

THE  
**COMPASS**  
ALLIANCE



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HELP HUBS



TAG TEAMS



RESOURCES



FIND YOUR WAY FIRST



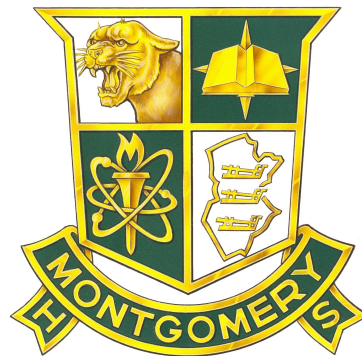
# ***The Compass Alliance Workshops: Intro To Inventor Tutorial Packet***

Presented by FRC 1403 - Cougar Robotics

*Montgomery High School*

***Saturday, September 21, 2019***

***9:15 am - 10:00 am***



# 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.

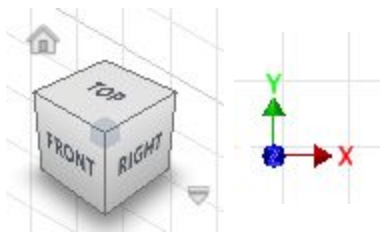
Shortcut keys. Throughout the course of this tutorial, I have included only a few of the many useful keystrokes that can increase the speed at which you use Inventor. Some others you may find helpful are listed here for your reference.

Sketching:	Modeling:	Assembling:			
D	Dimension	E	Extrude	P	Place
component					
X	Trim	C	Constrain	R	Revolve
L	Line	H	Hole		
F	Fillet				

## 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.

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



The X-Y-Z  
coordinate  
axis  
indicator

change the  
working

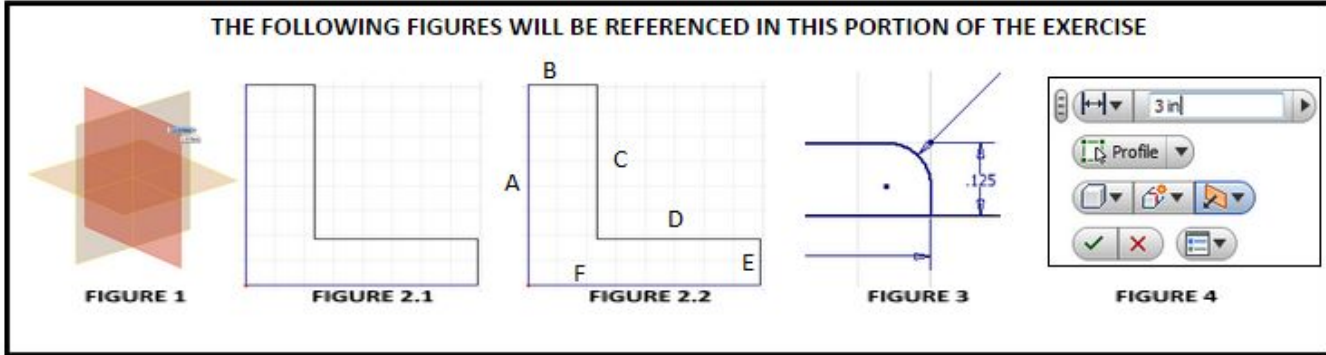
## SHORTCUT KEYS:

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 I: CREATING A PART

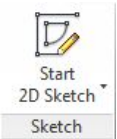
1. Double click on Autodesk Inventor Professional 2020 on your desktop.
2. Once Inventor has loaded, click on **New** located in the top left corner of the screen.
3. In the **New File** window that opens, double click on **Standard.ipt**. You have now started a new part.



4. To begin, you must first create a new sketch. Click on **Start 2D Sketch** in the top left corner. Select the **XY Plane** highlighted in dark orange in **FIGURE 1**. Notice that the toolbar on the top has now switched to the **Sketch** Toolbar.
5. In the Sketch toolbar, click on the **Line** tool.
6. To use the line tool, click once where you want the line to begin, and then click again where you want it to end. Notice that after ending the first line, the line tool will start the next line at the endpoint of the first line. To start a line at the origin, move your cursor over to the yellow point in the middle of your workspace. You will notice the origin turn a bright green color when you are over it. At this point, you can click once to select it. Now, using the line tool and starting at the origin, make the shape shown in **FIGURE 2.1** above.

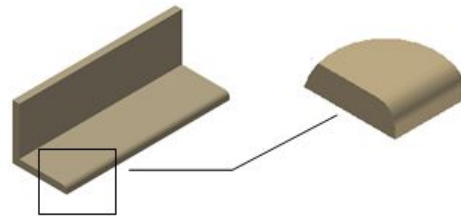
*NOTE: It is OK if your shape does not appear exactly as shown in **FIGURE 2.1**, however you should have the same general shape. Your sketch will look more like the shape in **FIGURE 2.1** after you dimension it in step 7 below.*

7. In the Sketch toolbar, click on the **Dimension** tool
  - Move the pointer over a line to highlight it, and then click the line to create a dimension. Move your cursor away from the line you selected, and then click again to place the dimension. A new box will open and allow you to edit the new dimension.
  - Type the new dimension in the box and click on the green checkmark to create it.
  - Dimension the shape as shown in **FIGURE 2.2** above, where **A=1"**, **B, E=0.125"**, **C, D=0.875"** and **F=1"**.
  - When you are done dimensioning, click on the green check mark in the sketch toolbar, or right click and select **OK**.
8. Now click on the **Fillet** tool at the top of your screen.
  - The **2D Fillet** menu will appear. Set the radius of your fillet to be **0.075"**.
  - Click on the lines labeled **"D"** and **"E"** in **FIGURE 2.2** above. Your sketch should now look like **FIGURE 3**.
  - Right click & select **"Ok"**.
9. Exit the sketch by right-clicking, and selecting **Finish 2D Sketch**. The **Sketch** toolbar changes to the **3D Model** toolbar
10. Click on the **Extrude** tool located at the top of the **3D Model** toolbar
  - Because you only have one sketch present in this part file, Inventor automatically selects the profile you have just created as the one that will be extruded.
  - Change the distance to **3"** as shown in **FIGURE 4**.

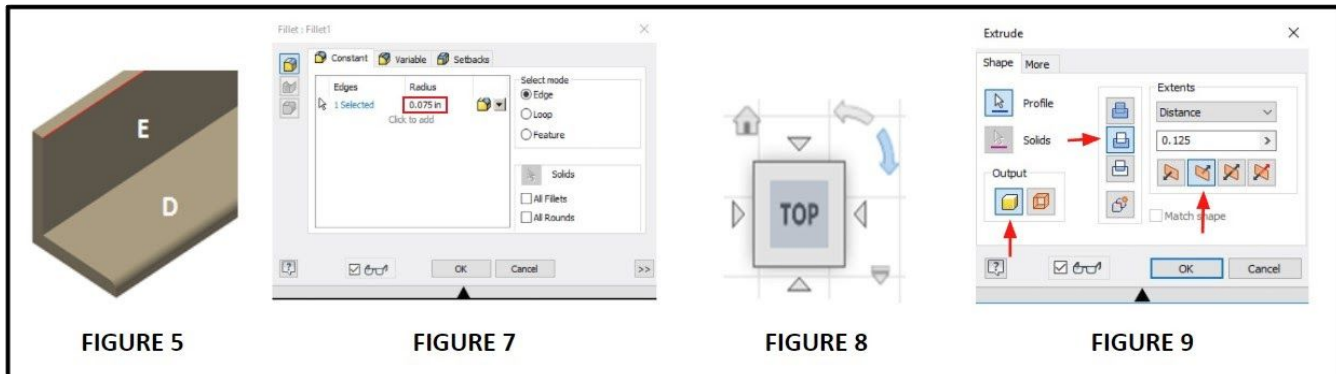


- Click the green check mark, or right click and select **OK**

11. Your part should now look like **THIS** →



NOTE: Only the bottom edge of the part should be rounded as seen in this detail view



The next step will be to create two holes on sides **E** and **D**, so that we can attach this part to others in a later assembly.

- You can also add a fillet once a sketch has been extruded. Select the **Fillet** tool from the **3D Model** toolbar. Click on the edge indicated in **RED** in **FIGURE 5** above. (This is the edge corresponding to the intersection of sides “E” and “D” in **FIGURE 2** on the previous page)
  - In the Fillet Menu shown in **FIGURE 7** above, dimension the fillet to be 0.075”.
  - Click OK add the new fillet.
- Now Right-Click on the face labeled “D” in **FIGURE 5**, and select **New Sketch**.
  - Using the **VIEWCUBE**, orient the part so that the rounded edge of face **D** is facing downward, and you are looking straight down at face **D** as displayed in **FIGURE 6** (on the next page) by clicking on the arrow highlighted in blue in **FIGURE 8** above.
  - Using the **Circle** tool, draw two circles anywhere on the part, as seen in **FIGURE 6**.
  - Click **Dimension** and select the center of the left circle and edge **(A)**. Edit that dimension to be 0.75” and repeat it with the right circle and edge **(B)**. Then click on the center of one of the circles and bottom edge **(C)** and edit the dimension to be 0.5” and repeat that process for the other circle.
  - Dimension the circles to have a diameter of 0.25”.
  - Exit the Sketch

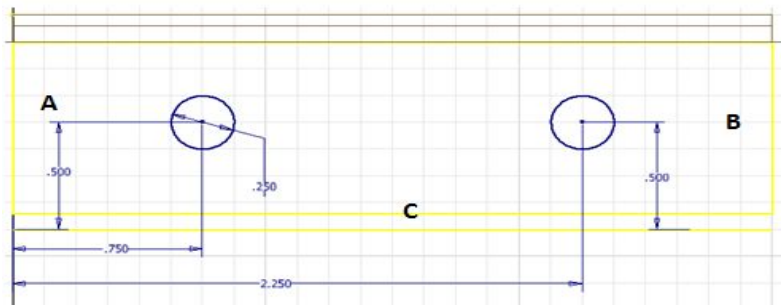
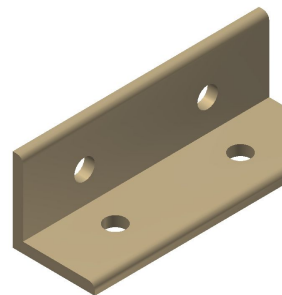


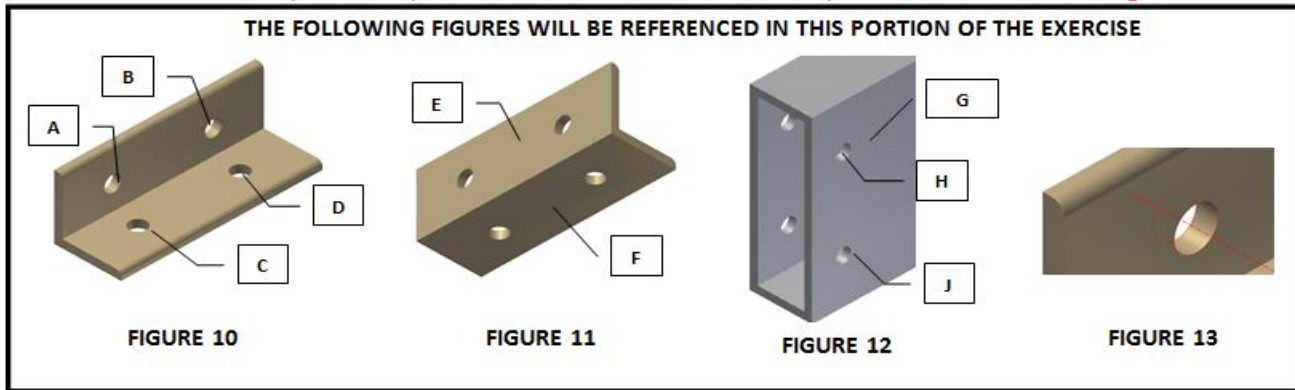
FIGURE 6

- Click on the **Extrude** tool, and click inside both circles to select them.
  - Select the **Cut** option as seen in **FIGURE 9** above. These options are highlighted in BLUE as being selected.
  - Click the green check mark to cut the holes.
- Repeat steps 13 through 14, but now use the side labeled as “E” in **FIGURE 5**
- Your part should look like **THIS** →



## SECTION 2: CREATING AN ASSEMBLY

\*There is a folder on your desktop home screen labeled “**TUTORIAL PART FILES**” that you will need to access during this section\*



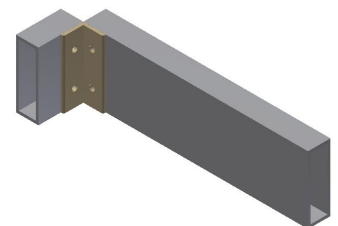
1. Go to **File** → **New** → **Assembly**, or alternatively if you were to click on the **New** button, you would select **Standard.iam**
2. Click on **Place**, and then navigate to the **TUTORIAL PART FILES** folder on the desktop.
  - Double click on the part labeled **Tutorial Part1**. Click once to insert it into the document.
  - Repeat this for **Part 1**, and **Tutorial Part2**, double clicking to select the part, and once in the document to place it.
  - Once you are done placing the above components, press the **Escape** key to exit the **Place** tool.
3. Click on the **Constrain** tool. This is the tool that will be used to mate or to create a relationship between components. This will allow for the parts to be bound together so they don't move or only move within certain parameters of each other.

*During this step, you will need to rotate your view so that you can select the different faces necessary to mate the components together. To rotate your view, you can either press and hold **F4** and click and drag with your mouse, OR by using the viewcube.*

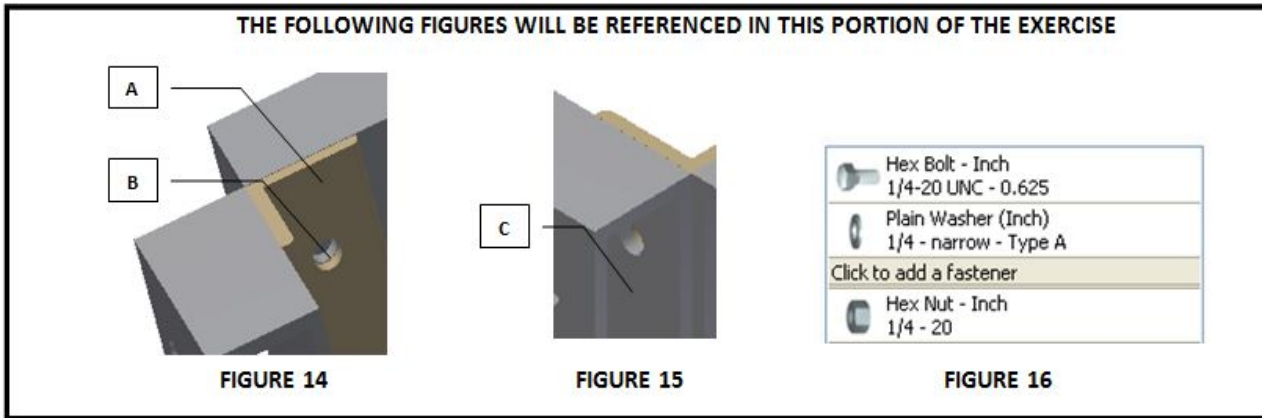


- **STEP 1 – Joining “Part1” (FIGURES 10 & 11) and “TUTORIAL PART1” (FIGURE 12)**
  - i. To make a concentric mate between two holes, move your cursor over the first hole so that it is highlighted with a dashed line running through the center as shown in **FIGURE 13** above, and then click once to select the first hole, and repeat for the second hole. **\*\*IMPORTANT: in FIGURE 13, the dashed line is shown in red. You may have a different color dashed line. This is OK, what is important is that you have the DASHED line running through the center of the hole when you select it. DO NOT SELECT THE OUTER EDGE OF THE CIRCLE.\*\***
  - ii. Make a concentric mate to join Hole **A** in **FIGURE 10** with hole **H** in **FIGURE 12**. Click **Apply** in the **Constraint** window, or hit the **Enter** key to apply the mate.
  - iii. Make a concentric mate between hole **B** in **FIGURE 10** and hole **J** in **FIGURE 12**. Apply the mate to complete it.
  - iv. To mate part faces together, click once on the first face, and then click once on the second face. Mate face **E** in **FIGURE 11** and face **G** in **FIGURE 12**. Apply the mate to complete it.
- **STEP 2 – Joining “Part1” and “TUTORIAL PART2”**
  - i. This is done using the same methods used in **Step 1**

Your final assembly should look like **THIS** →



## SECTION 3: CREATING A BOLTED CONNECTION



1. While you still have your assembly open, click on the **Design** tab located next to **Assemble** in the top left of the screen, and then select **Bolted Connection**. The **Bolted Connection Component Generator** opens.
  - Under the **Placement** menu, click on the drop down menu, and change the selection from **Linear** to **Concentric**.
  - For the start plane, select face **A** in **FIGURE 14**. For the circular reference, select hole **B** in **FIGURE 14** **\*\*IMPORTANT\*\*** make sure you select the **INSIDE FACE** of hole **B**! This is done the same way you selected the hole when making a concentric mate. For the termination point, select face **C** in **FIGURE 15** above
  - For this bolted connection, go to the diameter drop down menu and select 0.25 in.
  - Next, select **Click to add Fastener**, change the category to **Hex Head Bolt**, and select **Hex Bolt - Inch**.
  - Again, select **Click to add Fastener**, scroll down to row 4, and select **Plain Washer (Inch)**
  - Select the second available option for **Click to add Fastener**, change the category from **Washers** to **Nuts**, and select **Hex Nut - Inch**.
  - The selected options for your bolt should appear as in **Figure 16** above.
  - Click **Apply**, and select **Ok** when prompted to name your file. Your bolt will now appear in the assembly.
2. Save this as **BOLTED ASSEMBLY I**  
*\*You may notice that every time you select a component of the bolted connection you are building, that a green "wire frame" outline of that component will appear on the assembly as a preview of what you are doing.*



## SECTION 4: DRAWINGS

### BASIC STANDARDS

*Drawings are an essential part of the Design Team's contributions to the build process. 3D CAD Models can be used to create drawings which are used for creating the physical part. It is important to have drawings standards to avoid confusion and inconsistency.*

#### RECOMMENDED UNIVERSAL STANDARDS:

- Every drawing is made to fit a standardized size for printing.
- Drawings should always use a unique template which has a proper title block. Teams should have customized templates that showcase their team logo and FRC involvement.
- The Template Title Block should include areas for:

Part name	follow team's naming standards/part catalogue
Drawing author	build team to have a person to consult with questions about the part
Date of drawing creation	prevent use of outdated drawings
Scale of drawing	attempt to maintain accuracy of drawings
Revision number	prevent use of outdated drawings and allows for comparison of different part iterations as it is modified

#### FRC TEAM 1403 DRAWING STANDARDS:

- A standard drawing is 11" x 17" with a landscape orientation.
- Always use the specific FRC Cougar Robotics Drawing Template. The template includes both the Team 1403 Cougar Robotics logo and the FRC logo in the title block.
- It is customary to use a minimum of 3 views of the part, 2 of them being 2D views for measurements and 1 being a 3D projected view to give a sense of what the part looks like.
  - More than 3 views can be used to properly define a part
- Annotations must be consistent
  - Dimensions must be accurate to 3 decimal places. See reference **A** in **FIGURE 39** on the next page.
  - Centers of holes must be marked and the dimensions should start from there (reference **B**).
  - Always dimension the measurements from the same corner to make the part easier to create/cut (reference **C**).
  - If multiple holes in a part share the same diameter, make sure to put in the "TYP" note next to the dimension (reference **D**).
- A revision note should be added after the drawing is initially made, then changed for the amount of times the drawing has been changed or modified no matter how small or large the modification
- Always follow the part-naming system to make it easier to catalogue and organize the parts.
  - Format should be: *Source/Vendor.SubassemblyCode.PartTypeCode.RRevision#[Descriptive Part Name]*
  - (e.g.) 1403.DT.11.R2 [2 x 1 x 24 in. Drive Train Block]
- Figure 39 on the following page shows an example of a standard Team 1403 drawing

# FIGURE 39



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

Figure 40

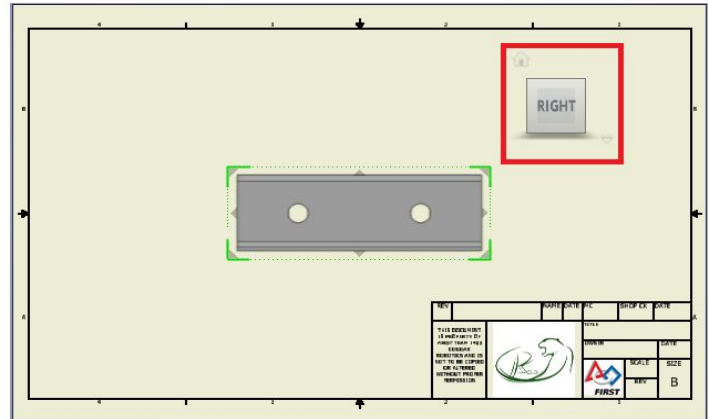
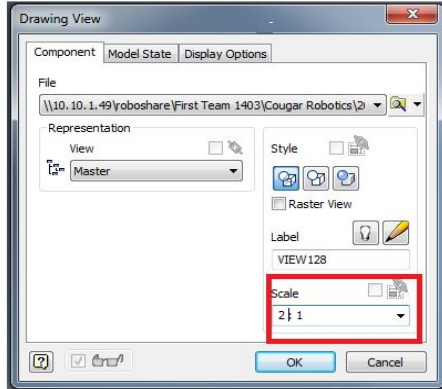
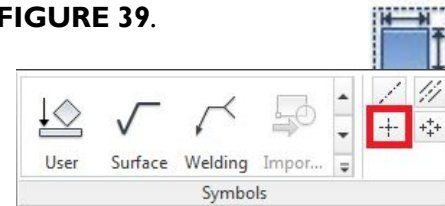
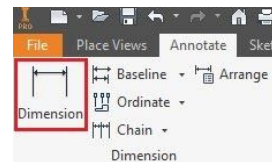


Figure 41

## CREATING A DRAWING

1. Click on **BASE** in the upper left corner. Then, click on the button to the right of the file directory and open **PART I** (saved wherever you saved it during Section 1).
  - Note: If a part is open in another tab, it will automatically be selected.
2. The part will appear in the center of the screen in the front view. Adjust it so that it is in the **Right** view. In the bottom right of the window, you can adjust the scale to change the size of the drawing. Set the scale to 2 : 1. Your window should look like **FIGURE 40**.
3. Click on **PROJECTED** in the upper left corner.
  - Moving the mouse around the view will allow you to change the angle from which the part is viewed.
4. Create at least 3 views (use **FIGURE 39** as a reference), then right click and select OK.
5. Click on the **DIMENSION** tool to begin annotating.
- With this tool, you can annotate various measurements and dimensions the same way you would dimension a sketch.
- Annotate the drawing to match **FIGURE 39**.
- (Note: when annotating circles, select the outer rim so that a green dot **does not** appear)
6. Use the **CENTER MARK** tool in the **Symbols** section of the toolbar (highlighted in red below) to indicate the centers of the holes. Mark the two holes in the center view just like in **FIGURE 39**.
7. When dimensioning the circles, make sure to add "TYP" to indicate when holes in the part share the same dimensions
8. Make sure to annotate any fillets as well (add "TYP" to indicate fillets with the same dimensions)
9. You now have enough dimensions annotated to recreate the part.
- Your drawing should look similar to **FIGURE 39**.



***NOTE: Make sure to make enough annotations so that somebody can recreate the part using the dimensions!***