

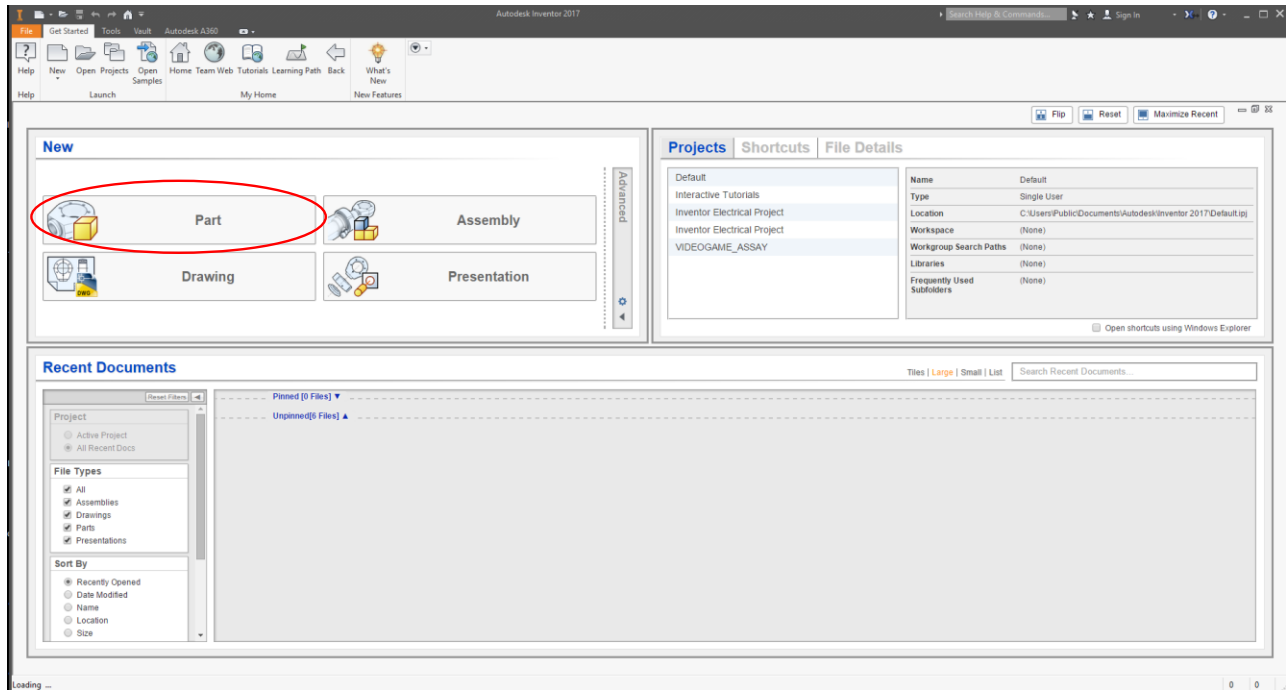
# AUTODESK INVENTOR BASICS

## INVENTOR BASICS

Autodesk Inventor is a CAD (computer-aided design) application for creating 3D digital prototypes used in the design, visualization and simulation of products. It is a parametric and feature based solid modeling tool and it allows you to convert the basic 2D sketch into a solid model using simple modeling options.



## GETTING STARTED



- **PART ( .ipt files )**

When you open a Part file you are in the part environment which means that with the commands you can manipulate sketches, features and bodies that combine to make parts. Most parts start with a 2D sketch. A sketch is the profile of a feature and any geometry required to create the feature.

- **ASSEMBLY ( .iam files )**

In Autodesk inventor you place components that act as a single functional unit into a separate document. Assembly joints and constraints define the position and behavior of these components. When you create or open an assembly file you are in assembly environment. You can insert parts into an assembly or create new part files within the assembly environment.

- DRAWING ( .idw, .dwg files )

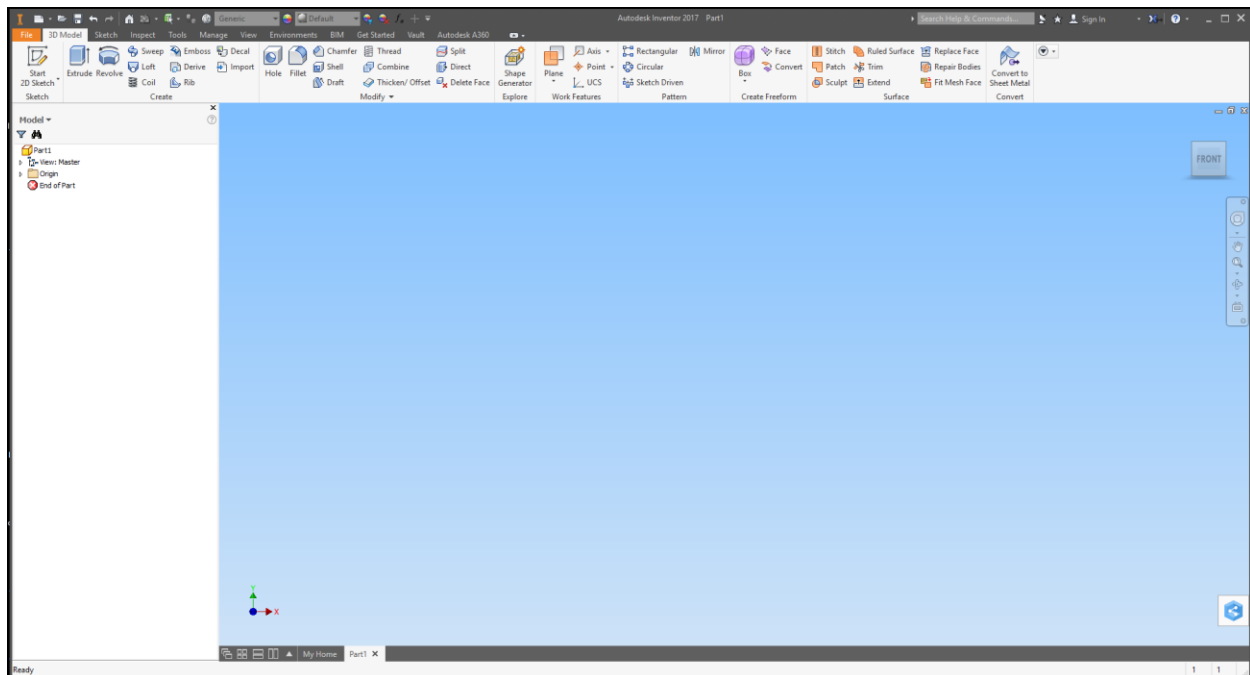
After you create a model, you can create a drawing to document your design. In a drawing, you place views of a model on one or more drawing sheets and you can add dimension and other annotations. A drawing that documents an assembly can contain an automated parts list. By default the drawing updates automatically when you edit the components.

- PRESENTATION ( .ipn files )

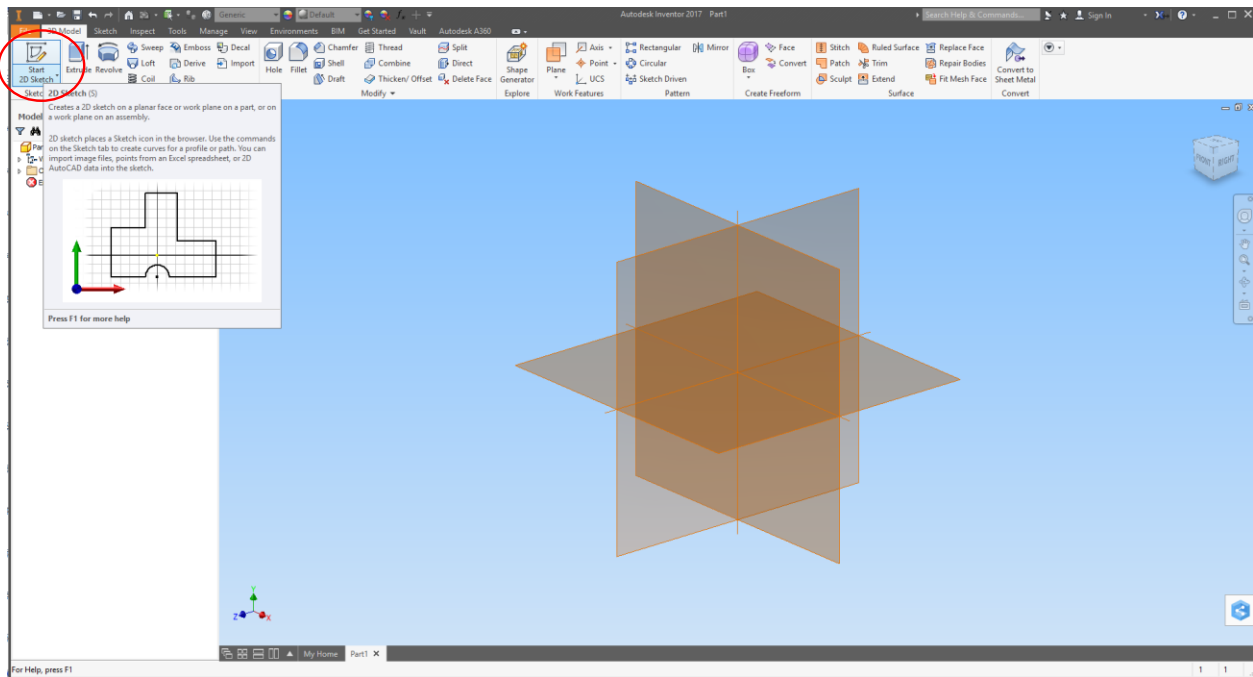
You can use a presentation file to create an exploded view of an assembly to use in a drawing file or create an animation which shows the step by step assembly order. You can save the animation to a .wmv or .avi format.

## GETTING STARTED: CREATE A PART FILE

Selecting part in Inventor drawing window will open a part environment in which you can get started with your model.

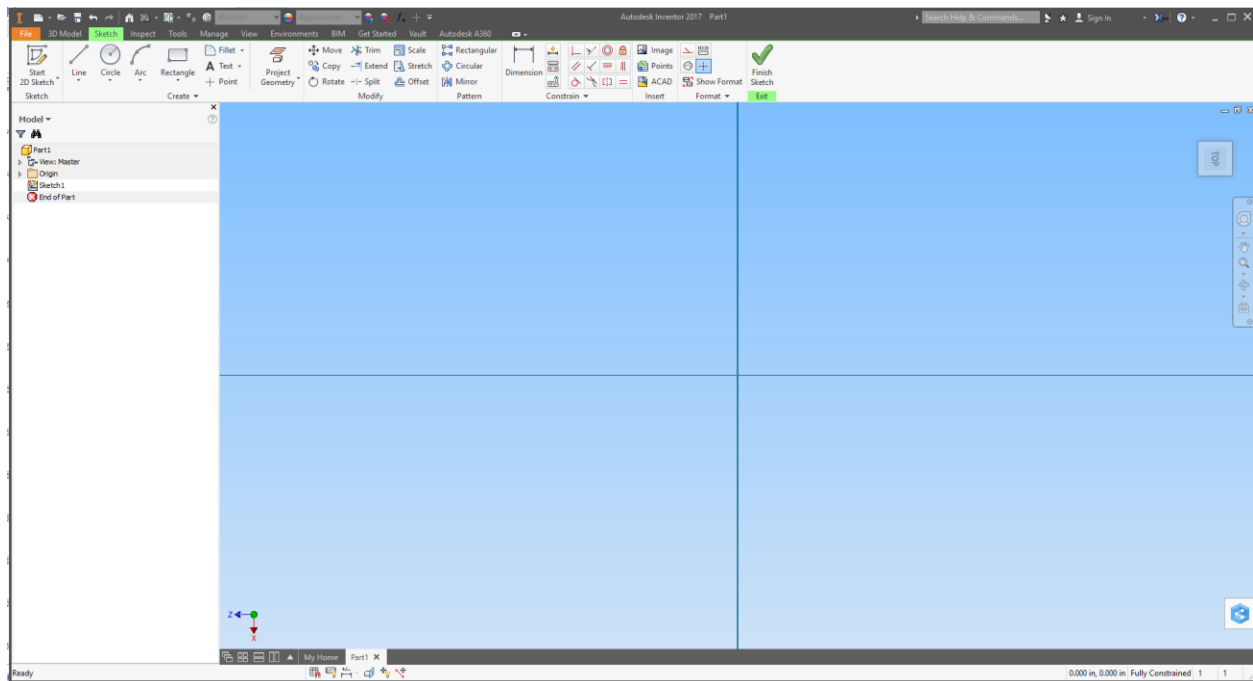


Drawing window



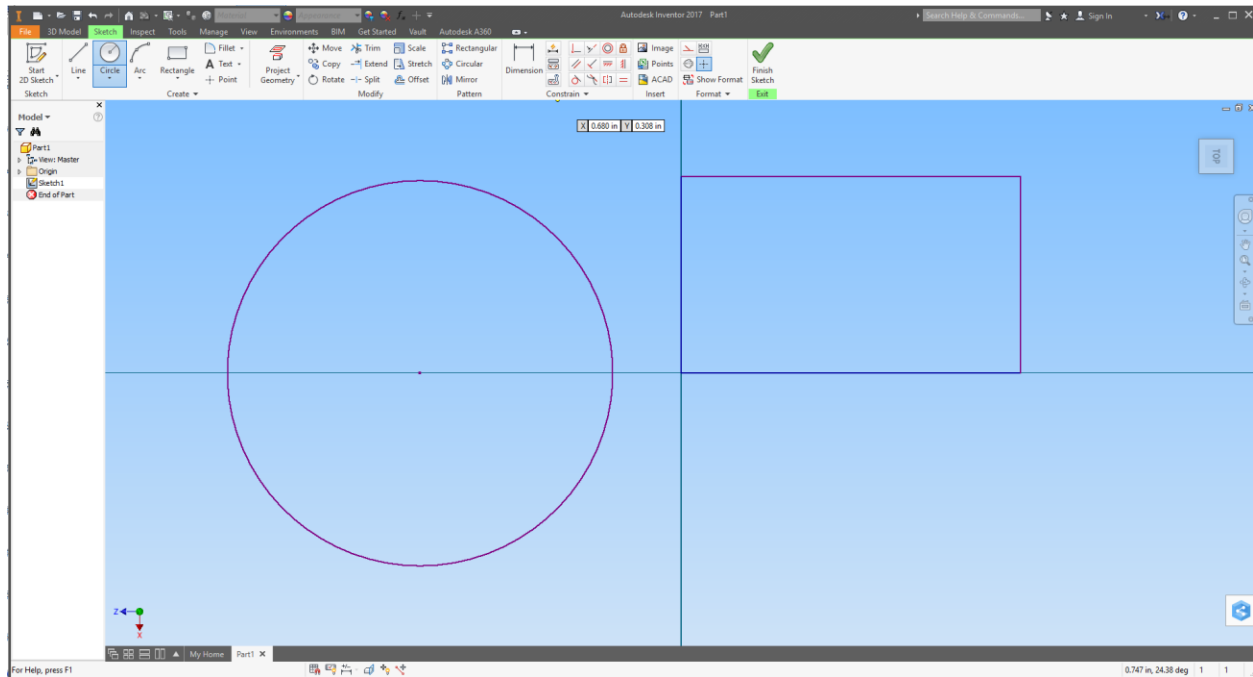
The first thing you need to do is to draw a sketch . By selecting the command “ *start 2D sketch*” Inventor will ask you to select a working plane (xy, xz or yz) on which you would like your model to be drawn.

Inventor will bring you to the selected plane and you are now ready to start the sketch

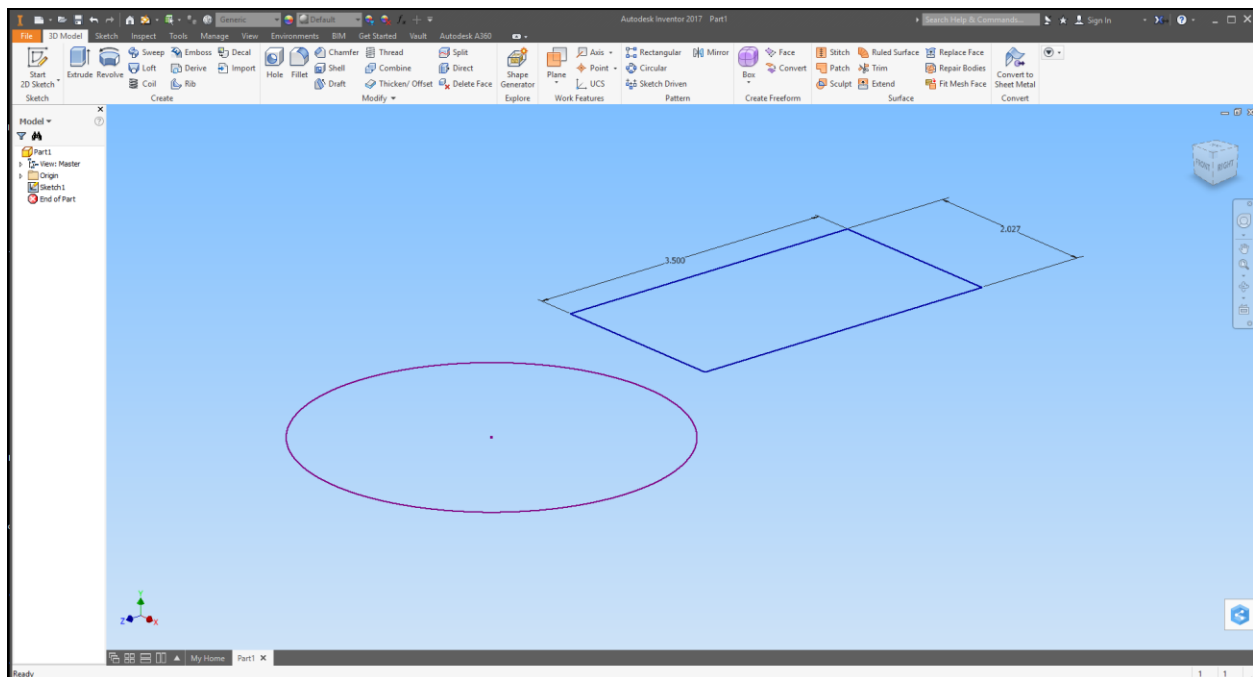


Selected plane

Using the shapes command you can create lines, circles, arcs, rectangle and polygons. Using the command “dimension” easily change their sizes which can be displayed in the drawing window.

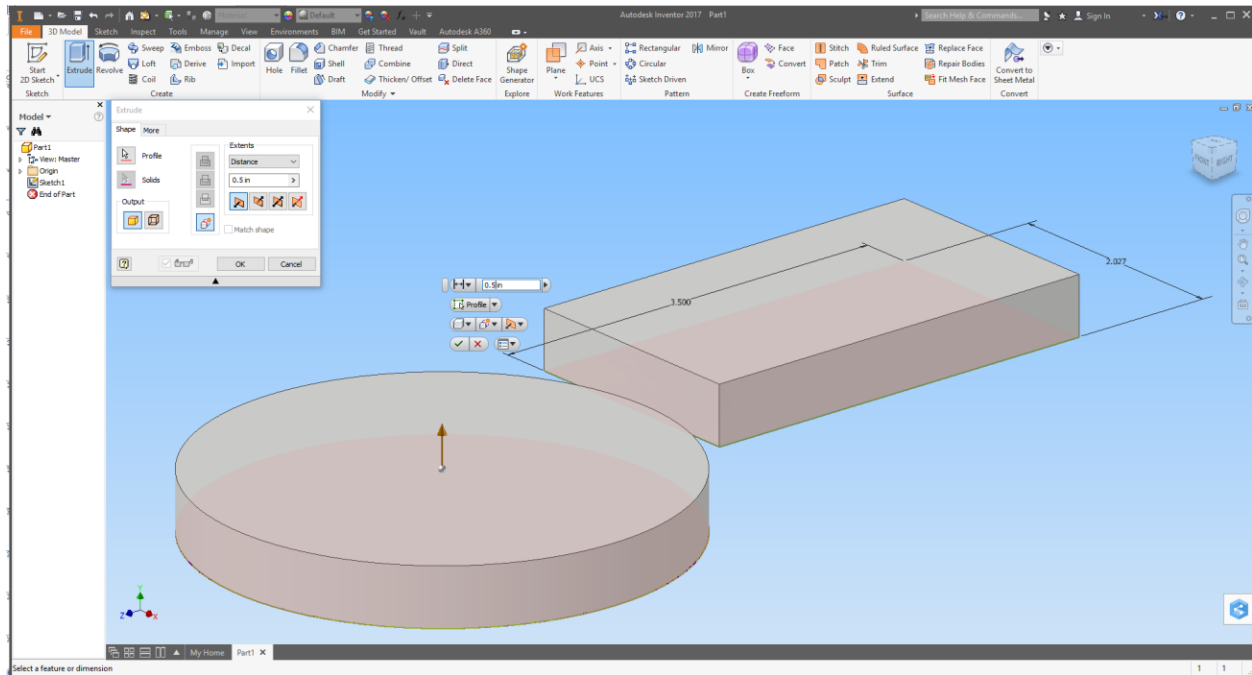


Once the sketch is finished you can exit the sketch mode by selecting “finish Sketch” which will bring you to the selected plane view.



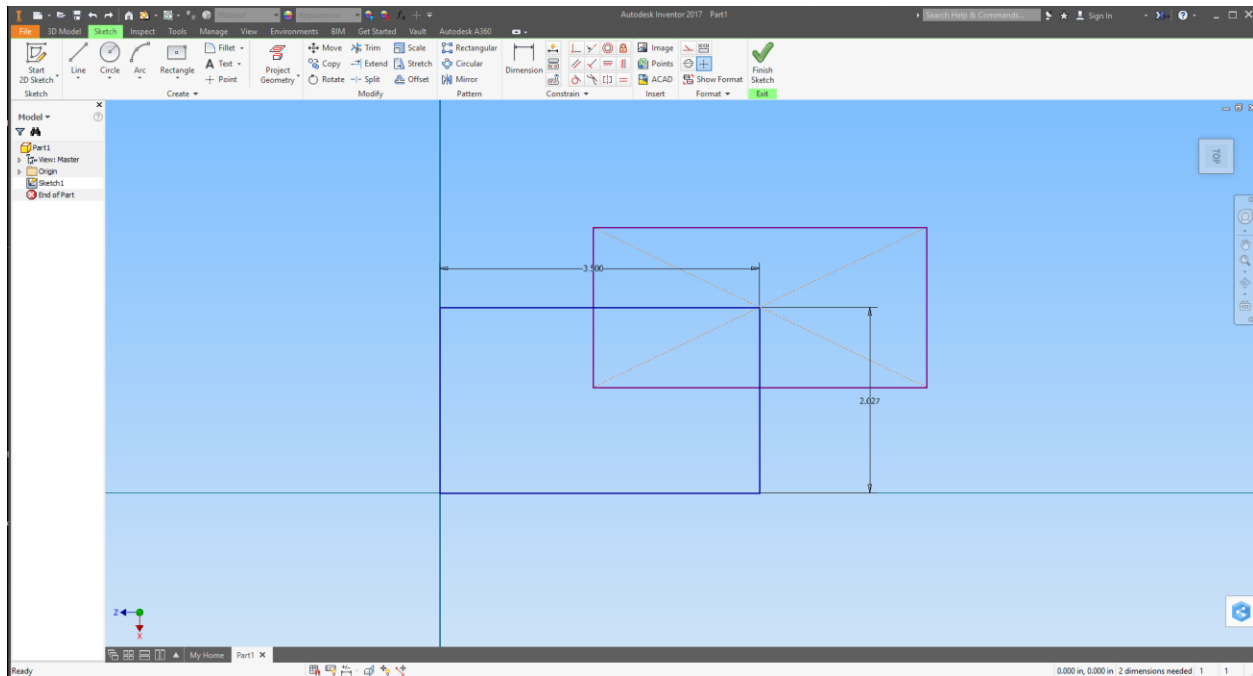
Sketch view on the chosen plane with sizes annotation

To create a solid model of your sketch you can use the command “Extrude” and select how much and in which direction you would like your sketch to be extruded.

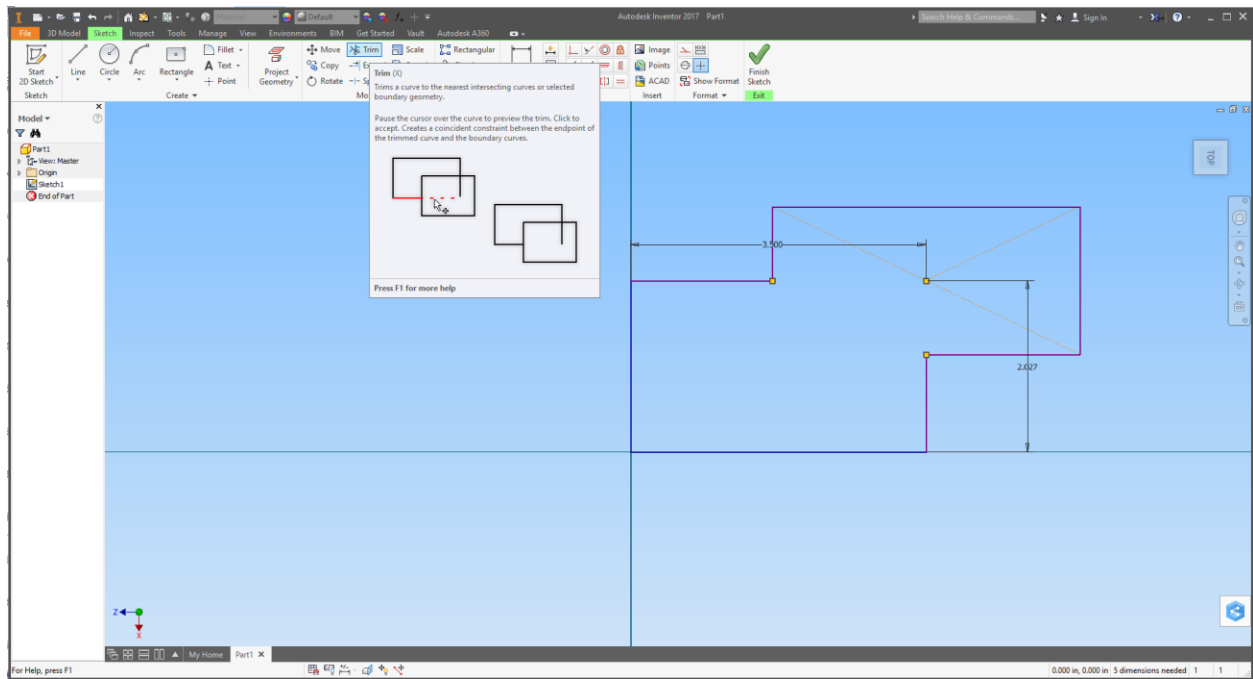


The view cube displayed at the top right corner and the tools on the navigation bar will help to control the view and orientation of the model.

You can also intersect shapes and use the trim tool to delete the overlapping lines creating a new shape.

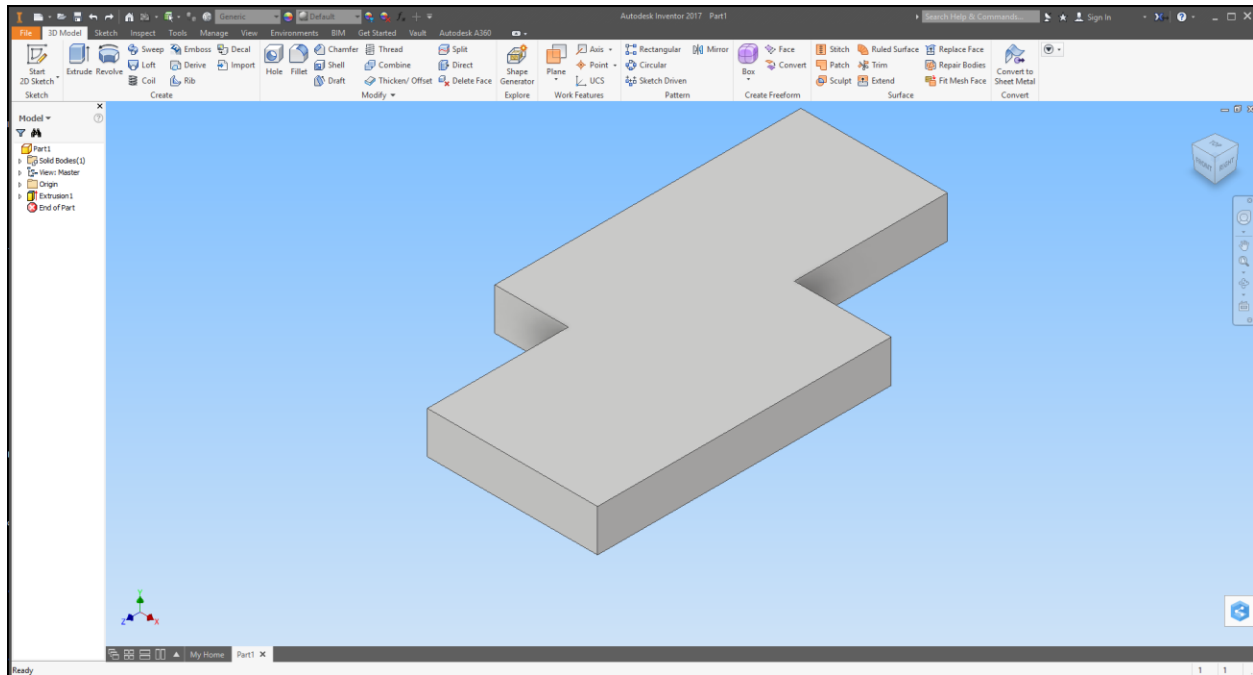


Intersecting shapes

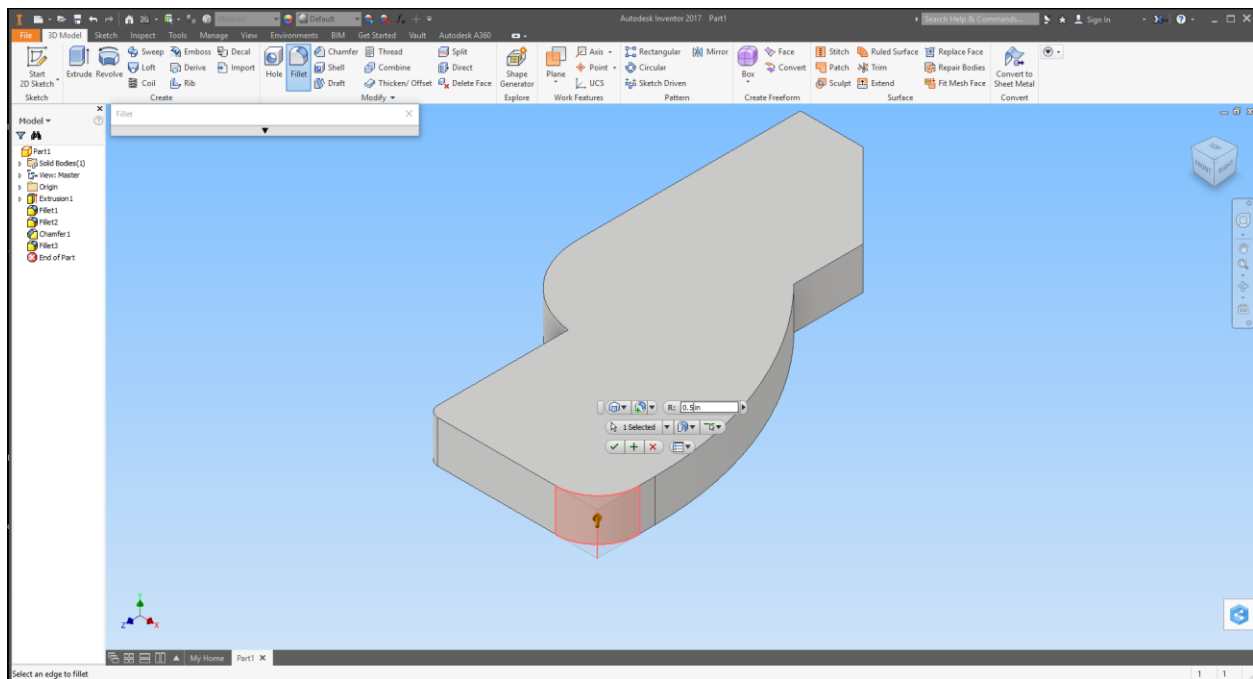


Trim command

After having extruded the intersected shapes you can modify the edges by using commands like “Fillet” and “Chamfer”.



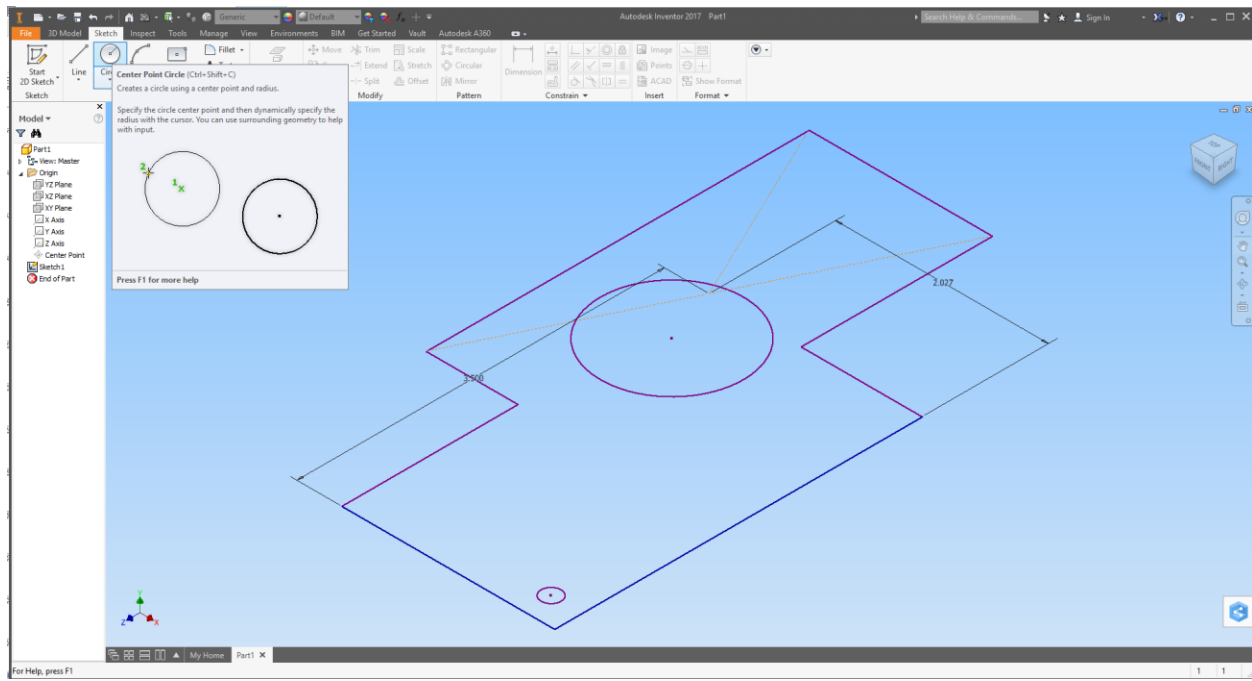
Extruded sketch



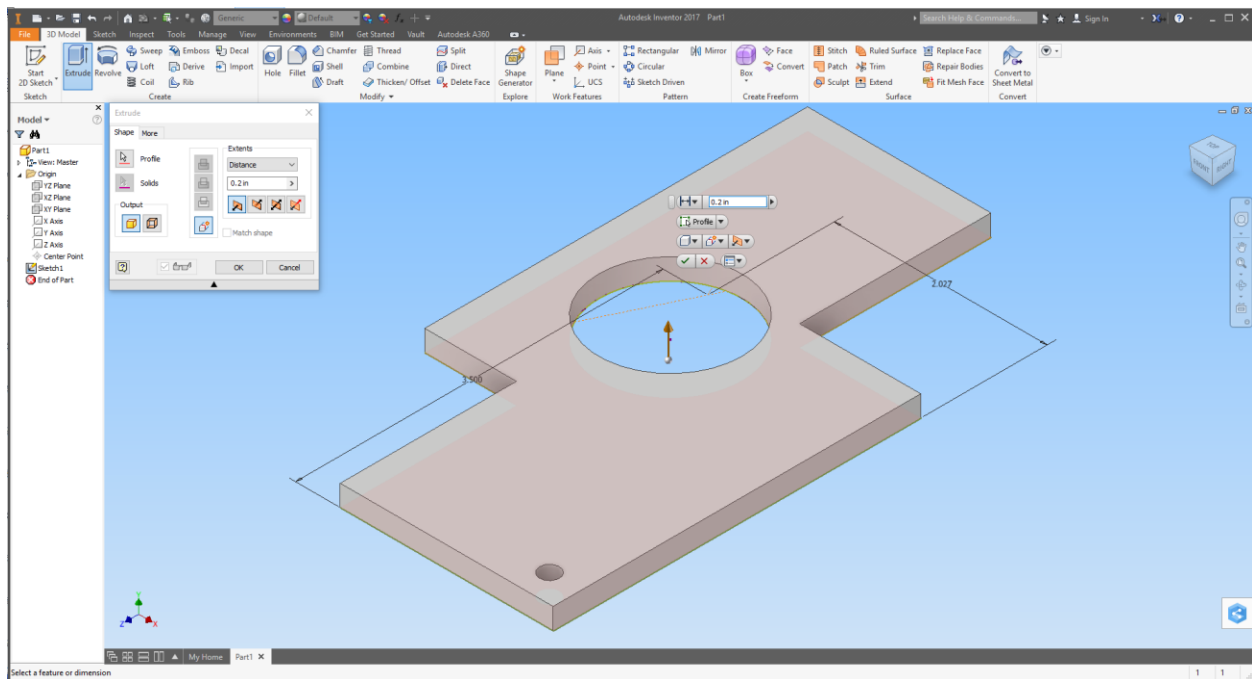
Fillet and chamfer examples



To create holes in your model you can either draw them in the sketch and then extrude the whole shape or create a second sketch on the surface of your model and extrude it on the opposite direction. Creating sketches will allow you to modify the part adding details and complexity.



Holes included in the first sketch



Extrusion of the whole shape including the holes

## EXERCISE: YOUR FIRST PART FILE

Using the tips given by this brief introduction to Autodesk Inventor try to design the base of your robot (we will cut the base out of blue, light blue or white acrylic sheets, thickness 5 mm)

## FROM INVENTOR TO LASER CUT

Create a PDF file of your model using the following steps:

- Save the .ipt file you have just created
- Open laser\_cut\_template .idw
- Place your .ipt on the drawing (open the .dwg file called Inventor\_LaserCut\_Template)
- Select Base and place your design on the drawing
- Select the right orientation using the view cube
- Scale it 1:1
- Select all – Annotate – format – laser cut (the drawing lines will become red)
- Export PDF
- In the options select vector resolution to be 1200 DPI
- Save the file