Autodesk Inventor

Software Instructions

Version #	1		
Created by	Logan	**Date**	13/10/20
Reviewed by	James T	**Date**	13/10/20

Software Instructions: Autodesk Inventor

Acknowledgments

We would like to acknowledge the following references used to compile these instructions for students:

- https://www.youtube.com/watch?v=msXjD96ayIY
- https://www.youtube.com/watch?v=iCnVZrzz1VI
- https://www.youtube.com/watch?v=7KjkCIlJmvA
- https://www.youtube.com/watch?v=LBKu0HF-hOM
- https://www.youtube.com/watch?v=teyRroZcX7E
- https://www.youtube.com/watch?v=89PX745HTfU
- https://knowledge.autodesk.com/support/inventor/learn-explore/caas/CloudHelp/cloudhelp/2019/ENU/Inventor-Help/files/GUID-08D6DD13-3066-4210-9F19-5F7E8DB8AD5D-htm.html

Software Instructions for Autodesk Inventor

Creating your project:

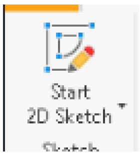
Inventor has 4 main file types you can create: Assembly, Drawing, Part, and Presentation. The main things to focus on for creating a new project will be the Part and the Assembly type. A part file is rather self explanatory. It holds a single component in your project. If you were designing a keyboard you'd have a part file for each key, for example. The assembly file is what combines all of the part files to create to final product.

When creating new parts or assemblies, before doing anything it's a good idea to File->Save As and name your file. It's also highly recommended to create a project file to store all of your part files and assembly files for easy reference later.

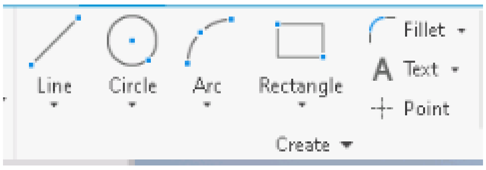
Sketches:

The first thing you'll do to create a part is to start a sketch. This is a 2D representation of some aspect of your part. A single part file can have as many sketches as you want, and it's common practice to keep each sketch as simple as possible so as not to create confusion and clutter for you and the software.

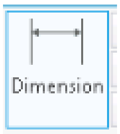
To begin sketching, click the "Start 2D Sketch" button and select a plane to work on. This will take you into 2D mode, allowing you to begin your 2D sketch of the part.



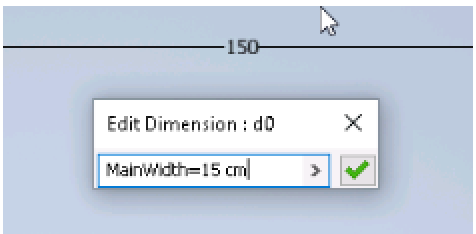
In the "Create" toolbar there is an assortment of tools you can use to begin your sketch, including Line, Circle, and Rectangle. When drawing your sketch. Avoid linking the sketch to the centre point of the plane, as this creates a constraint which can make things difficult in the future.

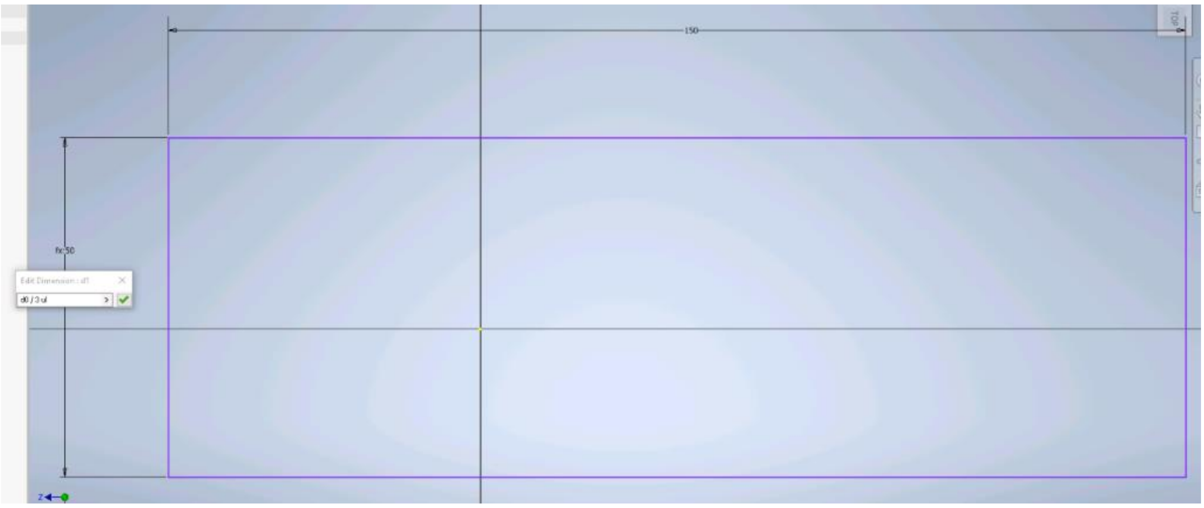


Once you've drawn a general shape for your sketch, you can use dimensions to create a more precise result.

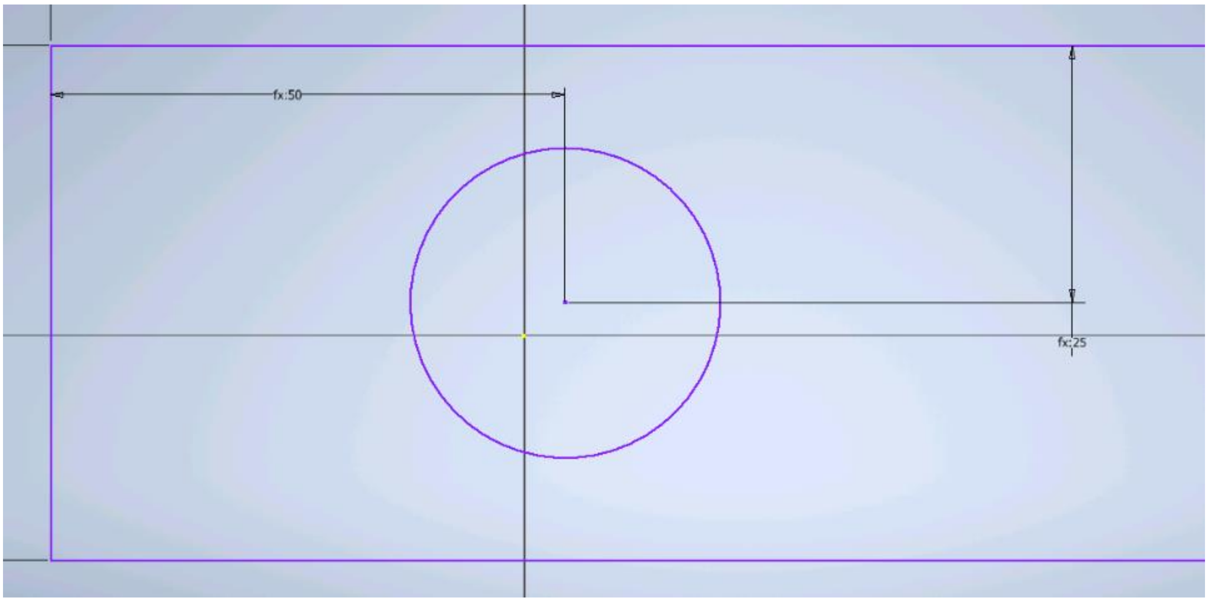


Using the dimension tool, you can click on an edge you'd like to specify a length for and describe that length. You can also name these dimensions for use later. When setting your dimensions, you can also set their values as functions of other dimensions such that altering the original will also change the other dimensions.

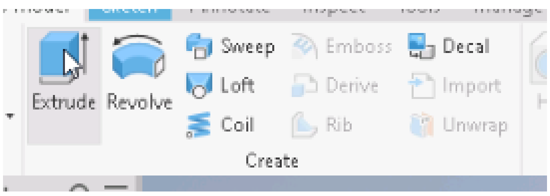




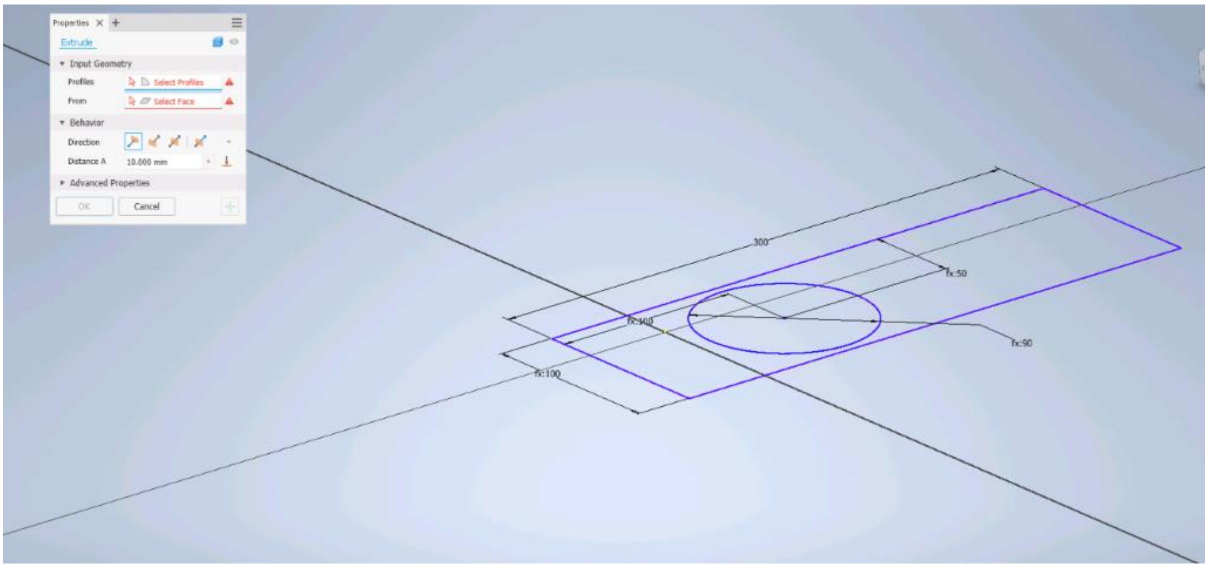
These dimensions can also be used between lines and shapes within a sketch to set distances or angles between them.



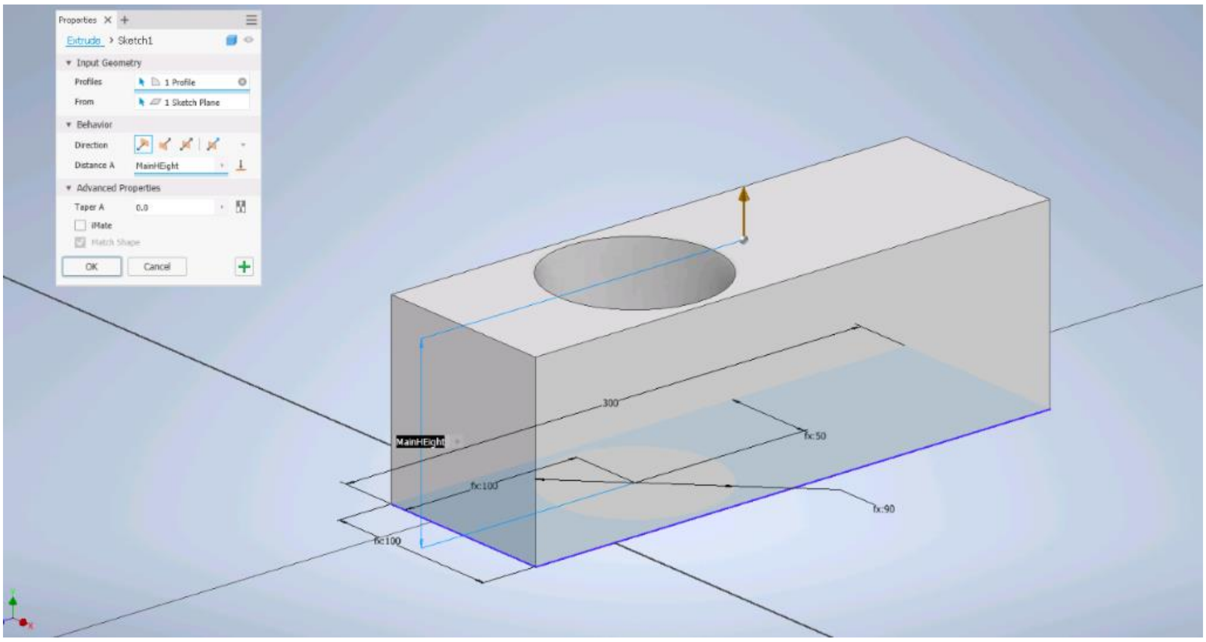
Once you have a sketch you're happy with, return to the 3D model tab. The Create toolbar has an assortment of options for bringing your sketch into 3D.



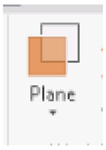
The extrude tool is the simplest and most common tool to use. It will present you with options such as which faces to extrude and which direction to extrude in.



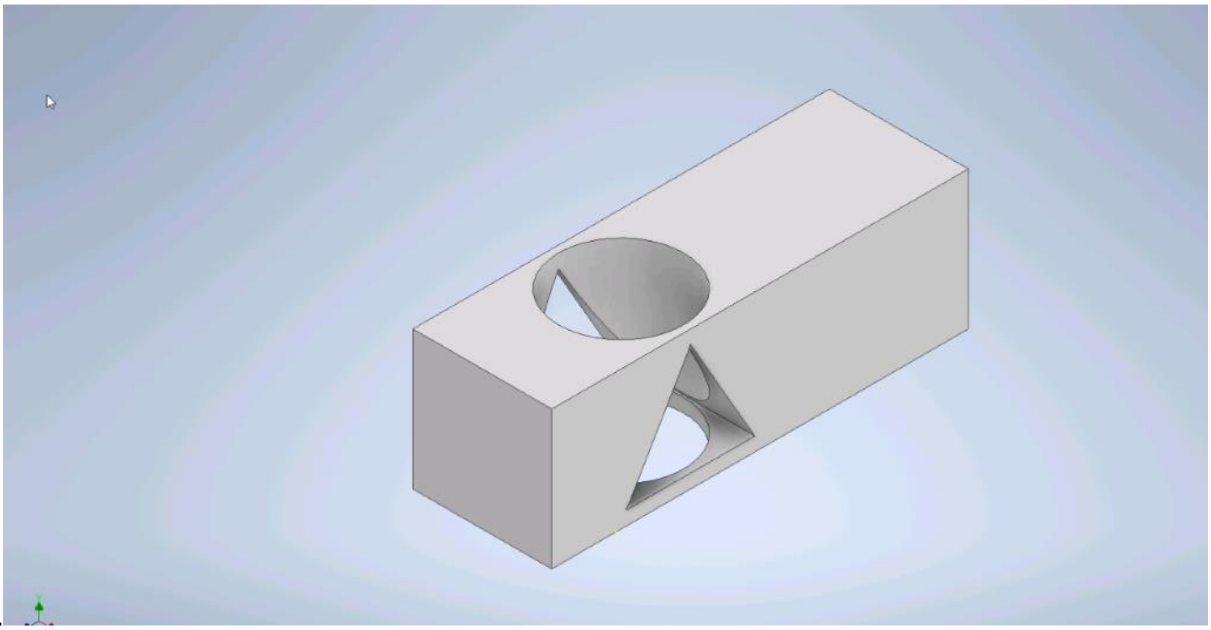
This will essentially extend your 2D sketch into a prism. This has similar options to the dimension tool as far as how far to extrude the object..



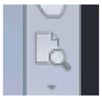
Once you click OK and finalize the extrusion, you can work on creating another sketch to further add to your part. Clicking on the sketch tool you'll be able to draw a new sketch on a face of your object. You can also create a workplane which can be placed anywhere in the sketch relative to your object.



This new sketch can reference dimensions from other sketches, which is why naming your dimensions can be very useful. Once you've finished your new sketch, you can then use the extrude tool to extend that sketch out of the object, use it to cut into the object, or use it to keep only where your sketch overlaps the object.



The Zoom tool is very useful. Clicking this zoom tool on the right will frame the whole object in the screen.

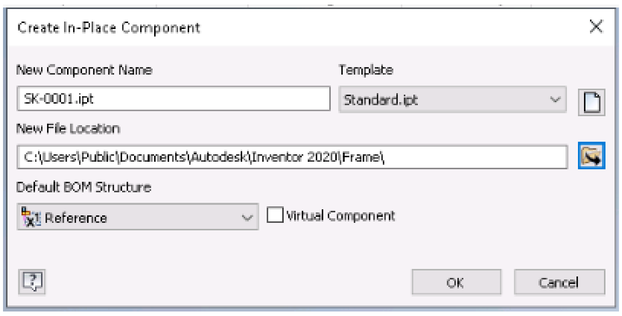


Creating a structure:

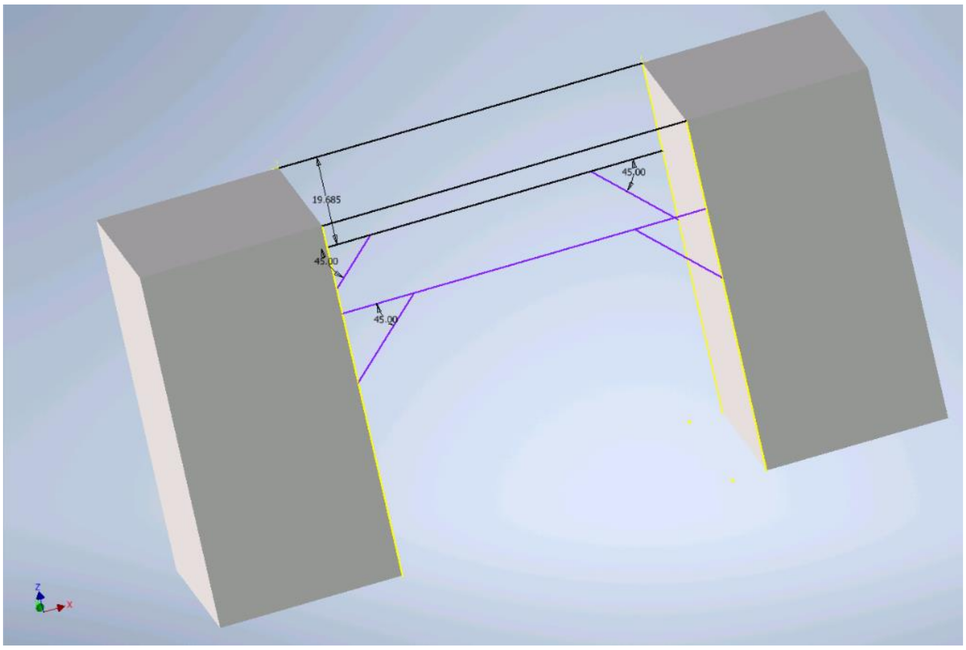
Now to begin working on creating a steel frame.

Step 1:

The first step is to design the shape of the frame itself. Open a new assembly and create a new component. Make sure the new component is set as a reference.

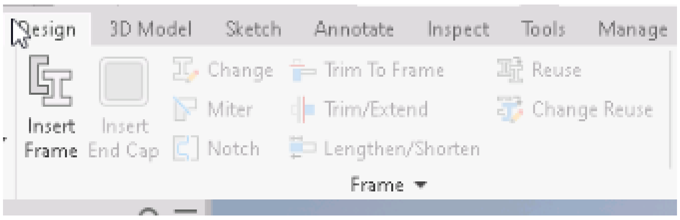


Create a new sketch to your desired dimensions as previously described. Extrude it out to create your 3D skeleton. Each edge on this skeleton will create a member in the steel frame. Feel free to add more sketches along the faces to create crossbars. This will be our example skeleton.

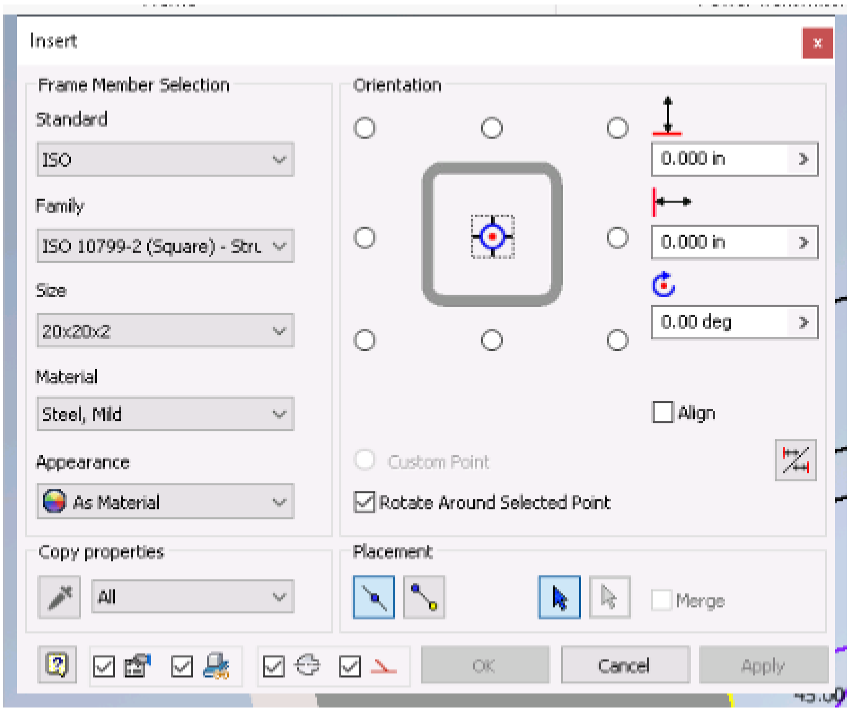


Step 2:

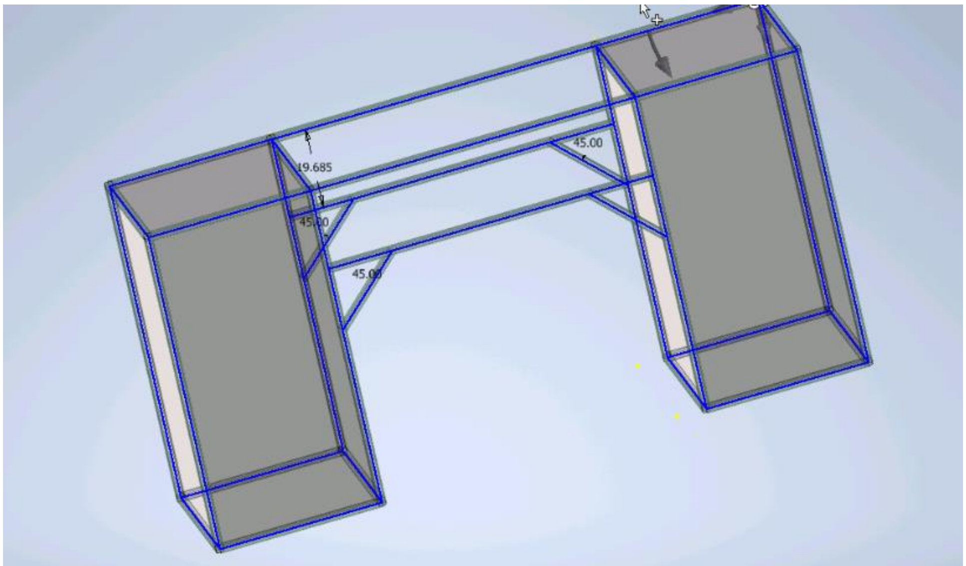
Now that we have a skeleton for our frame, it’s time to actually add the frame. Under the Design tab, you’ll find the Frame toolbar.



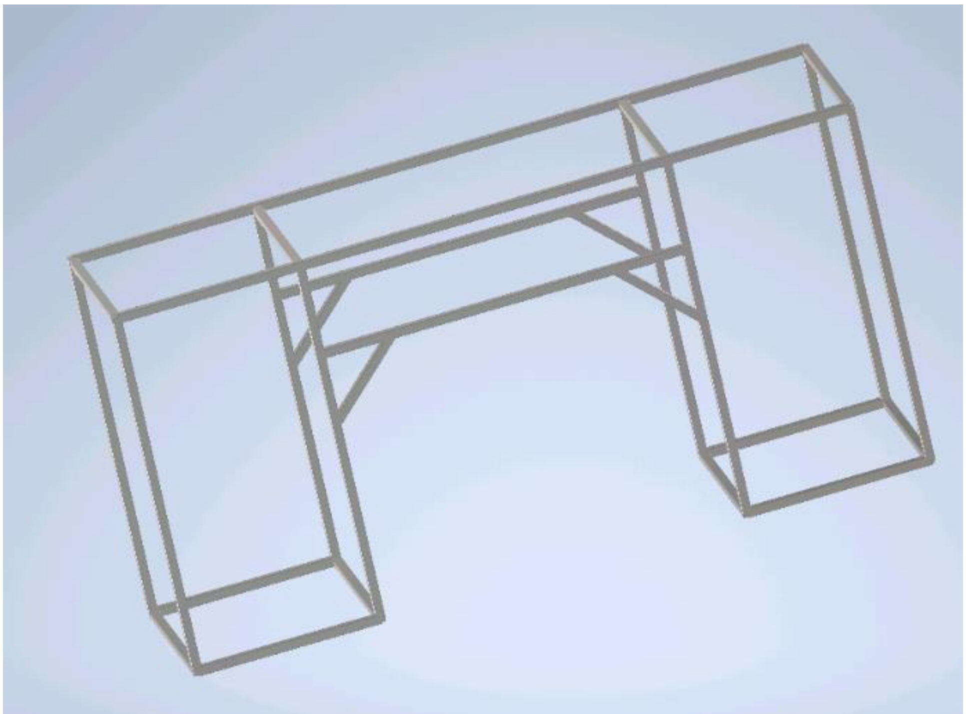
Clicking the Insert Frame button will open a menu which allows you to choose all of the aspects of your frame, from the standard used, to the size and shape of the member cross sections, to the material, to the angle and offset from the edge.



Fill this menu out with all of the relevent details you require, and once you’re ready, you can start selecting all of the edges you want to become members.

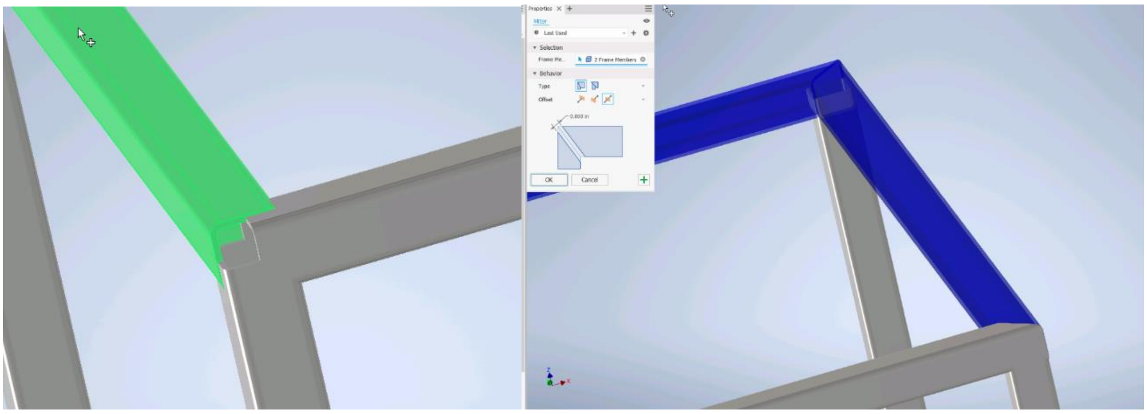


Once everything is selected, click OK and name everything in the new dialogue box to what you'd like your frame and skeleton to be called. After clicking OK again you’ll be presented with a list of file names. These are the names of each component file for the frame members. Feel free to name these files, but it isn’t recommended. After clicking OK one more time, the frame will begin to generate.



Step 3:

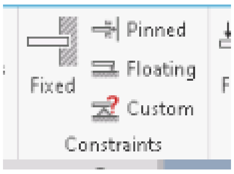
Once you have your frame generated, the corners will appear quite messy. Back in the frames tool bar there is an assortment of end treatment options. On corners, you can use the Miter option, the trim/extend option will allow you to select members, and then select a reference face, and the members will be trimmed to that face.



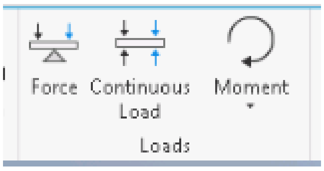
Frame Analysis:

Inventor provides the option to perform stress analysis on your components and frames. To perform a stress analysis on your frame, select the Frame analysis button.

After clicking Create Simulation, you will be presented with a selection of options and tools. The first set of tools to use is the constraints tool bar.



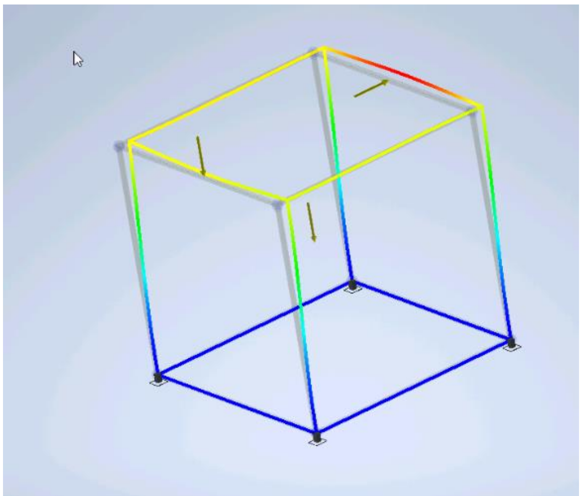
Select one of these options and click on the frame in the places you want them. These will be where the frame is attached to something. After your constraints are set, you can start using loads.



Select one of these and click where on the frame you'd like them to apply. Adjust the magnitude of the load as well as the direction to match whatever you want to simulate.

Once all of the parameters are set, you can click the Simulate button tocalculate the simulation. 

Which will result in something like this

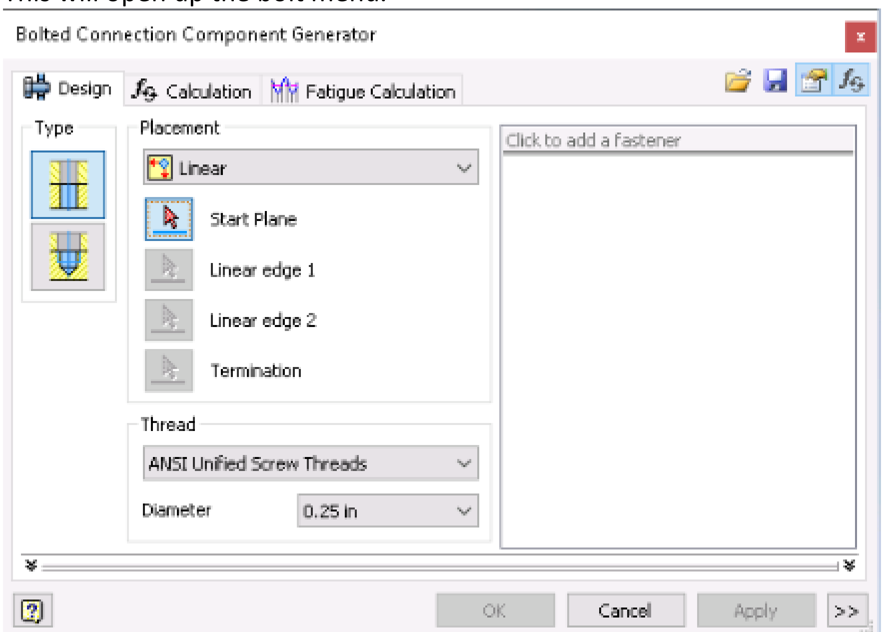


You also have the option to animate the simulation under the Result tool bar and can even record and save the animation.

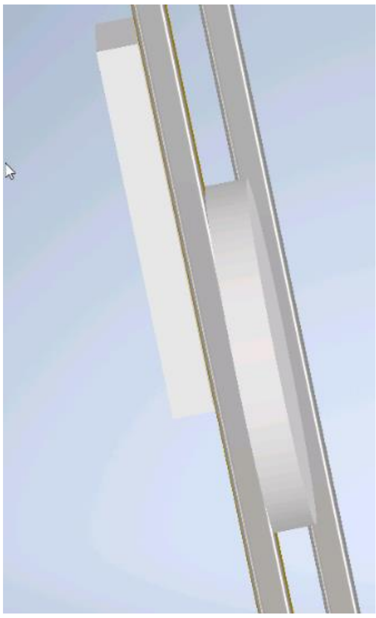
Bolted connections:

Inventor offers the capability of creating bolted connections between components. In the Design tab, under the Fasten tool bar, is the bolted connection button.

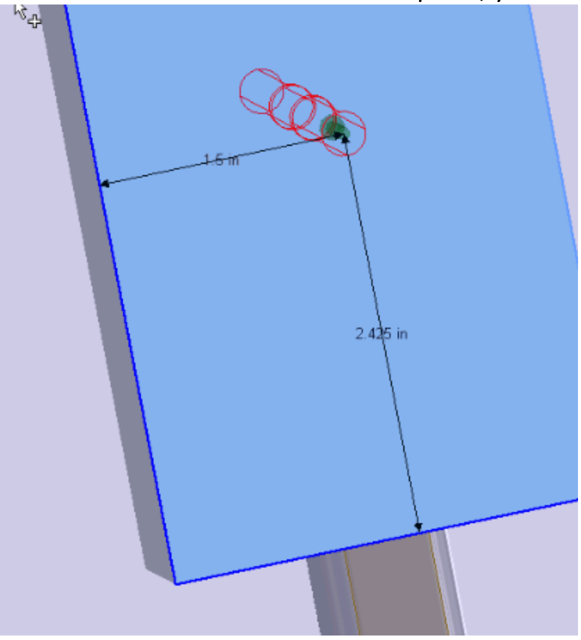
This will open up the bolt menu.



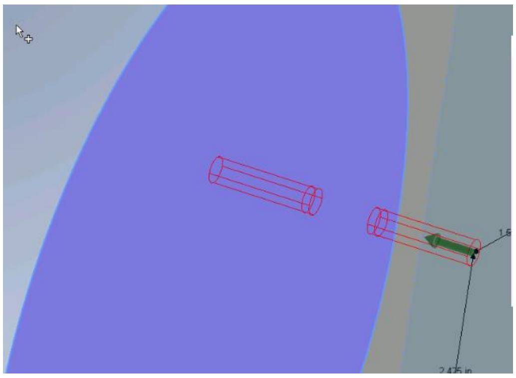
There are multiple ways to select the location of the bolt, in this example we will be looking at the linear option. Here are the components to be fastened.



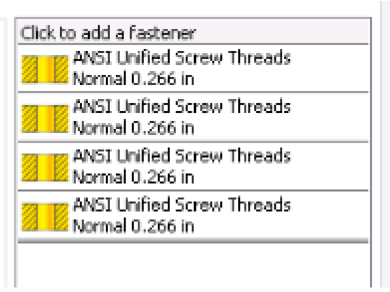
Firstly, select the starting face, this is where the top of your bolt will be. Under the linear option, you then



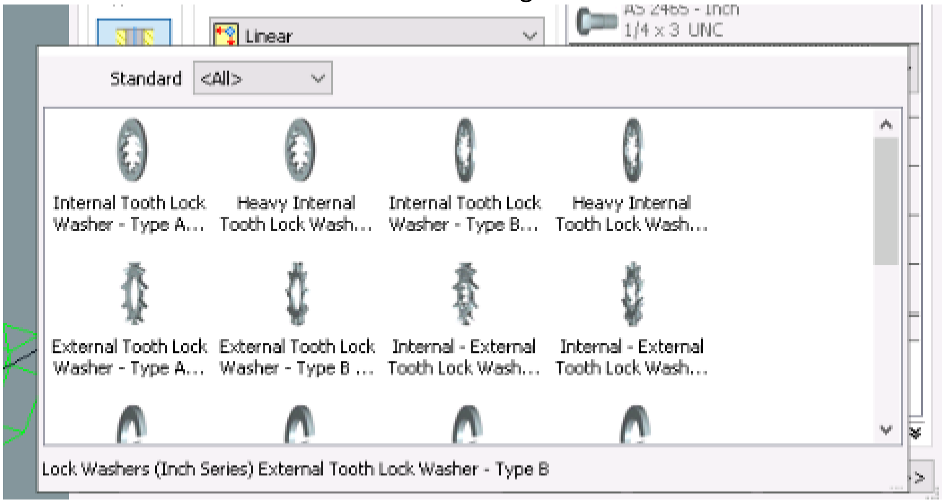
select two edges to reference the distances for the bolt.
Once you've selected the location for the bolt, you choose the face for the bolt to finish on.



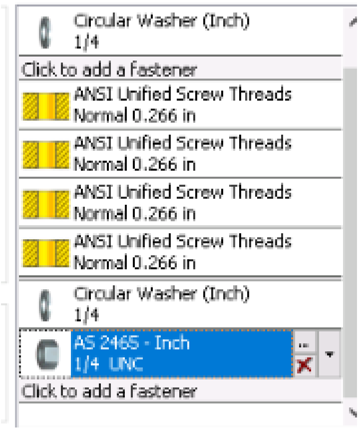
Now that the hole for the bolt is set, you can select the type of bolt ans fasteners to use.



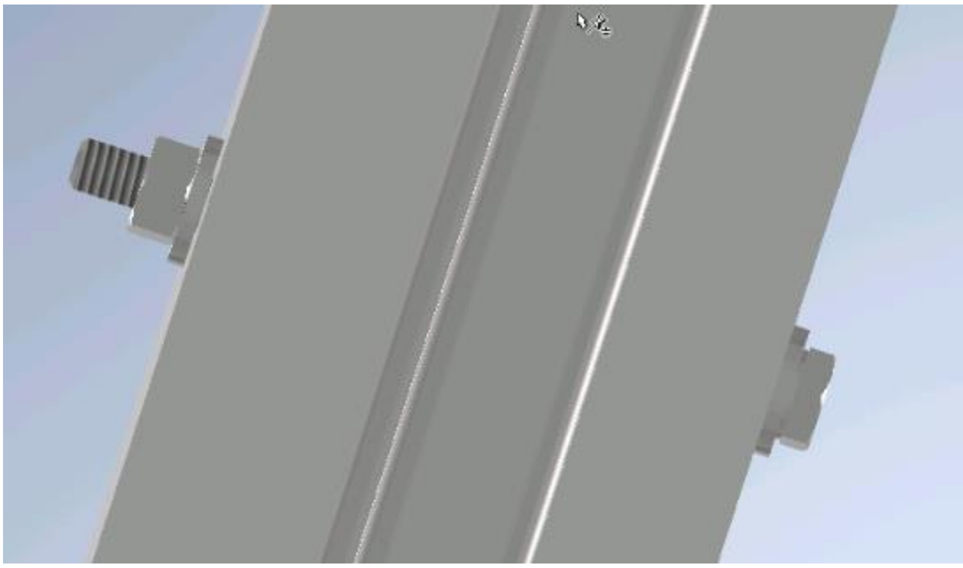
Clicking this will allow you to select the top of the bolt. Once that is chosen you can add another fastener there to choose a washer from the following menu.



The same can be done on the bottom of the bolt to select washers and nuts.



Now that everything has been chosen, select OK and the bolt will be added.



Exporting Files:

In order to export your part and assembly files, go to the file menu and select Export. There is an array of file type optoions to export your files including image, 3D PDF, PDF, CAD Format, Export to DWF, Export to DWG. Within each of these menus are more specific file options for export