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CAD Parts + SVN Intro

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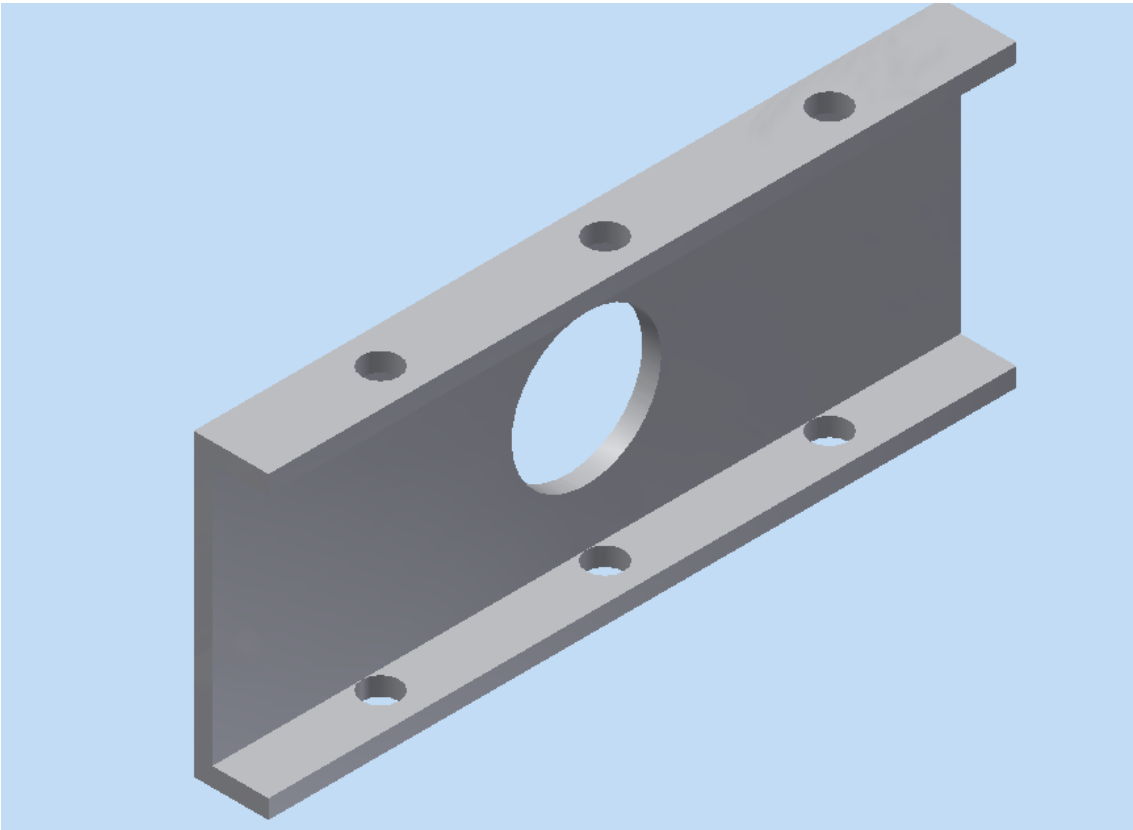
A C-Channel Part



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Make a Part File

- Click on the “I” in the top left
- Select the arrow next to “New”
- Select “Part”
- Save the File as “C_Channel_(your name).ipt”

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Sketches

- The most basic design is a sketch
- 3D models are derived from basic sketches using features like:
  - Extrude
  - Revolve
  - Hole

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Create a 2D Sketch

- Click “Create 2D Sketch” in the top left corner
- Click the (+) next to “Origin” on the sidebar and select “XY Plane”

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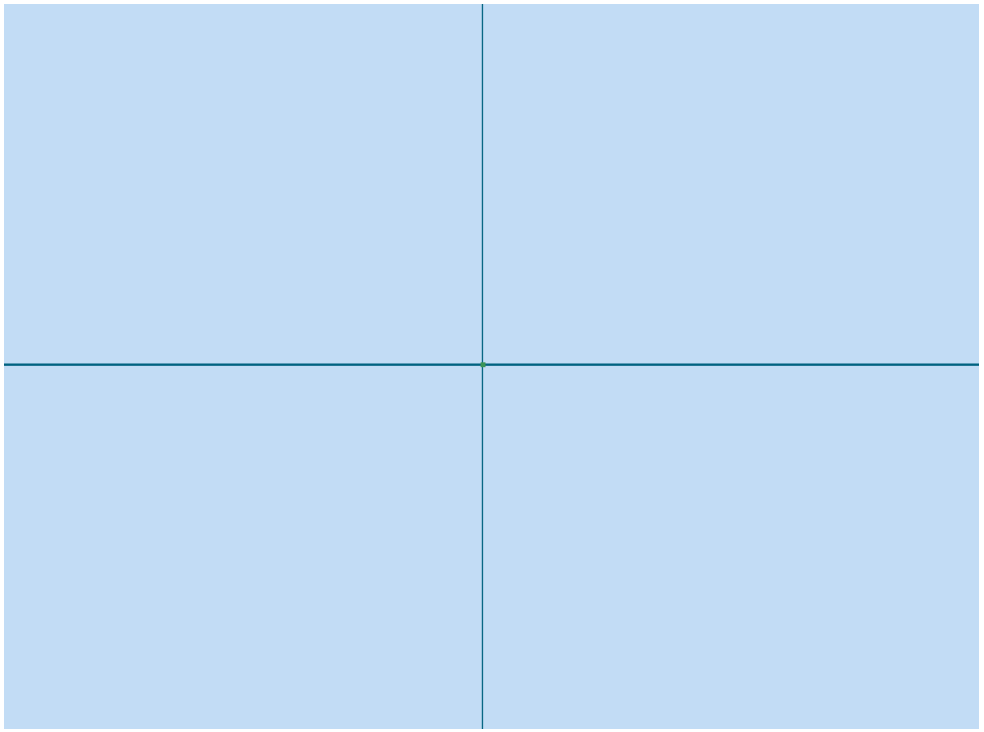
Sketch Mode

- You will enter sketch mode now in Inventor
- The two lines that you see are the X and Y axes



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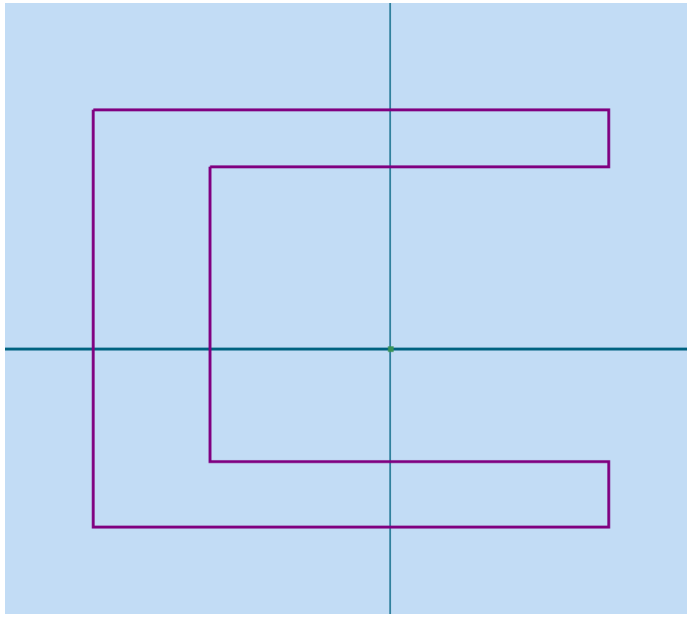
Make the "C"

- Start by Making Lines:
- Use the line tool in the top left to drag out perpendicular lines in the form of a general "C"



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Constrain!

- Use collinear constraint to make the outside lines of the flanges lie on the same line...click collinear, then the two lines

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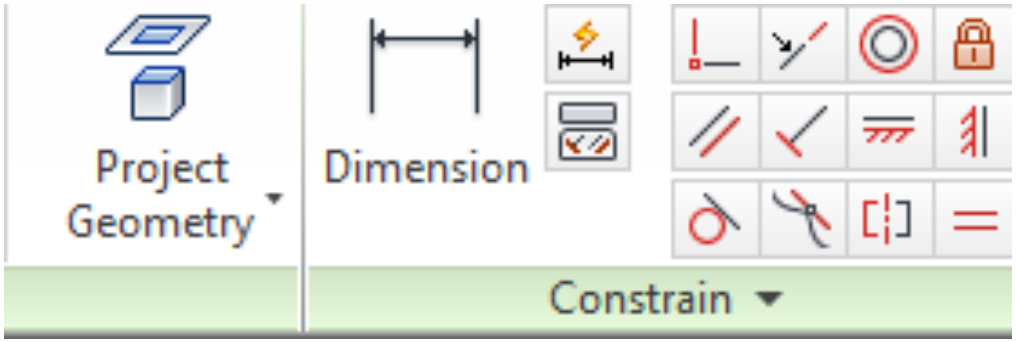
Dimensioning

- Add dimensions (measurements) to the drawing!
- Click the “Dimension” Tool
- Create Dimensions



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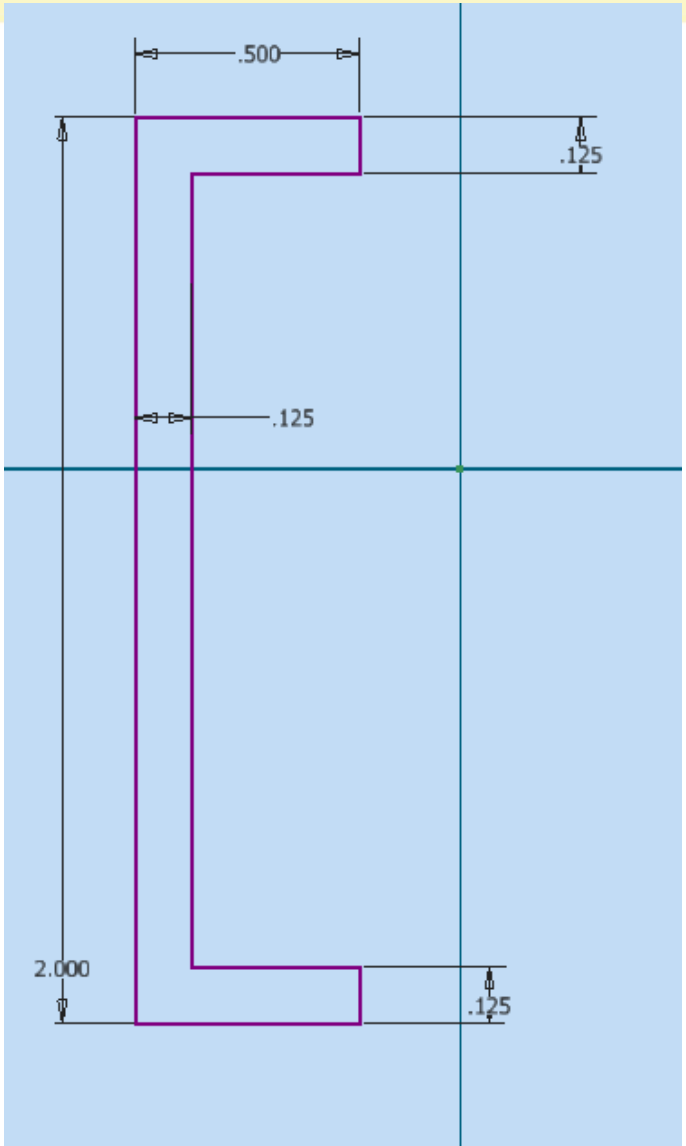
Sketch Dimensions



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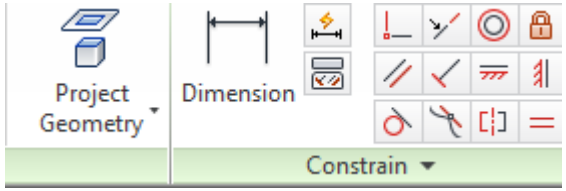
Constrain!

- Add Sketch Constraints to fix the locations of the lines in the sketch
- Project the x and y-axes onto sketch with Project Geometry



- Make top and bottom lines symmetrical on x-axis...click top, bottom, then x-axis

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Constrain!

- Use symmetry constraint again to make left and right sides symmetrical on the y-axis

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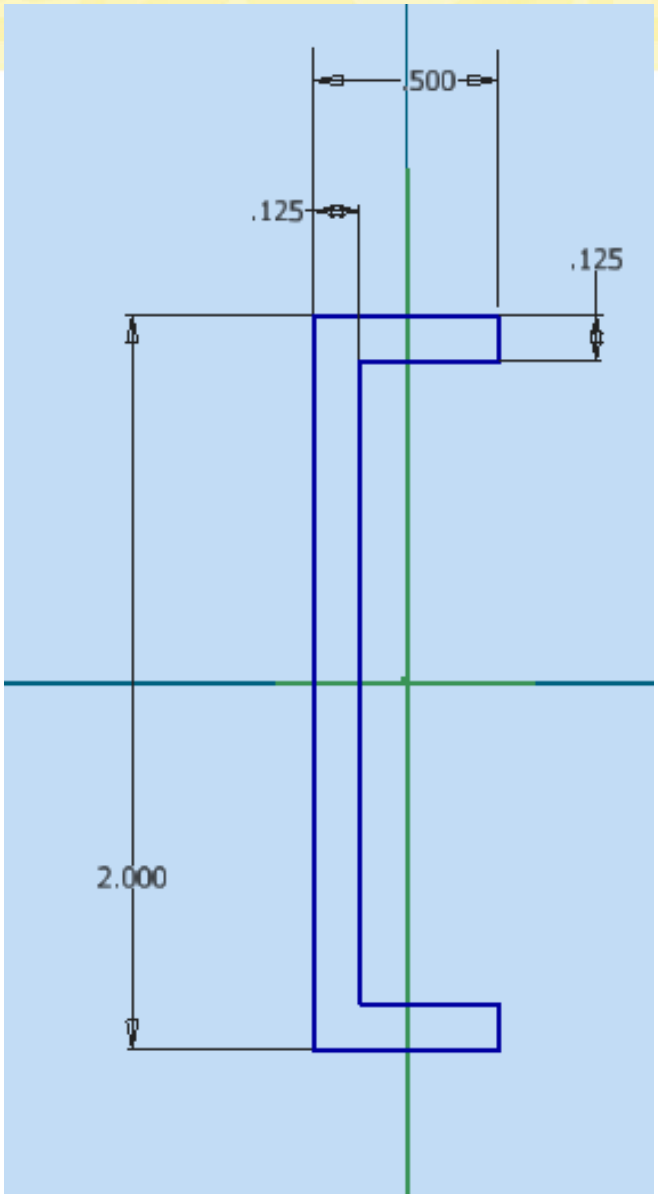
What it Should Be



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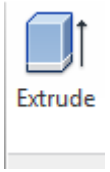
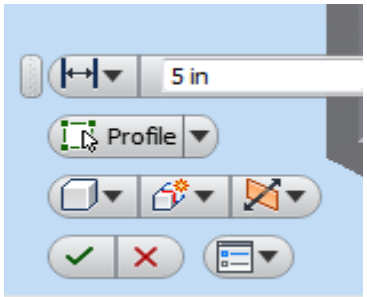
Make it 3D!

- Click “Finish Sketch” on the top bar

- We will now extrude the 2D sketch into a 3D part…
- Click “Extrude” on the top bar



- Enter “5” in the dimension field and select the two directional extrusion option



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A Second Sketch

- Rename the Extrusion-Click two times slowly on the extrusion name in the sidebar
- Make a hole on the side of the part.
- Rotate view to see side of channel and click face.
- Select “Create 2D Sketch”
- Make a circle on the center dot on this face using the circle tool
- Dimension the Circle as “1 inch”

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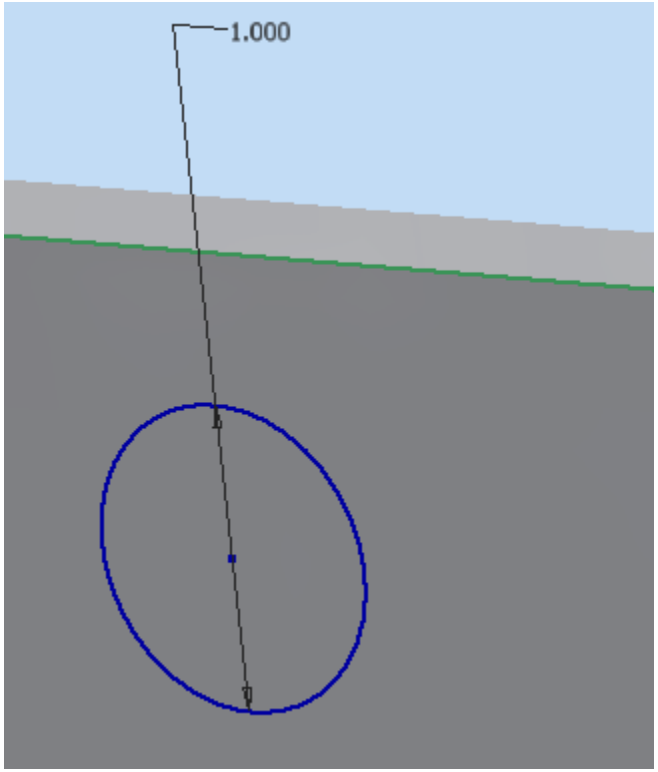
The Sketch



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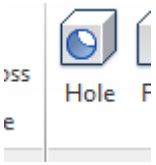
Make a Hole

- Click “Hole” in the top bar



- Click the center point on the sketch and dimension to hole to be 1 inch
- Select Termination “Through All”

- Rename the Hole in the sidebar

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More Holes

- Create a 2D sketch on the upper flange face
- Draw horizontal and vertical centerlines
- Draw 3 circles on the vertical centerline… one on the intersection, one on top, and one below
- Make the distance between the centers and the horizontal centerline symmetrical

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More Holes

- Add a dimension for the distance from the center of one hole to the horizontal centerline
- Enter “1.5 inches”
- Dimension one hole size as “0.25 inches”
- Make the holes equal with the equal constraint

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Make the sketch 3D

- Use the hole feature to make the holes
- Click on all three centers and dimension the holes as 0.25 inches
- Select Termination “Through All”
- Rename the Hole in the sidebar

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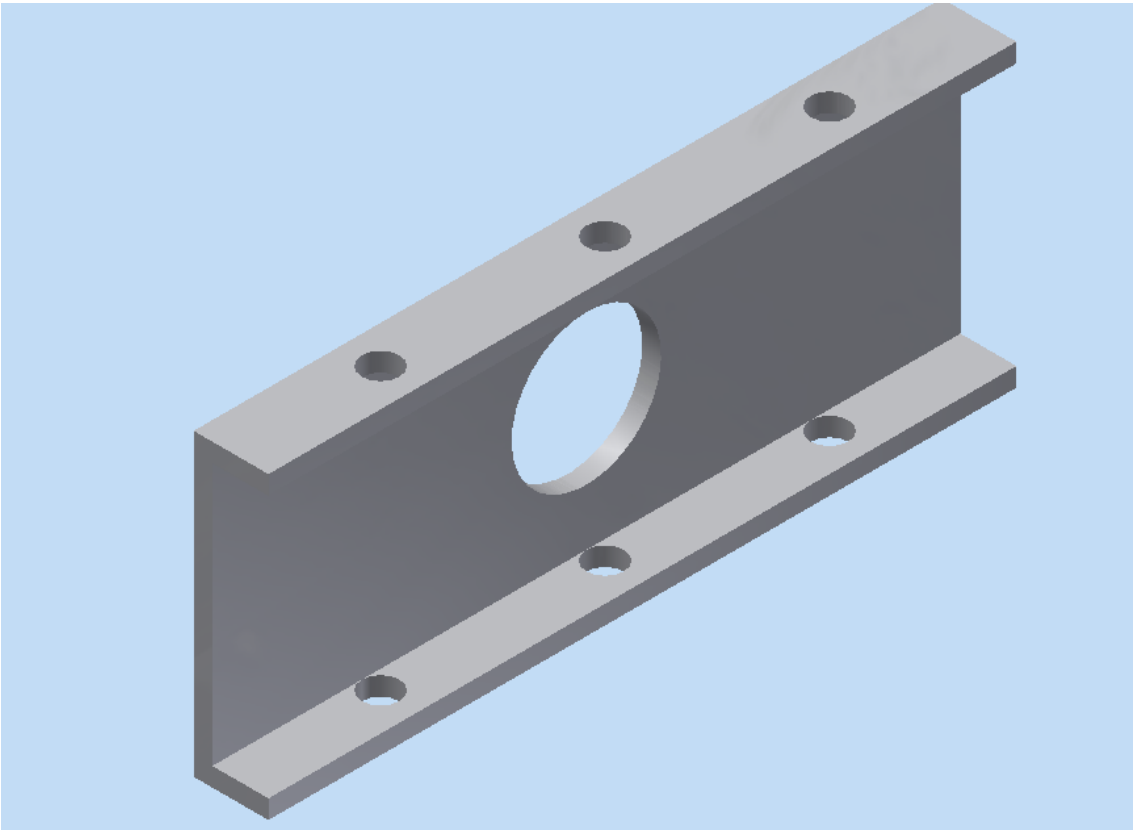
The Final Part



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You're Done!

Congratulations!