimensional constraints manually. Note the dynamic dimensions applied through this panel can be used.

- The Manage panel contains two tools: Delete Constraints to manually remove unwanted constraints and Parameters Manager to allow parametric equations be set up among dimensions.

- We will first use the Geometric Constraint tools to control the constructed geometry.
Applying Geometric Constraints

- In the Geometric Constraints panel, twelve types of constraints are available for 2D sketches.

**Coincident constraint:** Constrains two points together or one point to a curve.

**Collinear constraint:** Causes two lines or ellipse axes to lie along the same line.

**Concentric constraint:** Constrains two arcs, circles, or ellipses to the same center point.

**Fixed location constraint:** Constrains points or curves to a fixed location relative to the sketch coordinate system.

**Parallel constraint:** Causes selected lines or ellipse axes to lie parallel to one another.

**Perpendicular constraint:** Causes selected curves or ellipse axes to lie at right angles to one another.

**Horizontal constraint:** Causes lines, ellipse axes, or pairs of points to lie parallel to the X-axis of the sketch coordinate system.

**Vertical constraint:** Causes lines, ellipse axes, or pairs of points to lie parallel to the Y-axis of the sketch coordinate system.

**Tangent constraint:** Constrains two curves to be tangent to one another.

**Smooth (G2) constraint:** Create a curvature continuous (G2) condition between a spline and a line, arc, or spline.

**Symmetric constraint:** Constrains lines or curves to become symmetrically constrained about a selected line.

**Equal constraint:** Selected arcs/circles constrained to the same radius or selected lines to the same length.
1. Select the **Horizontal** constraint by clicking on the icon as shown in the figure.

2. Click on the line with the left-mouse-button to apply the constraint. The line is adjusted to the horizontal position as shown.

3. Select the **Coincident** constraint by clicking on the icon as shown in the figure.

4. Click on the larger circle with the left-mouse-button to apply the constraint.

5. Click on the left endpoint of the line with the left-mouse-button to apply the constraint. Note that may move the line or the circle to maintain the applied constraint.
Note that since both the circle and line can be moved, AutoCAD will move either geometry to satisfy the applied geometric property. We will take a different approach to have more controls in the editing of the geometry.

6. In the Quick Access toolbar, click the **Undo** button as shown.

7. Select the **Fix** constraint by clicking on the icon as shown in the figure.

8. Click near the left endpoint of the line with the left-mouse-button to apply the constraint.

   - Note the different constraint symbols next to the geometry.

9. Select the **Coincident** constraint by clicking on the icon as shown in the figure.

10. Click on the larger circle with the left-mouse-button to apply the constraint.

11. Click on the **left endpoint of the line** with the left-mouse-button to apply the constraint. Now the circle is moved and the alignment constraint is maintained.
Applying Dimensional Constraints

- In the *Dimensional Constraints* panel, a set of dimensional constraints are available; we can apply the dimensional constraints, such as Linear, Radial, or Angular dimensions, to control geometry.

  > By default, the **Show Constraints** option is activated, which will show all applied constraints on the screen.

1. Select the **Linear** dimension constraint by clicking on the icon as shown in the figure.

2. Inside the display area, right-mouse-click once to bring up the option menu and select **Object** as shown.

3. Select the **line** with a single click of the left-mouse-button.

4. Place the dimension **below the line** with a single left-mouse-button click as shown.

  > AutoCAD next expects us to edit the length of the line through the applied dimensional constraint.
5. Enter 96 as the new length of the line.

Note the length of the line is adjusted to the specified length.

6. Select the **Coincident** constraint by clicking on the icon as shown in the figure.

7. On your own, align the **center of the smaller circle** to the **right endpoint of the line** as shown in the figure below.

8. On your own, adjust the length of the line by **double-clicking on the dimension**. Experiment with different values and see how the geometry behaves as changes are made. Set the dimension back to 96 before continuing.
Additional Geometric and Dimensional Constructions

1. In the Ribbon tabs area, left-mouse-click once on the Home tab as shown.

2. Select the Line command icon in the Draw toolbar. In the command prompt area, near the bottom of the AutoCAD drawing screen, the message "line Specify first point:" is displayed. AutoCAD expects us to identify the starting location of a straight line.

3. On your own, create three connected line segments of arbitrary length just above the two circles as shown in the figure below.

4. In the Ribbon tabs area, left-mouse-click once on the Parametric Tools tab as shown.
5. Select the **Parallel** constraint by clicking on the icon as shown in the figure.

6. Select the line segment **on the left** as shown.

7. Select the **horizontal line** connected to the circle centers as shown.

- Note that although the selected line segment now becomes a horizontal line, parallel to the first line created, the line segment is no longer connected to the two other lines. Although it is feasible to apply additional Coincident constraints, a better option is to use the **Auto Constrain** command.

8. In the **Quick Access** toolbar, click the **Undo** button as shown.
17. On your own, apply the Perpendicular constraint to the line segment to the

32. Click on the **Home** tab in the **Ribbon** tabs area.

33. Click on the **3-Point arc** icon to activate the command.

34. On your own, create an arc of arbitrary size to the right side of the top horizontal line as shown.

35. In the **Ribbon** tabs area, left-mouse-click once on the **Parametric Tools** tab as shown.

36. Select the **Radius** dimensional constraint by clicking on the icon as shown in the figure.
37. Select the arc by clicking once with the left-mouse-button.

38. Enter 15 as the new radius of the arc. Note the geometry adjustment is done instantly.

39. Select the Coincident constraint by clicking on the icon as shown in the figure.

40. Select the top endpoint of the arc as shown.

41. Select the right endpoint of the top horizontal line as shown.

42. Select the Tangent constraint by clicking on the icon as shown in the figure.
43. Select the arc as the first object.

44. Select the horizontal line on the left as the second object.

45. Select the Coincident constraint by clicking on the icon as shown in the figure.

46. On your own, select the right endpoint of the arc and the horizontal line to set the constraint.
Using the **MIRROR** Command

1. Select the **Mirror** command icon in the **Modify** toolbar. In the command prompt area, the message “Select objects:” is displayed.

2. Use a *selection window* to select the objects on the upper portion as shown.

3. Inside the **Drawing Area**, right-mouse-click to accept the selection and continue with the Mirror command.

4. In the command prompt area, the message “Specify the first point of the mirror line:” is displayed. Pick the center point of the circle on the left. (Hint: Switch on the **Object Snap** option to assist the selection.)

5. In the command prompt area, the message “Specify the second point:” is displayed. Pick the center point of the circle on the right.

6. In the command prompt area, the message “Delete source objects? [Yes/No] <N>:” is displayed. Inside the Drawing Area, right-mouse-click and select **Enter** to retain the original objects.
Using the **TRIM** Command

- The Trim command shortens an object so that it ends precisely at a selected boundary.

1. Select the **Trim** command icon in the **Modify** toolbar. In the command prompt area, the message “Select boundary edges... Select objects:” is displayed.

   - First, we will need to select the objects that define the **boundary edges** to which we want to trim the object. If no item is selected, then all existing objects can be used as boundary edges.

2. Inside the **Drawing Area**, click once with the **right-mouse-button** to accept the default option and proceed with the Trim command.

3. The message “Select object to trim or shift-select object to extend or [Fence/Crossing/Project/Edge/eRase/Undo]:” is displayed in the command prompt area. Pick the **inside top segment of the circle on the right** as shown.

4. Pick the **left side of the arc** to trim the lower left portion of the arc.

5. On your own, trim the other circle to remove the inside portion.
6. On your own, continue to trim the horizontal lines on the right, and the drawing should appear as shown in the figure below.

Note that all constraints appear to be intact prior to exiting the Trim command.

7. 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.

Note that some of the constraints, both geometric and dimensional constraints, are removed as we exited the Trim command. The constraints are properties applied to the geometry, and the associated constraints will be removed when the geometry is edited/changed.
Using the Auto Constrain Command

1. Select the Auto Constrain command by clicking on the icon as shown in the figure.

2. Select all of the constructed objects by using a selection window as shown.

3. Inside the Drawing Area, right-mouse-click to bring up the option menu and select Enter to accept the selection and continue with the Auto Constrain command.

- Since the geometric entities were constructed precisely, AutoCAD is able to reapply the geometric constraints correctly; however, we will need to create the two radial dimensional constraints manually.
4. Select the **Radius** dimensional constraint by clicking on the icon as shown in the figure.

5. Select the **circle on the right** and place the dimension to the right as shown.

6. Hit the **Enter** key once to accept the default dimension.

7. On your own, move the objects forming the outside loop to layer **Object** and switch **ON** the **Display Lineweight** option. Your drawing should appear as shown in the figure below.
Creating and Constraining Additional Circles

1. Click on and switch to the **Home** tab in the **Ribbon** tabs and panels area.

2. Select the **Circle – Center, Radius** command icon in the **Draw** toolbar. In the command prompt area, the message “_circle Specify center point for the circle or [3P/2P/Ttr (tan tan radius)]:” is displayed.

3. On your own, create four circles of arbitrary sizes below the current drawing as shown.

4. Select the **Diameter** dimensional constraint by clicking on the icon as shown in the figure.

5. On your own, apply four **Diameter dimensional constraints** as shown in the figure below.
6. Select the **Concentric** constraint by clicking on the icon as shown in the figure.

7. Select the **two circles on the left side** to align the center points as shown.

8. On your own, repeat the **Concentric** constraint command and align the three circles as shown in the figure below.

9. Click on the **triangle symbol** below the icon of the **Linear** dimensional constraint to show more options.

10. Select the **Horizontal** dimensional constraint by clicking on the icon as shown in the figure.
11. Select the center point of the arc on the left and the center point of the diameter 10 circle.

12. On your own, set the horizontal distance to 48 as shown.

13. On your own, use the Vertical dimensional constraint command to align the center of the small circle and the center point of the arc on the left as shown.
Control the Display of Constraints

1. Click on the **Hide All Dynamic Constraints** icon with the left-mouse-button.

![Image of Hide All Dynamic Constraints icon]

- Note all dimensional constraints are removed from the screen.

2. Click on the **Hide All Constraints** icon with the left-mouse-button.

![Image of Hide All Constraints icon]

3. On your own, use the **Show** and **Show All** commands to display the applied geometric constraints.

![Image of Show and Show All commands]

- Note that the procedures involved in using *parametric drawing tools* are very different than traditional drafting techniques. Although both approaches can be used to achieve the same results; the *parametric drawing* approach provides the more powerful functionalities for design revisions and modifications.
The Implicit Geometric Constraint Approach

- In AutoCAD 2015, a parametric option is also available in the Status toolbar area, the Infer Constraints option. The new option allows us to use Implicit Geometric Constraint approach and apply constraints at the same time the geometric entities are constructed.

1. Activate the Infer Constraints, Ortho and Object Snap options by clicking with the left-mouse-button on the first icon in the Status toolbar area as shown.

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

3. To the right side of the completed drawing, start from the top and create the five line segments that are perpendicular/parallel to each other. Notice the associated coincident constraints and perpendicular constraints are automatically applied.

4. Click on the 3-Point arc icon to activate the command.
5. Create a 3-point arc with the starting and ending points connected to the two horizontal lines as shown.

6. In the Ribbon tabs area, left-mouse-click once on the Parametric Tools tab as shown.

7. Select the Tangent constraint by clicking on the icon as shown in the figure.

8. Select the arc as the first object.

9. Select the horizontal line on the left as the second object.
10. Select the **Linear** dimension constraint by clicking on the icon as shown in the figure.

11. Inside the display area, right-mouse-click once to bring up the option menu and select **Object** as shown.

12. Select the **bottom horizontal line** with a single click of the left-mouse-button.

13. Place the dimension below the line and enter **96** as the new dimension of the line.

14. On your own, repeat the above steps and create the addition dimensions as shown.

15. Select the **Fix** constraint by clicking on the icon as shown in the figure.

16. Click on the **bottom left corner** of the sketch with the left-mouse-button to apply the constraint.
17. In the Ribbon tabs area, left-mouse-click once on the **Home** tab as shown.

![Ribbon tabs](image)

18. Select the **Mirror** command icon in the **Modify** toolbar. In the command prompt area, the message “Select objects:” is displayed.

![Mirror command icon](image)

19. Select the three objects that are on top as shown.

![Three objects](image)

20. Inside the **Drawing Area**, right-mouse-click to accept the selection and continue with the **Mirror** command.

![Drawing area](image)

21. In the command prompt area, the message “Specify the first point of the mirror line:” is displayed. Pick the **left endpoint of the bottom horizontal line**.

![Command prompt area](image)

22. In the command prompt area, the message “Specify the second point:” is displayed. Pick the **right endpoint of the bottom horizontal line**.

![Command prompt area](image)

23. In the command prompt area, the message “Delete source objects? [Yes/No] <N>.” is displayed. Hit the **[Enter]** key once to retain the original objects.

![Command prompt area](image)
24. Activate the [Arc] \(\rightarrow\) [Center, Start, End] command as shown.

25. Create the two arcs as shown in the figure below.

26. On your own, move the objects forming the outline of the design to the **Object** layer.

27. Switch on the **Line Weight** option in the status bar area.
28. On your own, create the four additional circles.

29. Complete the construction of the design by adding geometry/dimensional constraints to the circles.

➢ Note that the implicit geometric constraint approach is the preferred method used in most 3D parametric modeling software, such as the Autodesk Inventor software.
Review Questions: (Time: 25 minutes)

1. The main characteristic of parametric modeling involves the use of **constraints**. What are the two types of constraints used in AutoCAD?

2. What is the main difference between the traditional geometric method versus the use of parametric drawing tools?

3. List and describe three geometric constraint commands you have used in the tutorial.

4. What do the dimensional constraint commands allow us to do?

5. Will the geometric constraint symbols be printed when we print the drawing?

6. How do we turn off the display of the applied constraints?

7. Besides using a Colinear constraint to align two lines, what other options can we use to align two lines?

8. When will AutoCAD automatically remove some of the applied constraints?

9. Besides applying geometric constraints individually, what other option is available to constrain precisely constructed geometry in AutoCAD?

10. Can the applied constraints be manually removed? How is this done?

11. When a design is created or changed, a drawing will be in one of three states. What are the three states? What are the differences between the three states?
Exercises: (Time: 150 minutes)

1. **Indexing Base** (Dimensions are in inches.)

   ![Indexing Base Diagram]

2. **Positioning Spacer** (Dimensions are in inches.)

   ![Positioning Spacer Diagram]
3. **V-Slide Plate** (Dimensions are in inches.)

4. **Adjustable Support** (Dimensions are in inches.)
5. Sensor Mount (Dimensions are in inches.)