

IM322454

Transform AutoCAD Drawing to Intelligent Design Objects Using Mechanical Toolsets

Vinod Kumar Balasubramanian
Autodesk Inc

Sridhar Subramani
Autodesk Inc

Learning Objectives

- Learn how to simplify AutoCAD objects to adhere to drawing design standards
- Learn how to convert an AutoCAD table to an intelligent BOM
- Learn how to utilize Auto Dimension in Power tools to annotate drawing views
- Learn how to use the ballooning feature to create smart annotations

Class Summary

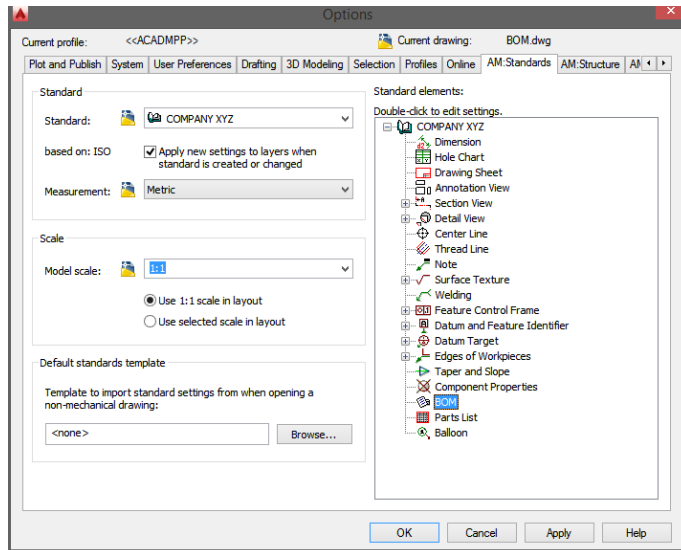
In this class, we'll introduce you to simple and powerful features in mechanical toolsets that help you transform your AutoCAD drawing into smart-drawing entities that can help increase productivity and efficiency. We'll do a simple walk-through of converting your AutoCAD tables to a smart BOM database. You'll learn an automatic dimensioning feature under power tools to make your drawing annotation simpler and faster. Later, we'll add intelligent balloons to the drawing views that you've created in AutoCAD using model documentation. Finally, we'll utilize the drawing design standards in AutoCAD Mechanical software to make your drawing views and annotations adhere to your company standards and create effectiveness to save costs and time with your drawing maintenance.

Description

In this exercise, you will create template drawings and new drawings using one of the template files. You will also set a new folder location for saving and accessing template files.

Part 1 – Create & Apply a new drawing standard in drawing

1. Open **Adsk_Company_standards.dwt**.
2. To change the active mechanical standard, do the following:
On the Application Menu, click Options.
In the Options dialog box, AM:Standards tab, on the Standard list, click **Autodesk**.
Click OK.

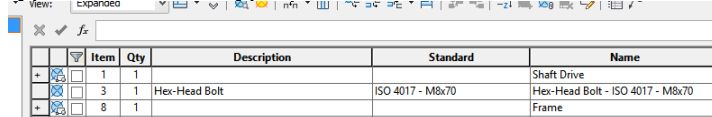


3. Under the measurement – Change the setting to Metric.
4. In Default Standards template – load the **Adsk_Company_standards.dwt**
5. Under Standard elements – Double click on Dimension style – Choose AU style from drop down
6. Select *“Force Power Dimensions to use this dimension style”*
7. Click OK.
8. Under Standard elements – Double click on Balloon.
9. It opens BOM Settings dialog. In BOM Settings choose Custom under default style and select a style.
10. Click OK.
11. Apply changes and Close the options dialog.

Part 2 - Export an AutoCAD table and import it as an intelligent BOM object

In this exercise, you will create import a simple AutoCAD table into Mechanical drawing and convert it as a smart and intelligent BOM.

1. Open **BOM.dwg**.
2. Type **AMSETUPDWG** and select the **Adsk_Company_standards.dwt**.
3. Continue to finish the setup.
4. This will import the standards set in Part 1 to BOM.dwg.
5. Select the table in the drawing and right click the table and export as table.csv.
6. Close drawing.
7. Open BOM.dwg.
8. Type **AMBOM > Type ADSKBOM** to invoke Bill of Materials in the drawing.



Item	Qty	Description	Standard	Name
1	1			Shaft Drive
3	1	Hex-Head Bolt	ISO 4017 - M8x70	Hex-Head Bolt - ISO 4017 - M8x70
8	1			Frame

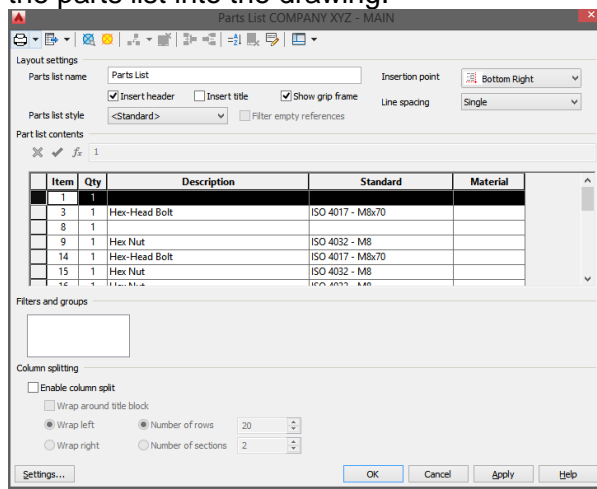
9. In the BOM panel – under BOM list – Select import and change file type to CSV.
10. Press Open. **It will automatically import all the table data in the BOM**

11. Save drawing.

Part 3 - Create Parts list for the drawing

In this exercise, we will associate part reference in the drawing with the BOM data and add a parts list table in the drawing.

1. Open **BOM.dwg**.
2. At the Command prompt, enter BOM name as ADSKBOM.
3. Press ENTER.
4. In the BOM, select Part reference (*blue circle object*) next to each object and associate them in the drawing.
5. At the Command prompt, enter **AMPARTSLIST**.
6. Click OK. The dialog box closes, and a preview of the parts list appears in the drawing area.
7. In the drawing area, click to indicate where to place the parts list. The program inserts the parts list into the drawing.



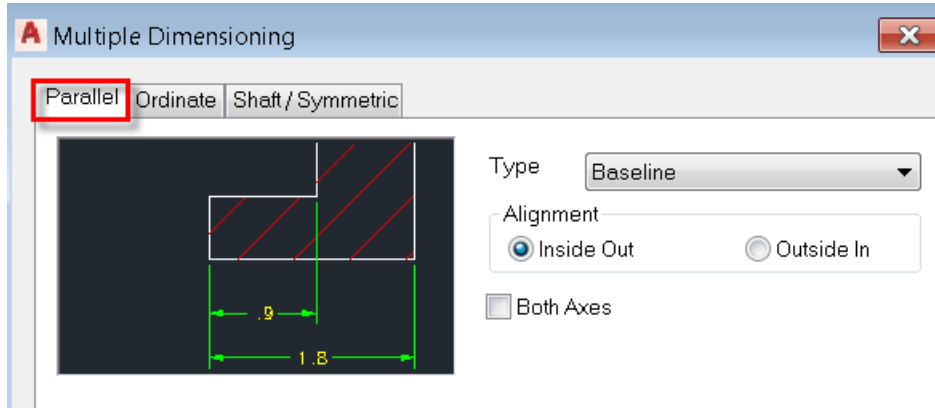
8. Select Insert Title to add name of the parts list.
9. Select Insertion Point to Bottom left.
10. Select line spacing to Double.
11. Enable column split – **use when parts list is lengthy**
12. Click Setting to Modify Parts List Color & Style.
13. Click Ok to save Changes.

Part 4 - Create quick power annotations in the Mechanical drawing.

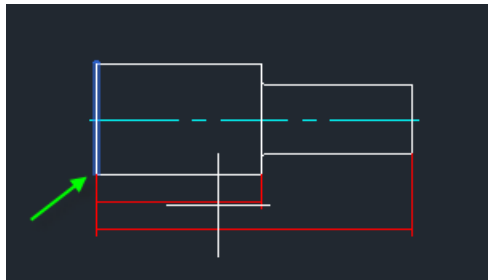
In this exercise, we will create a various type of power dimensions to the drawing that has standards and BOM updated in the previous segments.

1. Open **BOM.dwg**.
2. Switch to Annotate Tab>Dimension Panel>Select Multiple Dimension button.

3. Select Parallel Tab.

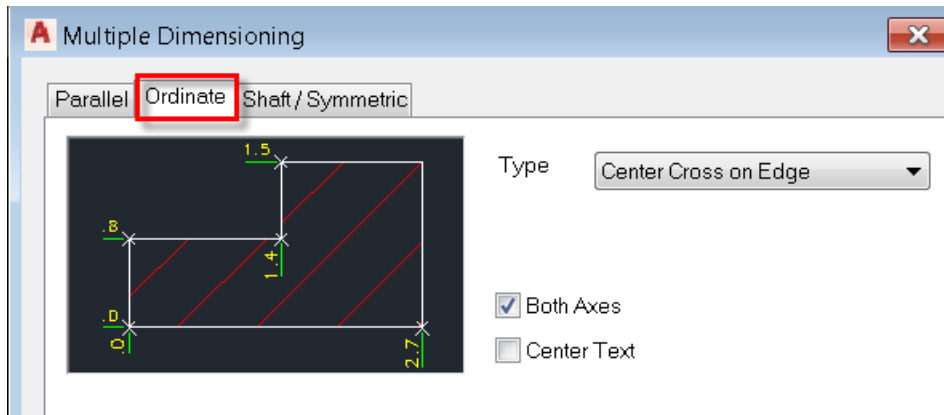


4. Select Type as Baseline or Chain.
5. For this example, select Baseline.
6. Select all objects in the drawing.
7. Pick bottom left hand corner as shown in the below image and press enter key.
8. Place the dimensions horizontally.



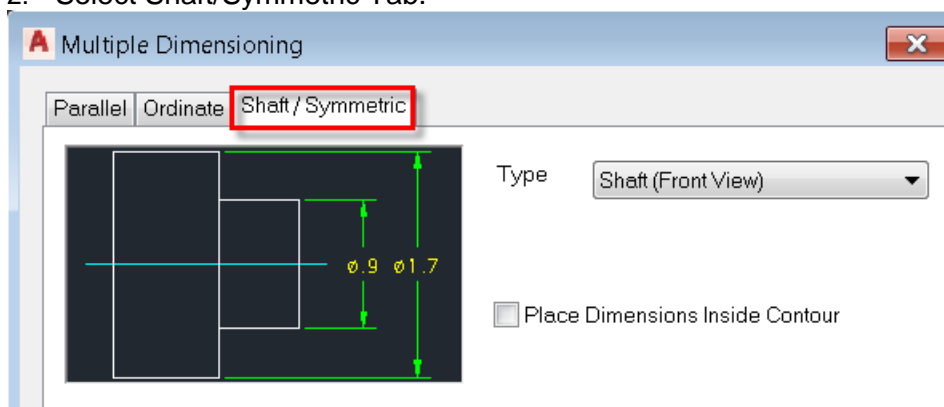
Note: You can enable “Both Axes” option.
This will automatically create both Vertical and Horizontal dimensions.

1. Switch to Annotate Tab>Dimension Panel>Select Multiple Dimension button.
2. Select ordinate Tab.



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

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




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



Part 5 – Add Balloon to the objects in the Mechanical drawing.

In this exercise, we will create simple balloons to the parts and connect them with the intelligent BOM created in earlier segments of the drawing.

To Create Balloon

1. Open **BOM.dwg**.
2. Click Annotate tab ► Balloon panel ► Balloons. 
3. In the drawing area, select a part reference or component (part or assembly).
4. In the drawing area, click to indicate the second point for the balloon leader, and continue until the balloon is in the location you want.
5. Press ENTER to place the balloon.
6. Press ENTER to end.

Auto Balloon:

1. In the Command line type: **AMBALLOON**.
2. Select Auto.
3. Select the view or list of objects in the assembly.
4. Press Enter.

To renumber balloon:

1. In the Command line type: **AMBALLOON_RENUM**.
2. Select Starting item number – Set to 1.
3. Enter Increment – Set to 1.
4. Select all the balloons in the drawing view.
5. Press Enter.

To arrange balloon in one leader:

1. In the Command line type: **AMBALLOON_COLLECT**.
2. Select a Part reference or all the balloons in the drawing view.
3. Press Enter.
4. Save BOM.dwg.

Thank you.

Speaker(s)

Vinod Kumar Balasubramanian is currently a senior technical lead for AutoCAD toolsets software. He is responsible for handling business escalations on support issues, preparing partners for new releases, creating high-quality knowledge content, and helping to improve total experience of product for customers.

Sridhar Subramani has more than 17 years of CAD experience, working as Product Owner and Principal QA Analyst at Autodesk, Inc. 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.

