

ES124329

Power Dimensioning in AutoCAD Mechanical Adding Power to the Mechanical Designs

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Learning Objectives

1. Learn how to create different dimension types using the Power Dimension tool
2. Learn how to insert fits and tolerances for dimensions
3. Learn how to lay out dimensions accurately
4. Understand how to use your custom dimension style in power dimensions

Description

[We've been dimensioning in AutoCAD software for what seems like a century. You will learn how the multipurpose AMPOWERDIM command automatically recognizes objects and defaults to the appropriate dimension type. Discover how to create multiple dimensions with minimal input, resulting in instant groups of appropriately spaced ordinate, parallel, or symmetrical dimensions. You will also learn how to automatically force overlapping dimensions to automatically space themselves appropriately. You can use the same command to place linear, angular, radial, diameter, baseline, and chain dimensions. Learn how to integrate tolerance and fit list information into the drawing. In a layout or scale area, power dimensions automatically adjust their linear scale based on the points or geometry selected.]

Speaker

Sridhar Subramani has more than 17 years of CAD experience, working as Principal QA Analyst, Product Owner and scrum master at Autodesk, Inc. A seasoned professional in software testing, he has also been actively involved in news groups of AutoCAD Mechanical software and AutoCAD Architecture software and resolved over 1350 issues reported by customers. Sridhar conducts in-house training in AutoCAD software and AutoCAD Architecture software. He has written several technical solutions that are published on the Autodesk support website.

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1. Create different dimension types

Power dimension creates multiple types of dimensions within a single command session.

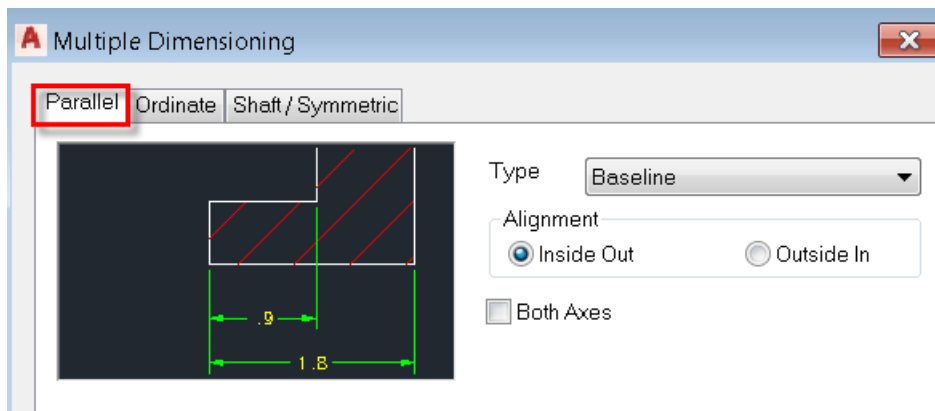
1.1 Recognize object type

1. Launch AutoCAD Mechanical
2. Open sample drawing Anno-1.dwg
3. Switch to Annotate Tab>Dimension Panel>Select Power Dimension button>Press enter key
 - a. Note: You may also enter command **AMPOWERDIM** at the command line
4. Select any object type like an inclined line where power dimension will automatically create a aligned dimension
5. Select a circle and power dimension will create a Radius/Diameter dimension type.
6. The different types of dimensions that can be created using Power Dimension are; *vertical, horizontal, aligned, and rotated linear dimensions, to angular dimensions, to radius, diameter, jogged radius, and arc length dimensions, to baseline and chain dimensions.* Help [Link](#)

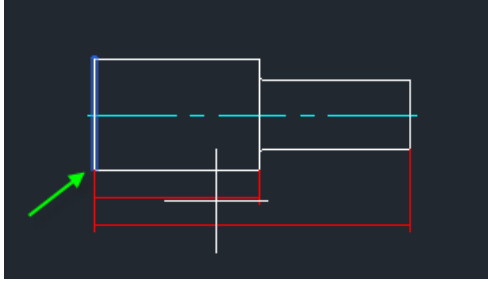
1.2 Create Multiple dimensions

1.2.1 Parallel Dimension

1. Open Shaft.dwg
2. Switch to Annotate Tab>Dimension Panel>Select Multiple Dimension button
3. Select Parallel Tab



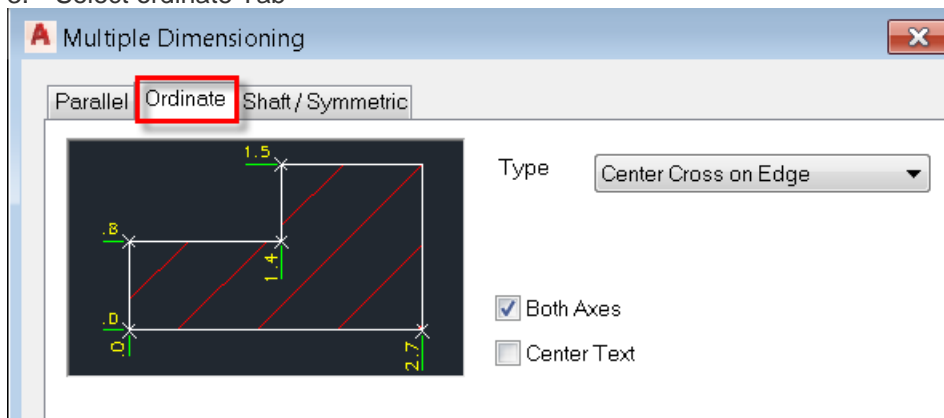
4. Select Type as Baseline or Chain
5. For this example, select Baseline
6. Select all objects in the drawing
7. Pick bottom left corner as shown in the below image and press enter key
8. Place the dimensions horizontally.



Note: You can enable “Both Axes” option
This will automatically create both Vertical and Horizontal dimensions.

1.2.2 Ordinate Dimension

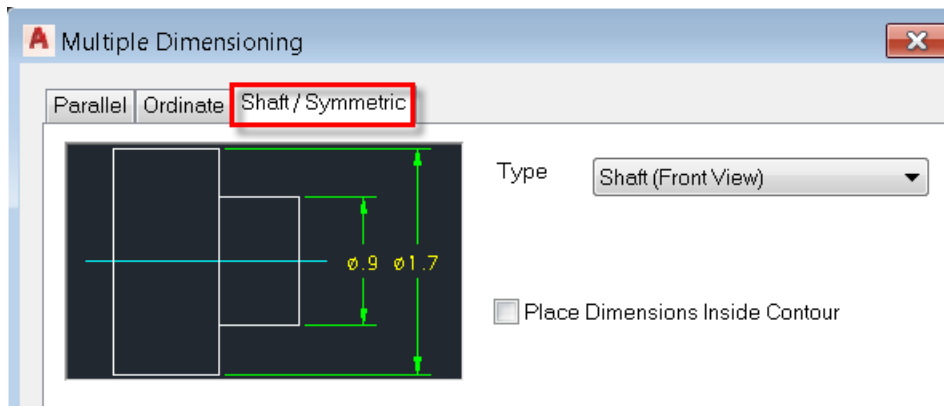
1. Open Shaft.dwg
2. Switch to Annotate Tab>Dimension Panel>Select Multiple Dimension button
3. Select ordinate Tab



4. Select Type as Center Cross on Edge
5. Enable “Both Axes” option
6. Select all objects in the drawing
7. Pick bottom left hand corner as shown in the below image and press enter key
8. Select a dimension type; e.g. Horizontal
9. Pick a point in the screen to place horizontal coordinates
10. Pick Another point to place Vertical coordinates.

1.2.3 Symmetrical Dimension

1. Open Shaft.dwg
2. Switch to Annotate Tab>Dimension Panel>Select Multiple Dimension button
3. Select Shaft/Symmetric Tab

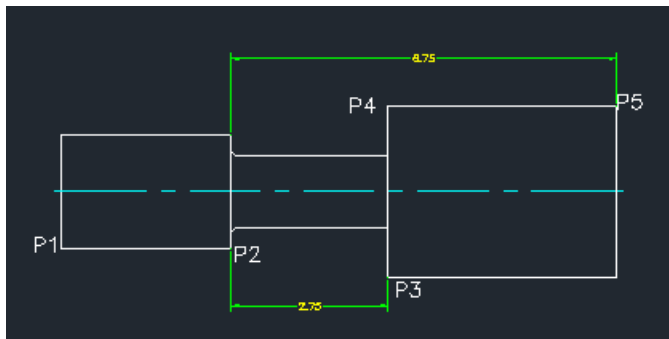


4. Select Type as Shaft (Front View)
5. Select all objects in the drawing
6. Pick a point on the center line and place the dimensions vertically.

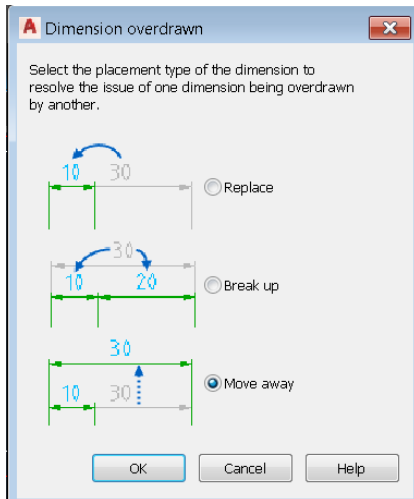


1.3 Force overlapping dimensions

1. Open Shaft-2.dwg
2. Switch to Annotate Tab>Dimension Panel>Select Power Dimension button



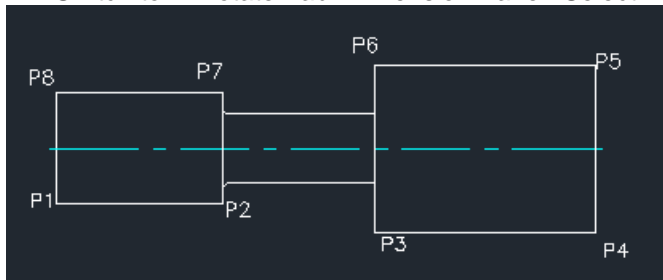
3. Pick Point P1 and P3
4. Move the mouse over the dimension and click
5. Note, you will see a dialogbox "Dimension overdrawn"
6. Select "**Replace**" Option



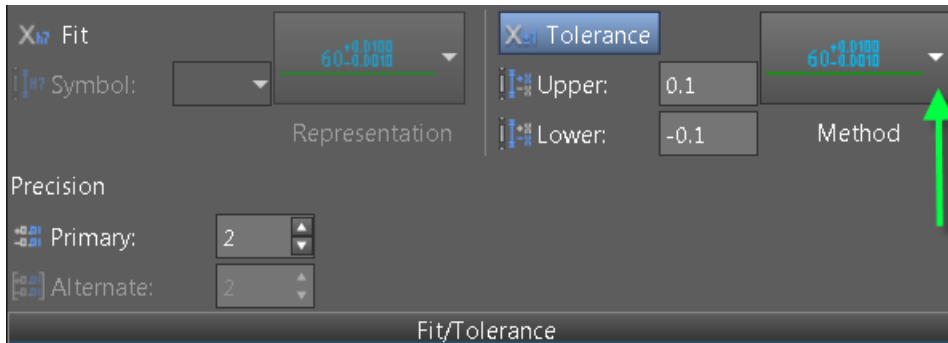
7. You will notice the existing dimension is replaced with a new dimension. (This will increase the productivity when there are many dimensions and you would like to replace one dimension with another one in the same location.)
8. Press enter key to create another dimension
9. In this step, pick P1 and P2 points>move the mouse over the dimension and click
10. Note, you will see a dialog box "Dimension overdrawn"
11. Select **"Move away"** Option
12. You will notice the existing dimension moves down, so that the new dimension is placed as a baseline dimension.
13. Press enter key to create another dimension
14. In this step, pick P4 and P5 points>move the mouse over the dimension and click
15. Note, you will see a dialog box "Dimension overdrawn"
16. Select **"Break up"** Option
17. You will see the existing dimension is divided in to 2 dimensions. (This will help user to quickly divide an existing dimension and create chain dimensions.)

2. Insert Tolerance

1. Open Tolerance_fits.dwg file
2. Switch to Annotate Tab>Dimension Panel>Select Power Dimension button



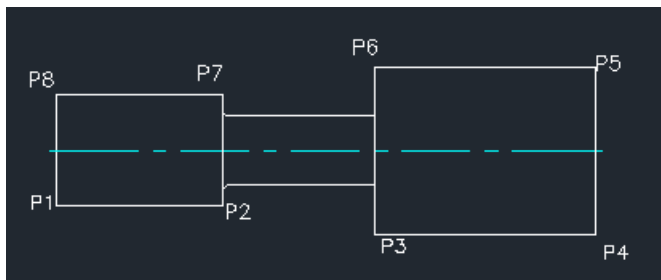
3. Pick Point P1 and P2 and Pick any point on the screen to place the dimension
4. In the Power Dimension contextual tab>Fit/Tolerance Panel>Enable 'Tolerance'
5. Enter the upper and lower deviation as 0.1 and -0.1 (Note for lower deviation, user need to manually enter the negative values)
6. Click Method dropdown



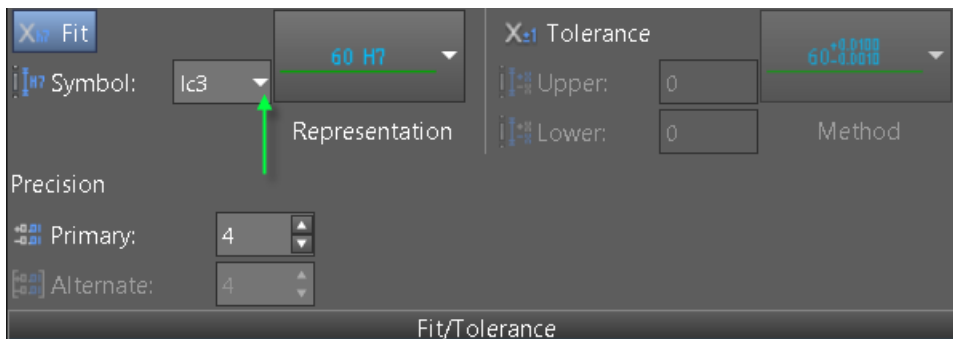
7. Select a method type as “AutoCAD Symmetrical/Deviation”
8. Click a point on screen to place the dimension
9. Let’s create another dimension with a different tolerance method
10. Pick Point P3 and P4
11. Click Method drop down on the Fits/Tolerance Panel
12. Select Tolerance: Symmetrical type
13. Place the dimension.
14. Press escape key to cancel the command.

3. Insert Fits

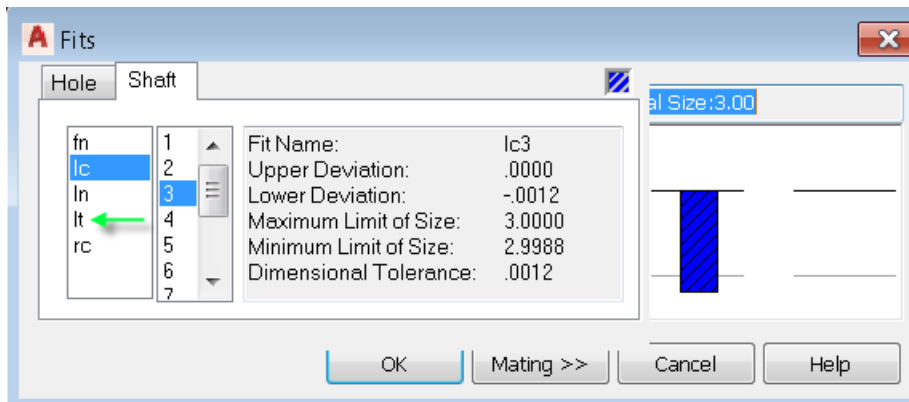
1. Open Tolerance_fits.dwg file
2. Switch to Annotate Tab>Dimension Panel>Select Power Dimension button



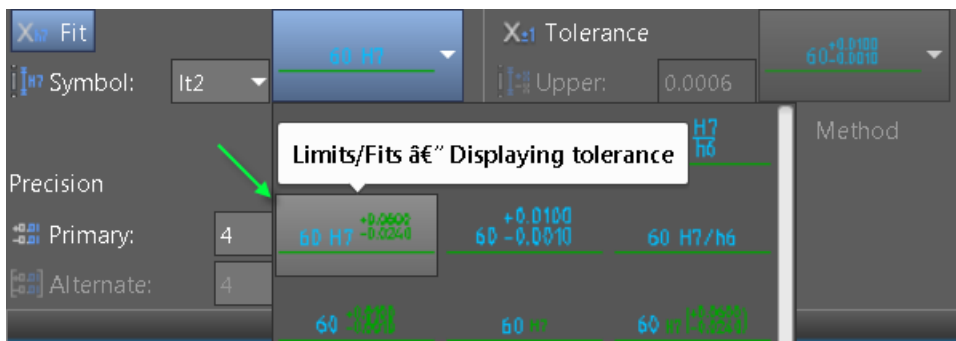
3. Pick Point P4 and P5 and Pick any point on the screen to place the dimension
In the Power Dimension contextual tab>Fit/Tolerance Panel>Enable ‘Fits’



4. Click the Symbol dropdown>Select Fit Dialog box



5. In the Fits dialog box>Select lt and 3 and click OK button
6. Pick a point to place the dimension.
7. Let's Insert another dimension with Fits
8. Pick Point P1 and P8 and Pick any point on the screen to place the dimension
In the Power Dimension contextual tab>Fit/Tolerance Panel>Enable 'Fits'
9. Click the Symbol dropdown>Select Fit Dialog box
10. In the Fits dialog box>Select lt and 2 and click OK button
11. Click Representation dropdown



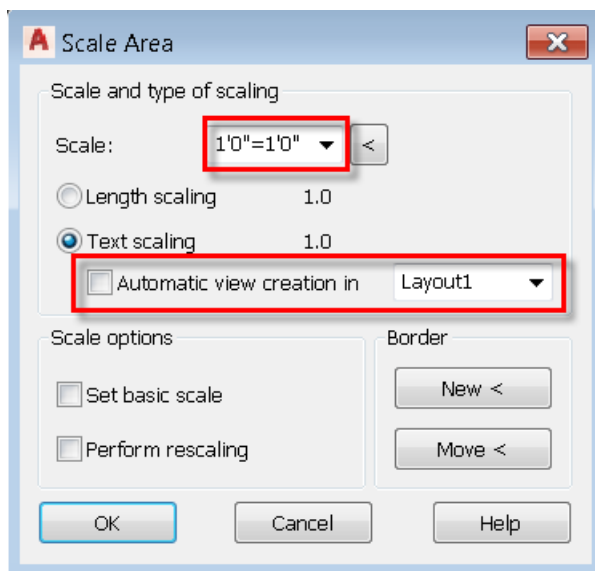
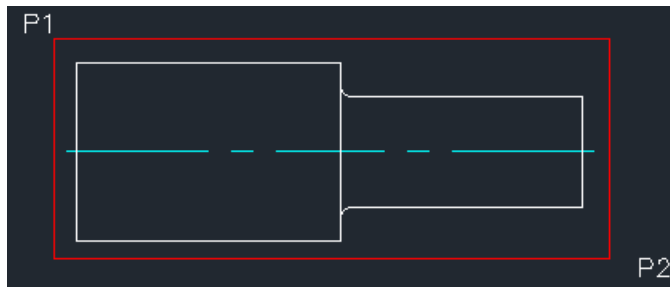
12. Select Limits/Fits Displaying tolerance button
13. Pick a point to place the dimension.
14. Save the drawing

3.1 Insert Fits List Table

1. Open the drawing Tolerance_fits.dwg file saved in the above step
2. Command: AMFITSLIST
3. Select New option
4. Place the Fits list anywhere in the drawing
5. Let's modify a dimension with Fits, so that the Fits table gets updated.
6. Double click the dimension on the right side of shaft
7. In the Power Dimension contextual tab>Fit/Tolerance Panel>Click Symbol
8. Select Fits Dialog box
9. Change the value to Lc and 3
10. Click OK
11. Pick any point
12. Note: You will be promoted to update the Fits table automatically.

4. Lay out dimensions accurately

1. Open Shaft-3.dwg file
2. Command: AMSCAREA
3. Pick near point P1 and P2 in the drawing area to select the objects for dimensioning

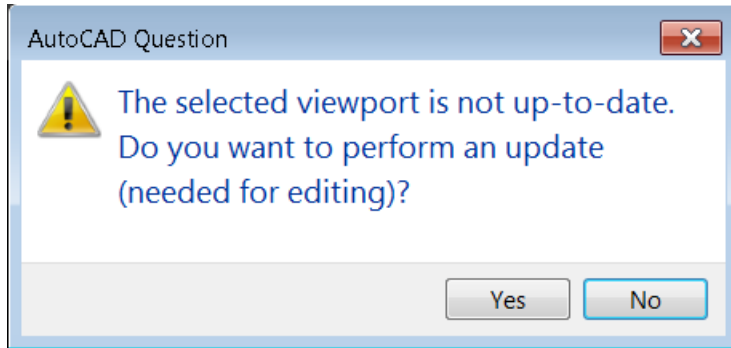


4. In the Scale Area dialog box>Select Scale dropdown>>select 1:2 Scale
5. Enable "Automatic view creation in" option to create a view in layout 1
6. Click OK in the dialog box
7. Place the view anywhere in the layout
8. Select Annotate Tab>Dimension Panel>Power Dimension
9. Create a horizontal dimension for total length of the shaft. Note down the value of dimension which is 5.75 inches
10. Switch to Model space
11. Measure the length of shaft which will match the value of dimension (5.75)
12. Repeat the above steps to create another view using Point P3 and P4
13. This time select the view scale as 1:4
14. Dimension the object in the layout and measure the distance.
15. Save the drawing.

Note: In the next session, let's update a view scale and verify the dimension values.

4.1 Updating View scale

1. Open the drawing saved in the above step
2. Switch to model Space
3. Double click on the red border for the first view
4. Change the view scale from 1:2 to 2:1
5. When prompted for rescaling>Select the objects in the red border>Click OK
6. Switch to Layout
7. Double click the viewport

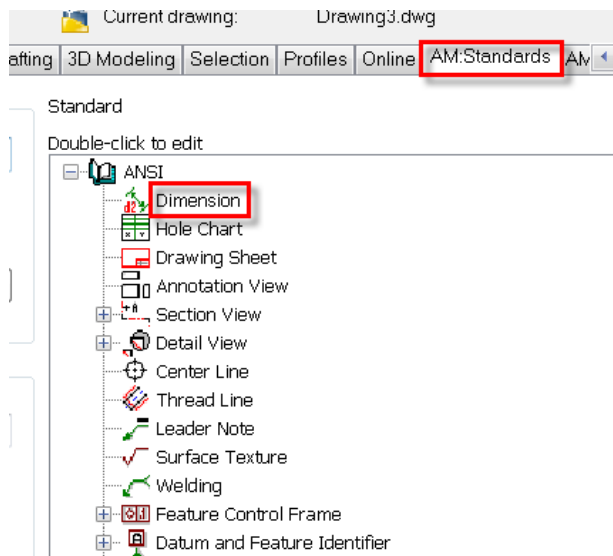


8. You see a task dialog to update the viewport
9. Select Yes
10. Notice the view is updated and the dimensions are automatically updated.

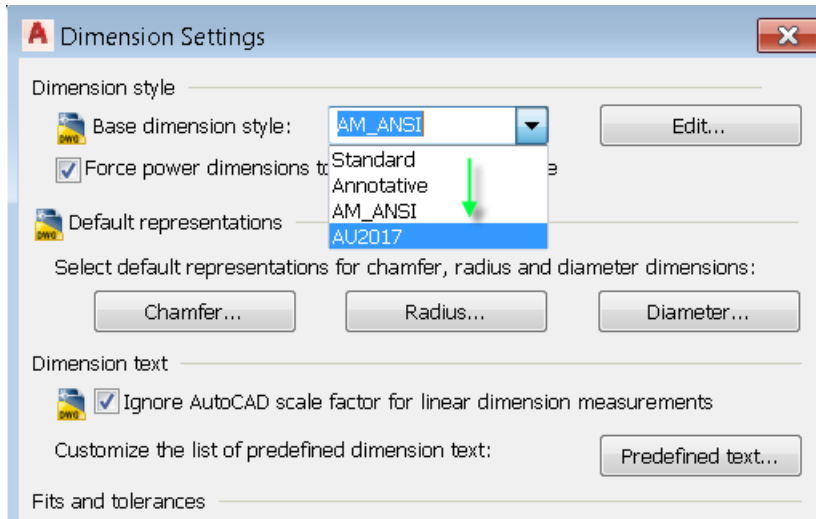
5. Custom dimension style

In this session, let's see how to enable company specific dimension styles.

1. Open a new drawing based on ANSI template
2. Create a new dimension style "AU2017" using DDIM command.
3. Command: Options>Switch to AM: Standards tab
4. Double click Dimension node



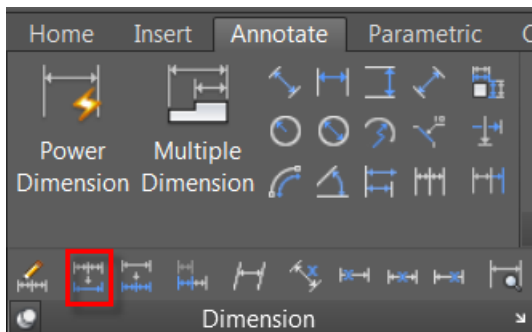
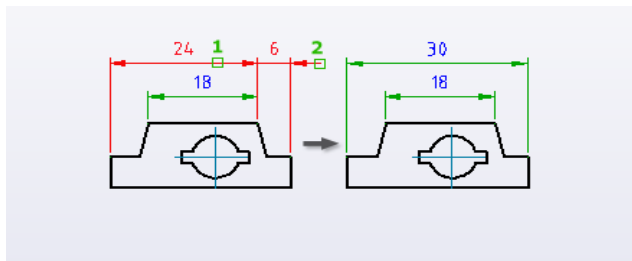
5. Set a base dimension style



6. Select AU2017. (Note: This dimension style is created before setting up)
7. Click OK.
8. Power Dimension will use AU2017 dimension style for all new dimensions.

6. Modifying Dimensions

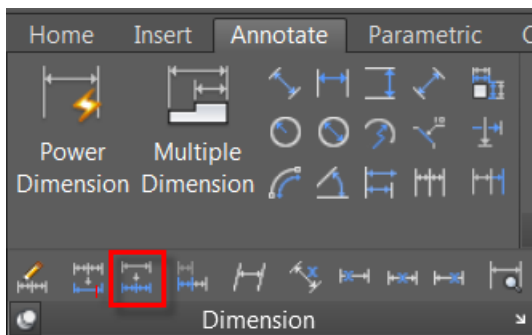
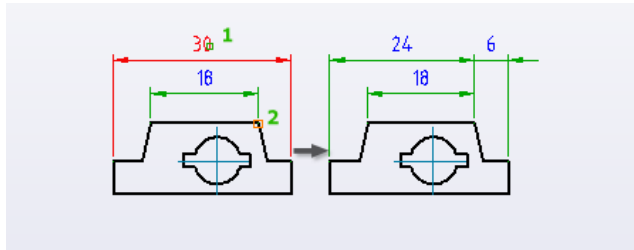
6.1 Dimension Join



1. Open Modify_dimension.dwg file
2. From Annotate Tab>Power dimension Panel>Slide out>Select Join Dimension button or enter command AMDIMJOIN
3. Select the 2 dimensions in the drawing under Join Dimension geometry

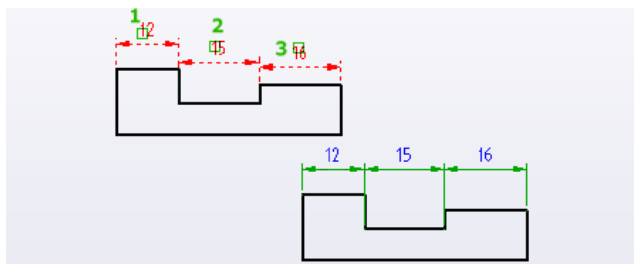
- This will help you convert 2 or more dimensions into a single dimension and place the dimension in the same location where the 2 dimensions were.

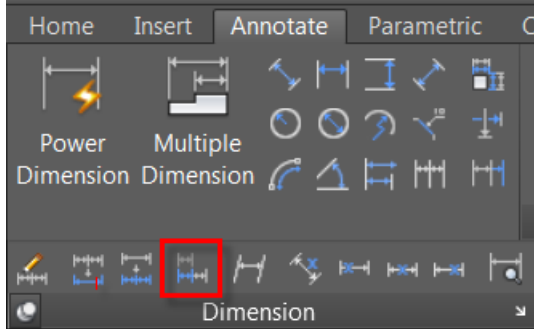
6.2 Dimension Break



- Open Modify_dimension.dwg file
- From Annotate Tab>Power dimension Panel>Slide out>Select insert Dimension button or enter command AMDIMINSERT
- Select the dimension in the drawing under Break Dimension geometry
- Pick the end point to first shaft
- This will help you insert another dimension into existing dimension or break a single dimension into 2.

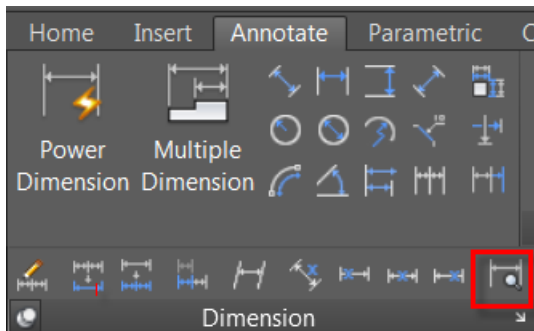
6.3 Dimension Align





1. Open Modify_dimension.dwg file
2. From Annotate Tab>Power dimension Panel>Slide out>Select Align Dimension button or enter command AMDIMALIGN
3. Select the first dimensions in the drawing under Align Dimension geometry as base dimension
4. Select the remaining 2 dimension and press enter key
5. This will help you align multiples dimensions in a single line like chain dimension.

6.4 Check Dimension



1. Open Modify_dimension.dwg file
2. From Annotate Tab>Power dimension Panel>Slide out>Select Check Dimension button or enter command AMCHECKDIM
3. The list of dimensions that are overridden are highlighted
4. Double click the dimension under Update dimension text
5. Hit back space to remove the override
6. Select "Primary dimension" button on the Ribbon as shown below.

