

# Autodesk Inventor Workshop Tutorial

*Hosted By: Team 1403 Cougar Robotics*

**Advanced Packet**



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## TROUBLESHOOTING & HELPFUL HINTS

Some common problems you may run into while using this program:

- Drawing Lines that are too long, or wanting to trim lines that are too long  
Option 1: Click on the segment of the line, or the line you want to delete, then press the **Delete** key  
Option 2: You can use the **Trim** tool.
- You want to **Undo** something you just did:  
Control-Z or undo button
- While creating a sketch: If you want to go back and edit a sketch, accidentally have exited the sketch you were working on, you can right-click on the sketch's name in the **Model** toolbar on the left side of your screen and select **Edit Sketch**. If the sketch hasn't been "used" for something like an extrusion or revolution, you can click on any part of the sketch and select the **Edit Sketch** option from the Mini-Toolbar.

Shortcut keys. Throughout the course of this tutorial, I have included only a few of the many useful keystrokes that can increase the speed at which you use Inventor. Some others you may find helpful are listed here for your reference.

### Sketching:

D Dimension  
X Trim  
L Line

### Modeling:

E Extrude  
C Constrain  
H Hole  
F Fillet

### Assembling:

P Place component  
R Revolve

### Where am I looking?

When you are building a part, you will need to look at faces and edges that won't be visible in your current view. Using the **ViewCube** seen below, you can select any one of 26 different views by clicking on the cube's edges, corners, and faces. The house icon will return you to your home or "isometric" view.



The X-Y-Z  
coordinate  
axis  
indicator

**VIEWCUBE:** used to change the view of the part you are working on.

### SHORTCUT KEYS:

F6: Will return you to the ISOMETRIC view.

F4: You can freely rotate your model by pressing and holding F4, and then clicking and dragging in your work area.

## SECTION 6: LOFTING

THE FOLLOWING FIGURES WILL BE REFERENCED IN THIS PORTION OF THE EXERCISE

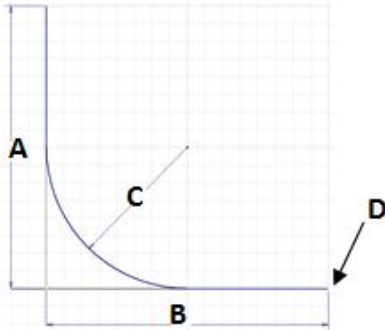


FIGURE 17

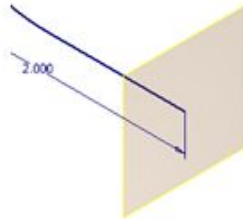


FIGURE 18

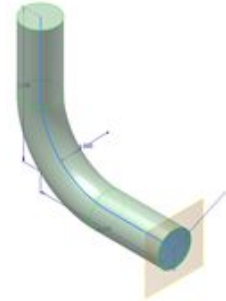
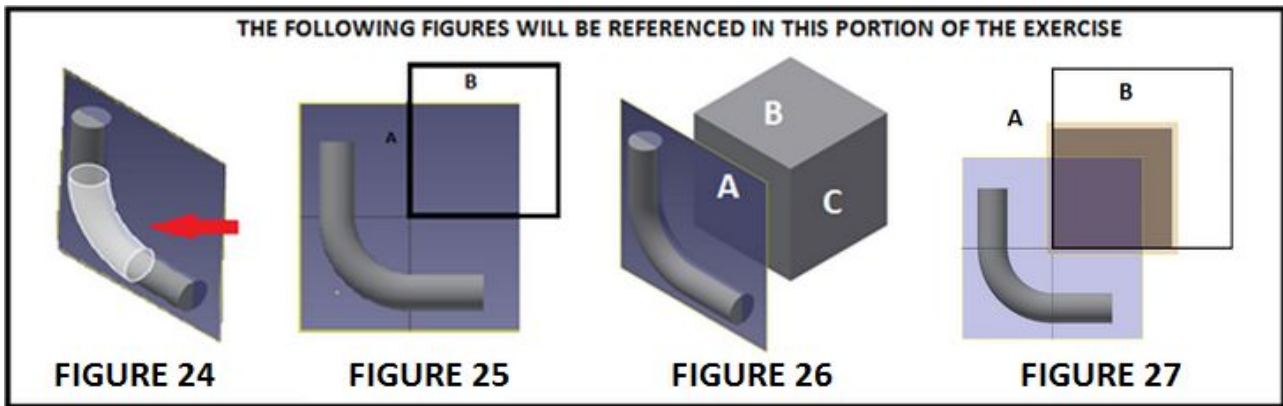


FIGURE 19

- Open a new part file. Start a **New Sketch** in the **XY** plane.
  - Draw two lines from the origin, with a fillet between them as shown in **FIGURE 17**, with dimensions **A=2"**, **B=2"**, and **C=1"**
  - Exit the sketch. Inventor brings you to the isometric view.
- On the **3D Model** toolbar, click on the drop down menu under the **Plane** tool
  - Select **Normal to Axis through Point**
  - Move your cursor over point **D** in **FIGURE 17**. The point should have an X marker. Click once on the point, and you will notice the outline of a new plane appear. Click again to create the new work plane perpendicular to and centered on line **B**. The plane should appear as in **Figure 18** above
  - Orient your view so you are looking **DIRECTLY** at the new plane. This can be done by right-clicking the edge of the plane and selecting **Look At**, or by using the ViewCube and clicking on the face labeled **Right**.
  - Create a new sketch on this plane. In the **3D Model** menu on the left side of the screen, you can right-click on the new plane (here it will be named **Work Plane 1**), and select **New Sketch**. Likewise, you can click on the edge of the plane and select **Create Sketch** from the Mini-Toolbar.
- Draw a circle with its center positioned **DIRECTLY** at the origin, with a diameter of 0.5"
- Exit the sketch and return to the isometric view by either pressing **F6** or using the ViewCube.
- Now click on the **Sweep** tool
  - If the circle you just drew is not already shaded blue, select it as the **Profile**.
  - Click on the first sketch you created. A faded shape should appear as in **FIGURE 19**
  - Click **OK**, and the sweep is complete.
  - If you wish to remove the yellow box surrounding the work plane you created, right click on the edge, and select **Visibility**
- Save this as **Part 3**

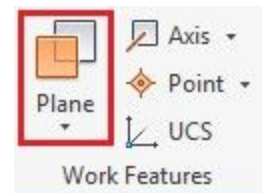


## SECTION 7: PLANES




### 7.1 WORK PLANES:

1. Open *Tutorial Part 3*.
2. Click on the **Plane** tool (Under *3D Model* and *Work Features*).
  - This will change the cursor to display a rectangle underneath the normal cursor
3. Move the cursor over the highlighted section as seen in **FIGURE 24** and click.
  - This will create a flat plane splitting through the curved section, which can be used to create a sketch when no face is present/usable.
4. Create a sketch on the newly created work plane by clicking **Start 2D Sketch** and selecting the yellow edge of the work plane.
5. Create a square as seen in **FIGURE 25**, with the initial point on the origin, and with dimensions **A=2"** and **B=2"**.
6. Finish the sketch and **Extrude** the square by 2".
7. Save this as **Part 5**. Your part should look like **FIGURE 26**.



### 7.2 OFFSET PLANES:

*Use Part 5 for this section.*

1. Click on the drop-down menu under the **Plane** tool, and select **Offset from Plane**.
 
2. Move the cursor over and click on face **A** in **FIGURE 26**.
  - A small menu will open, with a section allowing you to set the distance the plane is offset from the face
3. Offset the plane by 5".
  - This will create a plane 5" away from the surface of the cube, on which you can create a sketch.
4. Create a sketch on the newly created work plane.
5. Create a square with dimensions **A=3"** and **B=3"** with the initial point on the origin, as seen above in **FIGURE 27**.
6. Finish the sketch and **Extrude** the square by 3".
7. Save the part. It should look like **FIGURE 28** below.

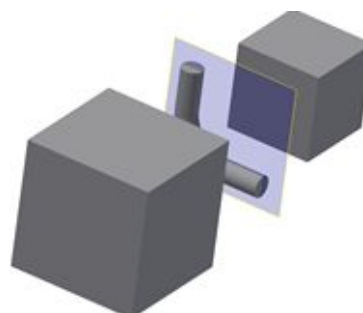
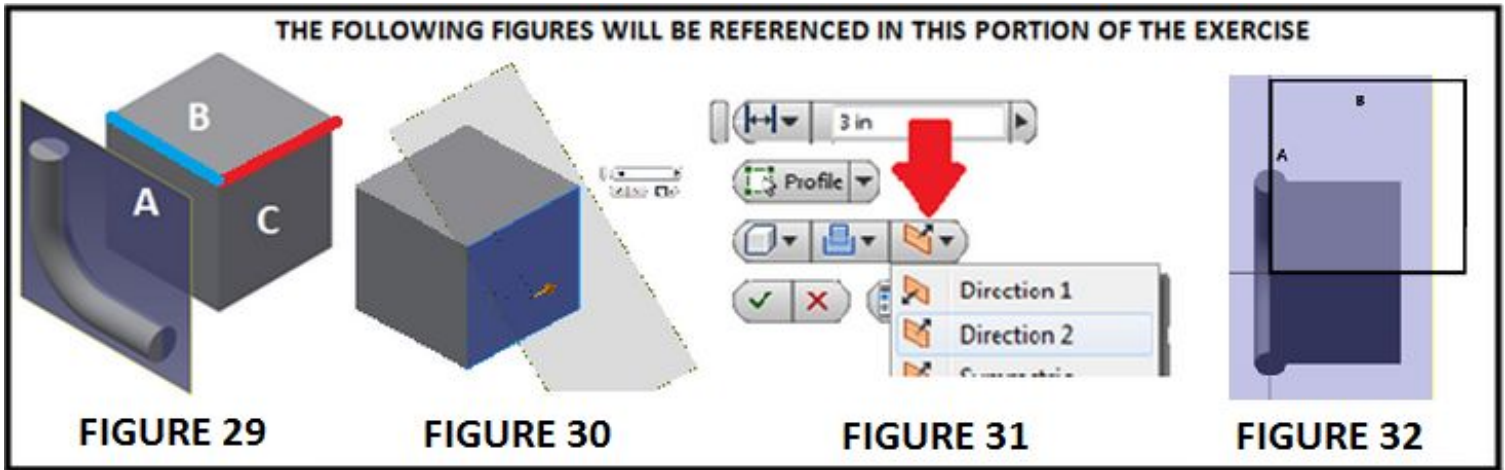


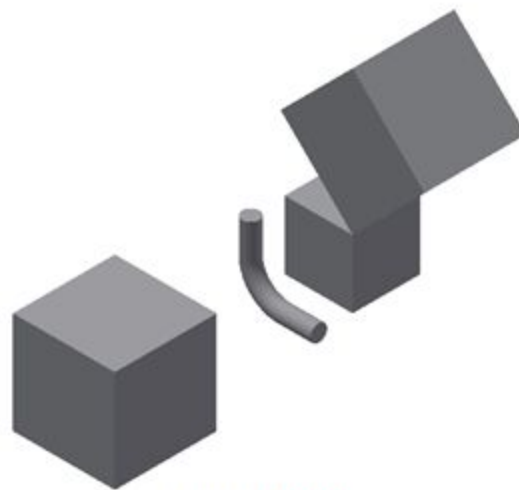
FIGURE 28



### 7.3 ANGLED PLANES

Use **Part 5** for this section.

1. Go to the **Plane** tool drop down menu and select **Angle to Plane around Edge**.
  - This will change the cursor to display a rectangle underneath the normal cursor.
2. Select the red edge in **FIGURE 29**, and then select face **C**.
  - A small menu will open up, allowing you to enter the angle at which the plane will intersect the edge.
3. Enter 135 degrees into the menu and press the enter key. A new plane should appear as seen in **FIGURE 30** above.
4. Create a square with dimensions **A=3"** and **B=3"** with the initial point on the origin, as seen in **FIGURE 32**.
5. Click on **Extrude** tool, open the extrude menu for the rectangle, and enter 3" into the dialogue box. , and using **FIGURE 31** as a reference, change the direction of the extrusion to **Direction 2**, and **Extrude** the rectangle a distance of 3".
6. Turn off the visibility of all work planes by right clicking the work plane and unchecking the visibility box.  Visibility
7. The final product of sections 7.1, 7.2, and 7.3 should look like **FIGURE 33** below.



**FIGURE 33**

## SECTION 8: CIRCULAR PATTERNING

THE FOLLOWING FIGURES WILL BE REFERENCED IN THIS PORTION OF THE EXERCISE

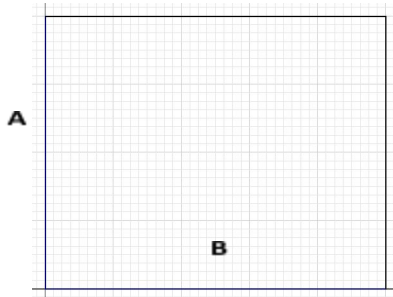


FIGURE 20



FIGURE 22

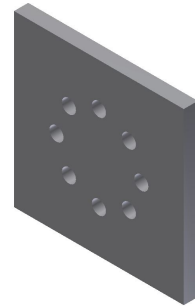



FIGURE 23

- Start a new part file, and draw a rectangle as shown in **FIGURE 20** with side lengths of **A=3"** and **B=3"**.
  - Extrude the rectangle to a thickness of 0.25"
- Right click on the top surface and select **New Sketch**.
  - Sketch the circles as shown below in **FIGURE 21**, with dimensions **A=1.5"** **B=0.75"** **C=0.25"** **D=1.0"** **E=1.5"**
  - To position the smaller circle to have its center vertically above the center of the larger circle, select **Vertical Constraint** as highlighted in **FIGURE 22** above. Then click on the centers of both circles.
  - Change the larger center circle to be a construction line. To do this, select the larger circle first, THEN click the **Construction** tool under the **Format** section located at the top right of the **Sketch** toolbar.
- Click on **Circular**  **Circular** to create a circular pattern
  - First click on the smaller circle to select it as the geometry you want to pattern. Then Click on the pointer next to **Axis** and click on the larger circle to select it as the axis to pattern the smaller circle around. Click **OK**
  - Exit the sketch

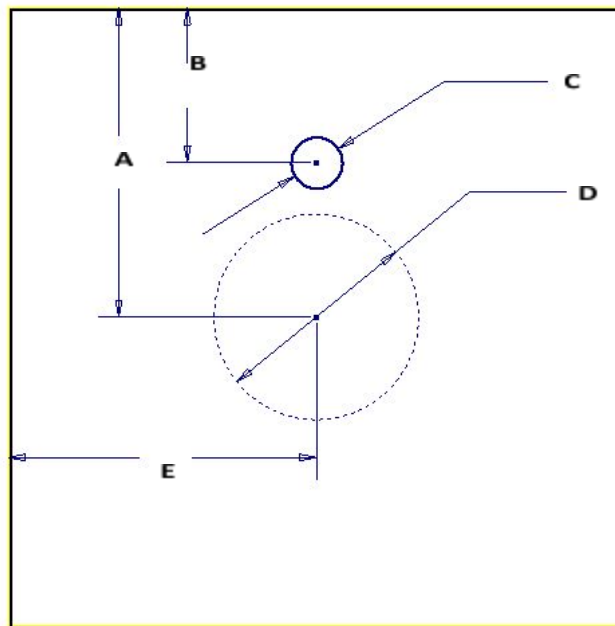
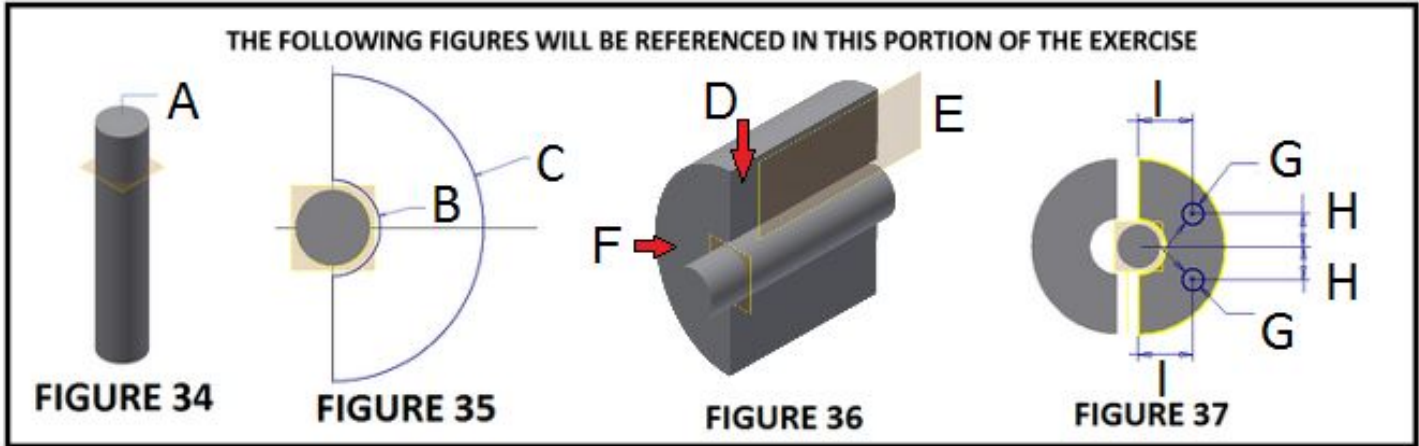


FIGURE 21

- Click on the **Extrude** tool, and then click in the center of each smaller circle. Extrude-cut these like you did for **Part 1**.
- Your part should appear as in **FIGURE 23** above.
- Save this as **PART 4**.

## SECTION 9: PROJECT GEOMETRY



The Project Geometry tool projects edges, vertices, work features, loops, and/or curves from the geometry of existing sketches or parts onto the current sketch plane. It can also be used to project a part's origin planes and axes.

1. Start a new sketch on the **XY Plane**.
2. Use the **Circle** tool and create a circle with a diameter of 1" on the origin .
3. Exit the sketch and **Extrude** the circle 5".
4. Using the **Offset from Plane** tool, create a plane inside the cylinder 1" from **Face A** as seen in **FIGURE 34**.
  - o Enter a distance of -1" in the dialogue box.
5. Create a **Sketch** on the newly created plane.
  - o To create a sketch on a plane, right click an edge of a plane and click **New Sketch**.
6. Under the **Sketch** toolbar, click on **Project Geometry**, and then click the edge of the cylinder.
  - An outline of the cylinder has been projected onto the created plane.
7. Using the **Offset**, **Line**, and **Trim** tools, replicate **FIGURE 35** as seen above, where
  - **B**=offset 0.25" from the edge of the cylinder (diameter of 1.25") and
  - **C**=offset 3" from edge of cylinder (diameter of 4")
8. Exit the sketch and **Extrude** the sketch 4" in direction 2.
9. Create a plane 0.25" away from **Face D** as seen in **FIGURE 36** (**Note that FIGURE 36 is rotated**).



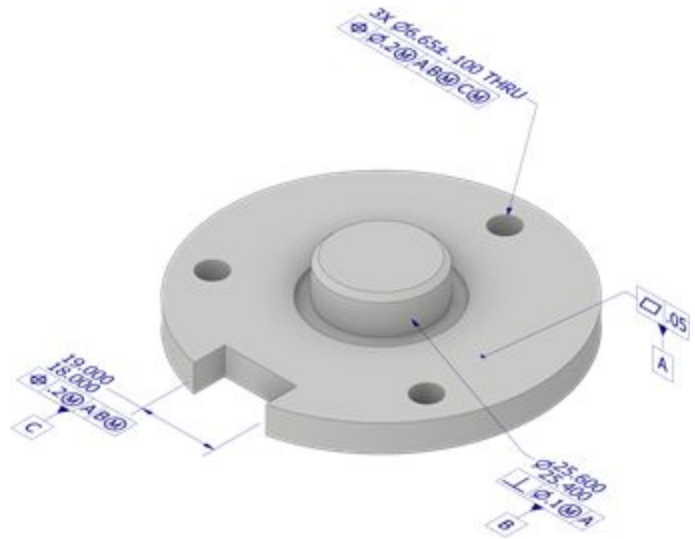
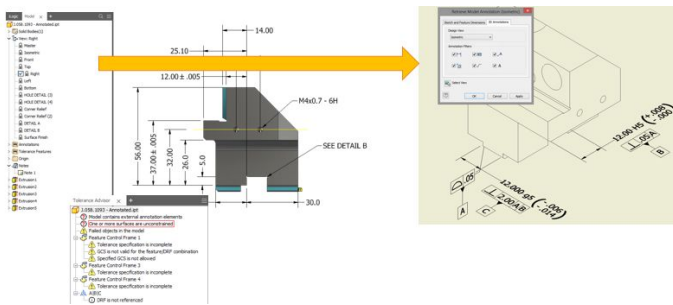
10. Select the object from step 8 and select the **Mirror** tool.
  - o Select **Plane E** (seen in **FIGURE 36**) and mirror the object.
11. Create a plane on side **F** as seen in **FIGURE 36** and project the edges of the object in a new sketch.
  - o Then, replicate the sketch in **FIGURE 37**, where **G**=0.5", **H**=0.75", and **I**=1.25".
12. Exit the sketch and **Extrude** the circles 1" into the structure.
13. Create a sketch on the opposite end of the figure.
14. Project the edges of the holes created in step 12 onto this sketch.
  - o This will transcribe the holes onto the new sketch.
15. Exit the sketch and extrude these circles out 2.5".
16. Your final product should look like **FIGURE 38** below.

# WHAT'S NEW IN AUTODESK INVENTOR 2018

## Model-based definition:

Model-Based Definition (MBD) is a powerful new set of tools for adding annotations, GD&T, and other manufacturing information directly to a 3D part. The integrated Tolerance Advisor checks the health of your tolerance scheme and lists potential problems, errors, and other information in the Tolerance Advisor browser.

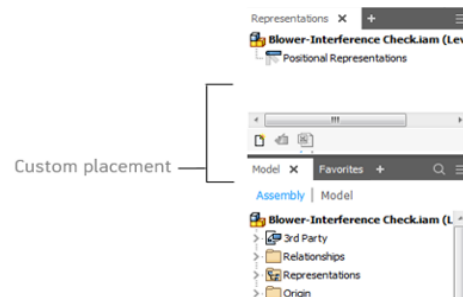
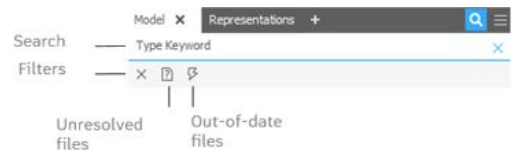
- Once your 3D model has MBD and 3D Annotations applied you can recover in your 2D drawing, export in a 3D PDF or STEP AP242.



## Browser Tools:

Save time with the exciting new browser tools.

- Find things quickly and easily in the current active document with the new browser search.
- In an assembly file, use the filter tools to narrow your search (not available in Inventor LT).
- Move and dock each tab to your preferred way of working. The following image is an example of side-by-side docking.

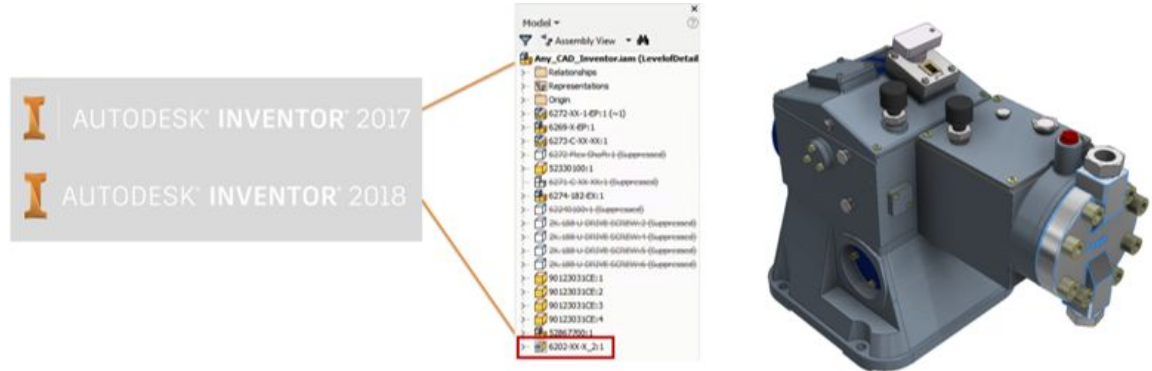




# 2018 FIRST Kickoff Event at Montgomery High School hosted by Team 1403 Cougar Robotics Autodesk Inventor 2018 Workshop taught by FRC Team 1403 Design Team

## AnyCAD:

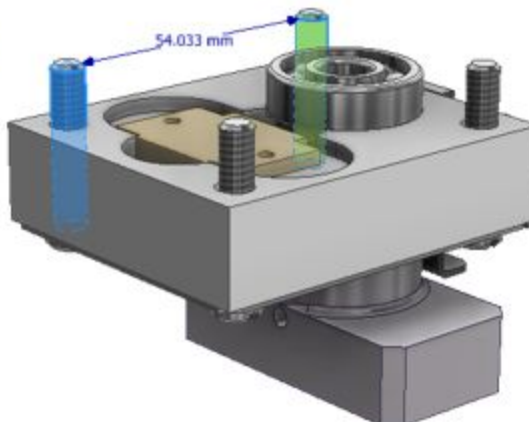
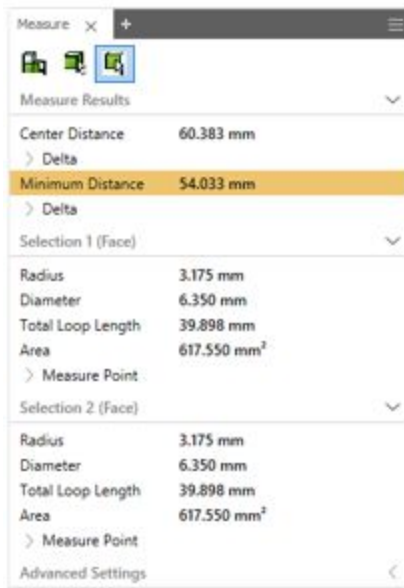
- With Inventor 2017.4, AnyCAD has been expanded to allow you to leverage Inventor 2018 files in your Inventor 2017 designs.
  - This means you can now work with others who are on versions of Inventor that are earlier or later than the version you are working in. To help you identify the version of an Inventor file, the Open dialog now displays Inventor version a file was last saved in.



## New and Improved Measure Workflows and Graphics

The Inventor measure tools and workflows have been greatly simplified and improved to enhance productivity. You can now:

- Use a single measure command!
- Perform all measure workflows from the new measure tool panel.
- Display rich information for each selection with a single click.
- Easily measure angles and identify measure selections in the graphics window.
- Easily identify measure selections in the graphics window.
- Additional productivity enhancements.



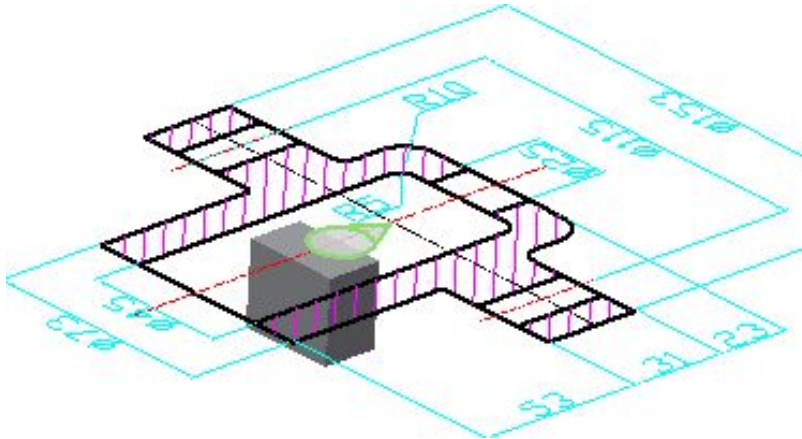
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## Autodesk Inventor 2018 Workshop taught by FRC Team 1403 Design Team

### DWG Underlay:

New DWG Underlay features make working with DWGs faster.

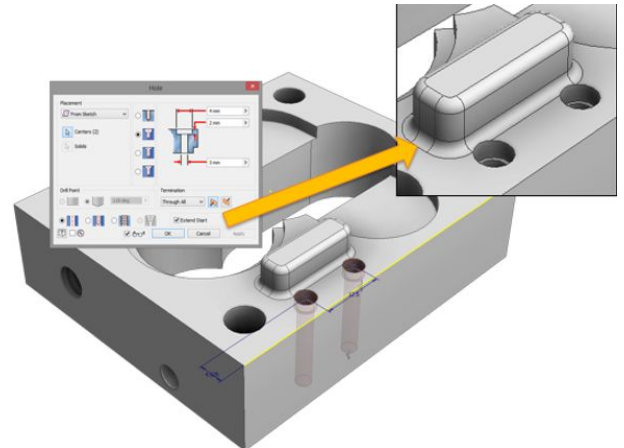
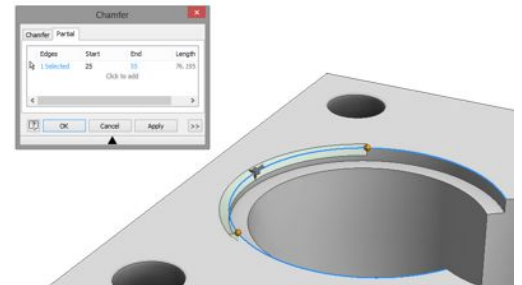
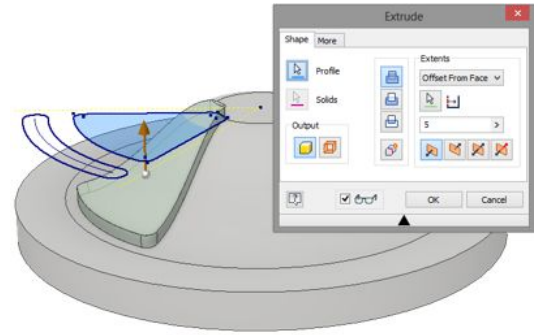
- Import your DWG file directly into an assembly using the Place Components command.
- Select multiple DWG Files when creating more than one underlay.
- Open an AutoCAD DWG file directly from the right-click context menu in your part or assembly model browser node in Inventor.
- Automatically project certain types of DWG geometry when you enable the new Application Options > Sketch > Autoproject edges during curve creation option.
- If the selected DWG Underlay geometry is within the XY, YZ, or XZ plane, the resulting joint is created on the same plane as the sketch in the same way a joint is created between 2D Inventor geometry.
- Create a joint on the same plane as the sketch in the same way a joint is created between 2D Inventor geometry: If the selected DWG Underlay geometry is within the XY, YZ, or XZ plane, the resulting joint is created on the same plane as the sketch.



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## Part Creation:

- Extrude:
  - Use the new Distance from Face option in the Extrude dialog box to start an extrusion from a face or workplane with a distance. Because Distance from Face creates one central entry point for modifying the geometry, no additional steps to make sketches or workplanes are needed.
- Chamfer:
  - You can now create a chamfer that does not require an entire edge. A new tab, Partial, is added to the Chamfer dialog box to support the creation of partial chamfers.
- Hole:
  - You can now create a symmetric hole type that extrudes in two directions. The Flip Termination button is replaced with Direction1 and Direction 2 buttons. The Symmetric distance is only available for the Drilled - Simple Hole - Through All hole type.
  - You can now specify zero (0) as a value for bore depth on Spotface holes. As a result, you can now place a spotface hole on the termination face of a body.
  - Use the new option, Extend Start, on the Hole dialog box to extend the start face of a hole to the first place where there is no intersection with the target body. The purpose of Extend Start is to remove a fragment resulting from the creation of the hole.



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## Presentation updates:

Presentation documents now provide more robust support for surface bodies:

You can:

- Use window selection to select multiple surface bodies.
- Use the Tweak command to move surface bodies.
- Use surface geometry to orient and position the Tweak triad.

