FIRST TEAM 2708
Lake Effect Robotics

CAD
TRAINING

Part 1 – Sketching and Making 3D Parts

Contents

Getting Started

Get Autodesk Inventor

Go to http://students.autodesk.com/ and make an account. Go to the Free Software section and choose Inventor Professional. Select 2017 for your version, English for your language and select the appropriate version of windows

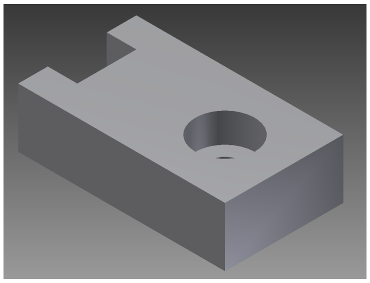
A quick way to figure out if you are running 64 bit or not is to navigate to your local drive C:/ (you can get there through my computer) and then look to see if you have a folder named Program Files (x86) and Program Files if so then you are running 64 bit windows if you only have one folder called Program Files then you are running 32 bit.

Download the file (it will take a while) and then click on it: "AutoCAD_Inventor_Suite_2017_Win_64bit.exe" It will ask for a location and then begin extracting files.

After the files have finished extracting navigate to the folder where you extracted them (most likely called "AutoCAD_Inventor_Suite_2017_Win_64bit" or 32bit) and click on Setup.exe. Wait for the program to launch then choose "Install Products" on the left. Press next. This screen should say "Select the Products to install" You only need Autodesk Inventor and the Content Center Libraries, deselect everything else. Accept the license agreement and then on the next page enter your name and serial key (obtained from the Autodesk student website, if you lose yours just login to your account and go to download inventor again, it will give you your key). On the next page you can configure your installation but the default settings should be fine. Press Install and wait while Inventor installs.

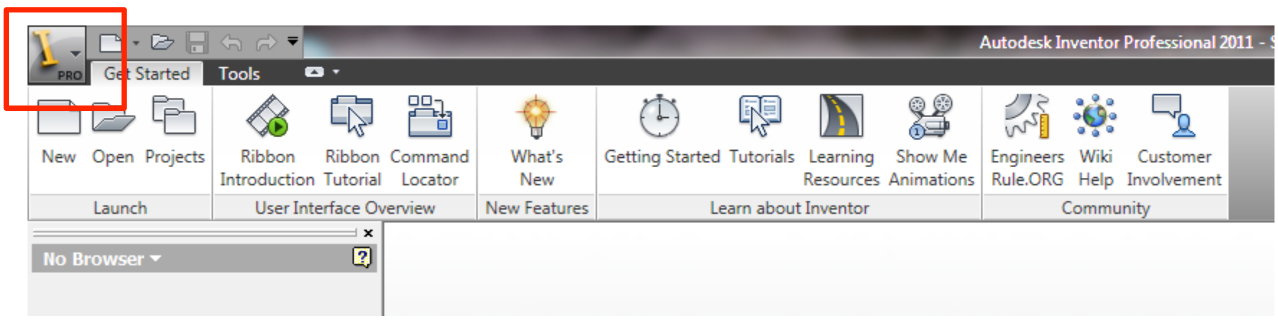
Afterwards you may delete the folder where you extracted the installation files ("AutoCAD_Inventor_Suite_2017_Win_64bit" or 32bit) as well as the downloaded files but you may want to keep them in case you need to install on a different computer.

Lesson 1:

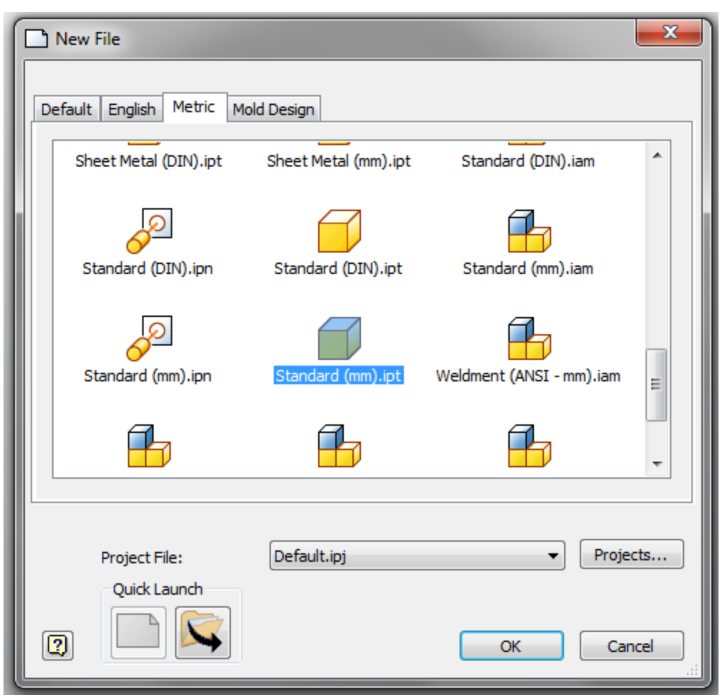


lesson1.ipt

Open Autodesk Inventor and from the drop down menu select "**New**"

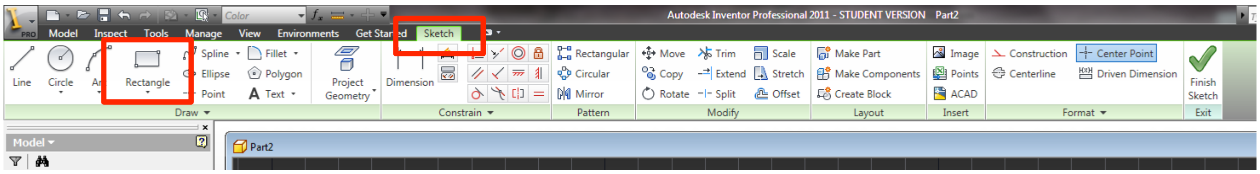


In the pop up menu select the "Metric" tab and scroll down to find "Standard (mm).ipt" Every CAD program has different file extensions, for Inventor parts are .ipt assemblies are .iam and 2D drawings are .dwg. For now only worry about .ipt. We are going to do these tutorials in metric but the units can easily be changed to imperial.

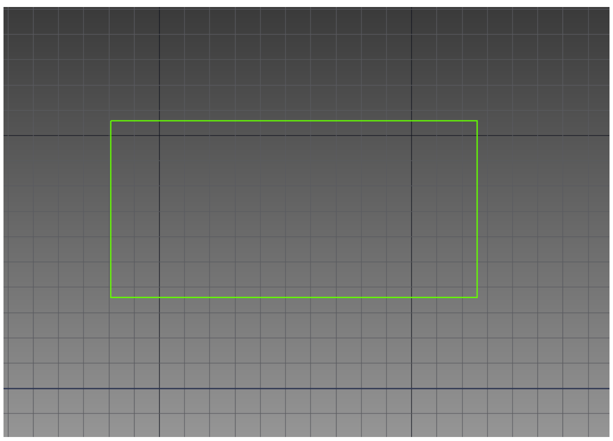


Now we have created a new part and Inventor has automatically created a new Sketch for us. Sketches are 2D drawings; the basic work flow of CAD is like this -> **create a 2D sketch then create a 3D feature from that sketch.**

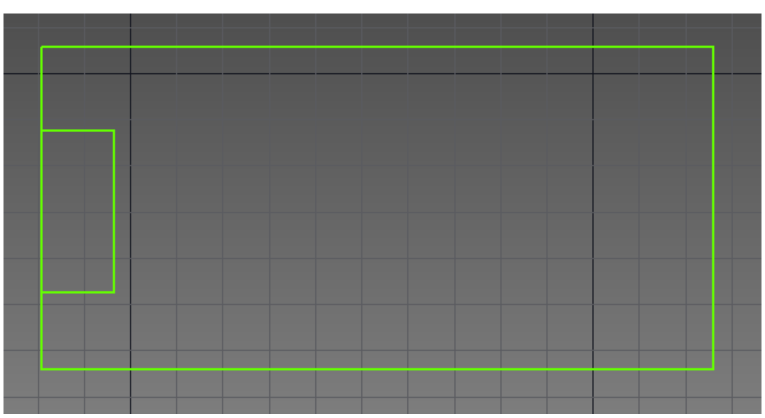
Take a moment to look at the ribbon interface across the top of the screen; we are in the sketch tab. If you are ever confused and can't find a tool make sure you are in the right tab.



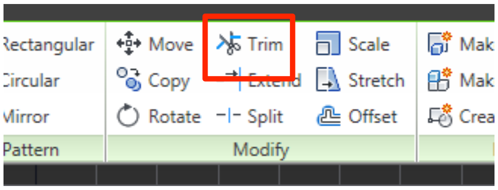
Now it's time to begin modeling our part. Select the rectangle tool and draw a rectangle, don't worry about its position or size etc. we'll take care of that later.



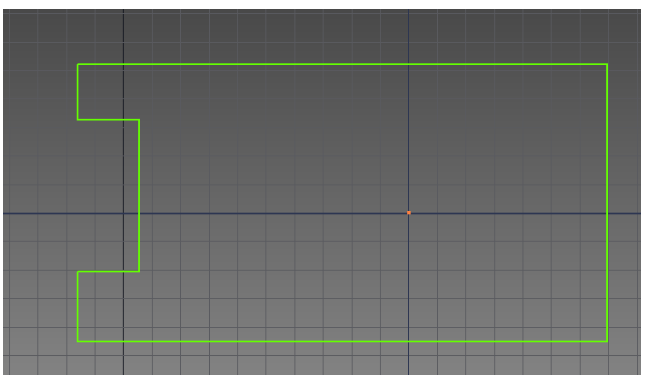
Select the Line tool and draw the following lines in your rectangle, again don't worry about exactly how they look for now. Make sure that your lines are connected to the rectangle at the end, your cursor should snap to the rectangle when you are close by. Also watch the symbols that Inventor is showing you as you draw, you'll probably notice the parallel and perpendicular symbols: this is Inventor creating constraints as you draw. Don't worry we'll learn more about constraints as we go.



Select the Trim tool and click on the line pointed to in the image below:

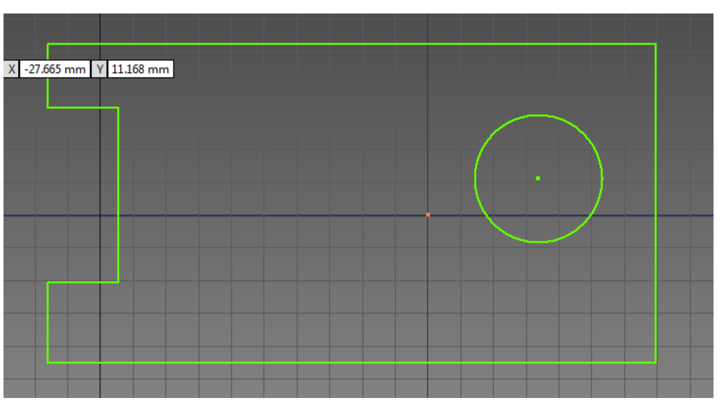


You should be left with this:



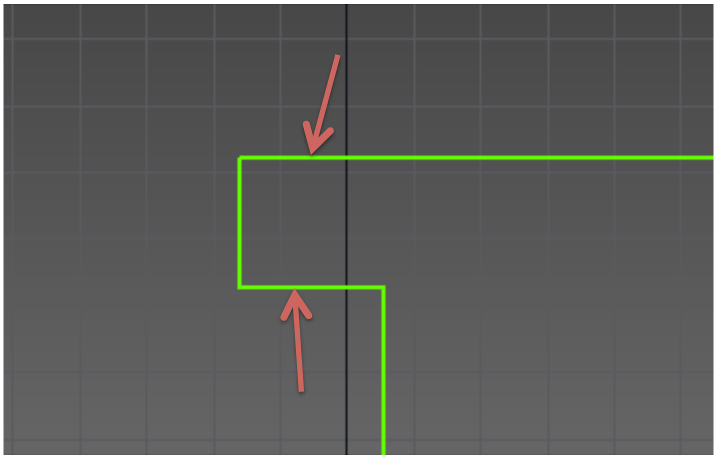
Trim removes lines between or after another line intersects. In this case it removed the portion between the two intersecting lines. Trim is very useful for creating geometry.

Next select the circle tool and draw a circle inside the shape we have made. This should be your result:



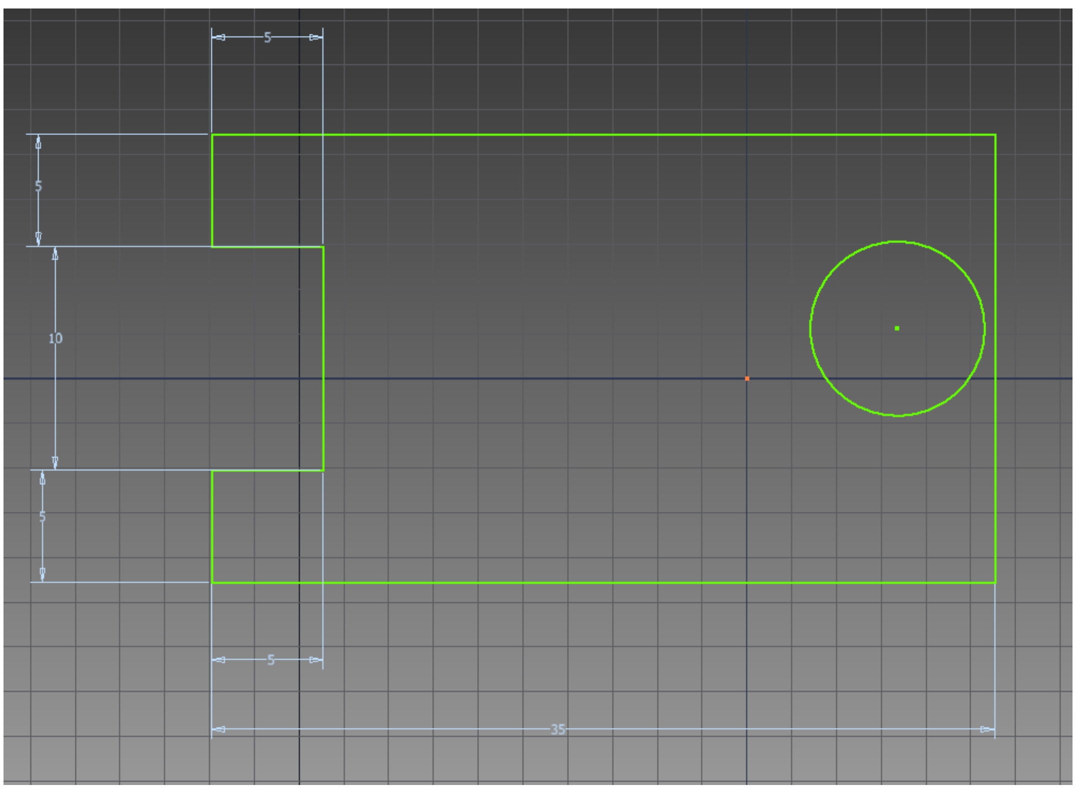
Before our sketch is finished we need to apply some dimensions and constraints. Dimensions force the given section to be a certain length and constraints for a certain geometric relationship.

You can add a dimension either by clicking on a line itself or by clicking on the lines on either side like the illustration below:

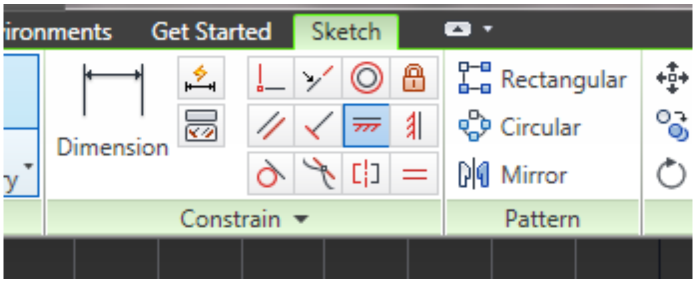


Clicking these 2 lines will allow you to dimension how much should be between them and because the other line is connected its length will change too.

Add these dimensions to your sketch:



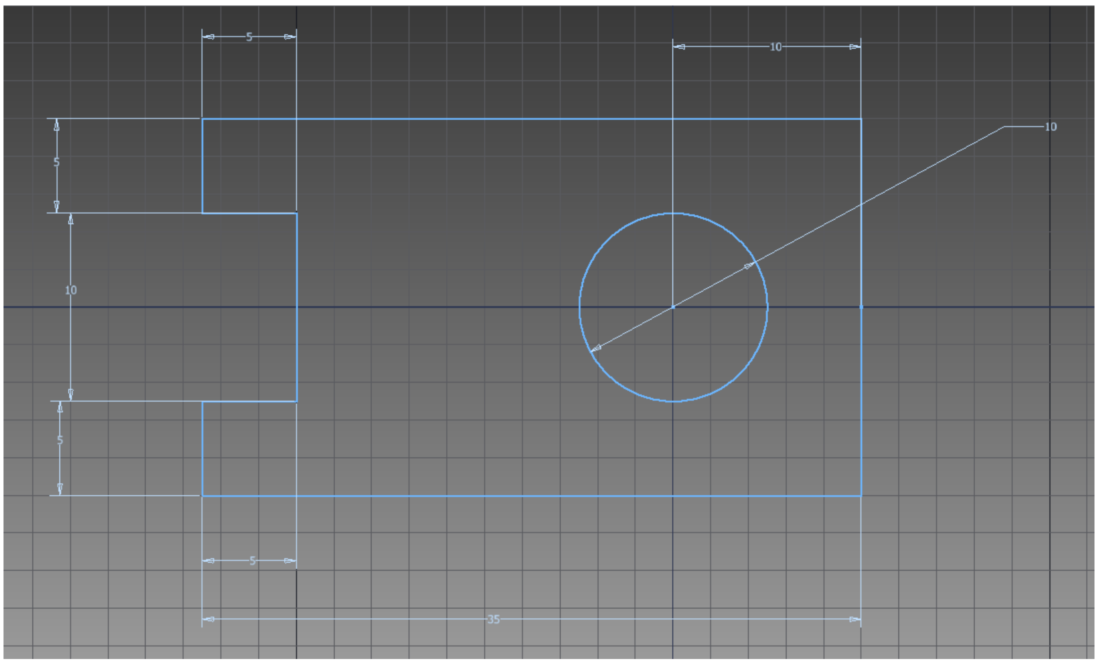
Next we are going to apply a constraint to the circle. Select horizontal from the constraints.



Pick the center of the circle and then carefully hover along the vertical line at the right end of the rectangle, when you pass the midpoint of this line your cursor will change to a green circle instead of a yellow one. Click to finish the constraint. Your circle should now be centered vertically in your shape. Apply a dimension of 10 mm between the center of the circle and the vertical line on the right and make the diameter of your circle 10 mm.

Finally lets apply a coincident (fancy way of saying in the same place) constraint between the center of the circle and the sketch origin (the intersection of the darker blue lines, it's orange).

The result should look like this:



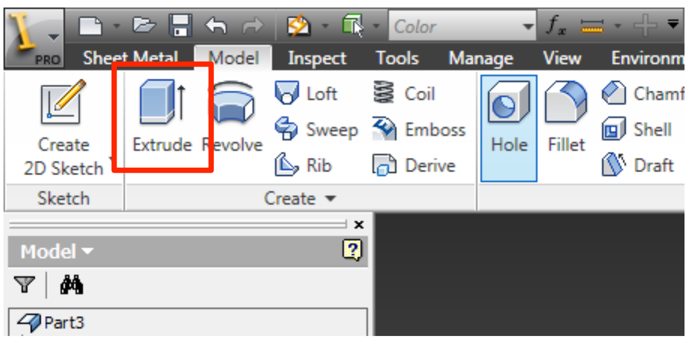
I would like to take a moment and mention a few things, first there is no right way to do things in CAD there are often many ways to accomplish the same thing. I recommend always using the simplest way you can think of. For example we could have used many complicated constraints to make this sketch but instead we used a lot of simple dimensions, this is good for 2 reasons: it was easy to do, and easy to understand if we have to come back to edit it later.

In this lesson so far we introduced 2 of the many constraints – horizontal and coincident. The remaining constraints are pretty self-explanatory just think about geometry class!

Also, notice how we only needed to dimension 1 side of the rectangle, that's because Inventor automatically created constraints for us that keep the sides equal to each other. If you try to dimension the other side, Inventor will warn you about an over-constraint – this happens when the shape is already fully defined with constraints and dimensions – in other words that side length is already determined it cannot be anything else or it would disobey the laws of geometry.

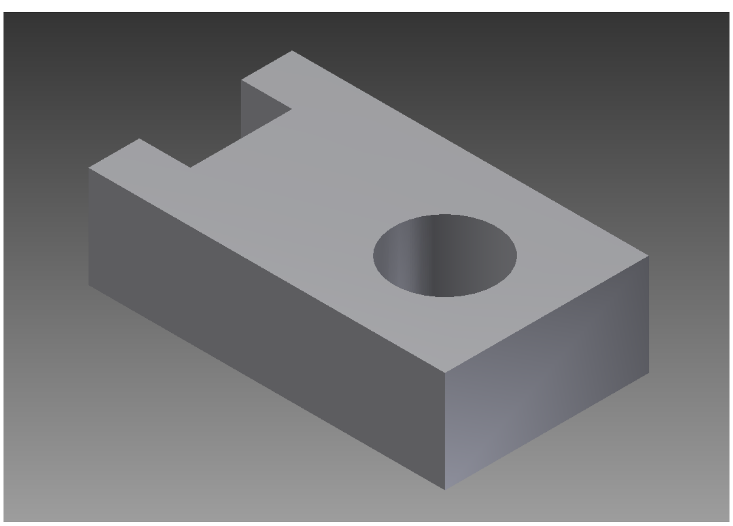
Don't worry about learning every tool in CAD: I nor any of your other mentors know them all; we just know the basics and when we need something else we look at what tools are available and play around until we get our result.

Let's finish our part. Click on finish sketch in the top right of the screen. Observe how the view changes to a 3D or isometric view. On the ribbon interface switch to model and then select the extrude tool.



Select the profile of our sketch (it will become shaded light grey), choose 10 mm for the distance and press okay.

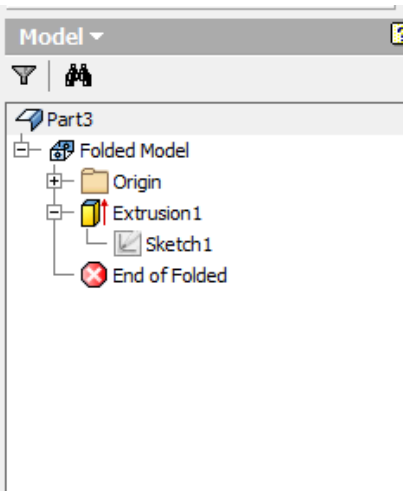
You should see this on your screen.



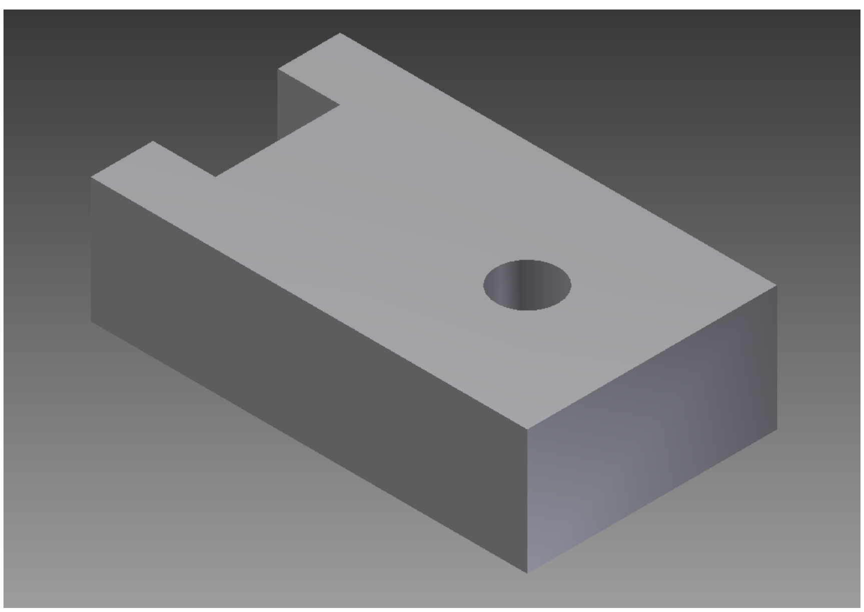
Now look at your screen on the left. A tree view is being created of how your part was made.

At any time you can edit any step of how your part was made, sometimes this works great and we can easily change the part other times when the part is really complicated it doesn't work so well. There is ways to fix this and work through it but for now let's just focus on the basics and we can help you with making changes to complicated parts.

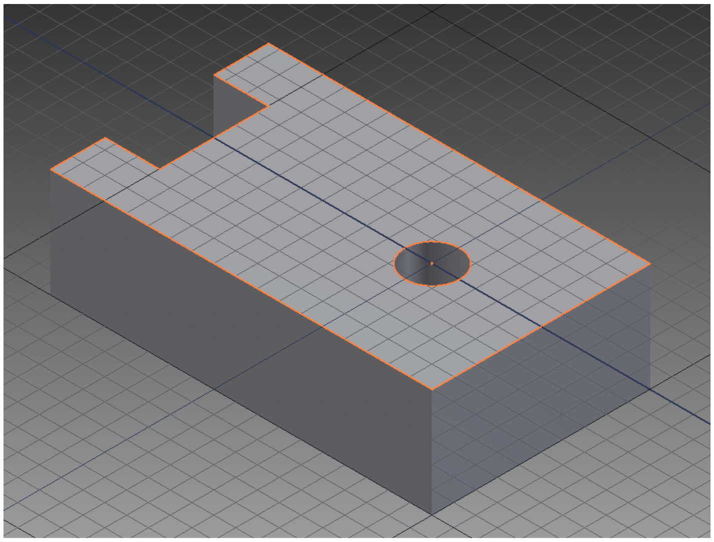
Click the plus next to Extrusion one and then right click on sketch 1 and select edit sketch.



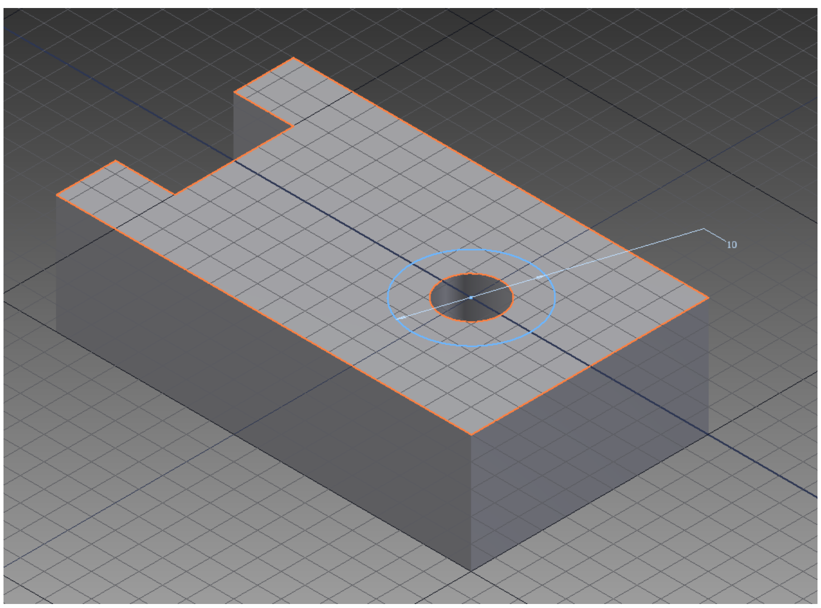
Change the diameter of the circle to 5mm. Click finish editing sketch. Observe the result:



Now let's make a new sketch. Click Create 2D sketch and click on the main surface of the part. Your screen should look like this:

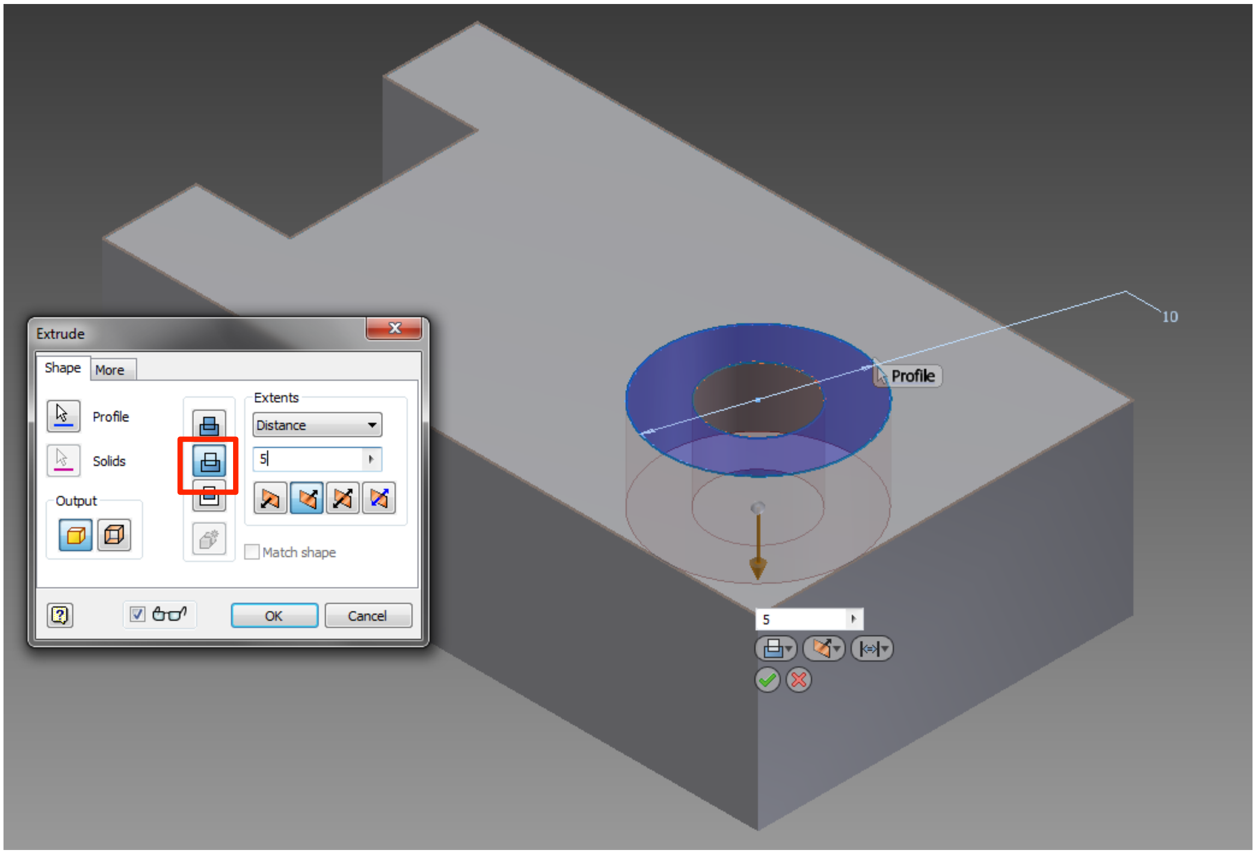


Click the circle tool and hover your mouse over the hole through the part. When it turns green draw a circle. Make the diameter of that circle 10 mm. Your part should look like this:

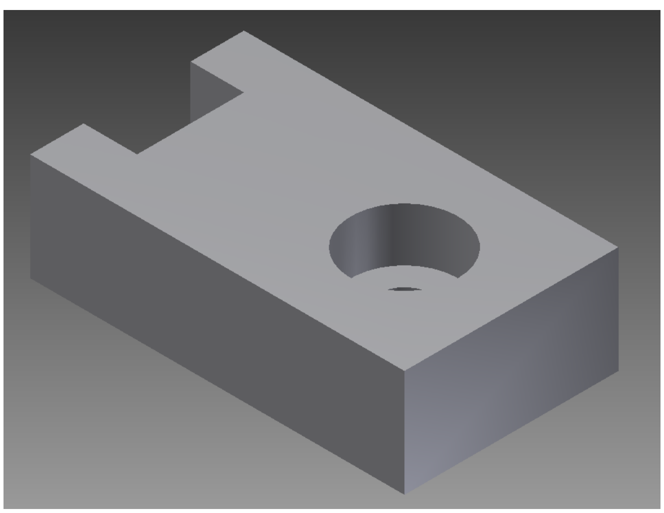


Finish the sketch.

Make sure you are on the model tab then select extrude. Select the profile of the new circle we just drew then from the extrude pop up. Select cut and make the distance 5 mm.



Click OK and look at your new part.



Save your part as “lesson1.ipt”. You’ll need to submit it later.

Before moving on, take a few minutes to familiarize yourself with looking at a part in 3D. You can access all the move/rotate/zoom commands from the dock bar on the right of your screen, however I recommend using the keyboard short cuts. Pressing on the roller wheel on your mouse will pan or move your view of the part around. Holding shift and clicking the roller wheel will let you rotate the view of the part and rolling the roller wheel will zoom in and out. You can also use the cube in the upper right corner to select certain viewpoint. It is important to note that you are not moving the part. Think of it as moving yourself around the part so you can see different things.

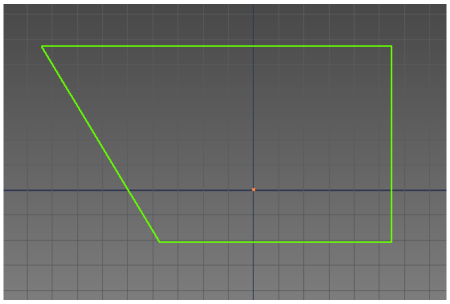
Congratulations you have finished lesson 1 and are well on your way to be **CADing robots!**

You might not believe me now but CADing is actually quite simple, there may seem like a lot of tools, and there are, but **95%** of parts are made using the simple **sketch, dimension, extrude and cut** process you just learned!

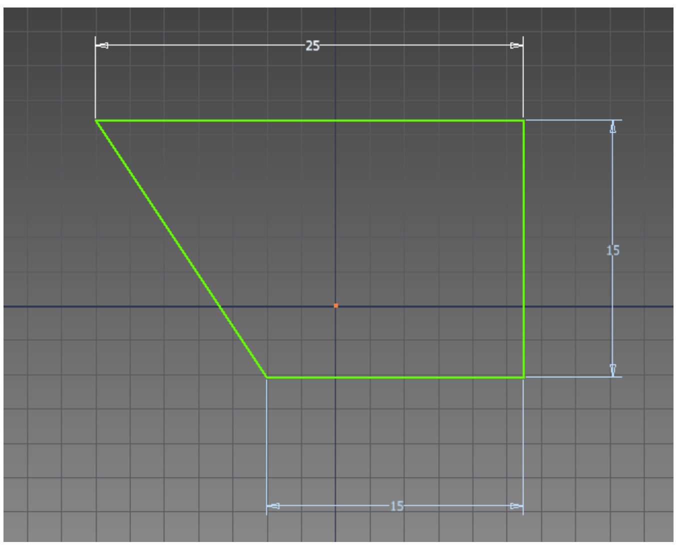
Lesson 2:

Open Inventor and start a new metric part.

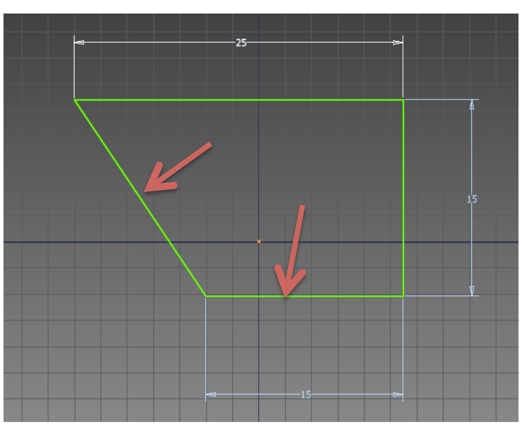
In the new sketch draw this shape (Don't worry about the exact shape we'll dimension it later).



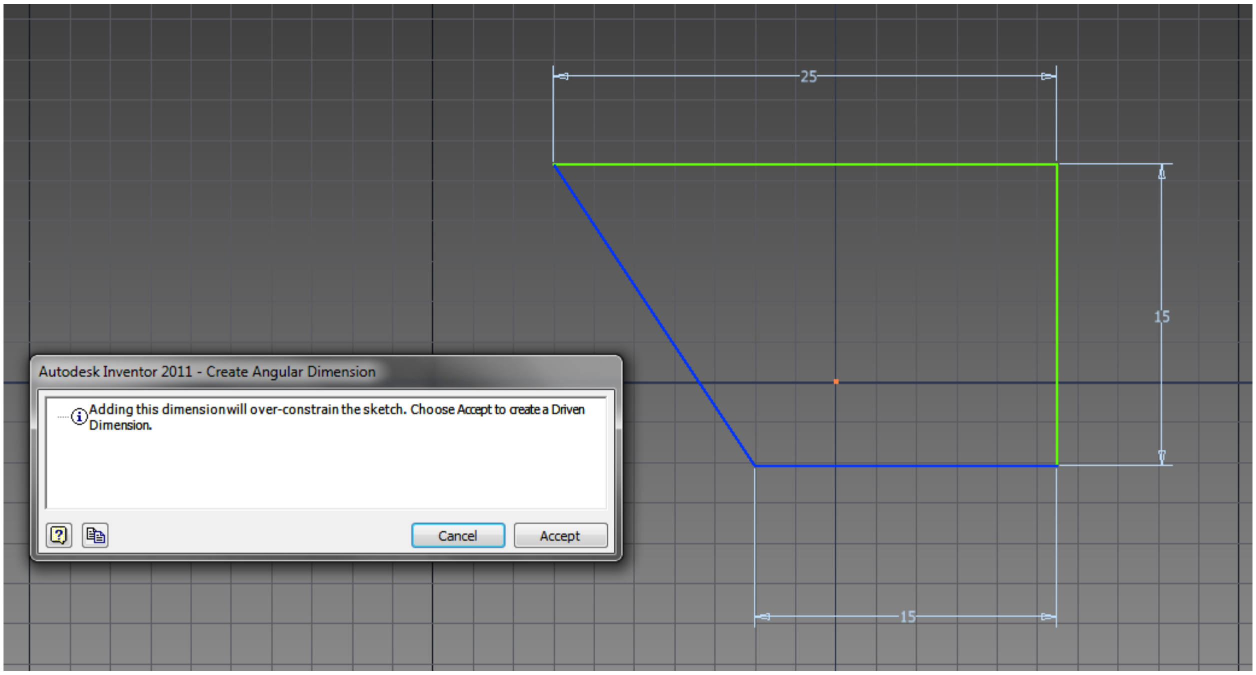
Now add these dimensions:



Now try and dimension the angle between these 2 lines by clicking on them:



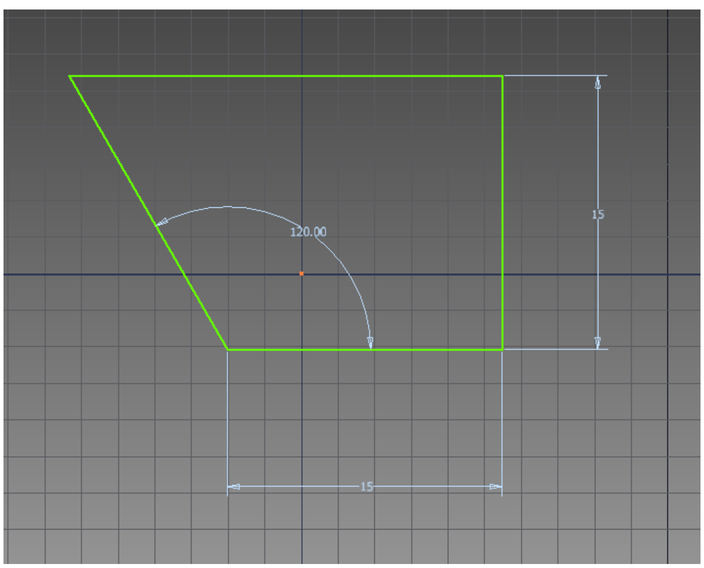
You should get this error:



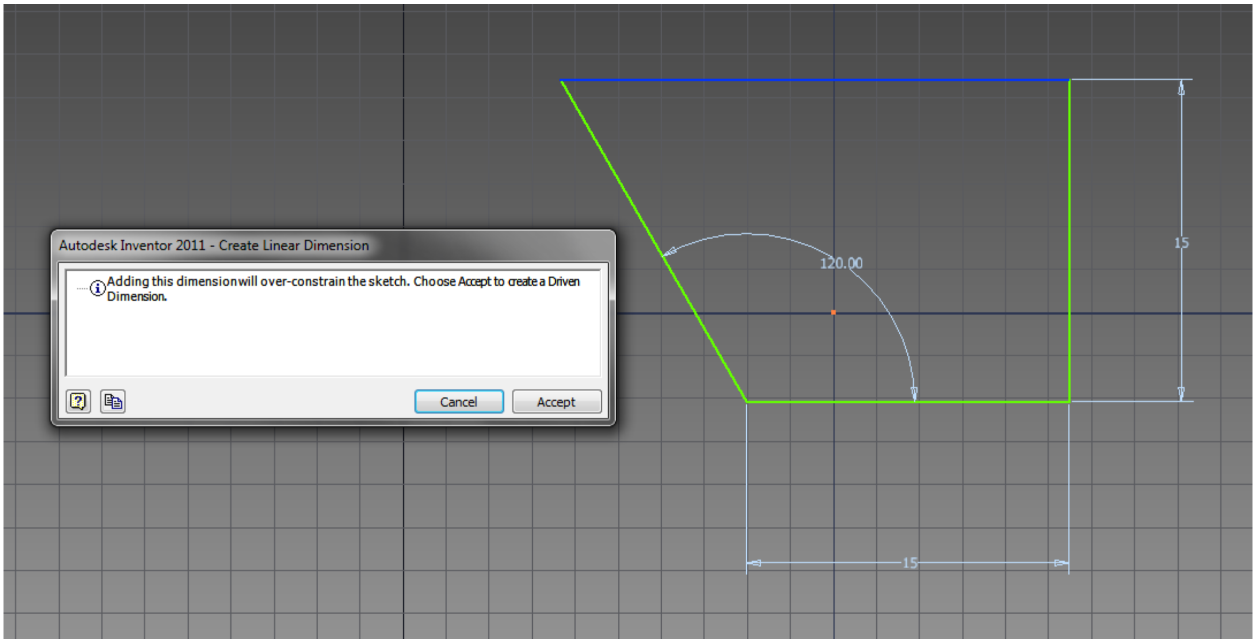
Because the other side lengths are dimensioned the angle must stay at its current value or else the geometry is impossible. A driven dimension will show up but it cannot be changed it just displays the dimension.

Click cancel and delete the 25 mm dimension of the top line (press esc to make sure you're not using the dimension tool still if you are having problems doing this, esc is the inventor cancel and resets your tool to be a plain cursor).

Now dimension the angle and make it 120 degrees:

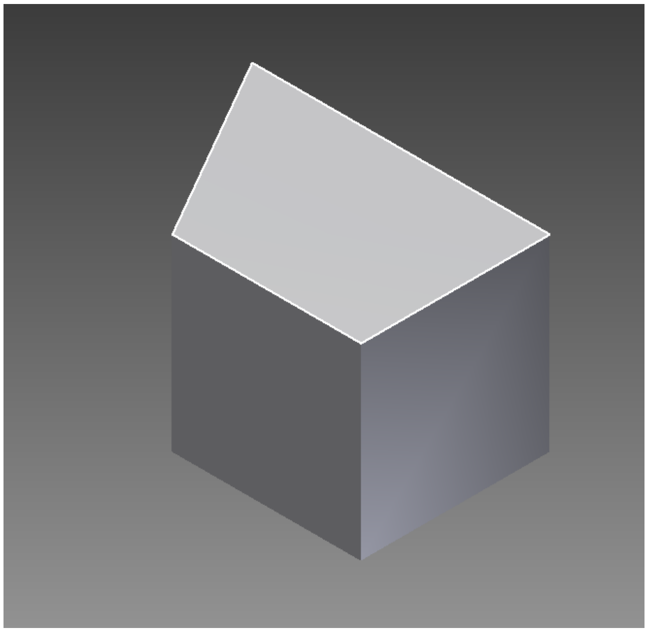


Now try and dimension the top line again, you should get this error again:

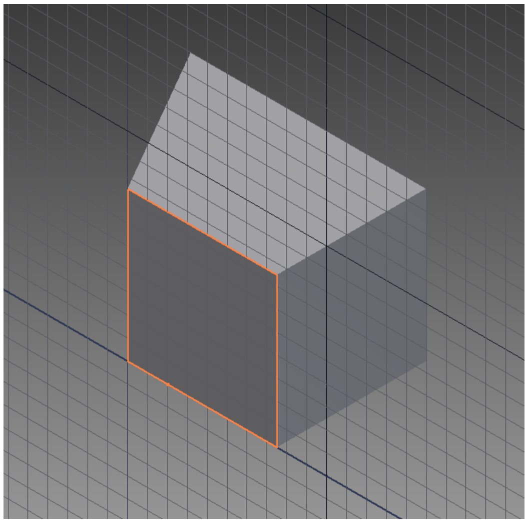


Hopefully this helps make sense of what over constraining the sketch means.

Press cancel and finish the sketch. Extrude your part 15 mm.

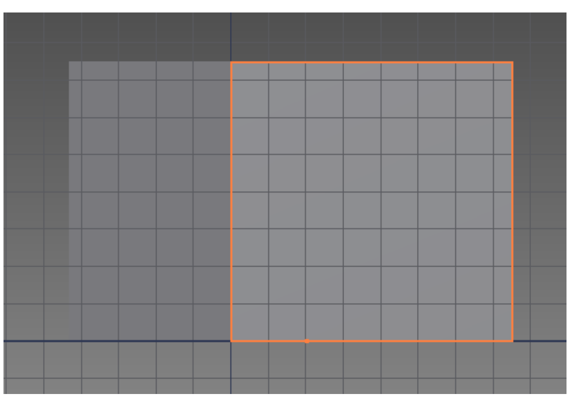


Start a new sketch on the front face:

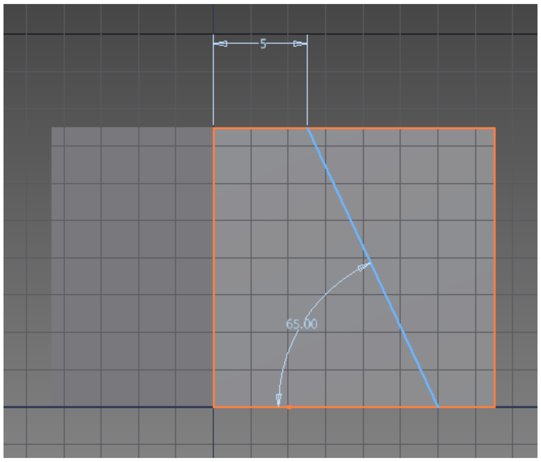


Click on the view tool bar and select view face, click on the sketch face.

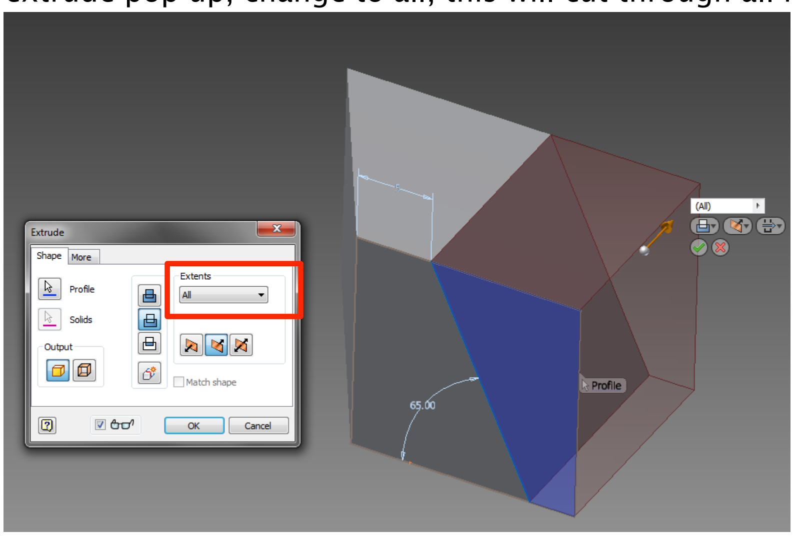
Your screen should now look like this:



Draw and dimension this sketch (it's only 1 line):



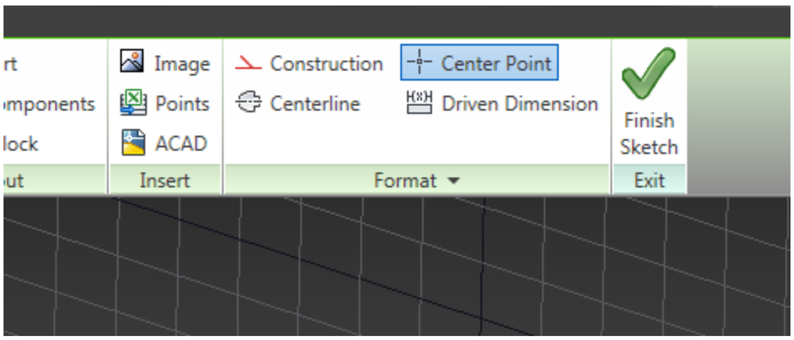
Finish the sketch and extrude cut the upper left profile away from your part. Instead of distance in the extrude pop up, change to all, this will cut through all material.



Result:

If you ever have difficulty selecting your profile for extrusion it is probably because your profile is not closed, one of the lines does not actually contact or connect with one of the others, think of how the fill tool in paint works.

Also, when you are creating more complicated sketches you may need to use "construction geometry" these are lines that you draw to help position and dimension other elements of your sketch. When you try and extrude though Inventor doesn't know the difference and it will make it hard to select the profile you want. Thankfully in the sketch mode you can select lines and make them into construction geometry so that Inventor knows the difference.



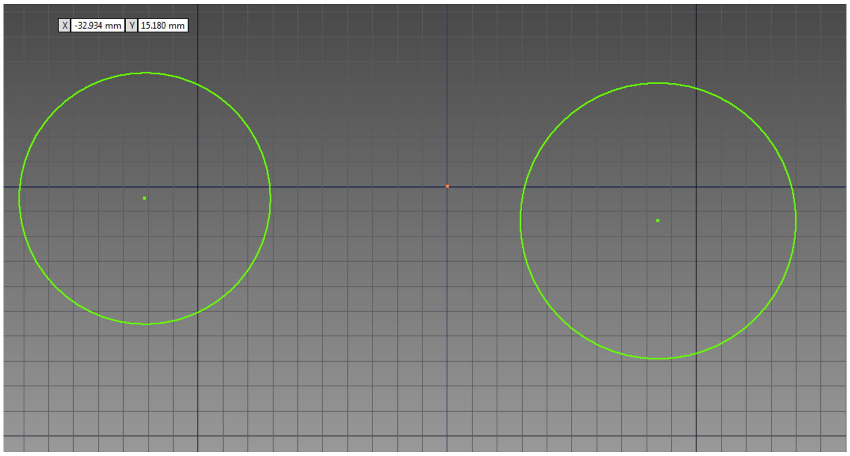
Save your part as lesson2.ipt.

Congratulations you've finished lesson 2.

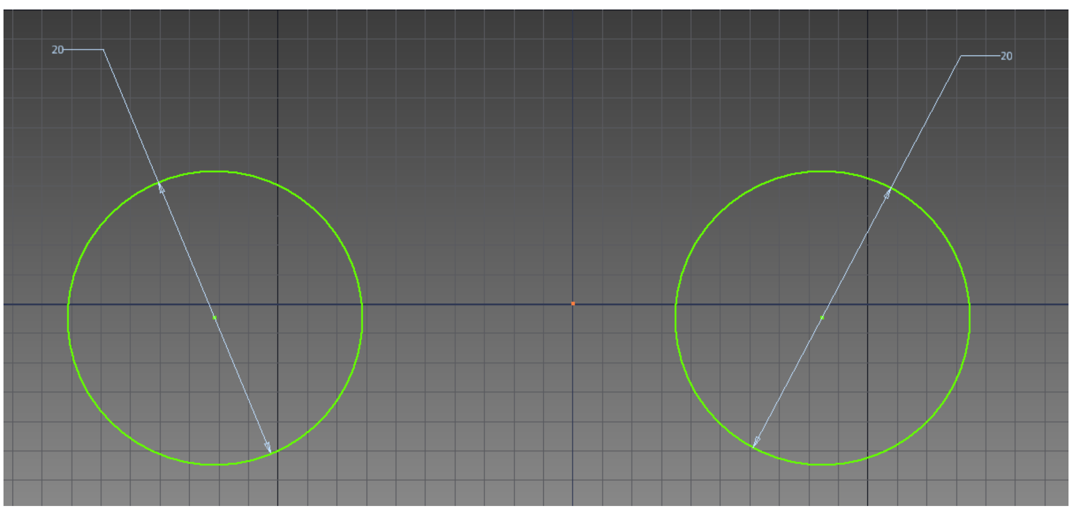
Lesson 3:

Open Inventor and start a new metric part.

Draw 2 circles a fair distance apart:

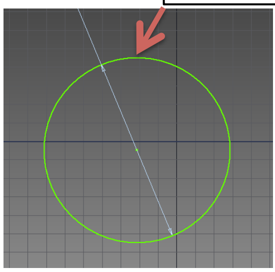


Dimension both of these circles to be 20 mm in diameter and then apply a horizontal constraint between the 2 circle centers:

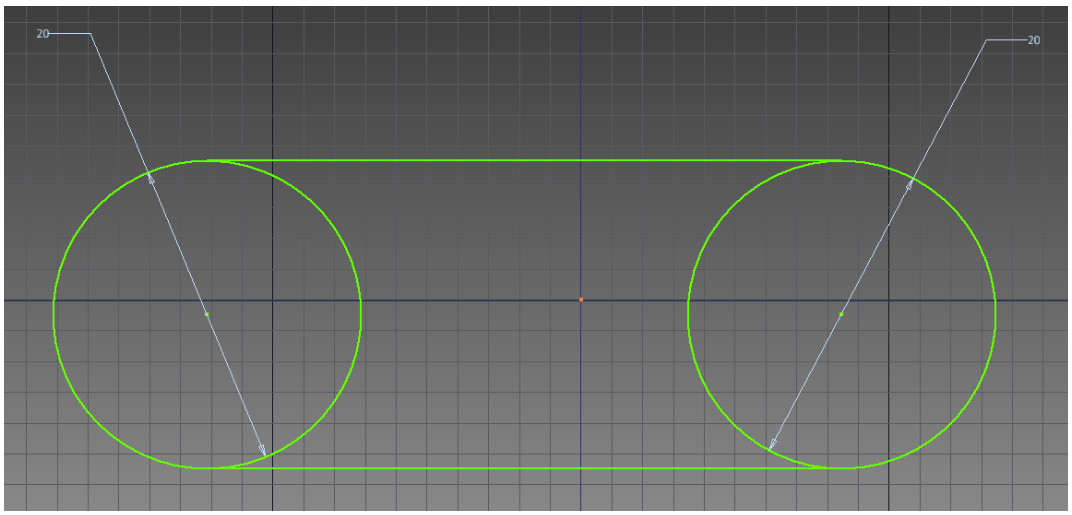


Select the line tool and hover on the top of the circle right above the center. Observe the tangent symbol that pops up and the dotted line that appears on its way to the other circle. Click and draw the line tangent from one circle to the other. Do the same for the bottom.

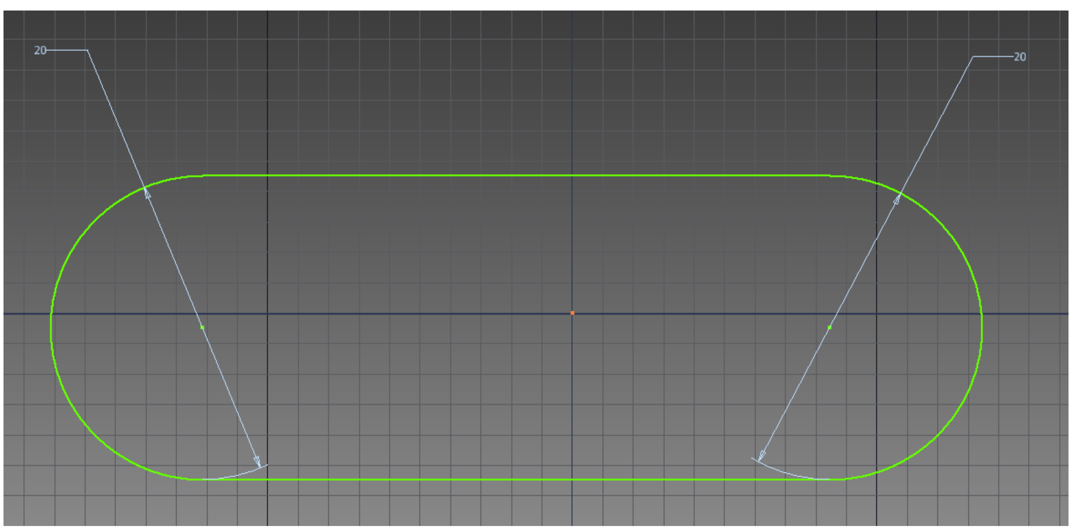
Hover and then click here



The result should look like this:

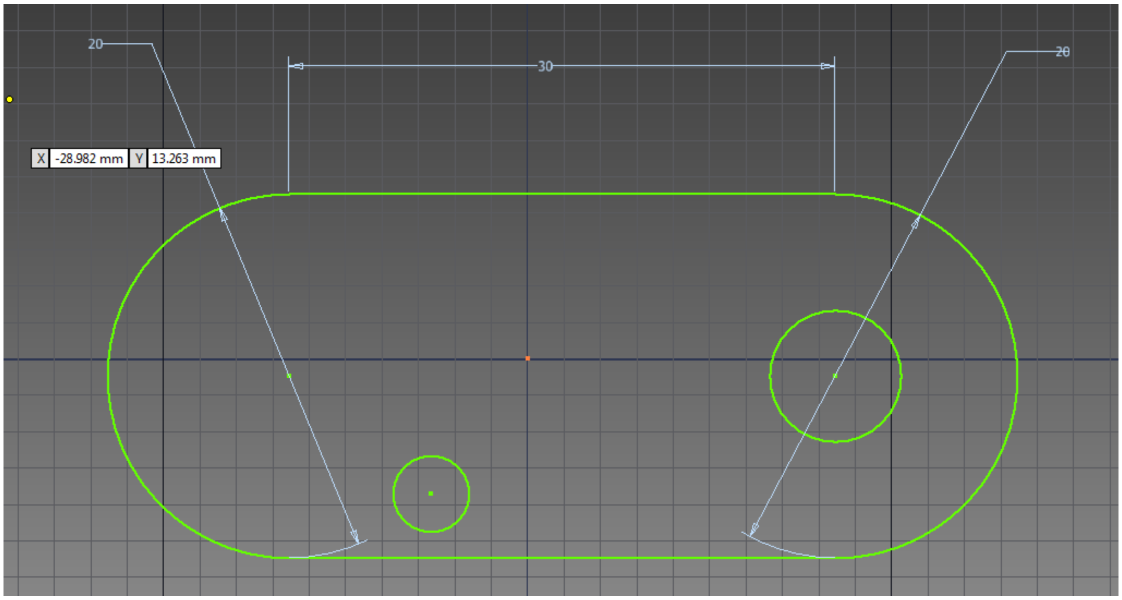


Now use the trim tool to get rid of the inner circles:

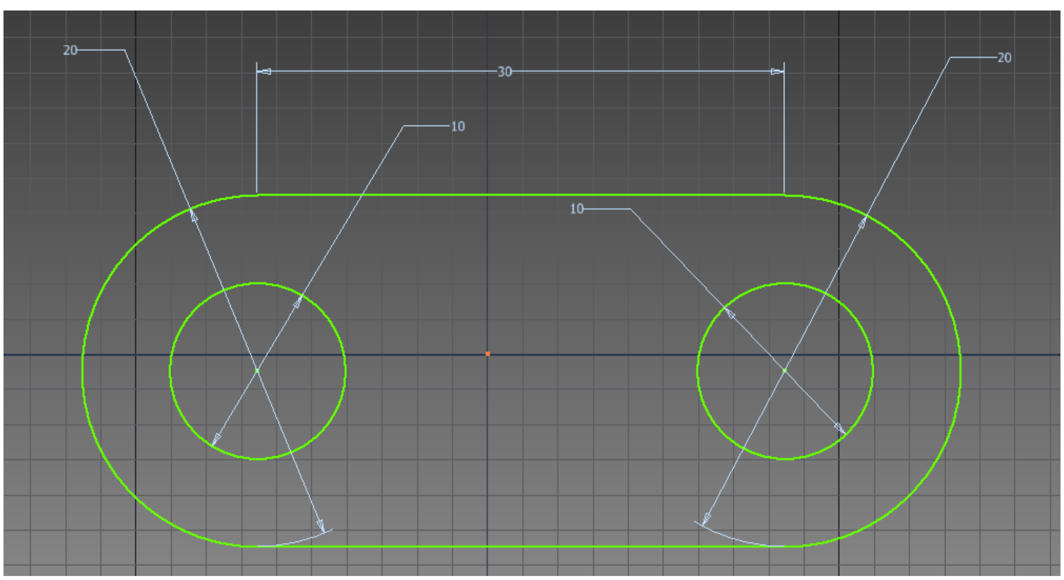


Add a dimension to the upper line of 30 mm.

We are going to draw 2 new circles, the first one just draw anywhere, for the second one snap it to the center of the arc of one of the end circles:

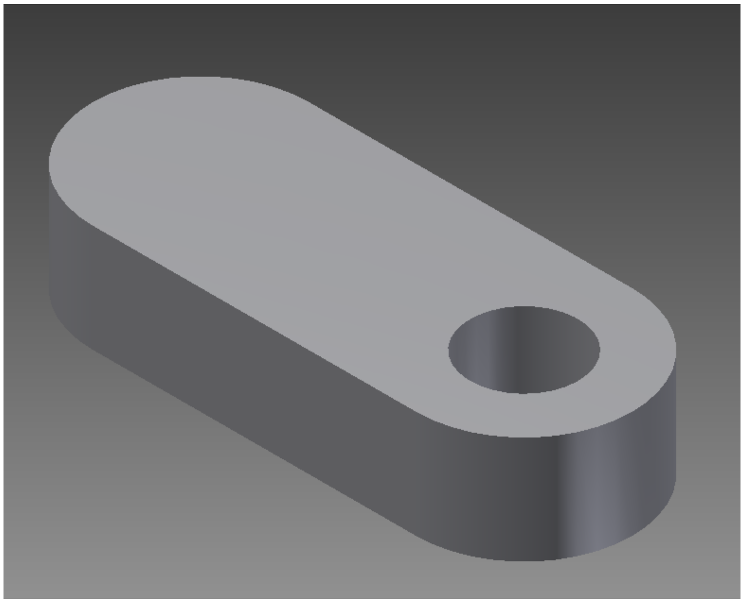


Now select the coincident constraint and constrain the first circle to the center of the other side's circle. Dimension both circles to 10 mm. This was done to demonstrate the difference between letting Inventor generate constraints as you go vs. adding them yourself later.



Select the circle on the right and change it to construction geometry. Finish the sketch.

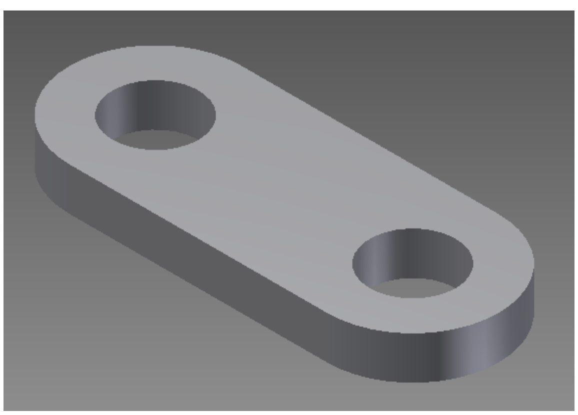
Open the extrude tool and observe that only the one circle cut out is part of the geometry. Extrude the part 10 mm.



Open the sketch again and return the other circle to non-construction geometry (click on the circle, then click on construction again). Extrude the part again.

Wait the part is the same!

Right click on the extrude feature in the model tree and click edit feature. Select the profile button and then while holding ctrl click on the second circle, this will deselect it from the profile. While you're in here change the height of the extrusion to 5 mm.



Save your part as lesson3.ipt.

On to lesson 4!

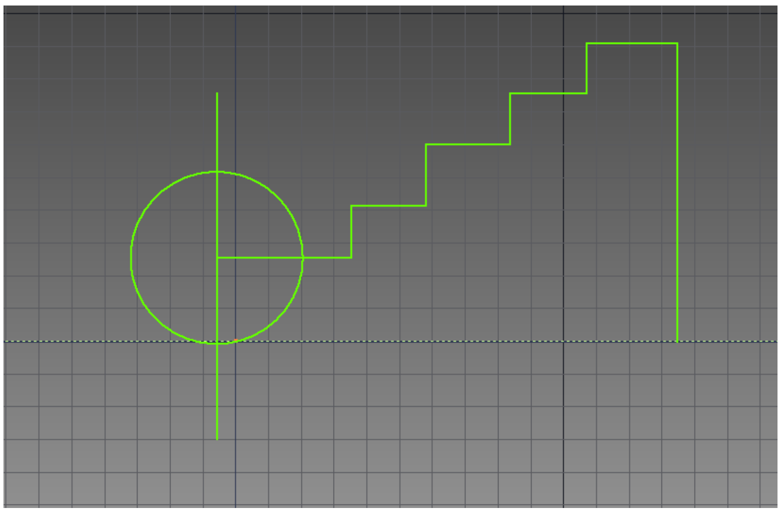
Lesson 4:

Open Inventor and start a new metric part.

In this lesson we are going to learn the second most common way to make a 3D part from a sketch – a revolution. To demonstrate this we're going to make a model of the Stanley Cup!

Start a new sketch and draw a long horizontal line, immediately make it construction geometry.

Then draw something like this:

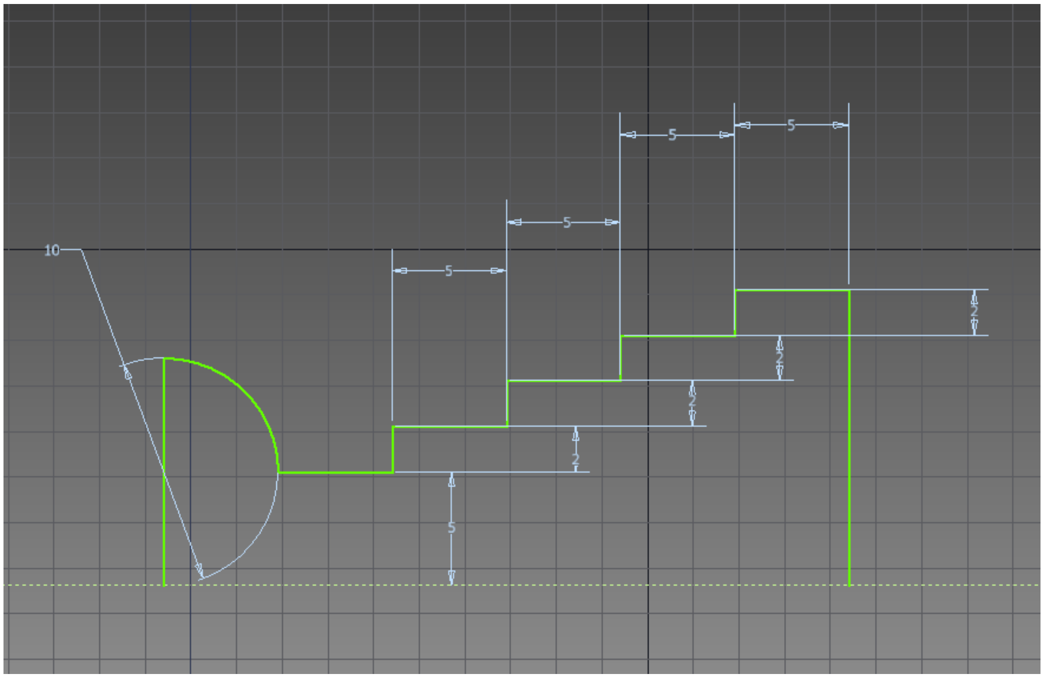


Note: Dotted line is construction geometry.

Notice how if you move your mouse to a point of interest (like the circle center) and then up it tracks your movement, this is to help you create a line from above that will pass through the center of the circle. Inventor will help you like this in lots of ways, you just have to get used to it.

Now let's do some clean up with trim and add some dimensions (you might need to delete the circle at the end, add the dimensions, and then redraw the circle):

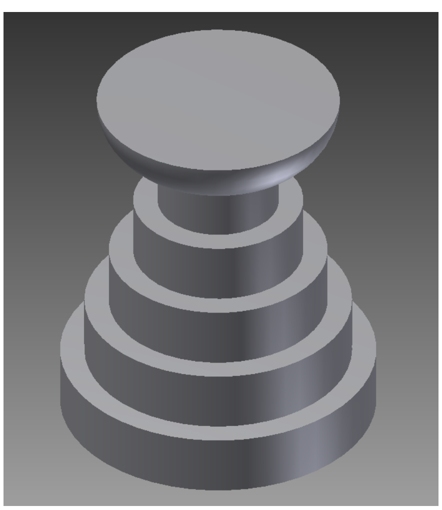
Also draw a line to close the bottom of the profile.



Finish your sketch.

Now instead of clicking on extrude this time pick revolve.

Pick the profile for the profile and then pick the original long construction line for your axis, this should be your result:



Doesn't look much like the Stanley Cup, but that's not important. Revolve is a useful tool especially for making shafts!

Save your part as lesson4.ipt

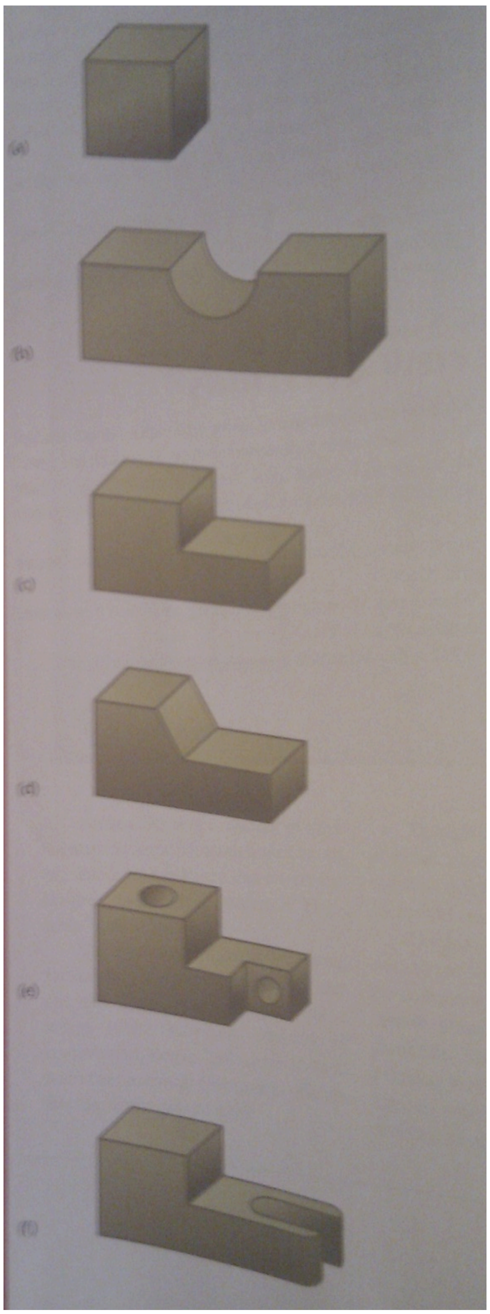
Lesson 5:

The best way to learn how to CAD is lots and lots of practice! You'll develop your own style and learn most by modeling lots of things!

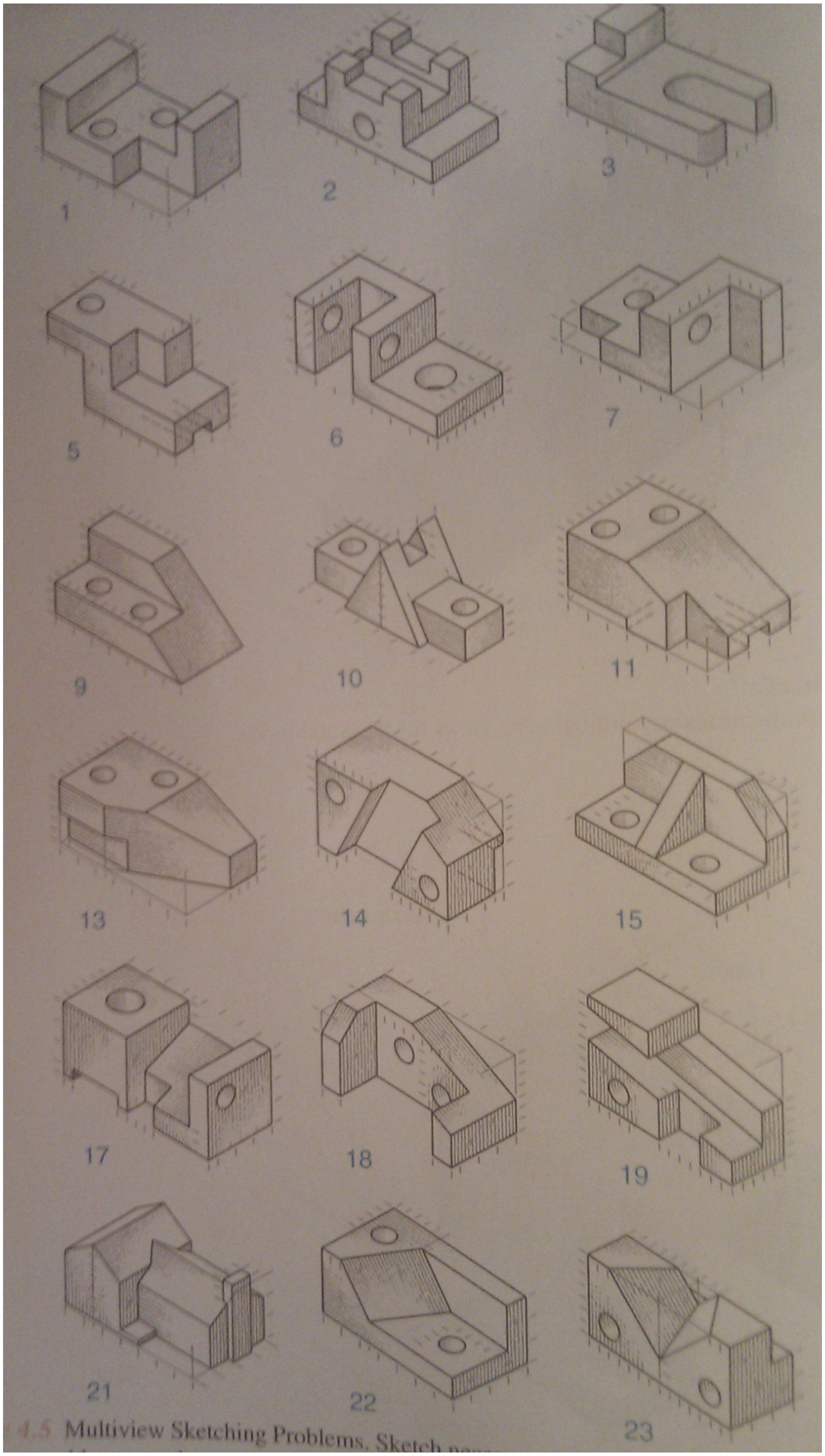
For lesson 5 you need to create solid models of the following parts, save them by the name indicated underneath the picture. You need to submit all these drawings to me to get your CAD certificate!

Practise by drawing all these simple shapes:

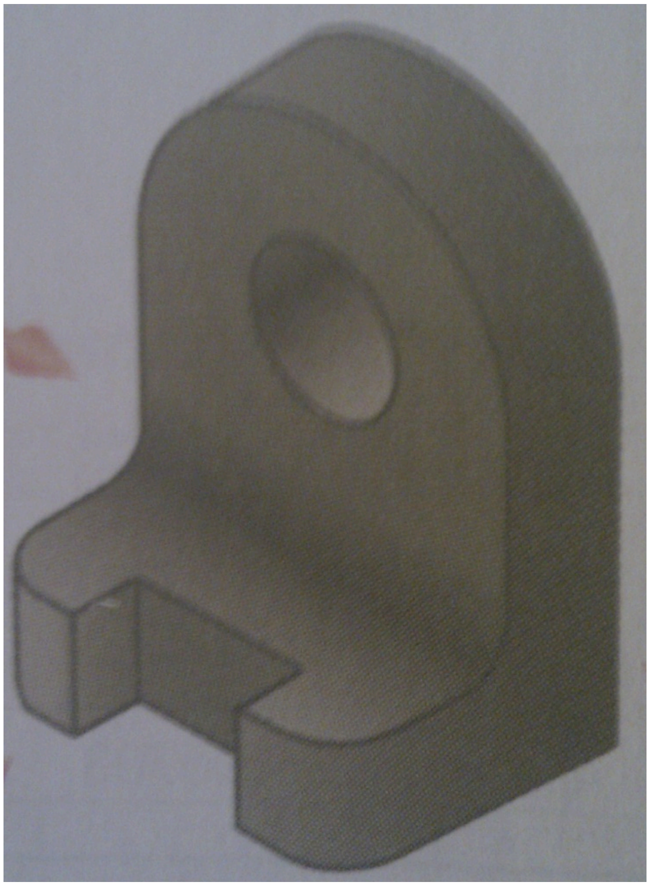
Just pick dimensions that make your model look close to the picture it doesn't have to be exact. Save the files using the naming convention simpleA.ipt through simpleF.ipt



Model 6 Parts from this page of your choice, again just pick dimensions that make it look similar not exact:

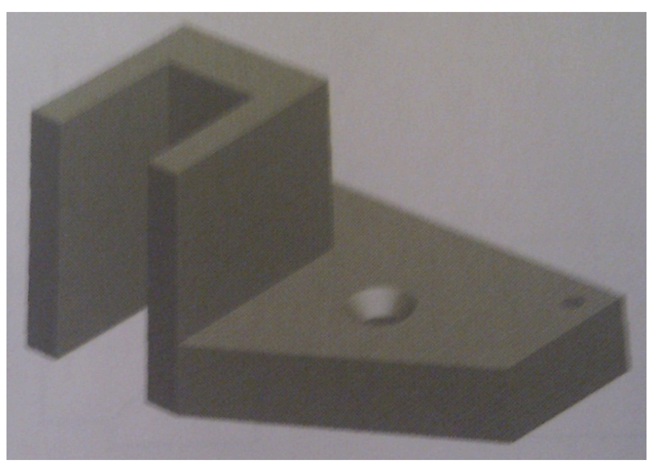


Model these 3 Blocks, again picking dimensions that make the part look right. Name files according to the name that appears under the image.



Block-A

Block-B



Block-C

Final Project:

Nice work on getting this far, you've almost earned your 2708 CAD certificate and I bet you've learned lots and can't wait to start CADing robots!

At this point you can submit all of your other files to the mentors for marking. To do this put all the files in a folder and create a zip folder by right clicking on that folder and clicking send to and then compression folder. Email this zip file to lake.effect.robotics@gmail.com

The last part of the CAD training is a mini project. You need to pick a real world object that you own or have easy access to that has more than 5 different parts to it (be creative and have fun with it!). You can also work in groups but your project must be more complicated then (number of group members x 5 = minimum number of parts) Once you have picked an object show it to a mentor or take a picture and send it to lake.effect.robotics@gmail.com to get it approved.

Once your project has been approved it's time to get to work. You need to measure and create accurate Inventor parts for each part of your object.

Remember what I said earlier, there are going to be many different ways to do something; no particular approach is the best so pick the one that is simplest and that you are most comfortable with. Remember 95% of parts are modelled using a simple sketch-extrude-sketch-cut and so on process. Also, if you can think about how the part would be machined from a block of material it might help you think about how to CAD it which will help when you are CADing parts we actually have to machine!

When you have built all the parts, create a new assembly in Inventor and add your parts to it. Try and see if you can figure out how to assemble your object using assembly constraints. A big part of CAD and of being an engineer is about figuring out how to use new tools using the internet, help documentation and your own brain. For this reason I have left out the instructions on how to make an assembly!

If you get stuck you can email lake.effect.robotics@gmail.com or talk to any of the mentors on Thursdays.

Good Luck!

What's Next?

We are planning on putting together another tutorial about how to take your 3D models and make 2D shop drawings with Inventor which we can take to machine shop and make the part!