FAB322732-L

Inventor Sheet Metal from Start To Press

Kelly Young
Autodesk

Learning Objectives

- Learn how to access and set up the sheet metal template for repeat use.
- Learn how to break down each of the sheet metal features and discover advanced techniques.
- Learn how to create flat patterns and export cut files for post processing.
- Learn how to document and dimension the part for shop fabrication.

Description

In this session, we'll start from scratch and create a sheet metal part to generate a flat pattern, export a DXF cut file, and create drawings with dimensioned bend lines. Discover best practices, things to avoid, and practical advice. Aimed at product designers and fabricators that use sheet metal parts looking to streamline their workflow through Inventor. Applicable occupations are CAD Manager, Design Engineer, Industrial Designer, Mechanical Engineer, Structural Detailer/Fabricator. Users should have an understanding of basic CAD skills.

Speaker

Kelly Young
Inventor Technical Support Autodesk

Experience primarily in manufacturing and design. Began AutoCAD use in high school. Majored in Computer Science at Western Oregon University and transferred to Arizona State University to study Architecture, finishing with a degree in Design Studies. Began drafting iron doors, gates, spiral staircases, and fire & water features. Became an engineer at Aerospace Contacts. Managed drawings for 53 screw machines, designed custom fixtures, multi-step drill bits, and specialty tooling. Worked for First Impression Ironworks transferring them from AutoCAD to Inventor with parametric modeling and Vault revision management. Using Vault Copy Design led to streamlining of manufacturing, limited errors in plasma cutting, ensured proper fitment, and allowed customers to visualize with 3-D renderings. Took a position as Research & Development Designer at U-Haul International. Joined Autodesk in June, 2017 working out of Portland, Oregon. Starting role of Inventor Community Forum Specialist and currently Inventor Technical Support.

Introduction:
Before you get started with any operations within Inventor, you should take extra care to make sure that all of your Settings and Templates are as you want before you start designing.

This isn't the most exciting part of the design process, but is imperative to successful use and limiting errors and headaches in the future.

Once we get settings dialed in, we will start designing parts within the Sheet Metal environment. This is where you should experiment, play, explore, and research advanced techniques. We will cover the basics and general shapes for this course and show how to generate Flat Patterns.

Moving forward, watch everything Inventor YouTube, everything. During my time as Inventor Community Forum Specialist I created a Collection of Screencasts showing different modeling techniques that might be interesting. Here is a great link repository, if you like getting into the weeds and customization here is the spot. And buy his book while you're at it, I did in 2010 and have had it on my desk since.

I would encourage you to get involved in the Inventor Forum, ask and answer questions. You will gain invaluable knowledge, confidence, and insights.

When you get ready to move to the next level, download this Collection of iLogic Models for Beginners. It will blow your mind as to the capabilities you will be able to accomplish.

Once we have an Assembly of Sheet Metal parts created we will enter the weldment environment and add our welds. For good measure we'll go through a few design accelerators to get you familiar with things like Bolted Connection and Frame Generator.

Now that we have the parts created, we need to document for our plasma technicians, press break operators, welders, coating finishers, and assemblers. Get to know these people in your company and have an active feedback loop for Quality Control. Know what tools they use and the limitations they have.

You will get frustrated, take a deep breath, look for solutions. Sometimes it is best to start over. The program will close unexpectedly, save often, every 3 features is a good rule. You will have incorrect plasma outcomes, wrong bends, misaligned holes, bad weld call outs, and parts that don't line up. The difference between an expert and a novice is not how many times you do this, but how frequent and if they are repeated.

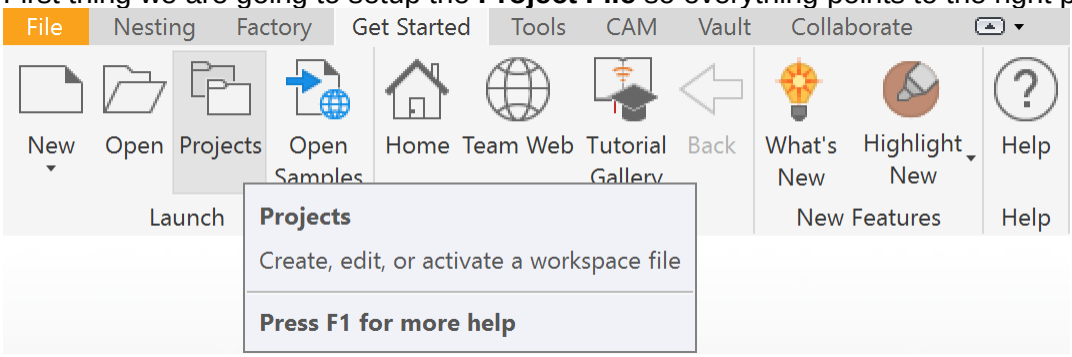
Learn from your mistakes. Take mental care in your actions and track the logic of your model. Keep it simple. There are design accelerators, but many times it is easier, quicker, and safer to just create a part and simply constrain it somewhere. Not all parts need to or should be parametric.

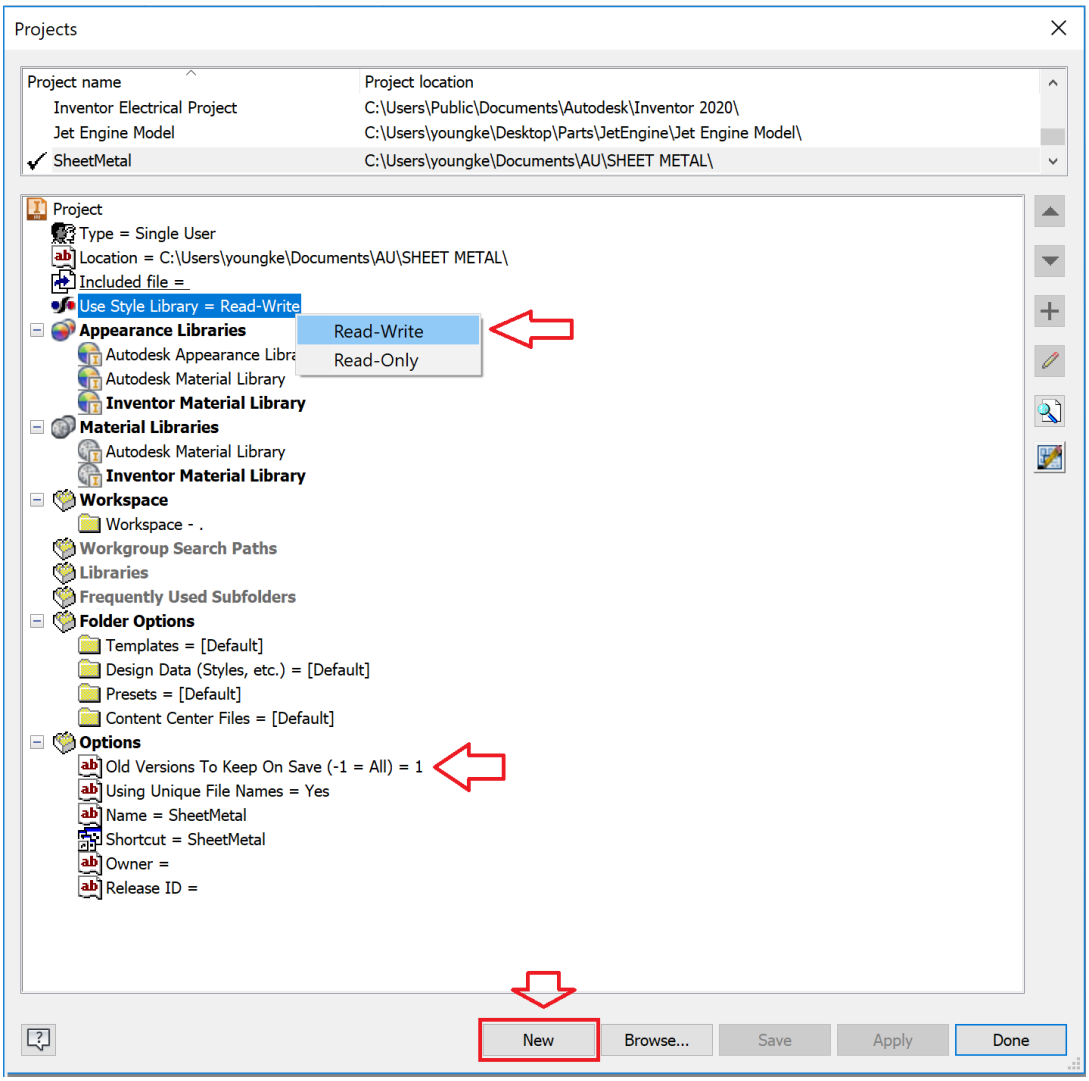
Time management and design intent are paramount for success when using Inventor.

Chapter 1: How to access and set up the sheet metal template for repeat use.

Fire up Inventor, let's go.

First thing we are going to setup the **Project File** so everything points to the right place.



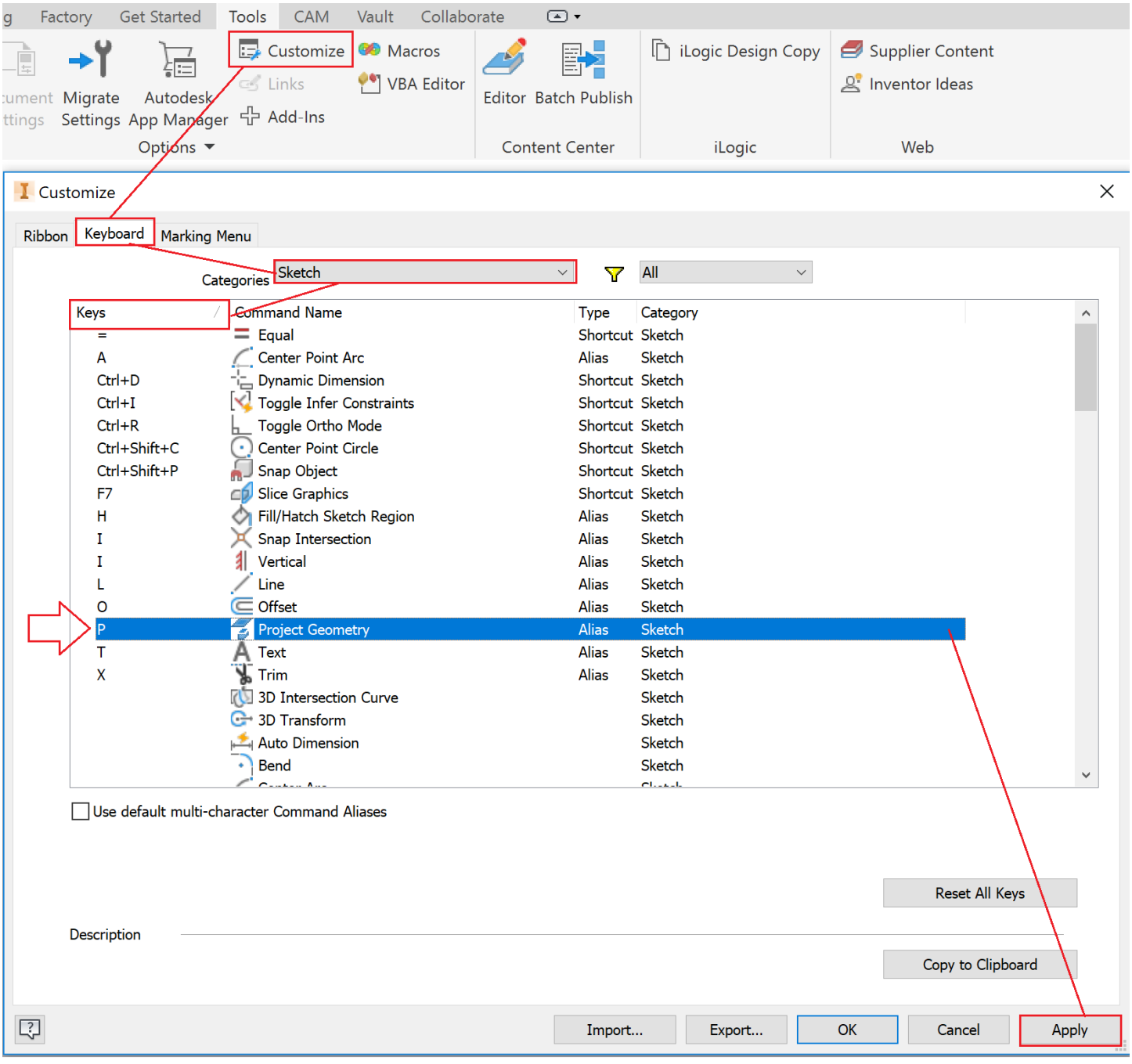


Select New > **Single User Project** > **Create** a custom name SheetMetal > Finish

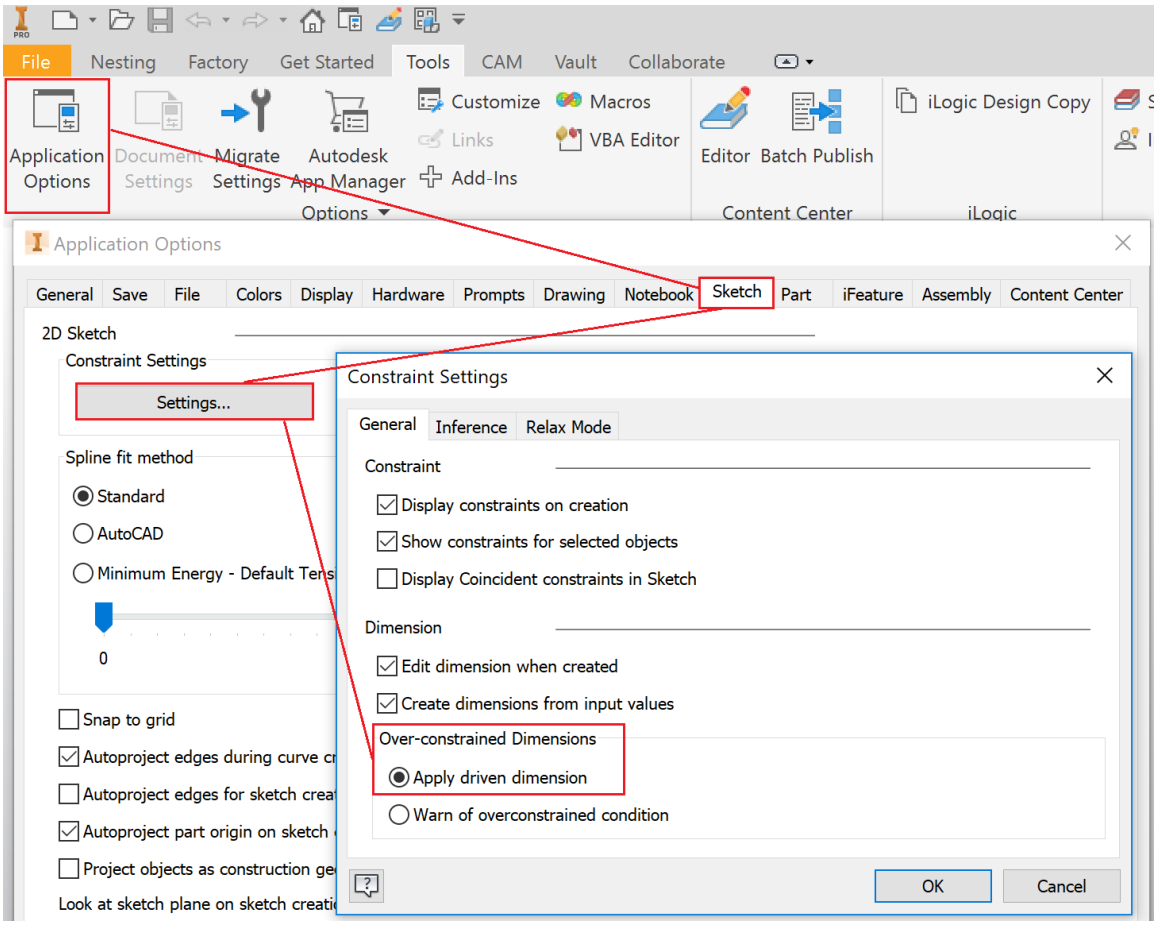
- If you have an existing **Project** that you want to use a **Library** from you can select it with Next.
- I would recommend keeping the 1 setting for **OldVersions** in case something happens.
    - **Always back up your files.** You will thank me one day.

Make sure the settings are **Saved** and **Active** with a check mark. Select **Done** or **Esc**.

Esc is your friend, you will use it often. Beware of hitting Esc instead of Apply or Save, this will happen and you will think you applied a setting but did not. Remember to pay attention, this will be a recurring theme.



This is where you setup **hotkeys**. If you've ever used AutoCAD you are itching to set these up. The one I would recommend immediately is setting **Project Geometry**, I always set to **P**, but you should make this your own as you progress and find what you use most often.

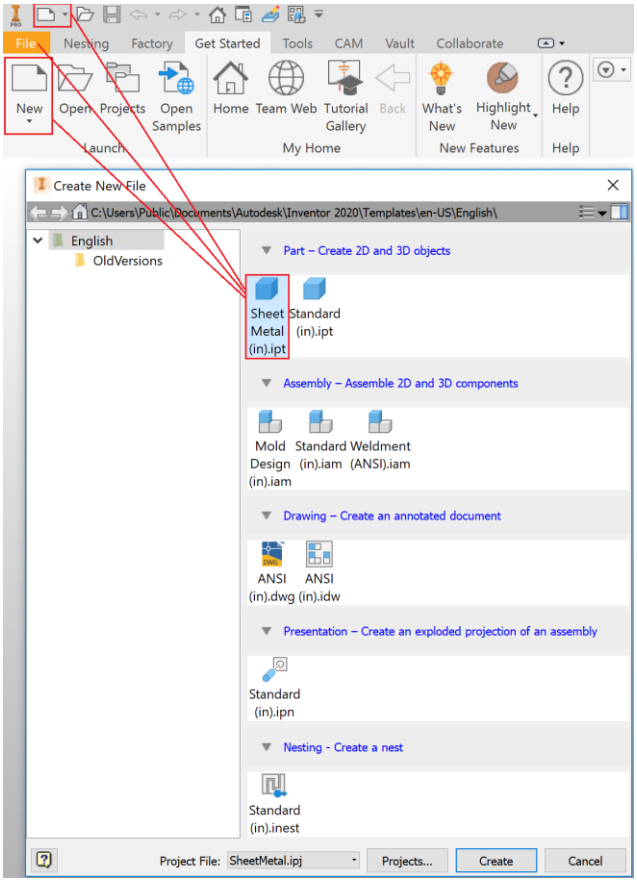


Pro Tip: Set **Apply Driven Dimension** so the pop-up doesn't show up. Leave it off until see it and know why it happens, but when you do, you'll want to go here and turn it off.

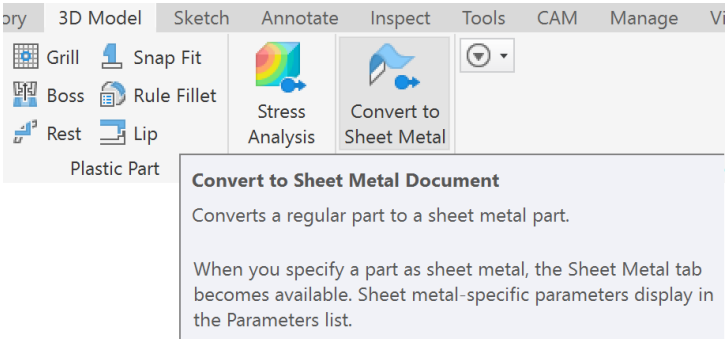
Time to get started with a New > Sheet Metal Part.ipt.

- Be familiar with the different file types unique to Inventor:
  - Part.ipt
  - Assembly.iam
  - Drawing.idw or Drawing.dwg (AutoCAD extension that works both ways!)
  - Presentation.ipn
  - Nest.inest

There are multiple ways to get to **New**, I like the little sheet up top as it never goes anywhere.

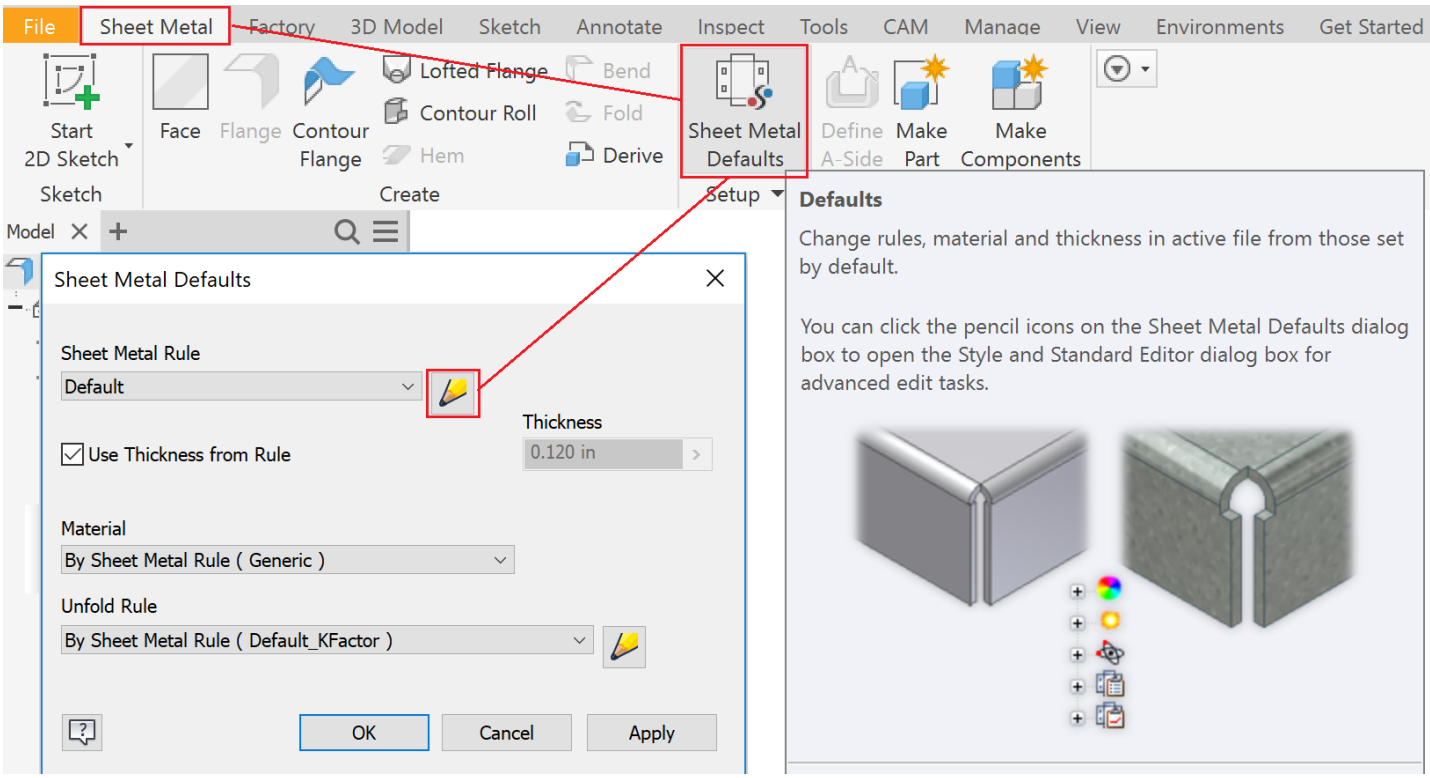


If you create a regular **Part** by mistake, no problem, you can **Convert To Sheet Metal**, and vise versa.

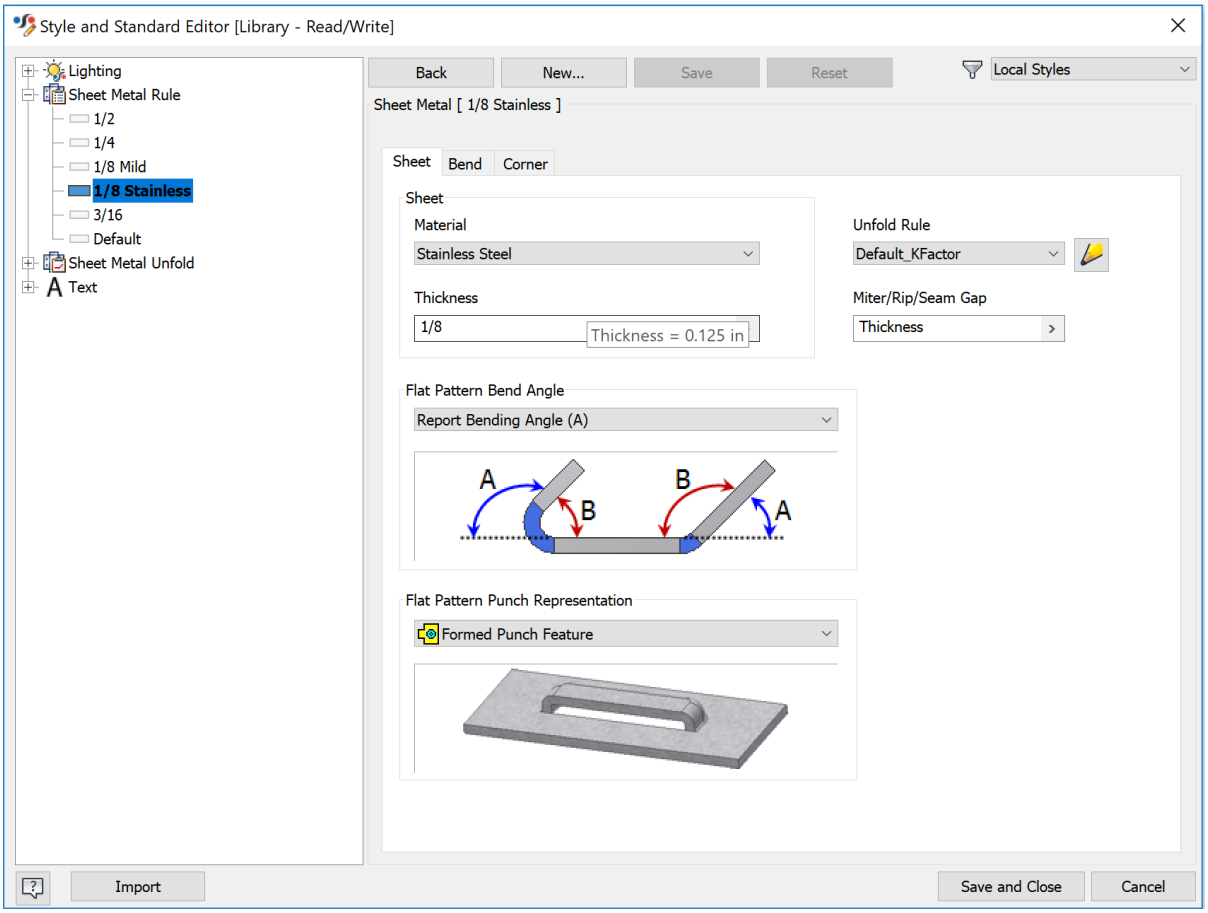


Let's setup the **Sheet Metal Defaults**.

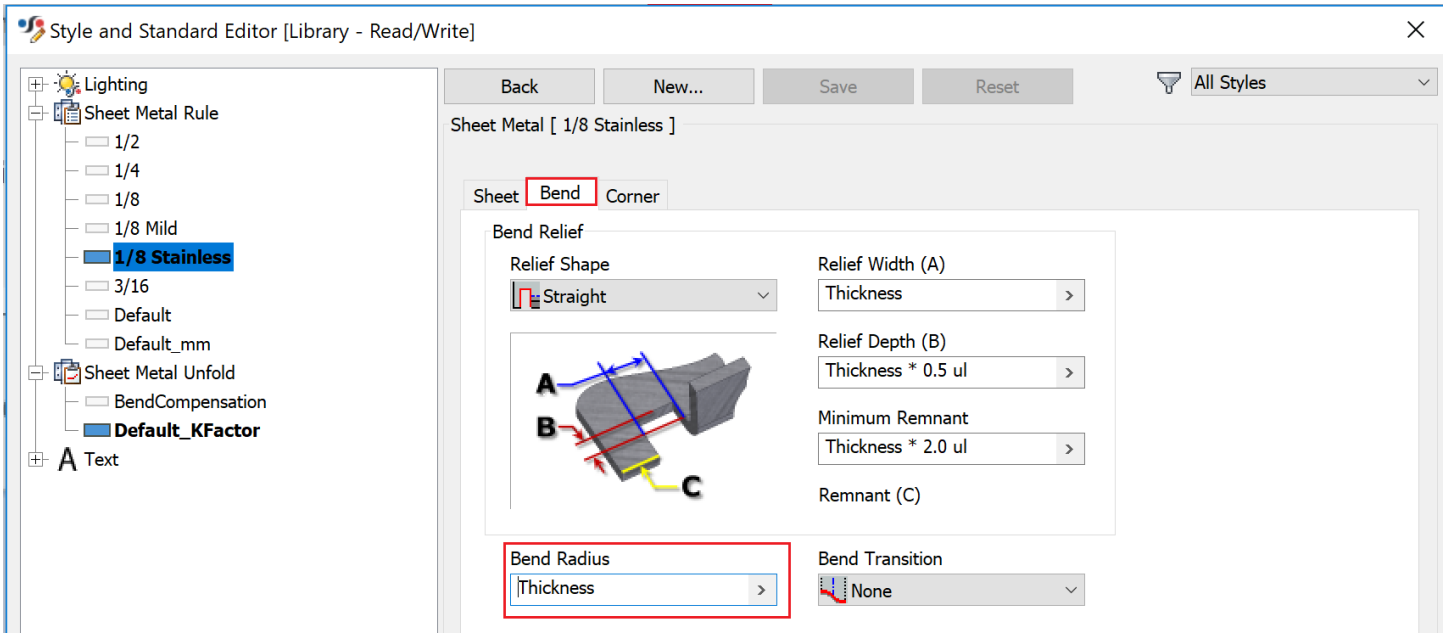
This step is critical and often overlooked.



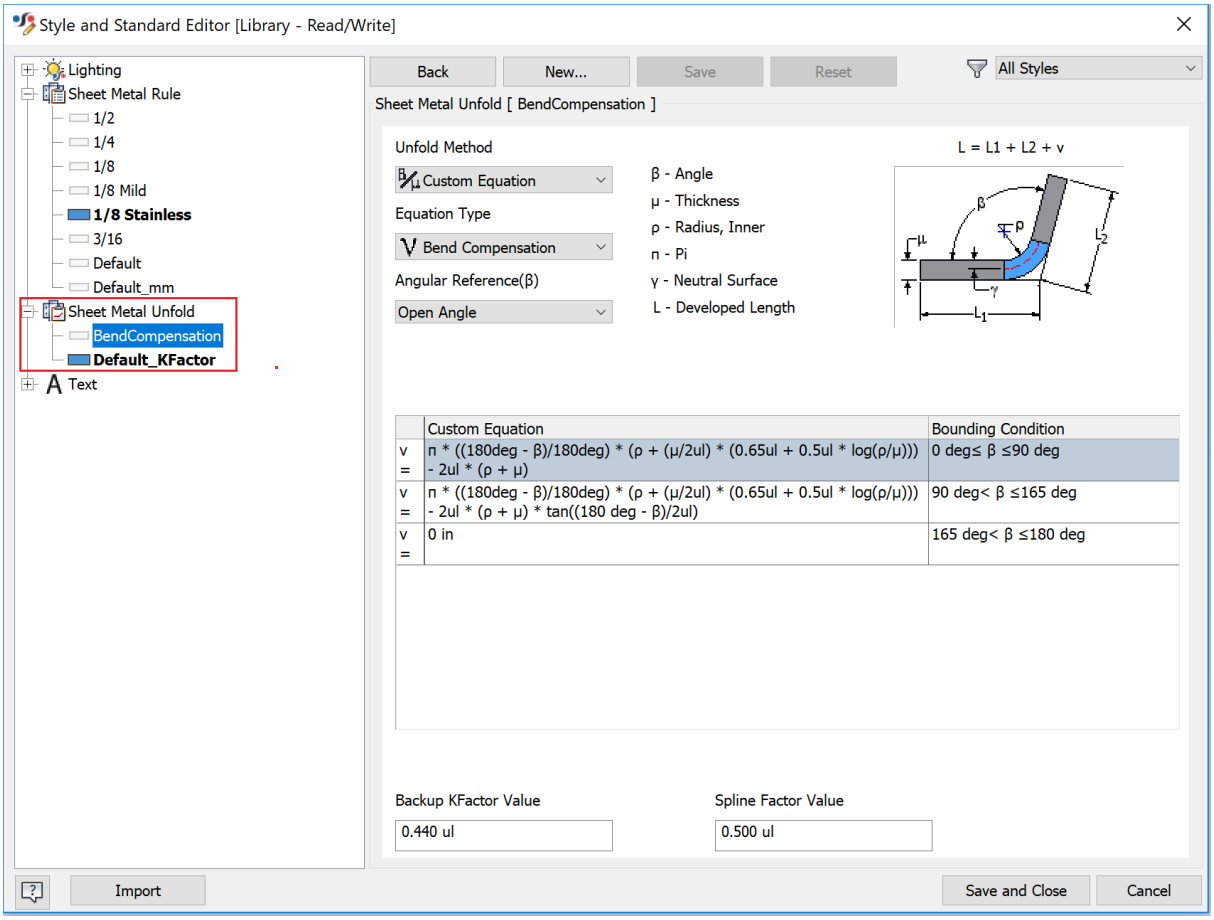
New… > 1/8 Stainless
Material = Stainless Steel.
Thickness = 1/8



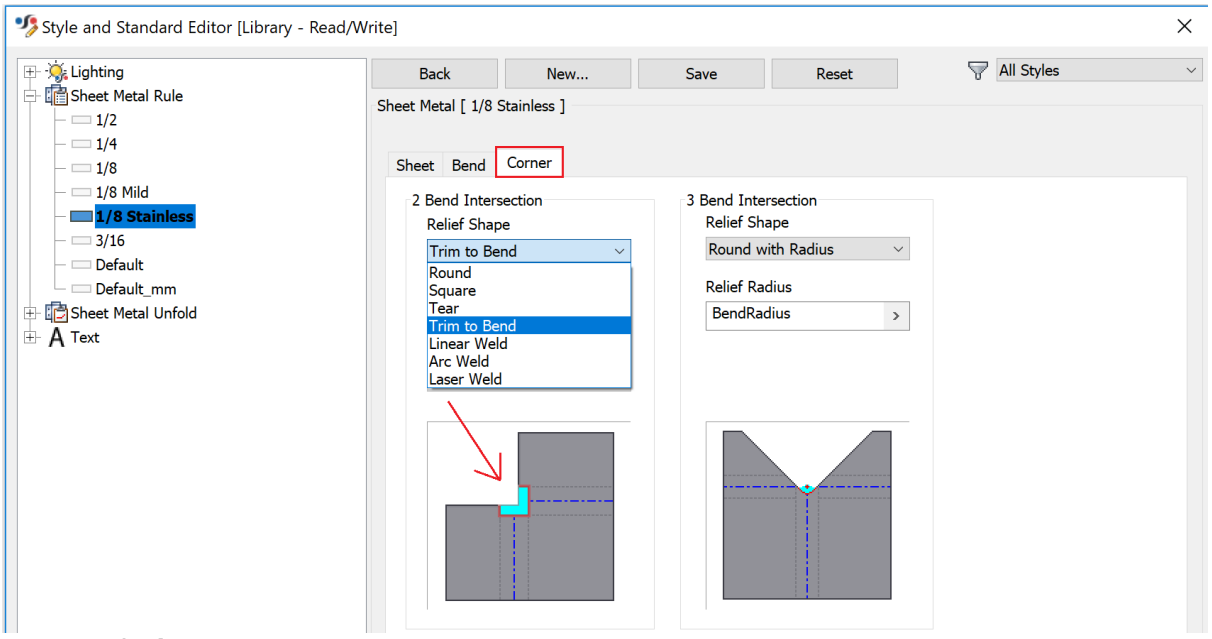
The **Bend Radius** is set to **Thickness** by default, but you should check with your press break operator to see what dies are available for use. You can define them as **User Parameters**.



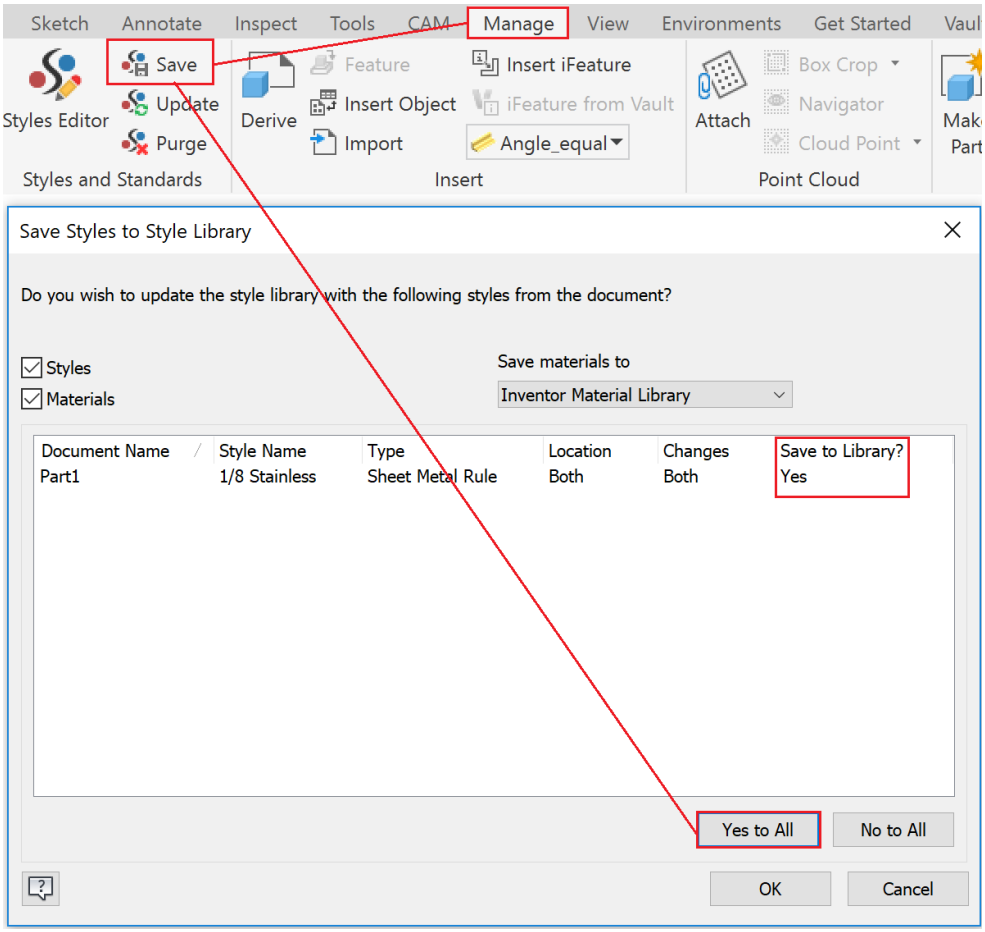
Depending on the outcome in the real world, after measuring the actual bend you might need to change the **Bend Compensation**. You can research K-Factor, Custom Unfold Equations, and Bend Allowance.



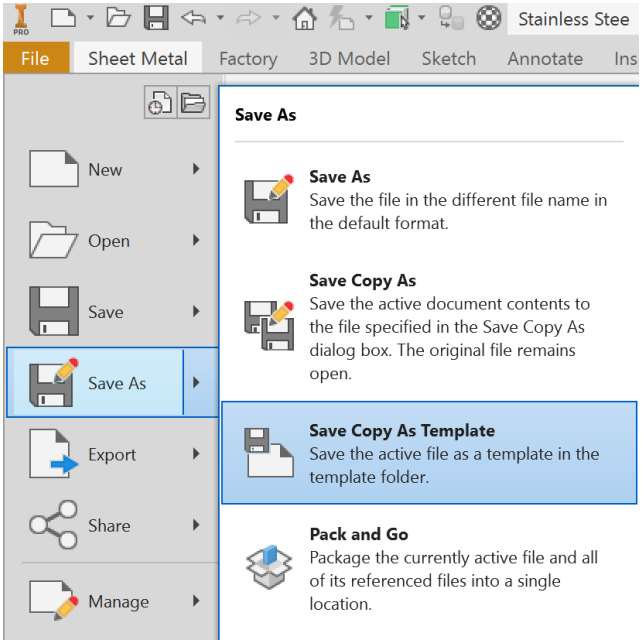
Corner will change the bend intersection relief. Depending on the Cutter and Press Break this will vary with the project scope.



Save and **Close**.



Now that we have written the **Sheet Metal Style** to the library it can be used in the future.

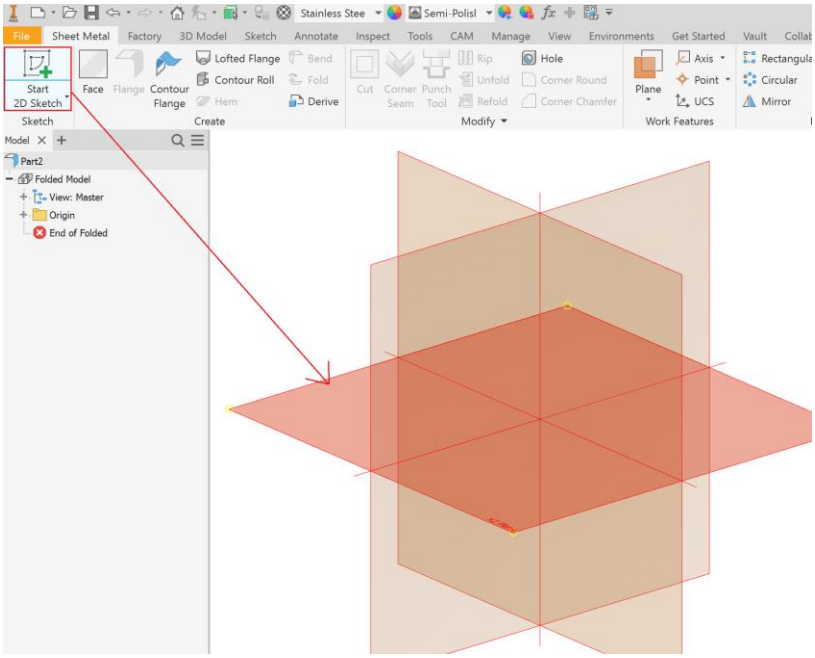


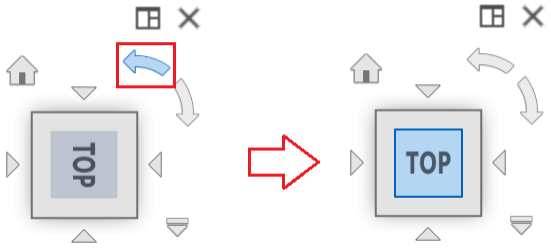
We have the settings for **Stainless Steel Material** and **Thickness** at 1/8. We can **Save Copy As Template** to ensure any new Sheet Metal part we create is set to this every time we start.

Chapter 2: How to break down each of the sheet metal features and discover advanced techniques.

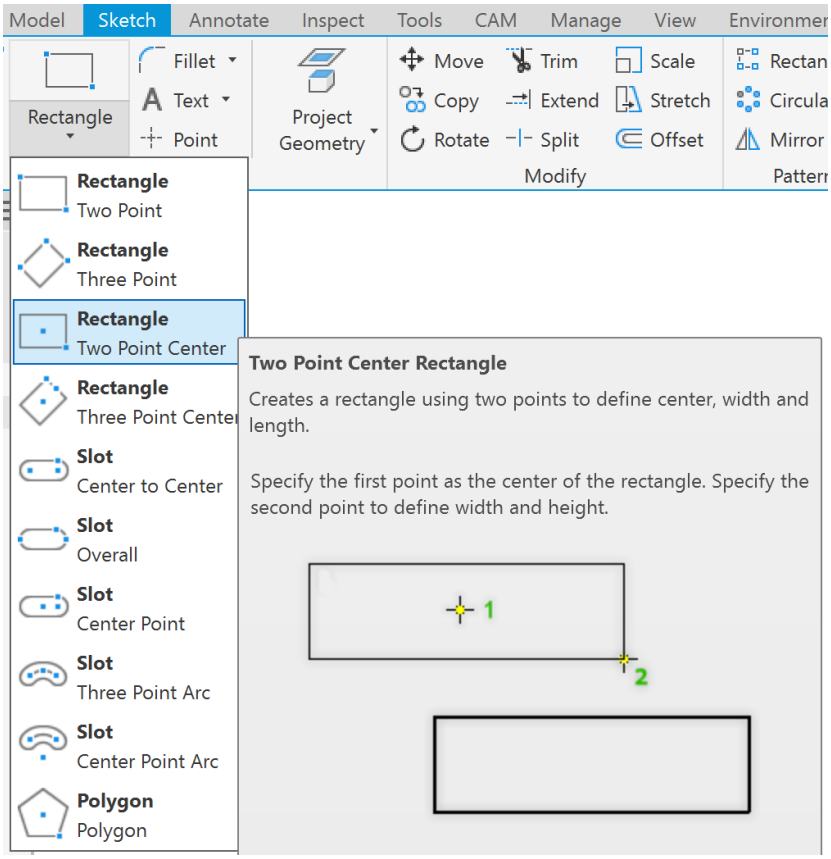
Time to start designing.

Start 2D Sketch:





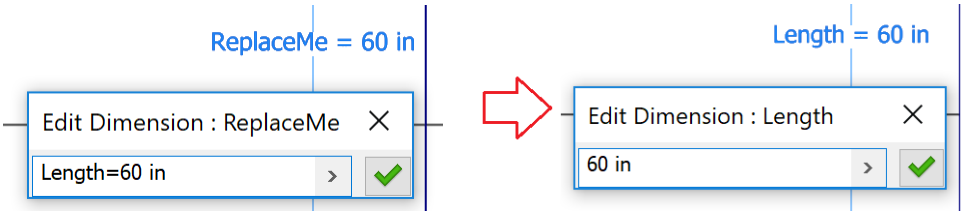
Rotate the **View Cube** so it is oriented normally to your view.



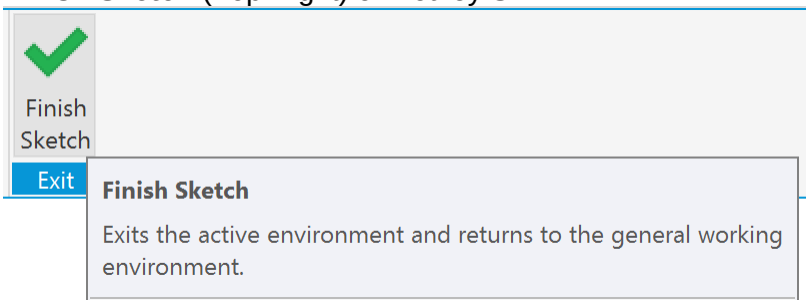
Select the **Origin Center Point** and draw a **Rectangle** without clicking a second time.

Type: **Width=36** > **Tab** > Type: **Length=60** > **Enter**
By typing in the **Parameter Name** we can then reference them easily later instead of having to remember where dimension d12 goes to.

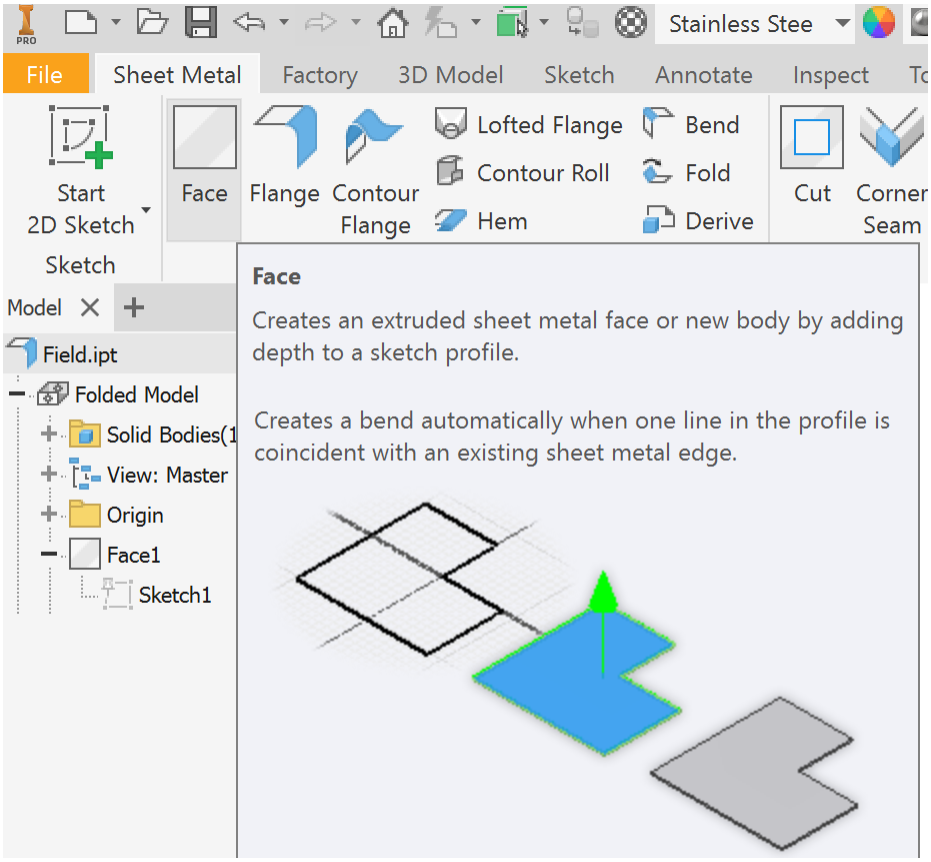
If you have already set a dimension you can always change it or set it after the fact by typing the NAME=



Finish Sketch (Top Right) or Hotkey S



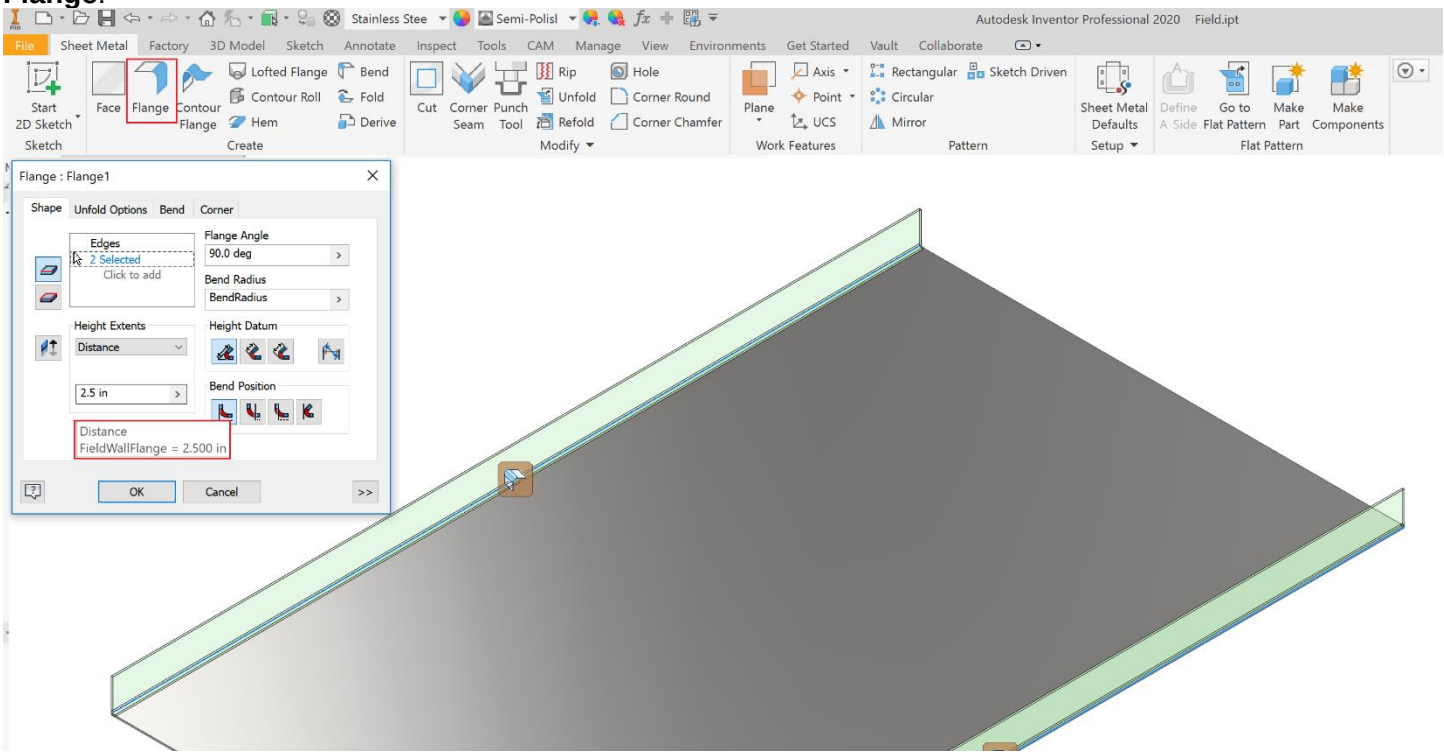
Face:



This automatically chooses the active **Sheet Metal Style** and uses the **Thickness** parameter.

- It in effect does the same thing as **Extrude** at a distance of **Thickness**.

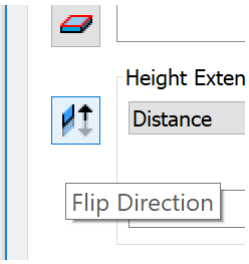
Flange:



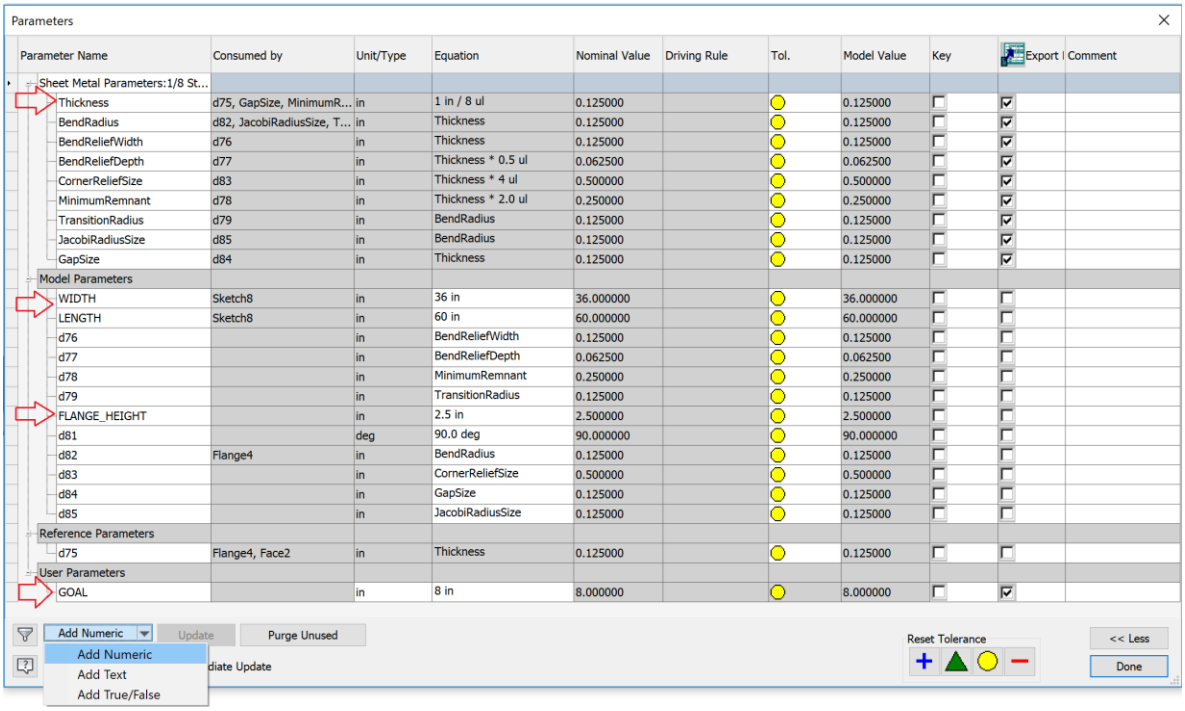
Set **FLANGE_HEIGHT=1.5**

Select top edge it will flip **Up**, bottom edge **Down**. You can manually **Flip Direction**.

- Use the **View Cube** to look at the **Front View** and zoom in on the bend.
- Change the **Bend Position** and **Height Datum** to see what they control.
- The other tabs will follow the **Sheet Metal Defaults** but you can override them here if needed.

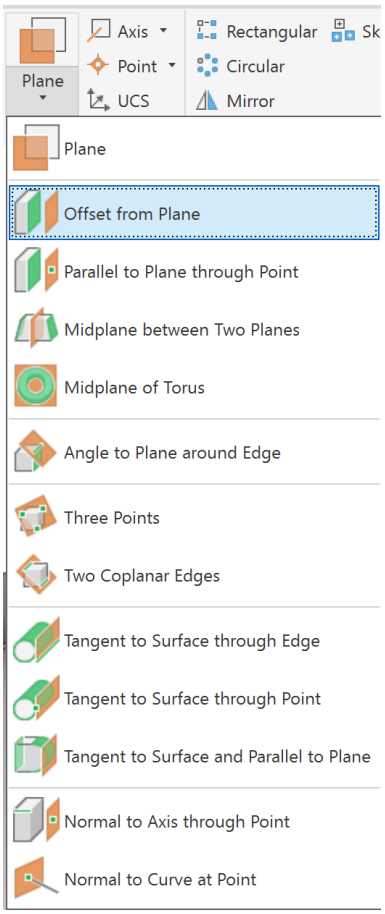


Access **Parameters** and **Add Numeric User Parameter** called **GOAL** set to 8in.

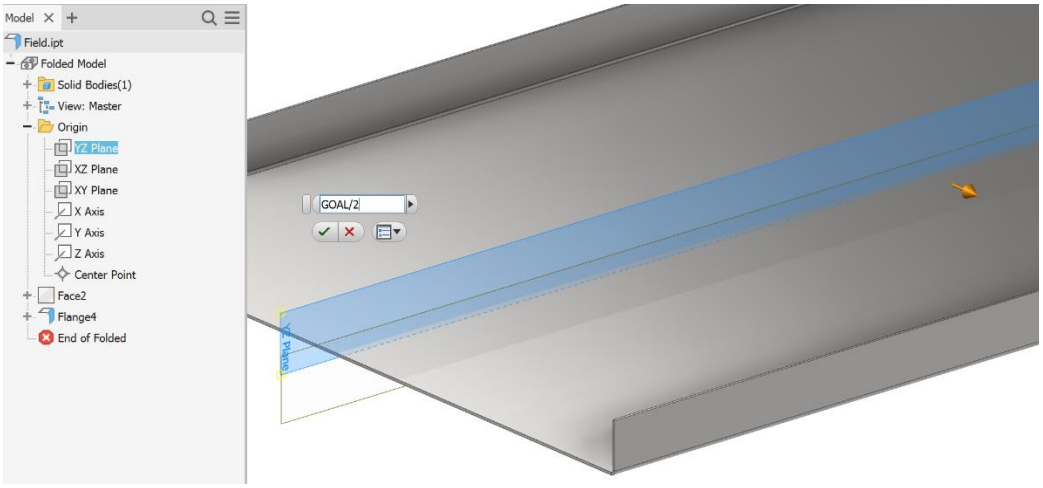


SAVE

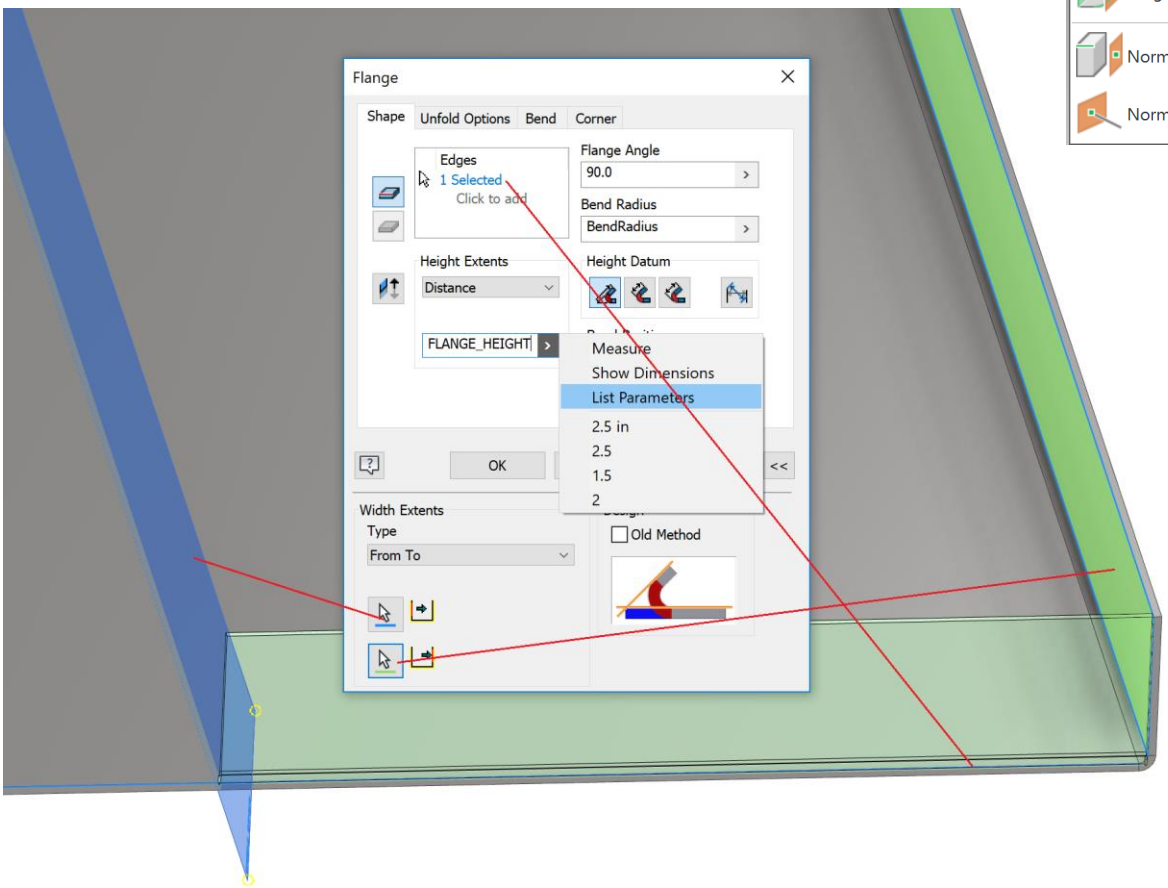
Setting up **Planes** is going to be a go to move for creating **Offsets** and known distances.



Offset from the **YZ Origin Plane** at a distance **GOAL/2**



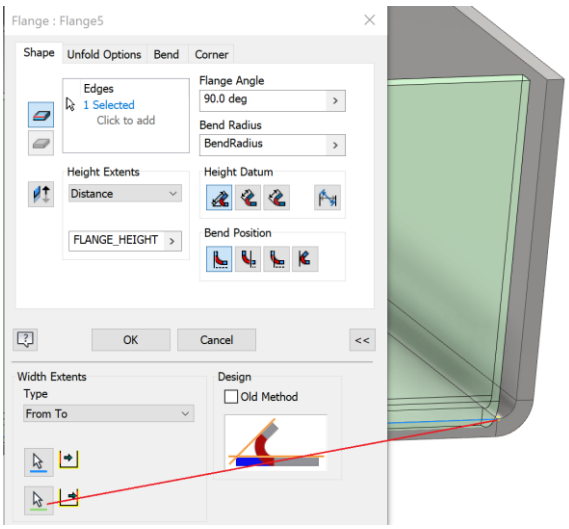
Flange:

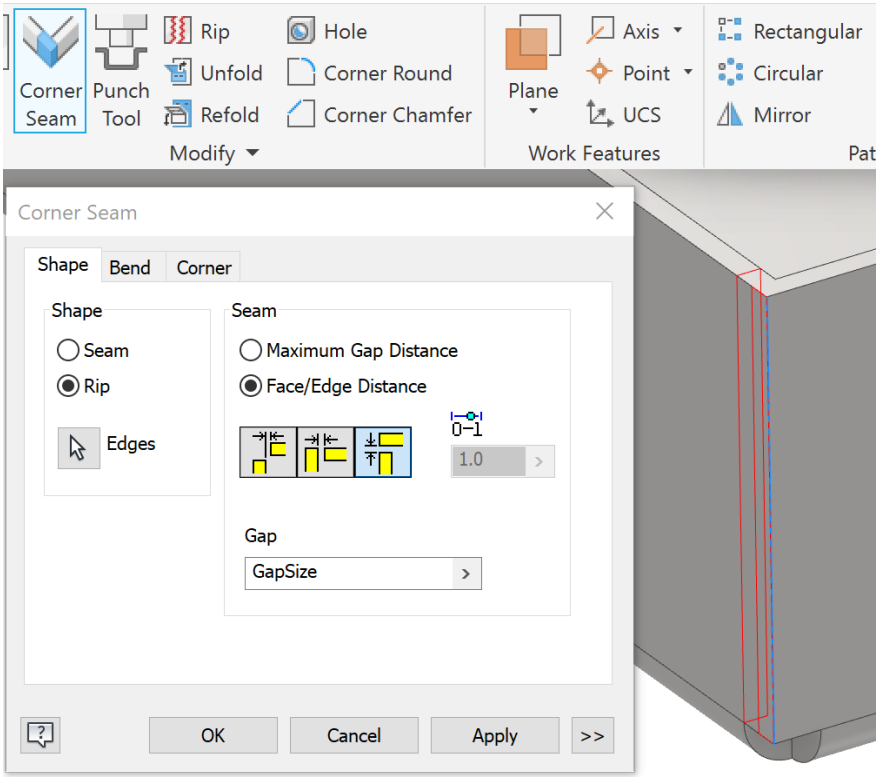


Expand the << at bottom right.
From: **XY Origin Axis**
To: the inside wall.
Keep the **Flange** height the same by using **List Parameters** to consume the same one as before.

You'll notice this combined the edge, which is not ideal. We did this on purpose to illustrate. You can either:

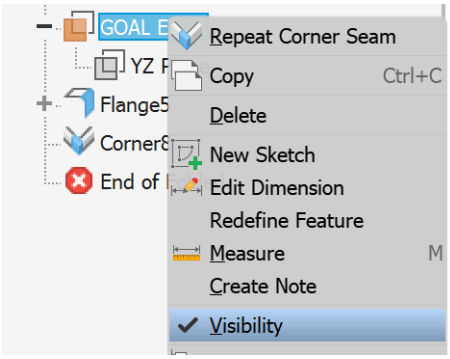
- Edit the **Flange** in the **Browser** and pick a different location for the **Green Pointer**. **Points** are able to be selected too.
- In this case, let's use the **Corner Seam** command.



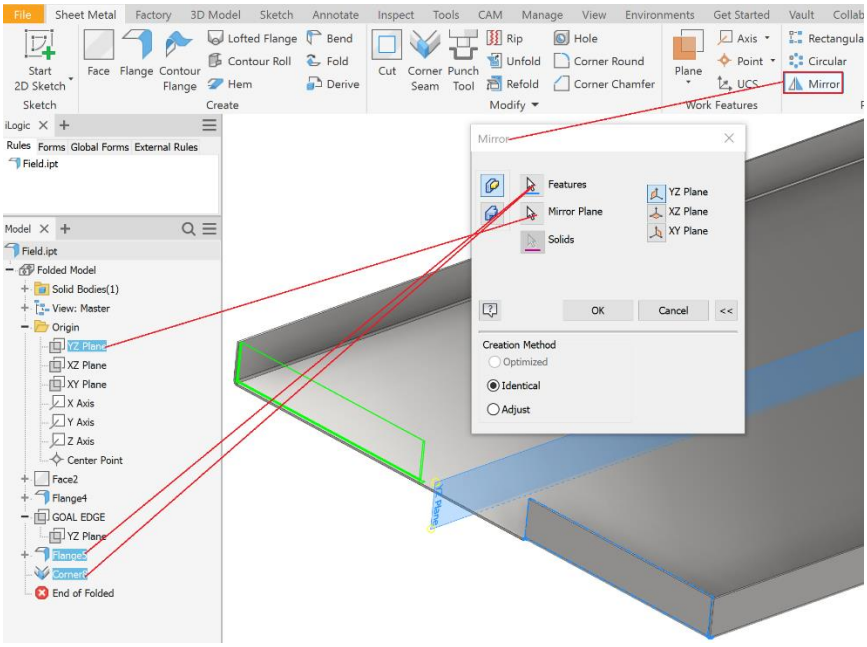


TIP: If you want the **Gap** to be smaller use **GapSize/2** and it will tighten that up. Discuss with your welders the preferred gap size for welding. This will change depending on material thickness so setup **Sheet Metal Defaults** accordingly.

Turn the **Visibility** of the **Plane** Off from the **Browser** or by RMB on the plane.

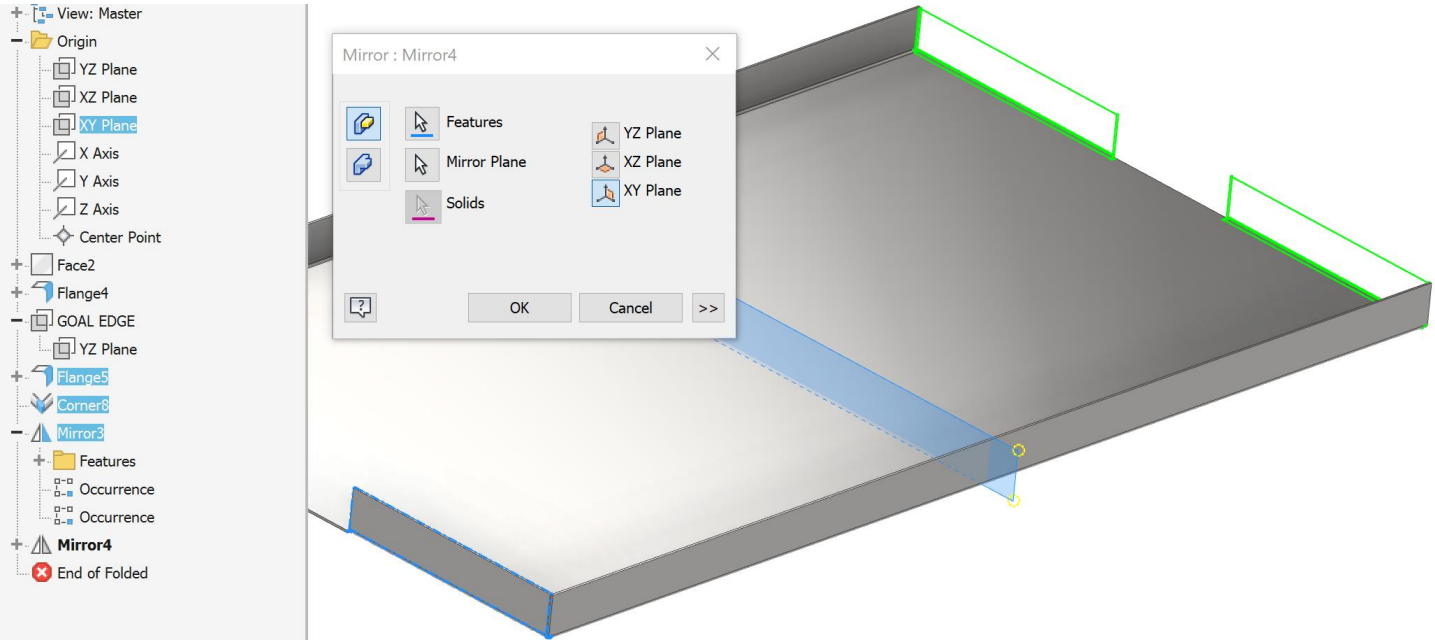


Mirror:



Select the **Feature(s)** from the **Browser** or in the window. If there are multiple things going on the **Browser** is easiest.

Repeat but this time use the **XY Plane**.

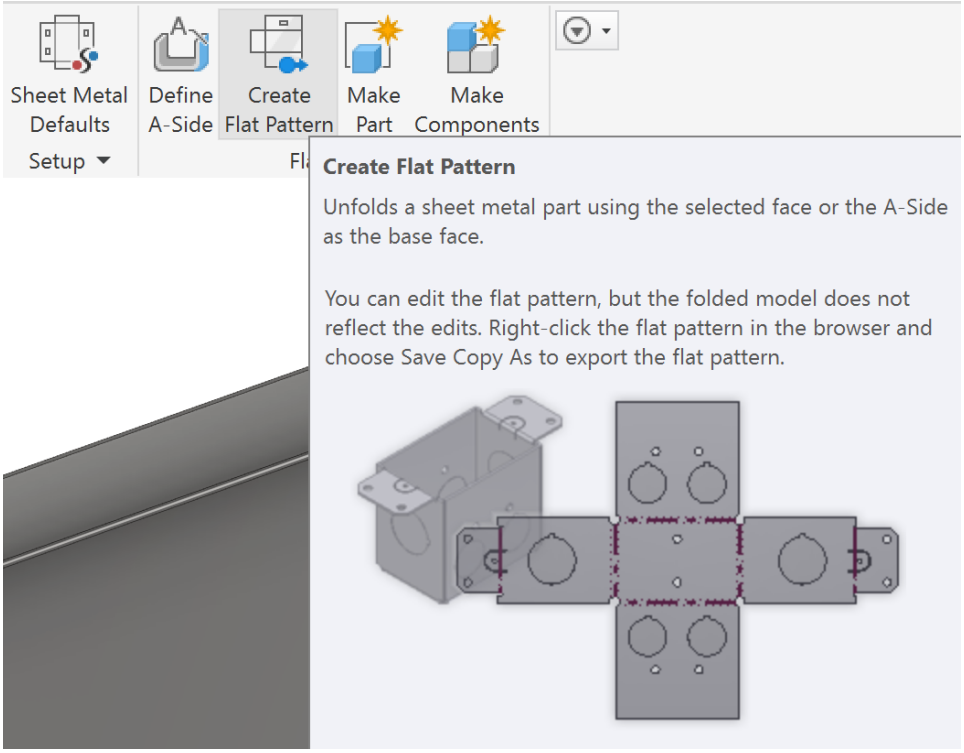


This is the nice thing about symmetric parts, mirroring is easy.

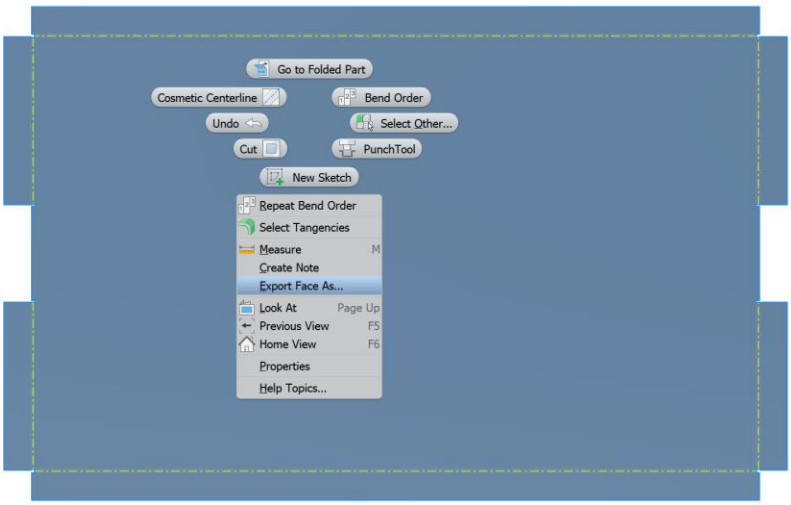
SAVE

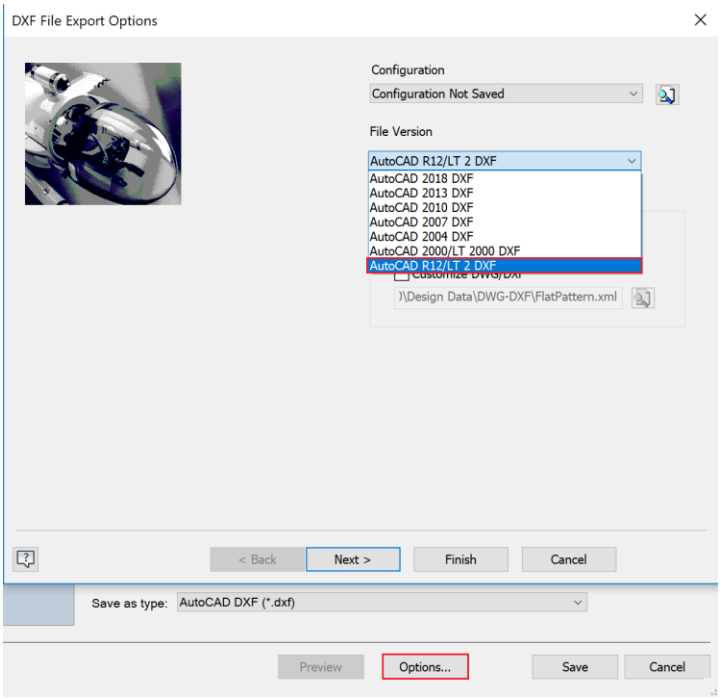
Section 3: How to create flat patterns and export cut files for post processing.

Create Flat Pattern



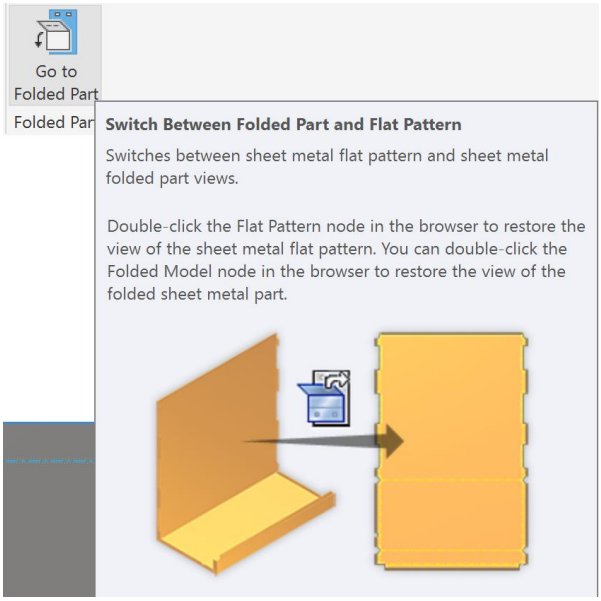
You can RMB on the face of the **Flat Pattern** and **Export Face As…** to create a **.dxf** and send for cutting. This is the easiest and fastest way to get a quick path out to the shop. If your machines are having problems reading the file, save as an older version within **Options...**

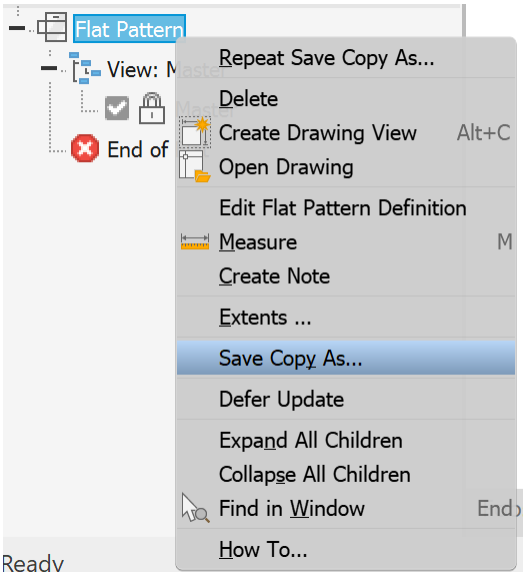




By RMB on the **Flat Pattern** in the **Browser** there are additional options to explore.

Return to the model at top right > **Go to Folded Part**



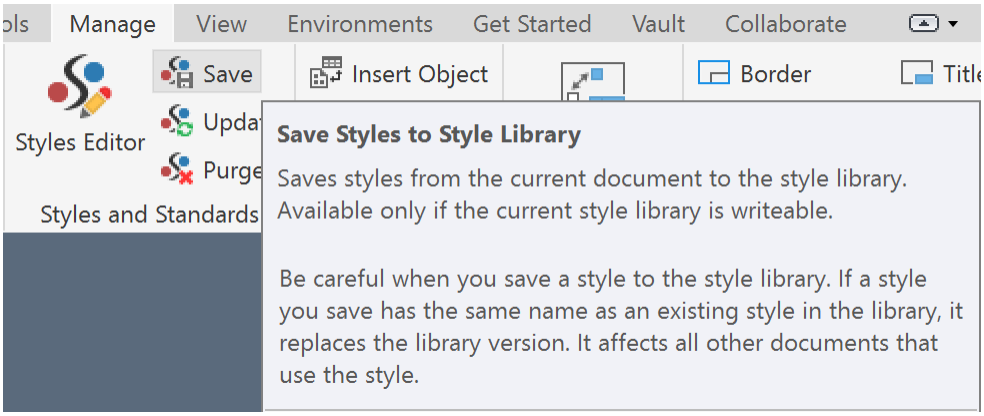


Now if we want to reference this part or use it for the **Nesting Utility** we can easily get to the **Flat Pattern**.
Section 4: How to document and dimension the part for shop fabrication.

SAVE. This is your mantra, keep saving after every successful endeavor.

Much like the **Sheet Metal Styles** we setup for the **Sheet Metal Template**, the **Drawing Template** setup is very similar. For this class we will just stick to the out of the box settings for Font Size, Dimension Style, and Border setup etc. There are other tutorials for that and should be decided what your company **Styles and Standards** are going to be.

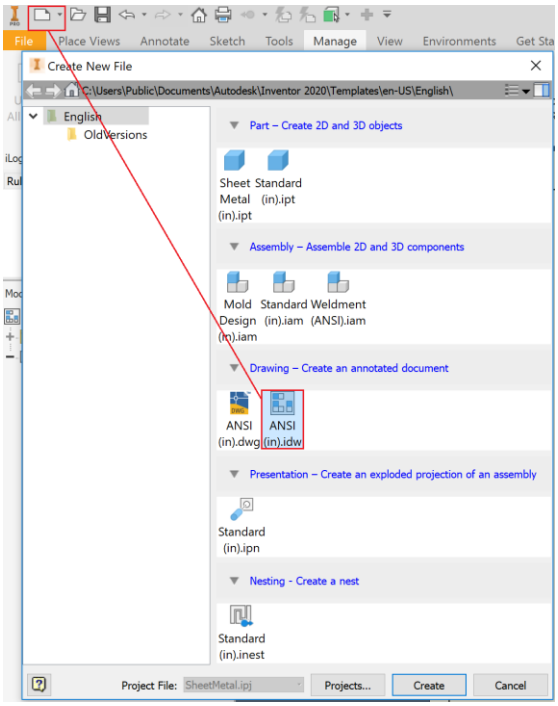
Set everything like you want through **Style Editor**, then **Save** to the **Style Library**.



After that you can **Save As** > **Save Copy As Template**.

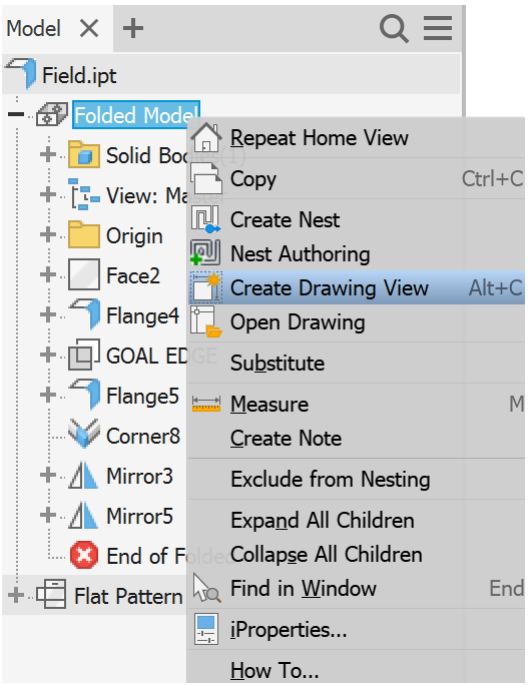
There are multiple ways to Create a New Drawings

- From the **New** icon
- **Ctrl+N**
- From the **Browser** RMB on the part icon > **Create Drawing View** which automatically loads the model and starts a **Base View** saving the step of creating one.



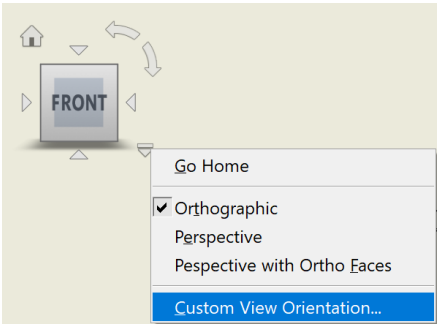
Notice that if you can also access **Create Nest** and **Nest Authoring** from here as well.

As long as the part is set to **Thickness** with a **Flat Pattern**, it will **Nest** easily.

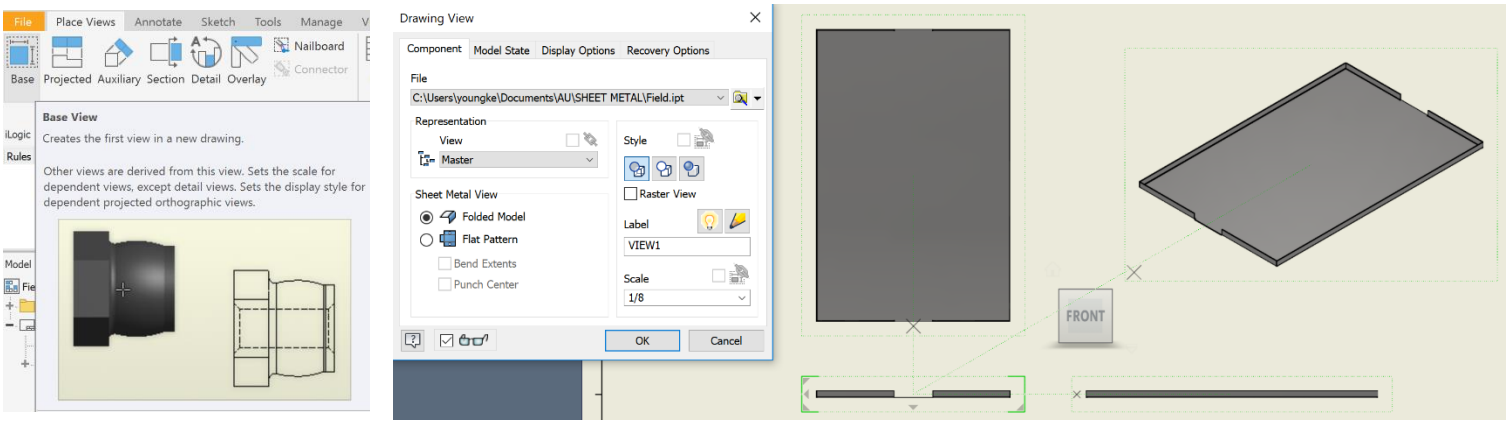


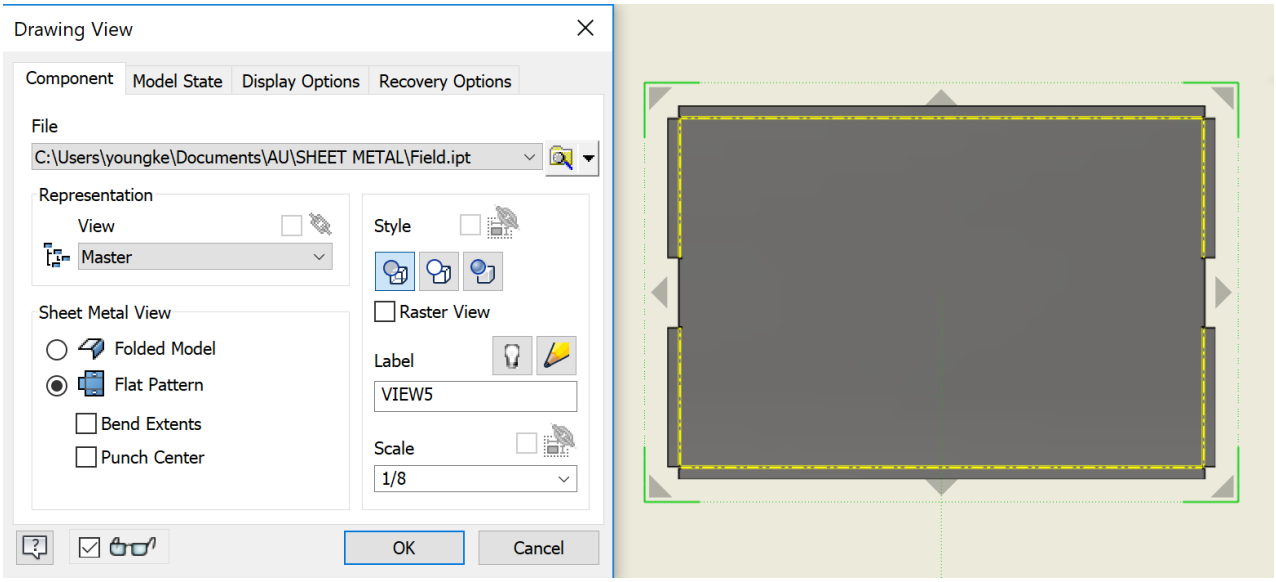
You can create as many base views as you want and have **Projected Views** showing Top, Side, Axonometric.

Pro Tip: When creating or editing a **Drawing View** if you click the little bottom right button to access **Custom View Orientation…** to orient as you like and **Return** to set that way.

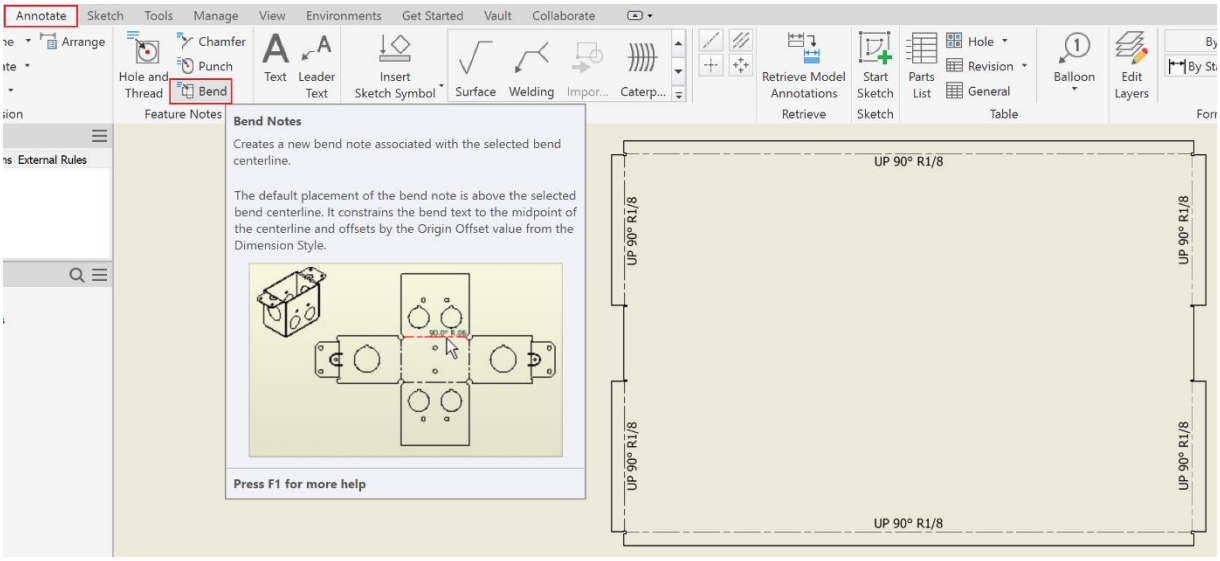


Create a new **Base View** > **Sheet Metal View** > **Flat Pattern**

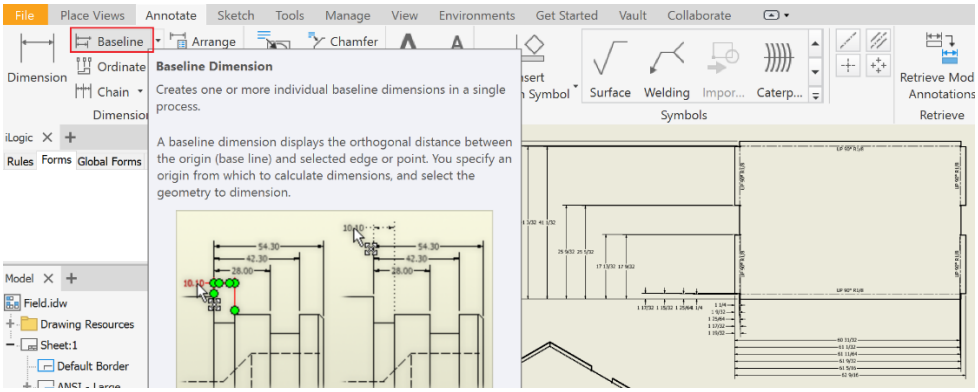


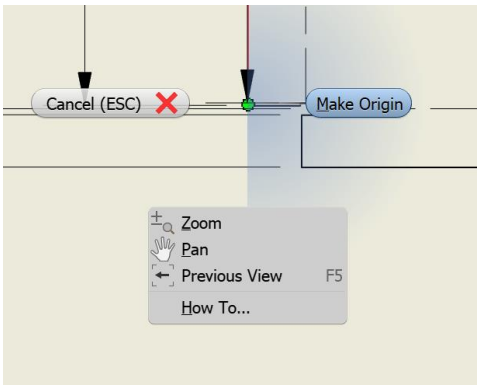


The bend lines show up as dashed, annotate the direction for the press brake with **Bend.**



Depending on your company's dimension standards, many use the bottom left corner as the Origin for inspection. The easiest way to quickly generate all dimensions is to create a Baseline Dimension.



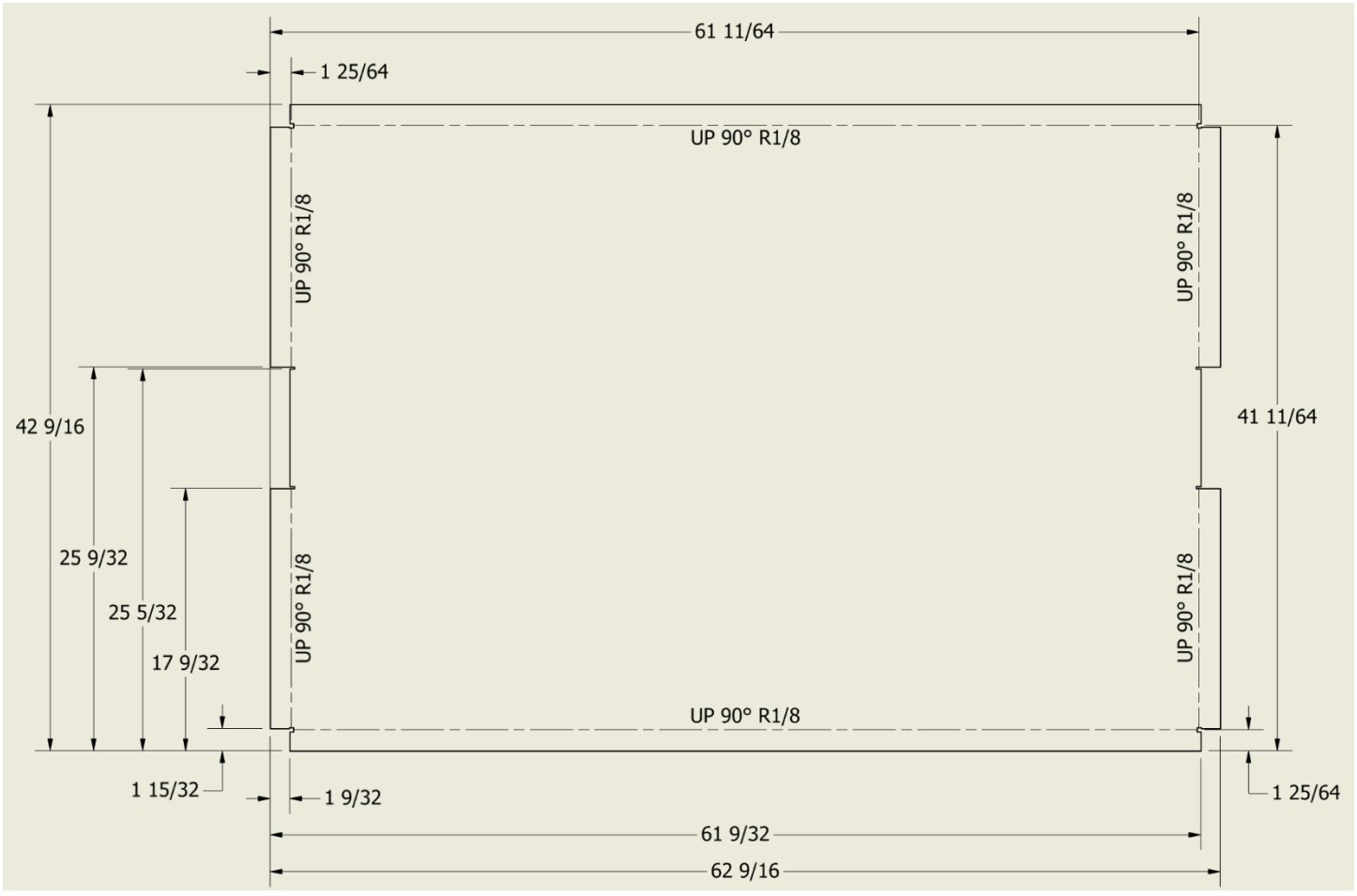


Window select the entire view > RMB > **Continue**

The available dimensions will populate in either direction you move. The origin might be in the middle of the part though instead of on the edge where we want it. To relocate zoom into the edge > RMB > **Make Origin**.

Place the dimension set by clicking, RMB > Create.

Repeat the process for the other direction. You now have dimensions going both directions but some of them you don't really want as they are just grabbing relief lines. Select these and delete them.



Select all the dimensions on a side and **Arrange**. It should look something like this after you tinker a bit.

- I like putting the bend dimensions on their own sides for ease of visualization and error proofing.

Pro Tip: You can add text and link Model and User Parameters in-line with text **Add Parameters**

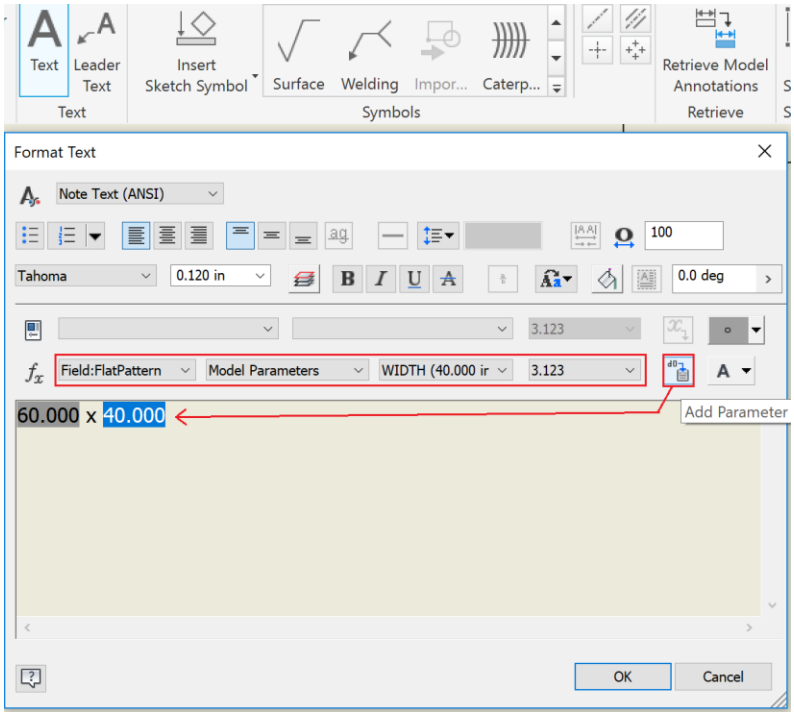
This is helpful for adding detail notes that reference important dimensions. You can also add it to **Leader Text** for easy callouts.

SAVE

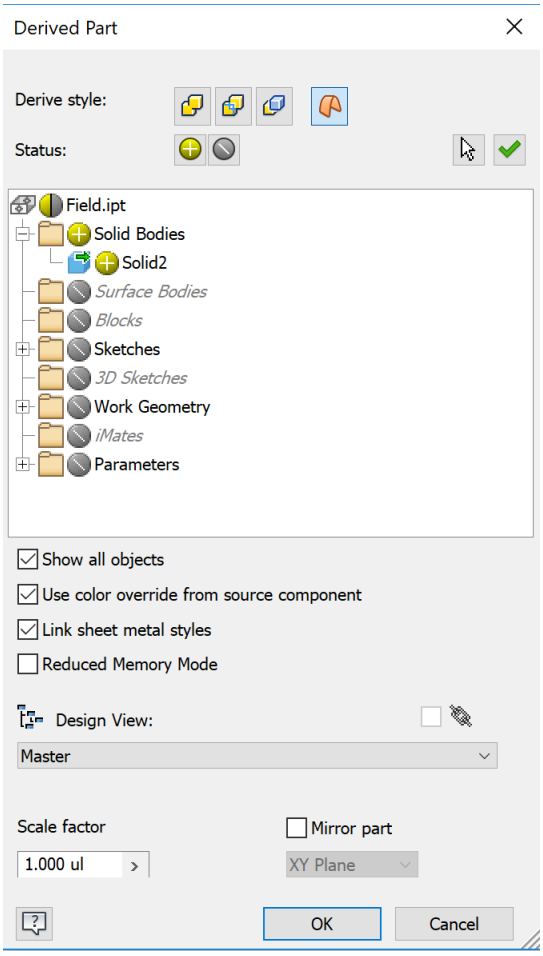
File > **Export** > **PDF**

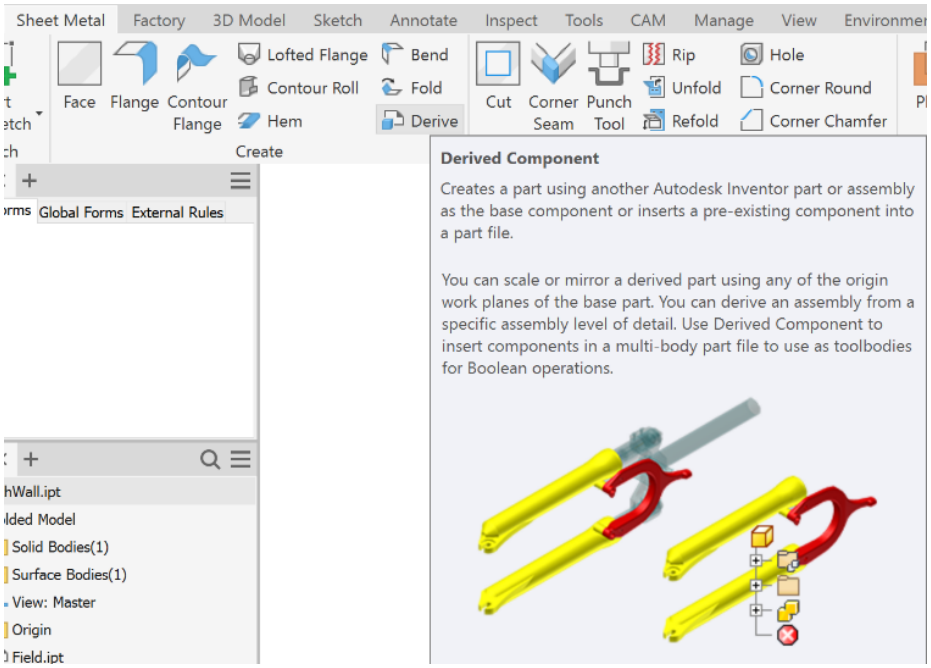
With these skills you can now create a Sheet Metal Part, generate a Flat Pattern, export a .dxf cutting path, and document for others to fabricate and then inspect.

Time for some advanced skills and helpful tricks.



Create New > **Sheet Metal Part**.





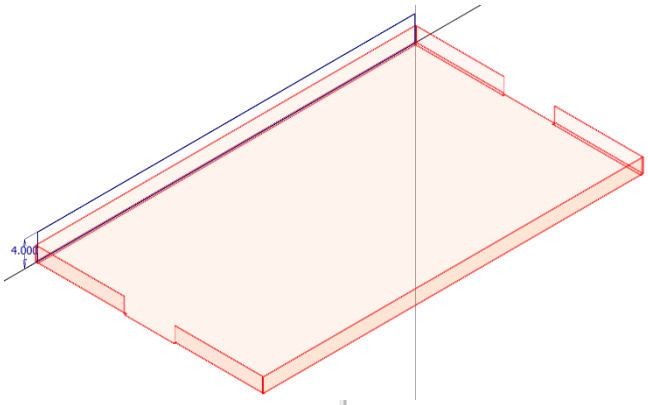
Derive

Select **Body as Work Surface** to use as a reference within this part, you will see it transparent in the preview.

There isn't anything else we need at this time, but you can expand and see what other things you can include.

Start 2D Sketch on the side surface wall.

Project Geometry of the edge and **Constrain** the rectangle to the ends.

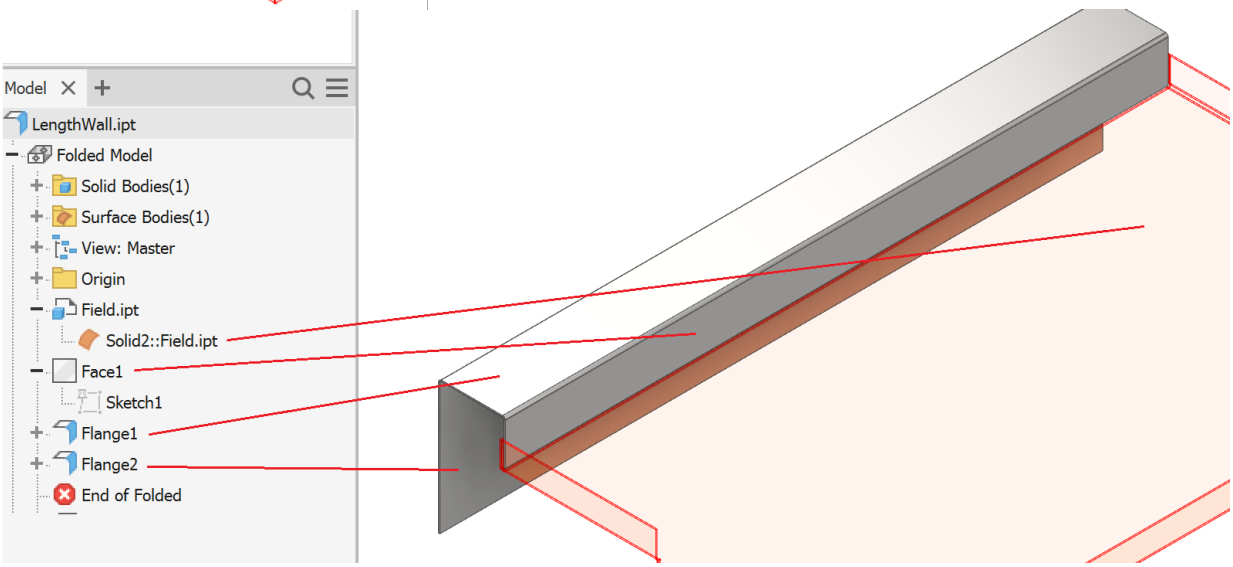


Set the height to **INSIDE_WALL_HEIGHT=4**

Finish Sketch.

Face > Should select the geometry of the rectangle.

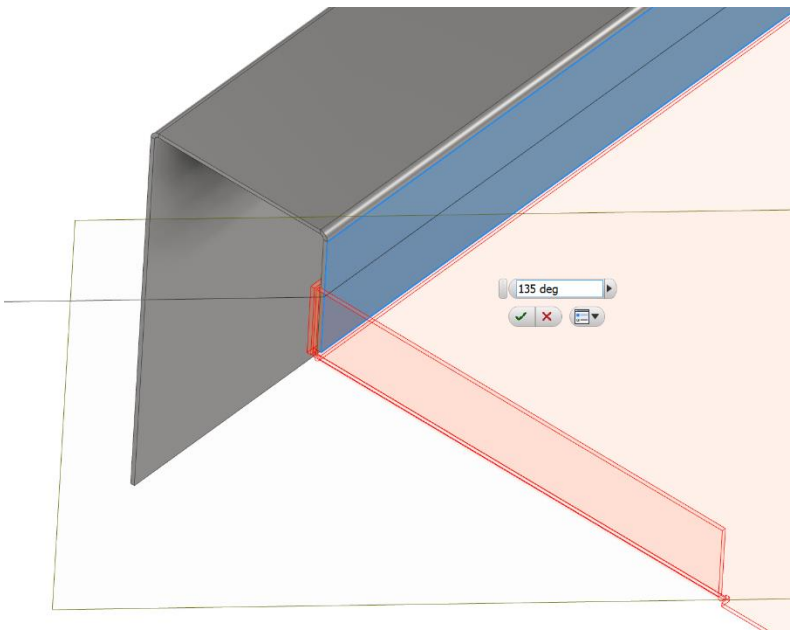
Flange: Make sure that your press break operator's die can fit within the bends otherwise it is impossible!



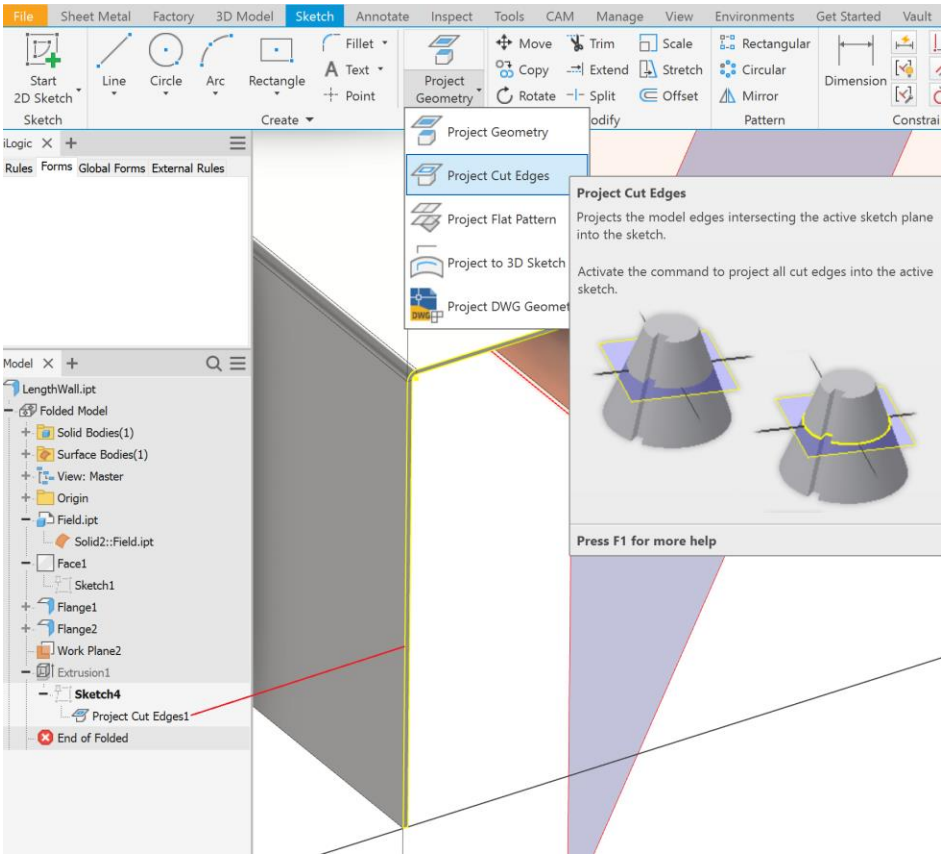
Flange1: **TOP_WIDTH=4**
Flange2: **OUTSIDE_WALL_HEIGHT=12**

Work Plane:

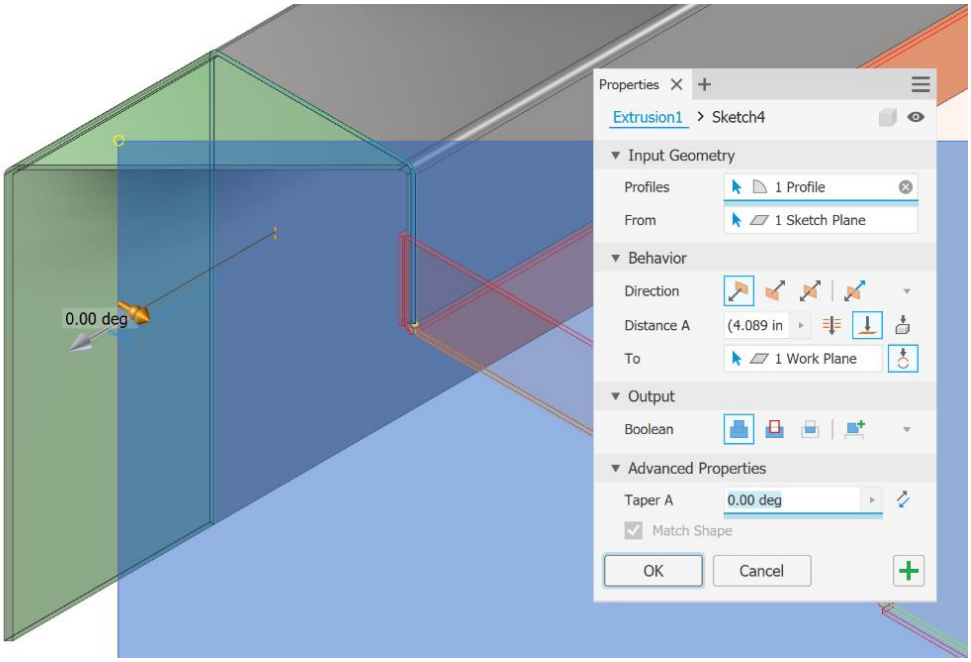
Select the inside edge and face, set to **ANGLE=135 deg**



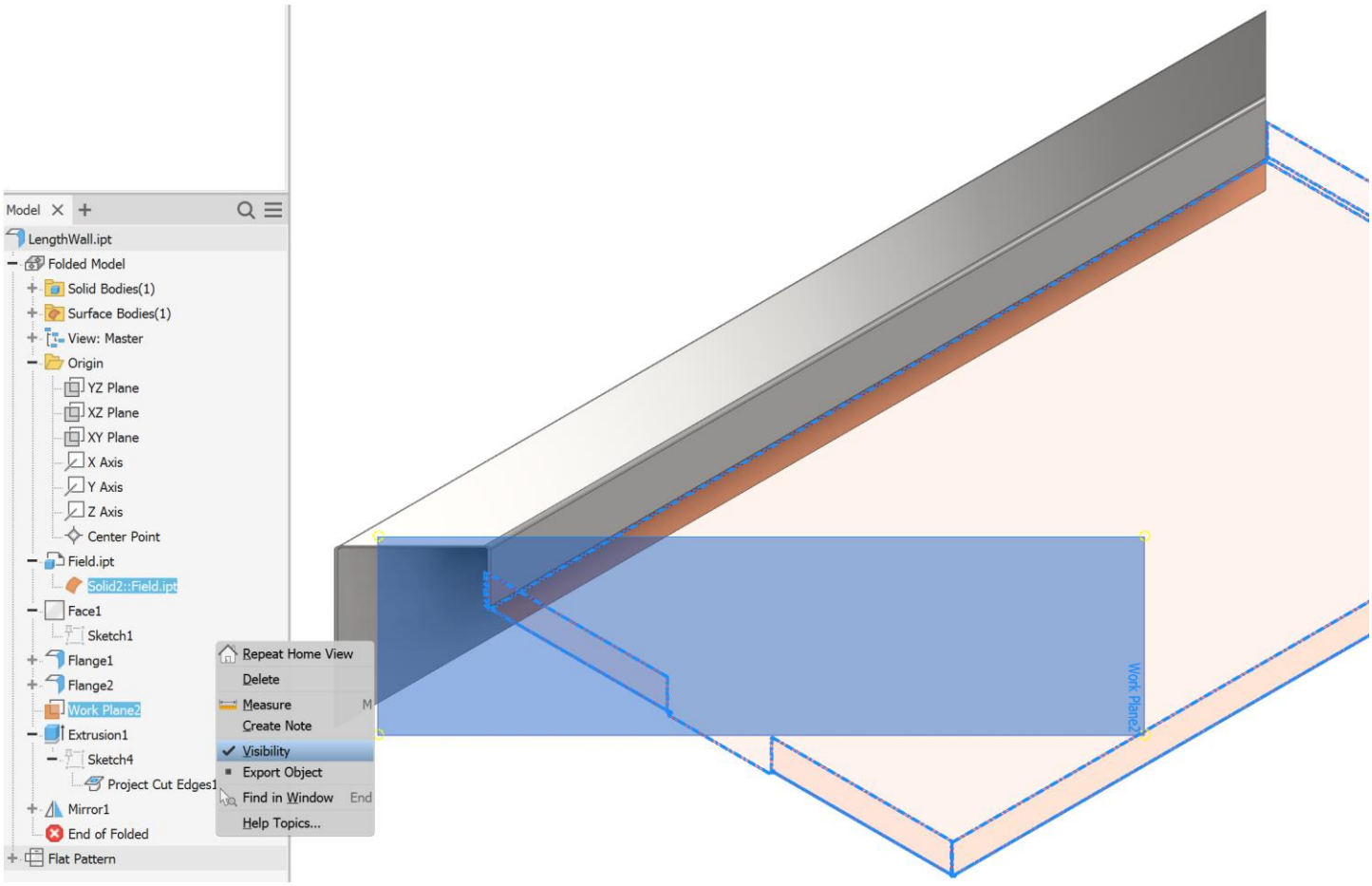
3D Model > **Start 2D Sketch** > **Project Cut Edges** > **Finish Sketch**



Extrude to the **Plane**.

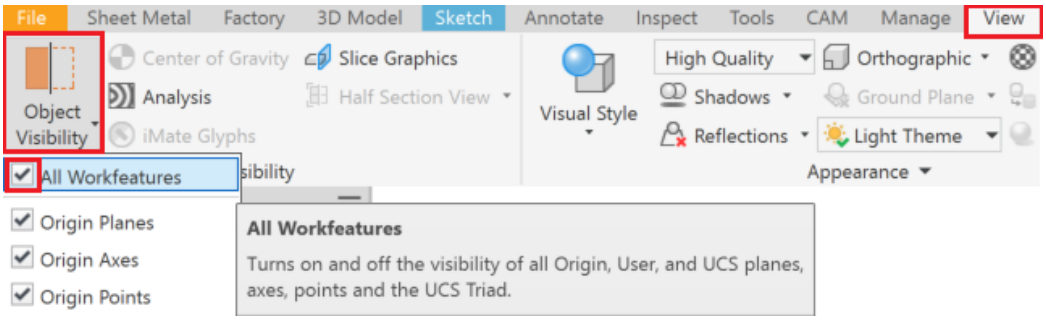


Mirror across the **XY Origin Plane**.



Turn off the **Visibility** of the **Work Plane** and **Derived Surface**.

If you are in an **Assembly** it is sometimes helpful to turn **Surfaces** off that you forgot about:



Notice the **Half Section View**, it is helpful when you need to get in somewhere tight. Don't forget to turn if off when you're done though, otherwise you can trip yourself up thinking it is a graphical error.

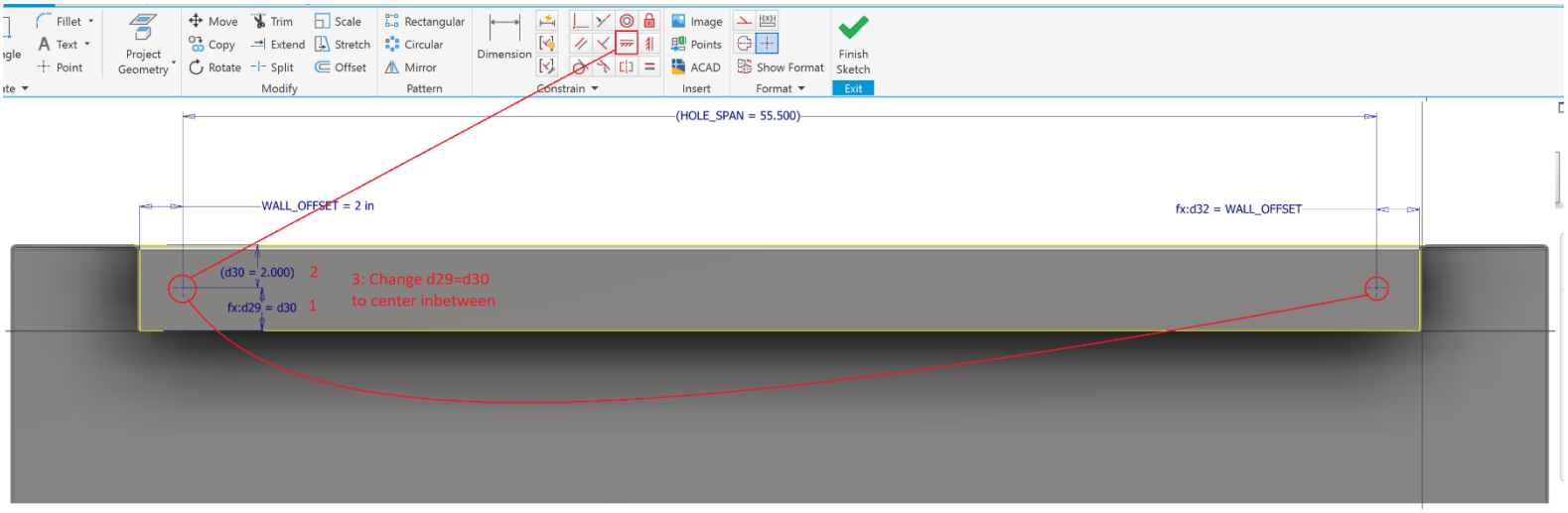
SAVE – Keep saving more often, get in the habit.

Time to add some holes, but we want it to be easily changeable for the future and able to stretch parametrically when the size changes.

3D Model > **Start 2D Sketch** on the outer wall face.

Project Geometry of the outer walls, lower edge, and upper edge.

- You might have to rotate the model slightly by holding Shift + Center Roll Button.



Create a **Point** on each side, approximately placed.

Dimension to the left wall and name **WALL_OFFSET=2**.

Dimension to the lower edge, the value does not matter as long as it is in the approximate placement.

Dimension to the upper edge, it will come in as a **Driven Dimension** (or you get a dialogue pop up).

Edit the lower edge dimension. Select the upper edge dimension. This will make them equal and centered.

Use **Horizontal Constraint** between the two **Points**.

Dimension the right **Point** to the right wall. Make the value **WALL_OFFSET** or click the left wall dimension.

Dimension between the **Points**.

- We will access **Parameters** later to rename the **Driven Dimension** to something helpful.

Finish Sketch.

Create **Hole**.

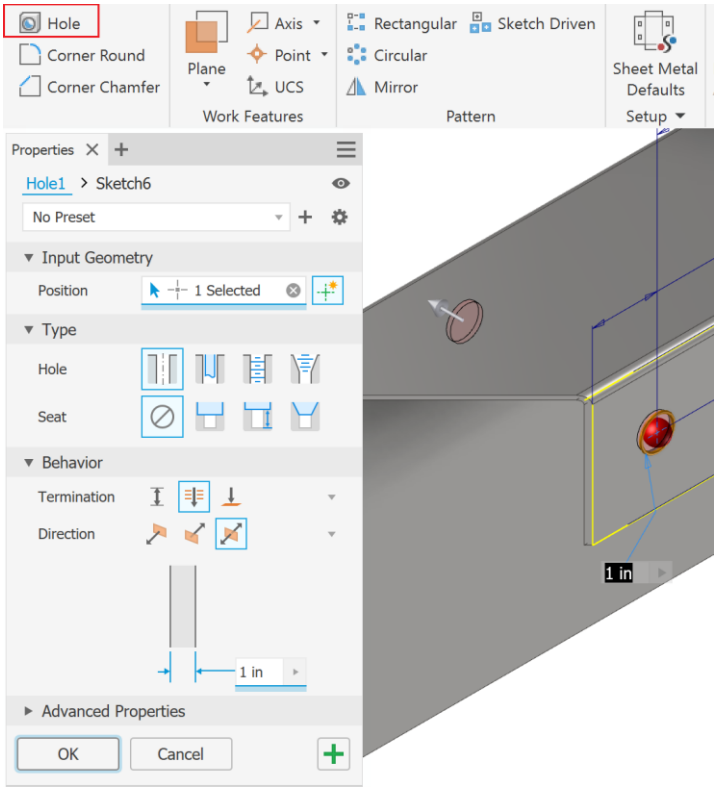
It will automatically select all of the **Points** visible.

Deselect the Right Hole by holding **Ctrl**.

Explore the **Options**, there are many. You can now save presets you use often in the top right **+** menu.

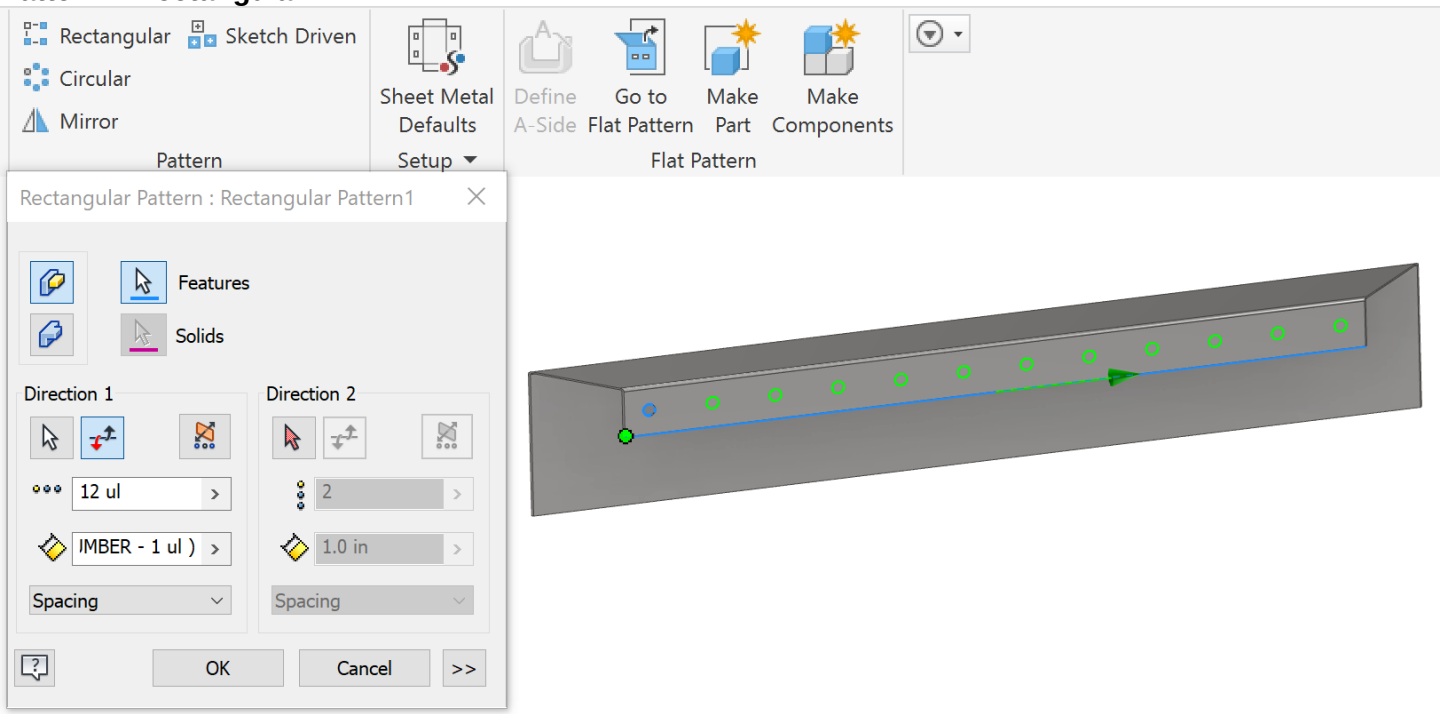
Type: **Through All**.

OK. SAVE



Access **Parameters** and rename the driven dimension (from between the points) to **HOLE_SPAN**.

Pattern > **Rectangular**



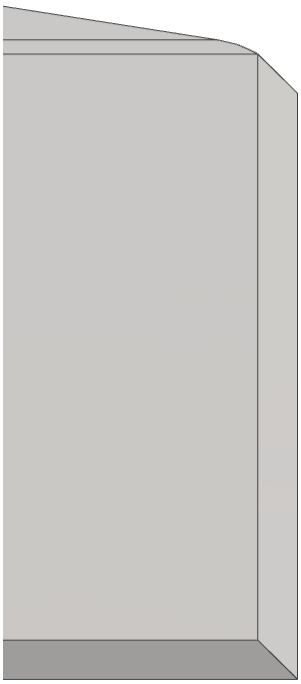
Set the Column Count to **RAIL_NUMBER=12**
Set the Column Spacing to **RAIL_COMPUTE=HOLE_SPAN / (11)**

Edit the **Rectangular Pattern** from the Browser.

Change the Column Spacing value to **HOLE_SPAN / (RAIL_NUMBER - 1)**

- We had to define the RAIL_NUMBER parameter before we can use it in the spacing equation.

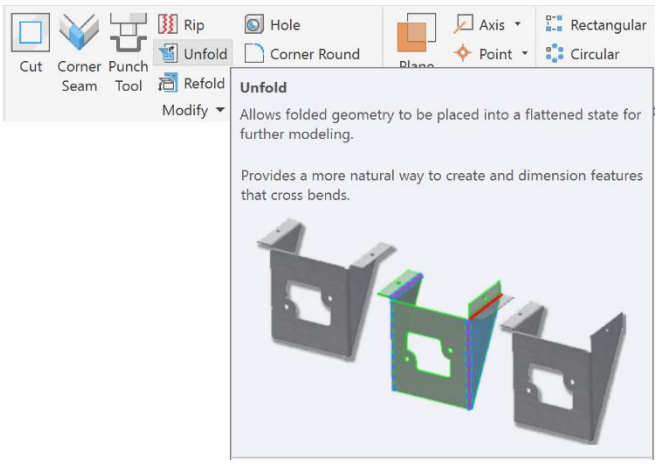
Now any time the length of the part changes, the holes will remain evenly spaced!



Create Flat Pattern.

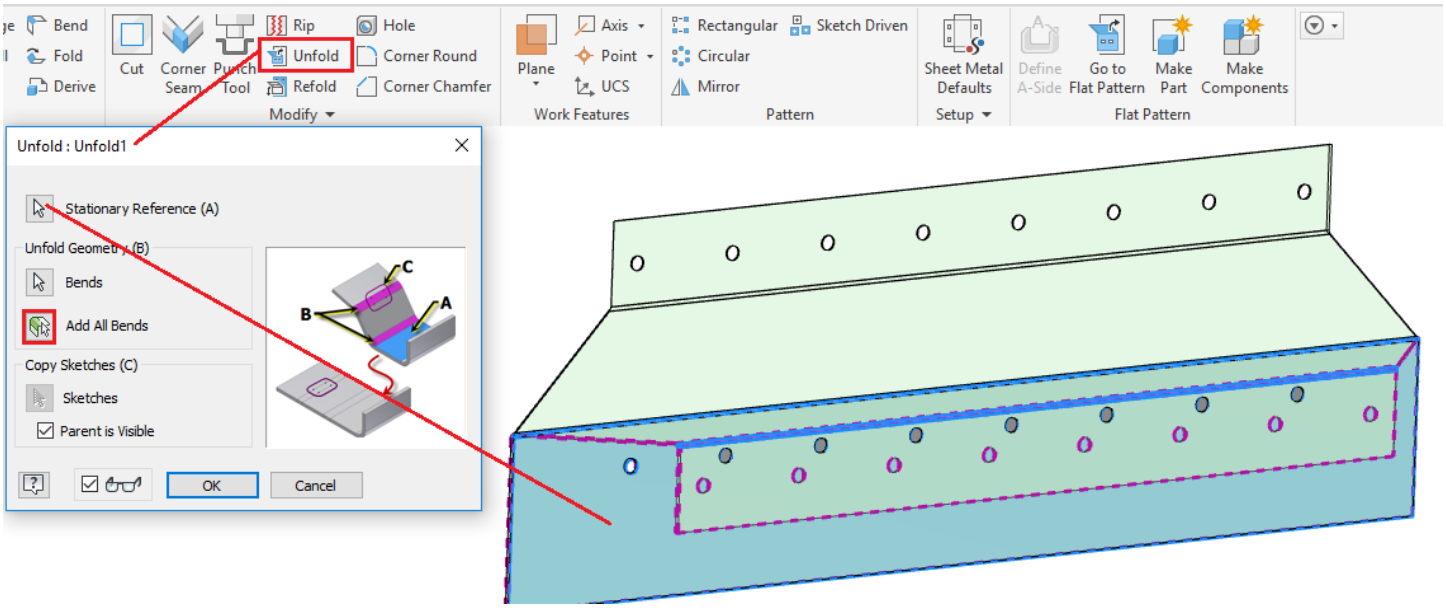
Turn the ViewCube to Bottom and you will notice that there is an edge that is at an angle!

We have illustrated this for things to beware of. You can easily make square, straight flanges all day, but when you want to get advanced shapes, you will run into this issue. Better to learn how to fix it now.



Return to the modeling environment by
Go To Folded Part.

Modify > **Unfold**



Start 2D Sketch on the flat face.

Use **Project Geometry** to highlight the angled edges on one side. Zoom in to ensure you get all the little remnants.

Finish Sketch.

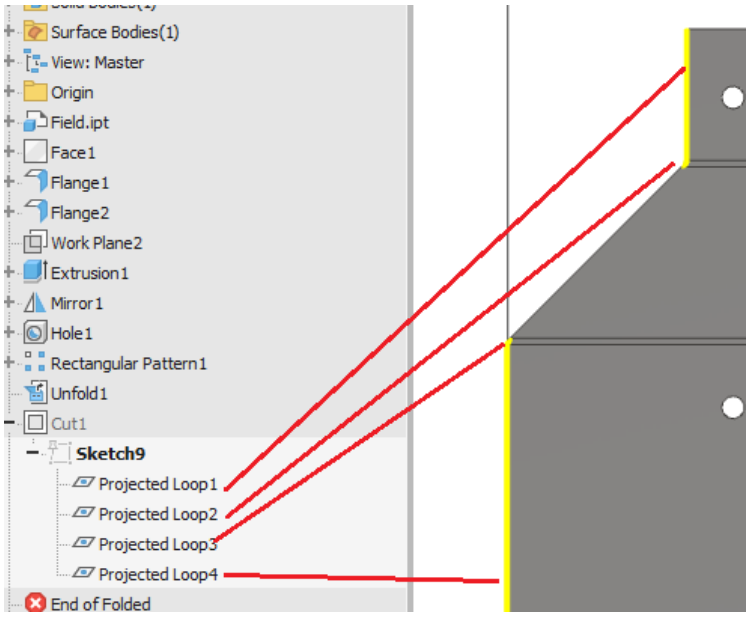
Cut at distance of **Thickness**.

Mirror the feature across **XY Origin Plane**.

Refold.

Select the same face for Stationary Reference and **Add All Bends** again.

Now if we inspect the **Flat Pattern** there are no angled edges!



We can safely **Export Face As…** to get our flat pattern.

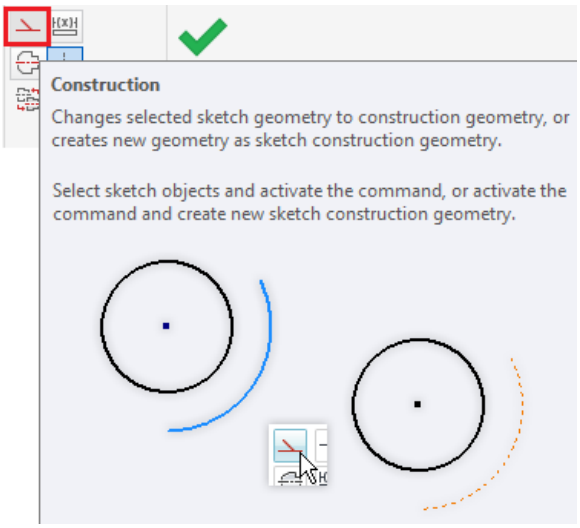
This is an example of using clean up techniques after you get the basic shape.

File > **Open** > **LengthWall-2.ipt**

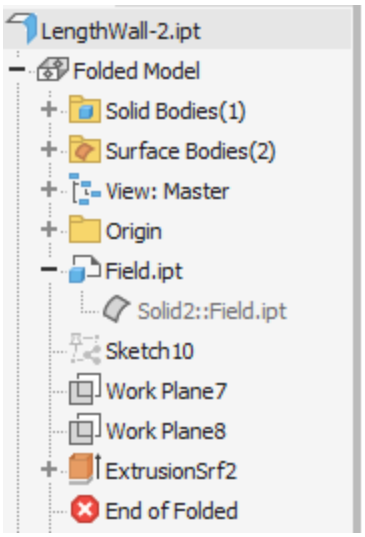
Edit Sketch10 to inspect a different method of creating the **Flange**.

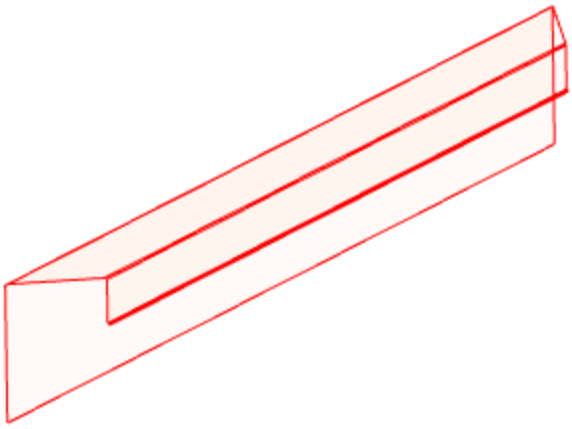
The profile was drawn and **Offset** at **Thickness** depth as a **Construction Line**. The corners are set to **BendRadius**.

There are the same angled **Work Planes** that we **Extruded To** before, but this time we will create as a **Surface**.



If the Field Surface **Visibility** is turned **Off**, we now have the Flange but as a thin **Surface Body**.



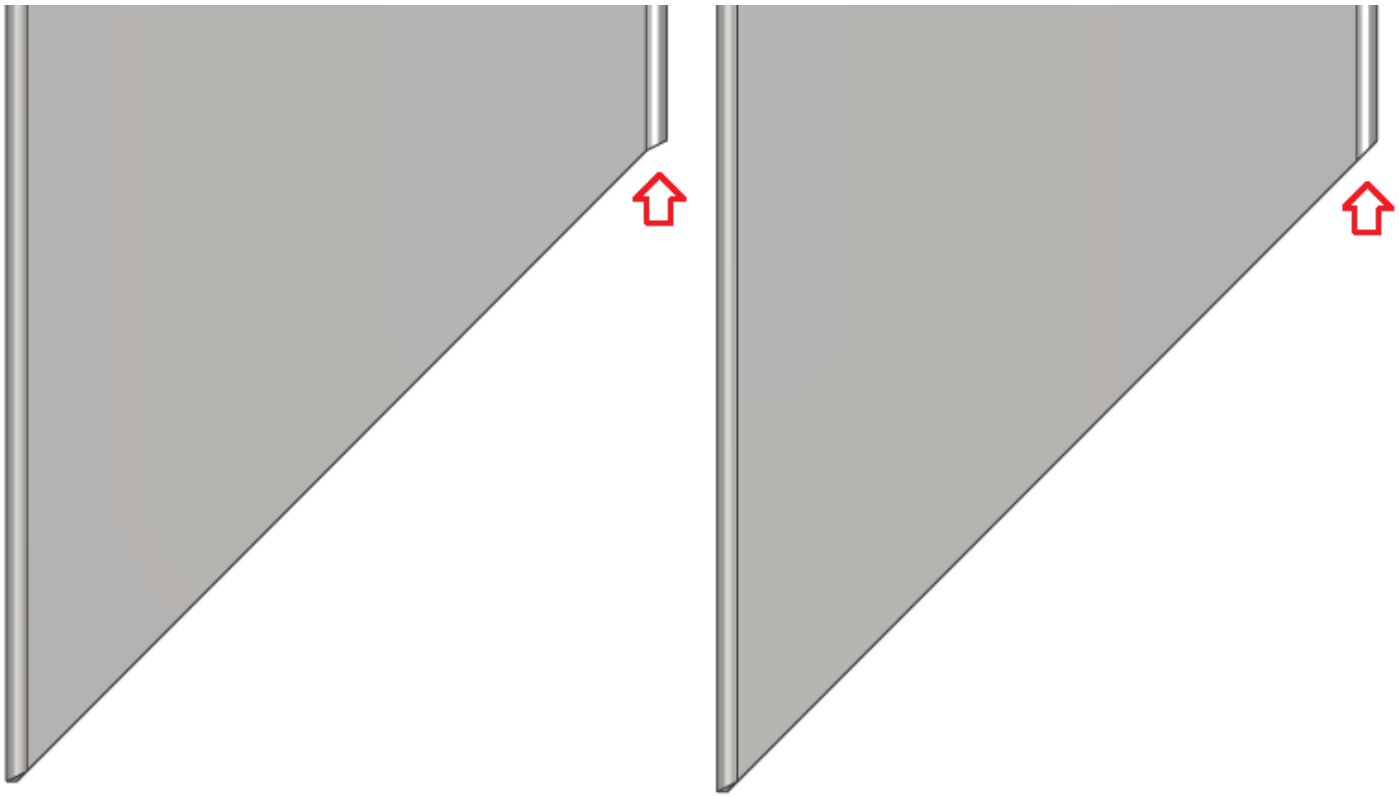


3D Model > Modify > **Thicken/Offset**

Select the **Surface Body** with **Quilt** at distance of **Thickness**.

It creates a perfect Flat Pattern that does not need to be trimmed or edited!

The shape is a little different than our other part though. These are the things that you will have to decide what you can live with, what is important, and what you can fudge to get a good end result.



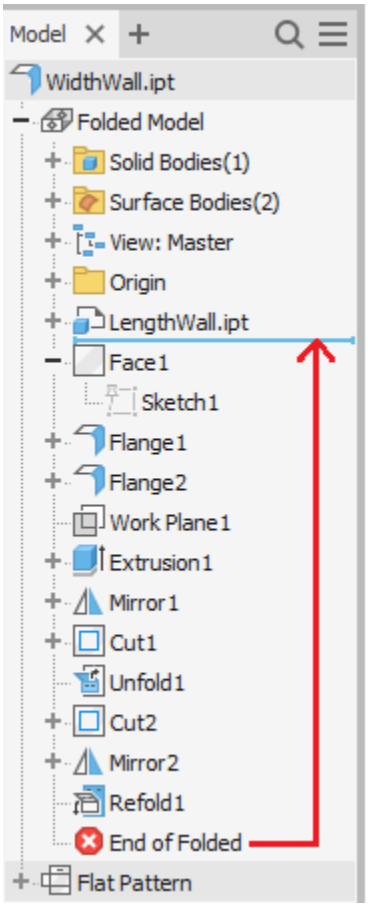
Pro Tip: Using the **Surface Thicken** at **Thickness Parameter** will get you out of a tough situation when you can't use the **Sheet Metal Tools** to do what you want.

The main problem people run into is when their **Material Thickness** does not match up to the **Sheet Metal Defaults Thickness** and the **Flat Pattern** will not compute.

For the shorter side, we will **Open WidthWall.ipt** and inspect the **Browser Tree** to see how it was built.

Getting familiar with inspecting parts will help you understand how things are designed.

Drag the **End of Folded** (typically shown as **End of Part** in non-Sheet Metal Parts) up to the top of the tree below the first Feature (LenghtWall.ipt Derive).



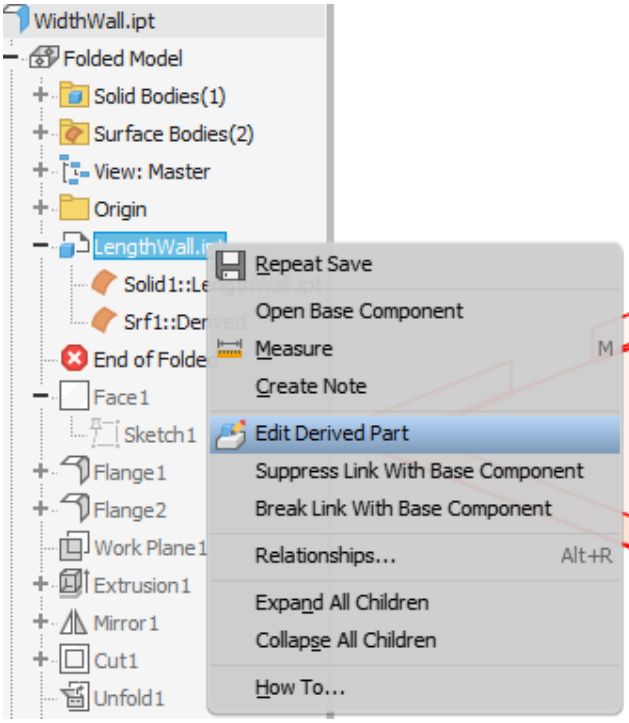
Edit Derived Part.

Take a look at what was brought in as references, dimensions, bodies, etc.

Drag the **End of Folded** down one step.

Inspect.

Repeat and you will notice it is the same workflow we did for the longer side.

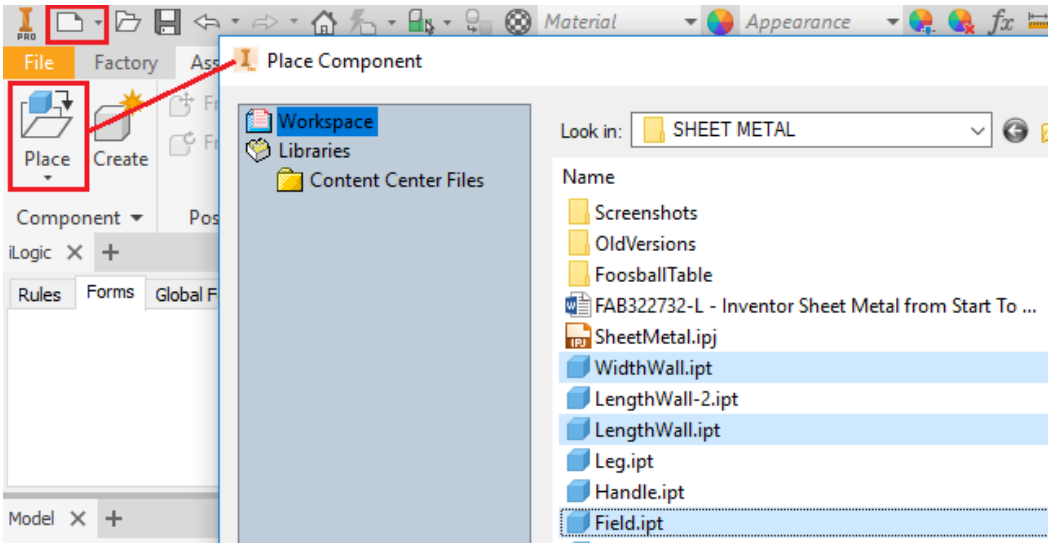


There is no need for the rail holes, but we do need to create a ball return.

Now we have the **Field.ipt**, **LenghtWall.ipt**, & **WidthWall.ipt**. Let's bring them together in a **Assembly.iam** from the top left **New** menu.

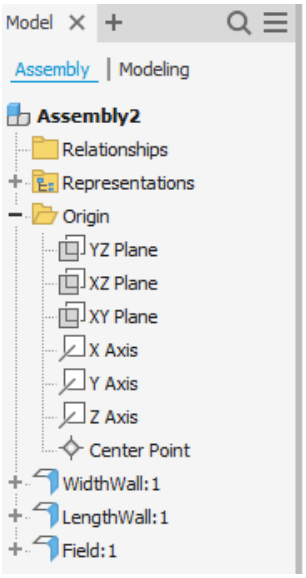
For this exercise we will be using the premade parts for ease of use.

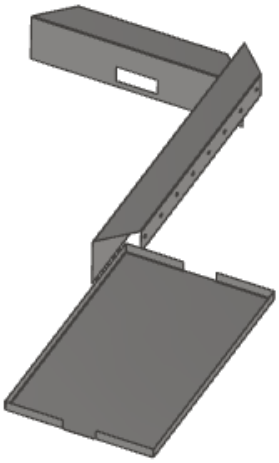
Component > **Place** > **Ctrl Select multiple files at once** > **OK**



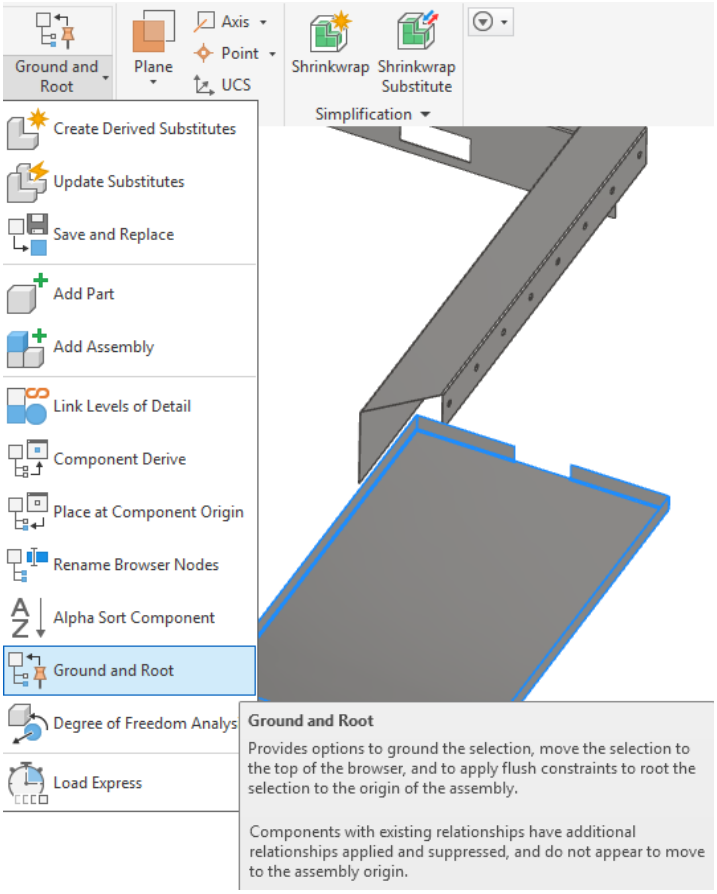
Click anywhere on the screen to place one set > **RMB** > **OK**

You should have something that looks like this:

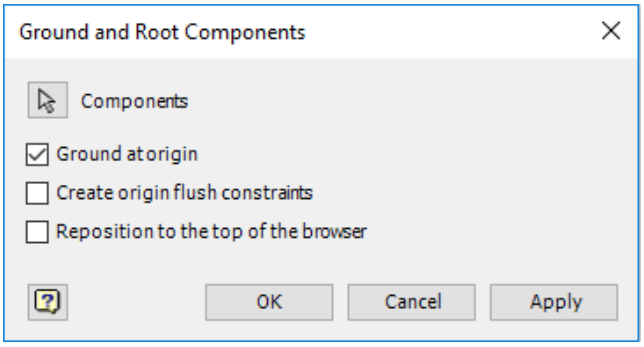




Select the **Field.ipt** from the screen or **Browser Tree**.

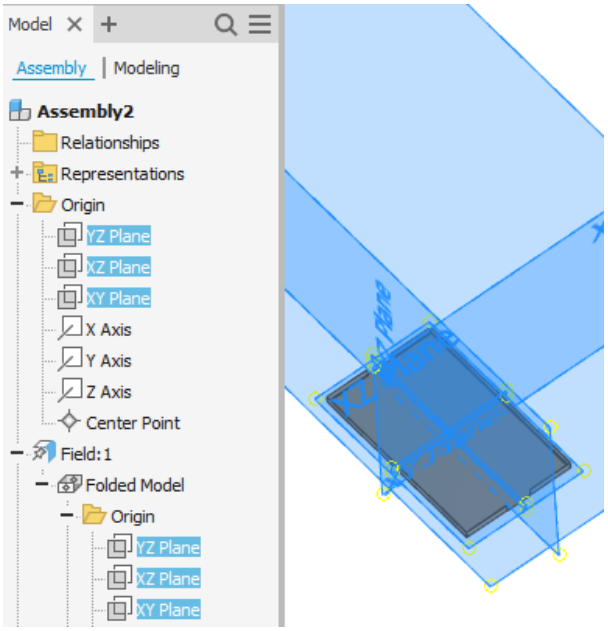


Assemble > Productivity > **Ground and Root**

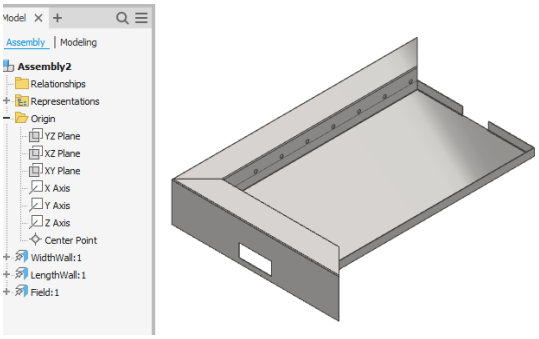


There is an option for Create origin flush constraints, but we will leave that off, just know it is an option.

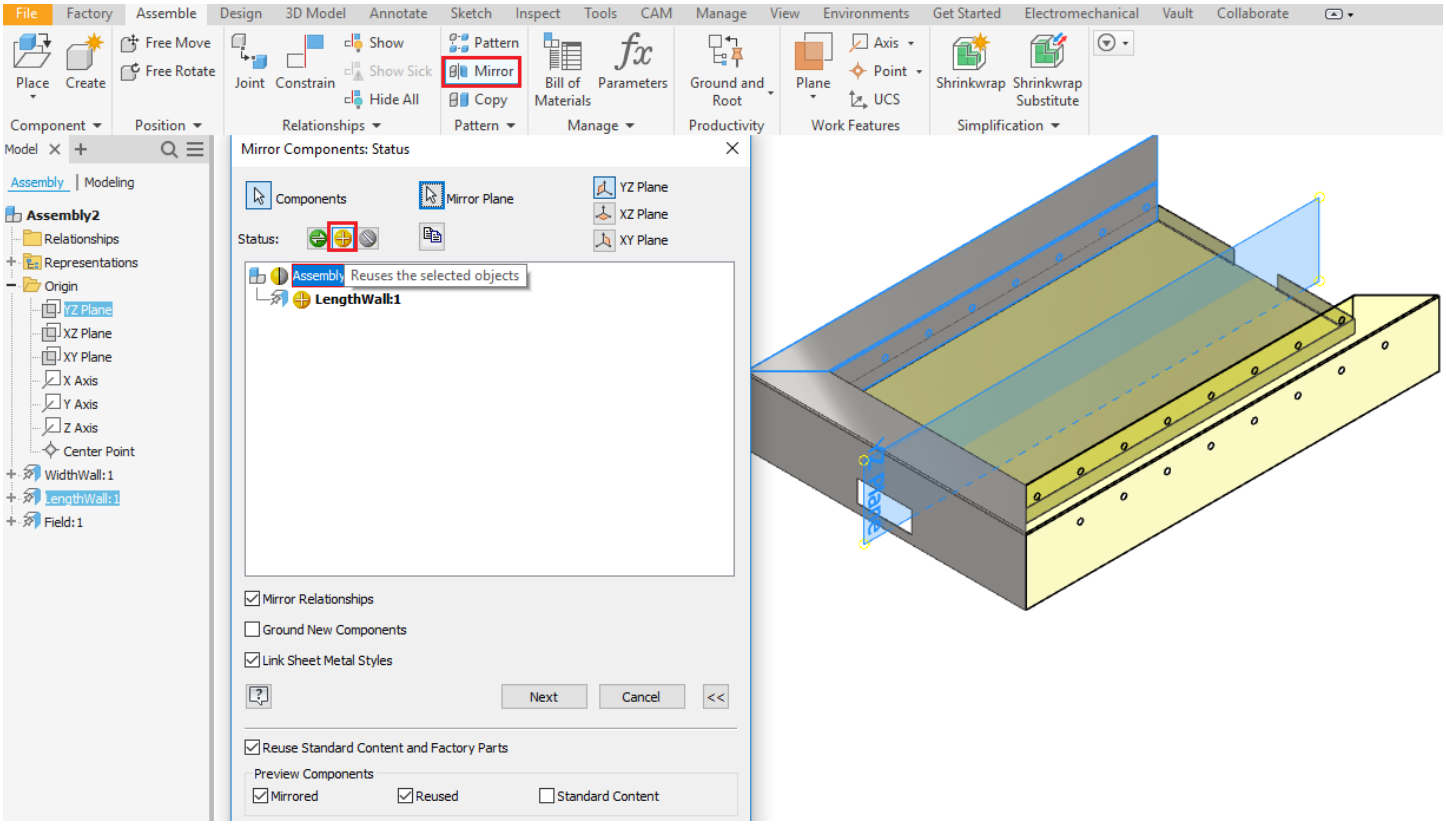
Boom – it locked it down to the origin, which if you check the **Assembly Origin Planes**, should be in the middle matching up with the **Part Origin Planes**.



Do the same thing with the other two parts. They should match up in the correct places, no constraints needed!



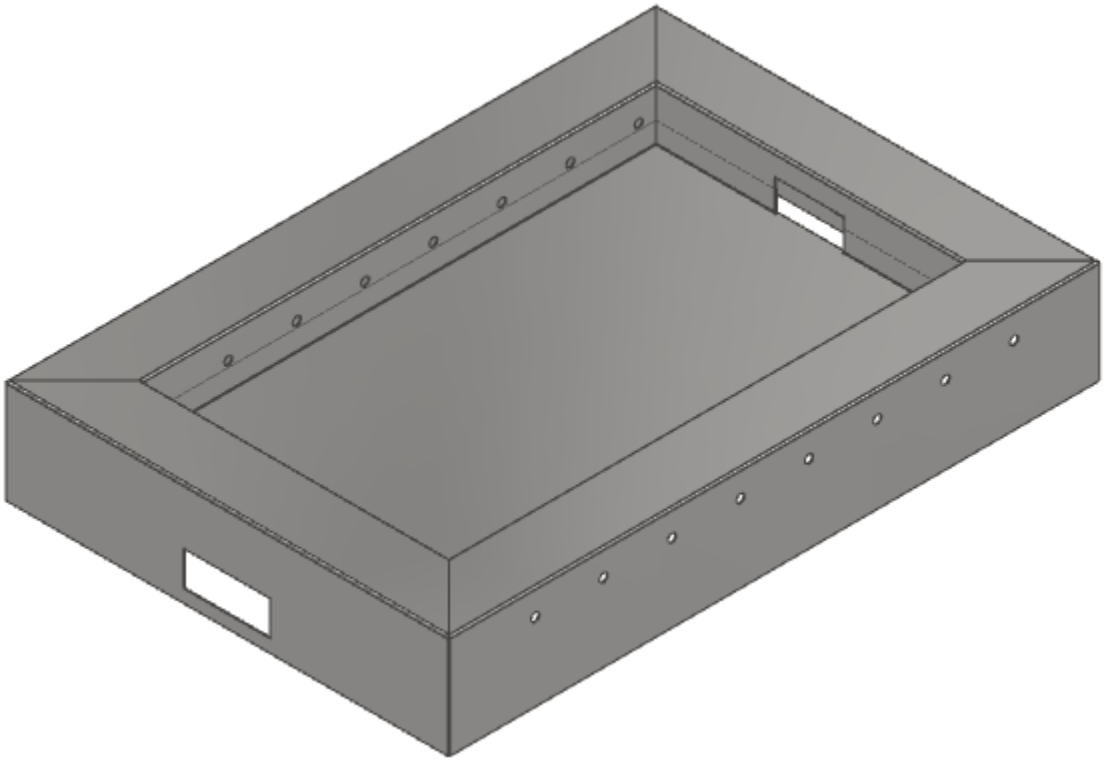
Assemble > Pattern > **Mirror**



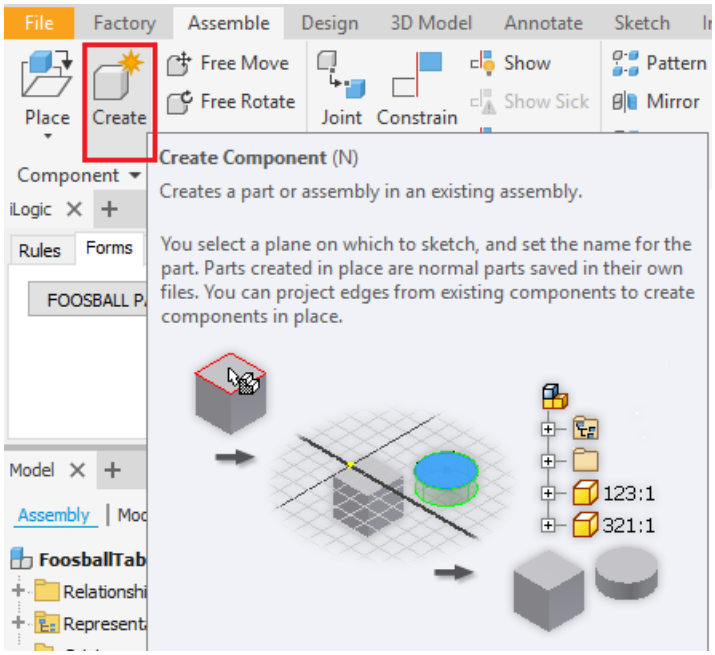
Select **LengthWall** and highlight **Reuse**. If you select the green button it will create a new part instead.

Use the **Assembly YZ Origin Plane**, not the Part. **Next** > **OK**.

Repeat for **WidthWall** along the **Assembly XY Origin Plane**.



Open > **FoosballTable.iam**



There are **Legs**, **BallReturn**, and **Braces**. Each of these parts uses techniques that you can inspect and figure out what is going on by dragging the End of Folded up from the Browser Tree.

You'll notice the Leg and Brace have **Adaptive** symbol with the red and blue arrows is On. The parts were created from inside the assembly environment and are directly linked to the geometry without using **Derive**.

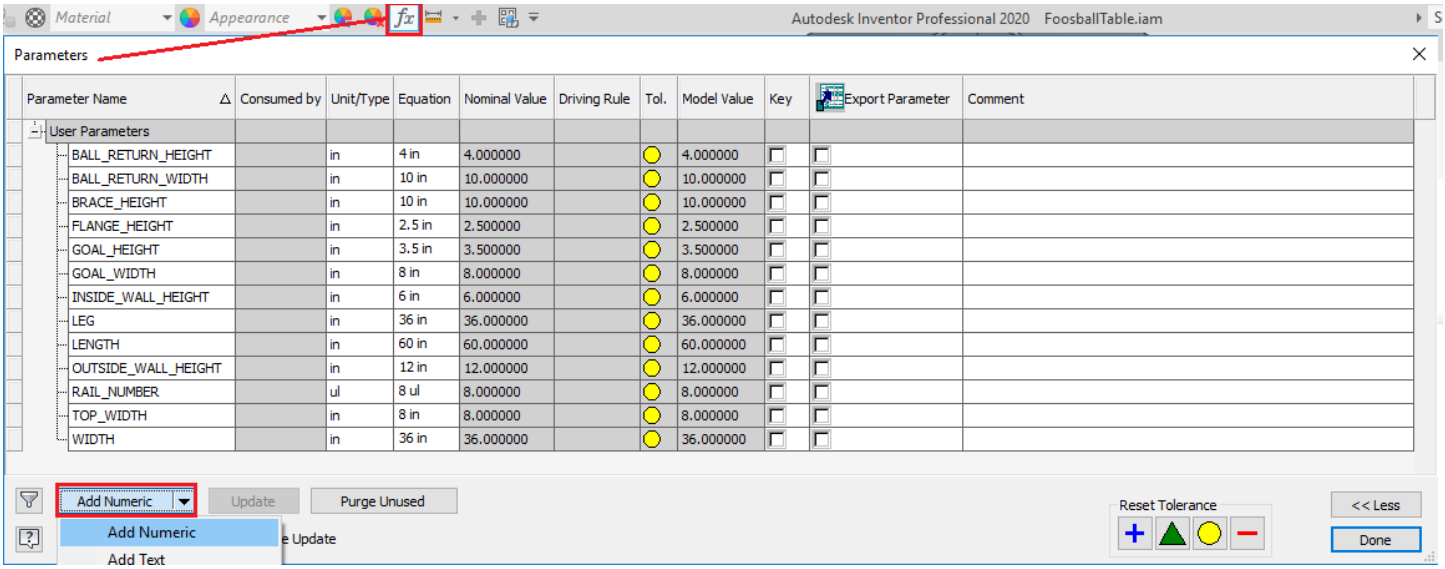
This has advantages of being fast and easy, but can easily become disjointed if the model geometry changes too much.

Personally, I use **Adaptive Parts** to define the shape of the model and then **turn Off** and **break any links**. This makes it not parametric anymore but saves headaches down the road.

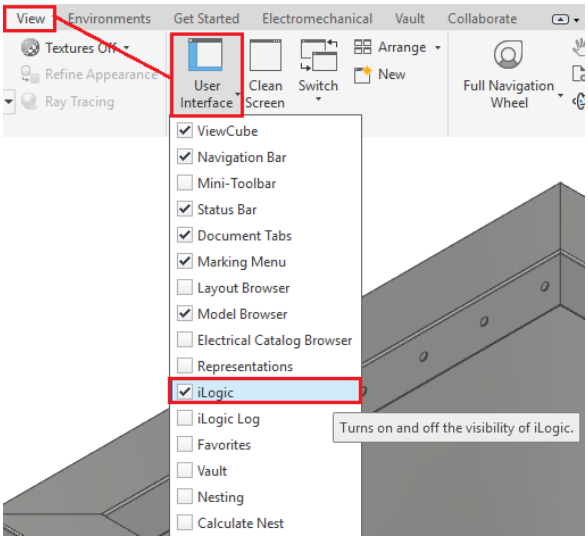
Pro Tip: Become more familiar with **iLogic**. Inspect all models at [Collection of iLogic Examples For Beginners.]

This is advanced use of Inventor, but I wanted to show you how to access and be able to look at the code and see what it is doing. This is where you want to get to someday so set your goals high.

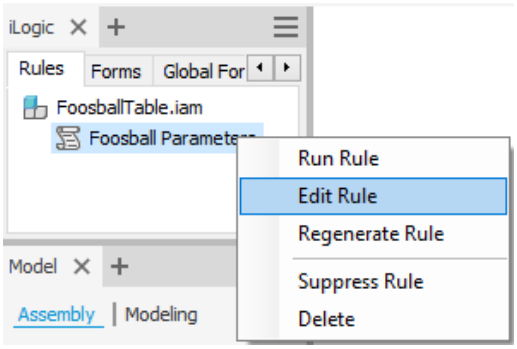
The first thing is to access the **Parameters** of the **Assembly** and create **User Parameters**.



View > **User Interface** > **iLogic**

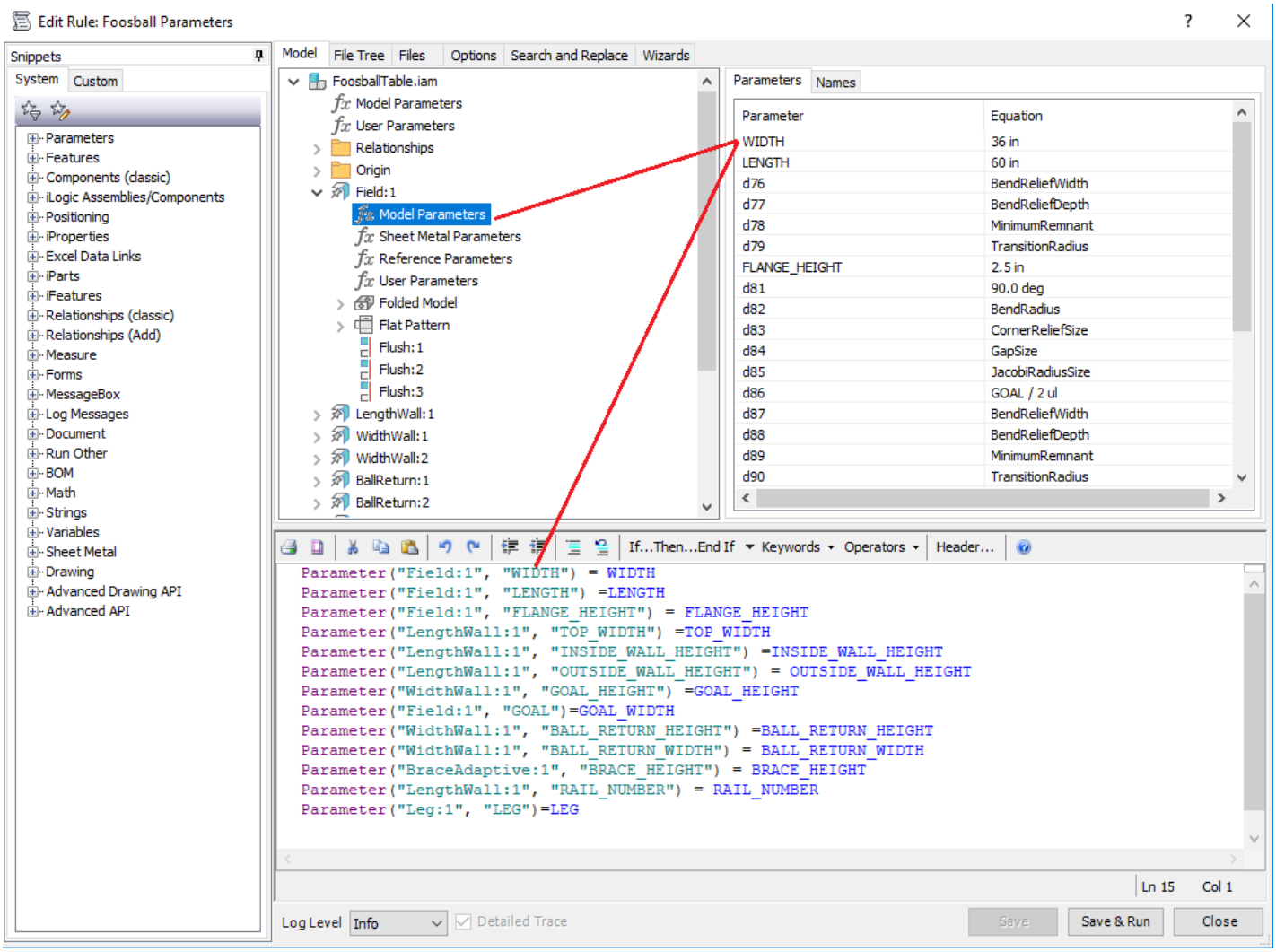


There will be a list of all the Parts in the Assembly along with the Parameters, Constraints, and other useful info.



If you select ModelParameters and double click on a Paramter at right it will populate the code below.

By simply putting =Width (which is a defined User Parameter) it will link the Part and you can drive it from the top level Assembly.

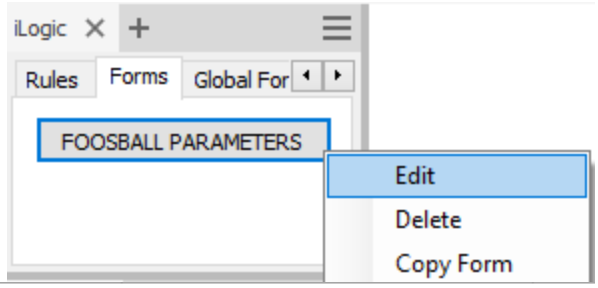


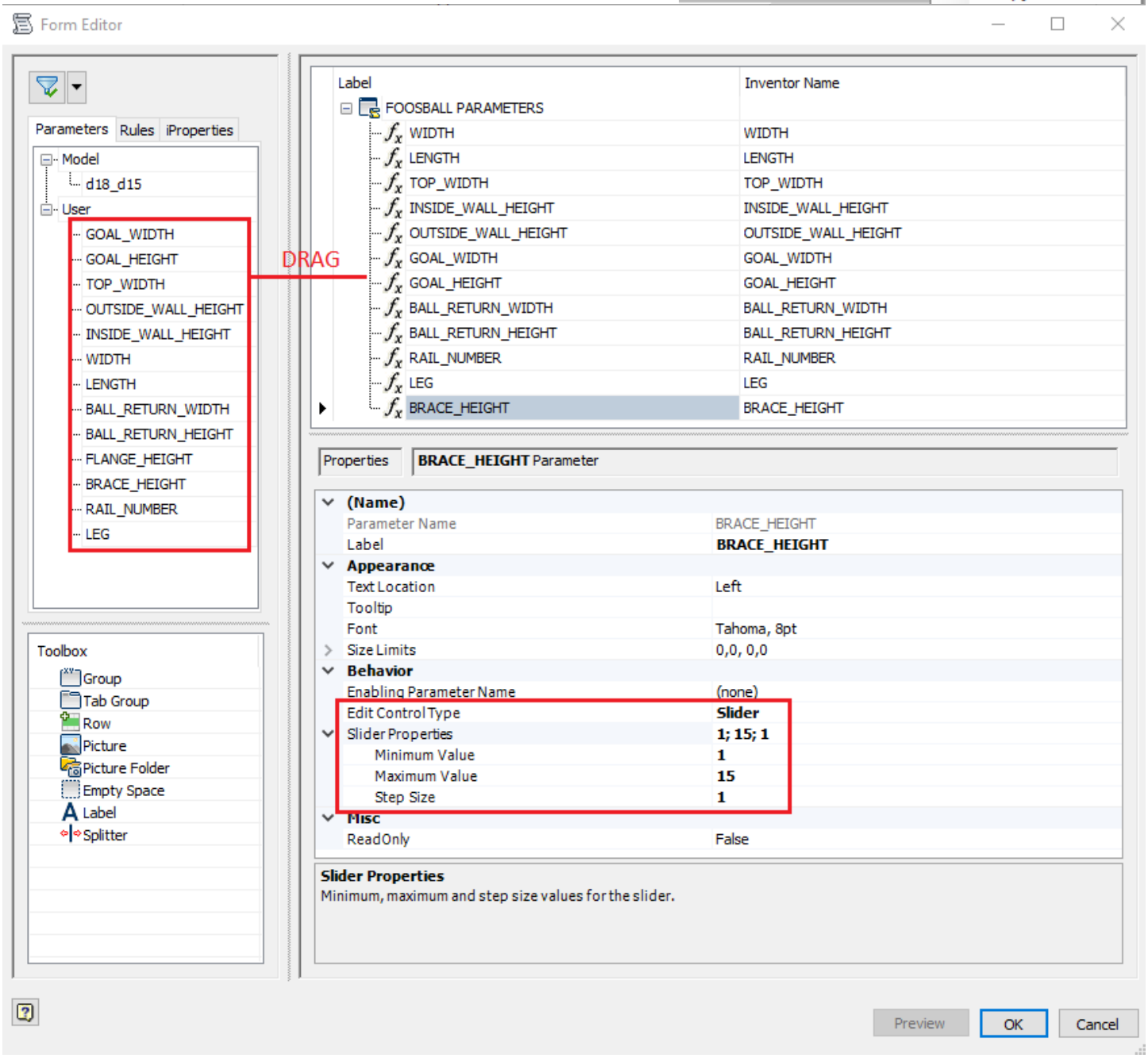
Find the **Forms Tab** > RMB > **Edit**

You can **Shift** select **User Parameters** and drag them to the Form at the right.

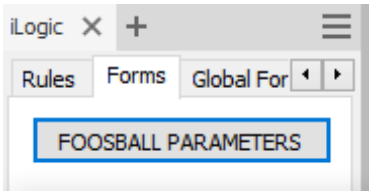
There are different options to explore like **Control Type: Slider**

You are now ready to drive your **Parametric model.**





Click on the **FOOSBALL PARAMETERS Form Button**



Play with the Values

Do not go too crazy big or small or it will break the model. Make small adjustments and see the updates.

You can setup a Slider with limits and increment values for easy visualization.

You now have a Foosball table parametric to any size.

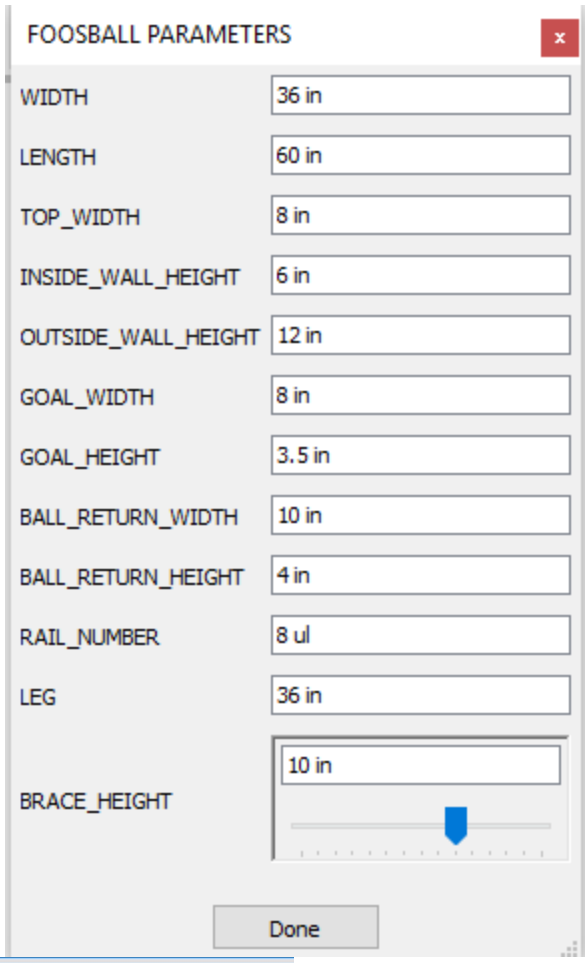
Taking it one step further, you could upload your model for **Configurator360** website use and have customers customize on the fly, make a .pdf, image, or get a quote.

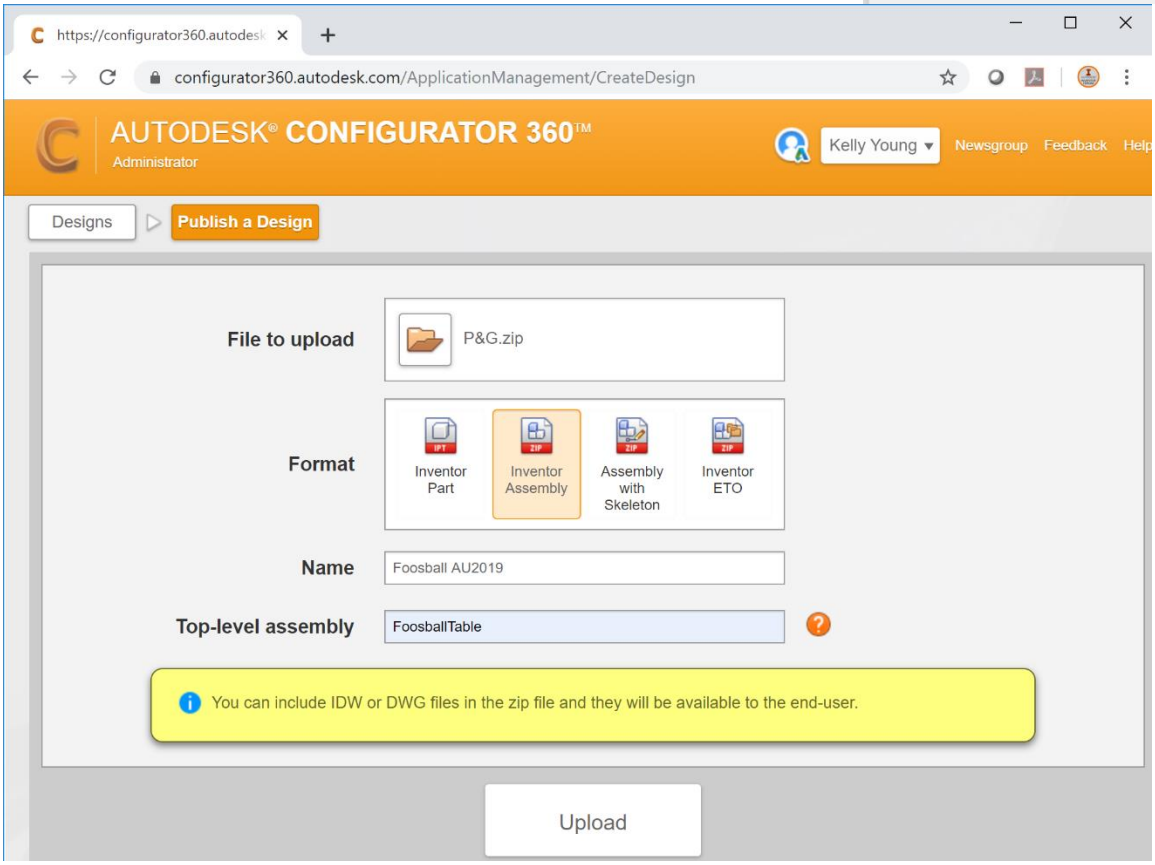
See this example site:

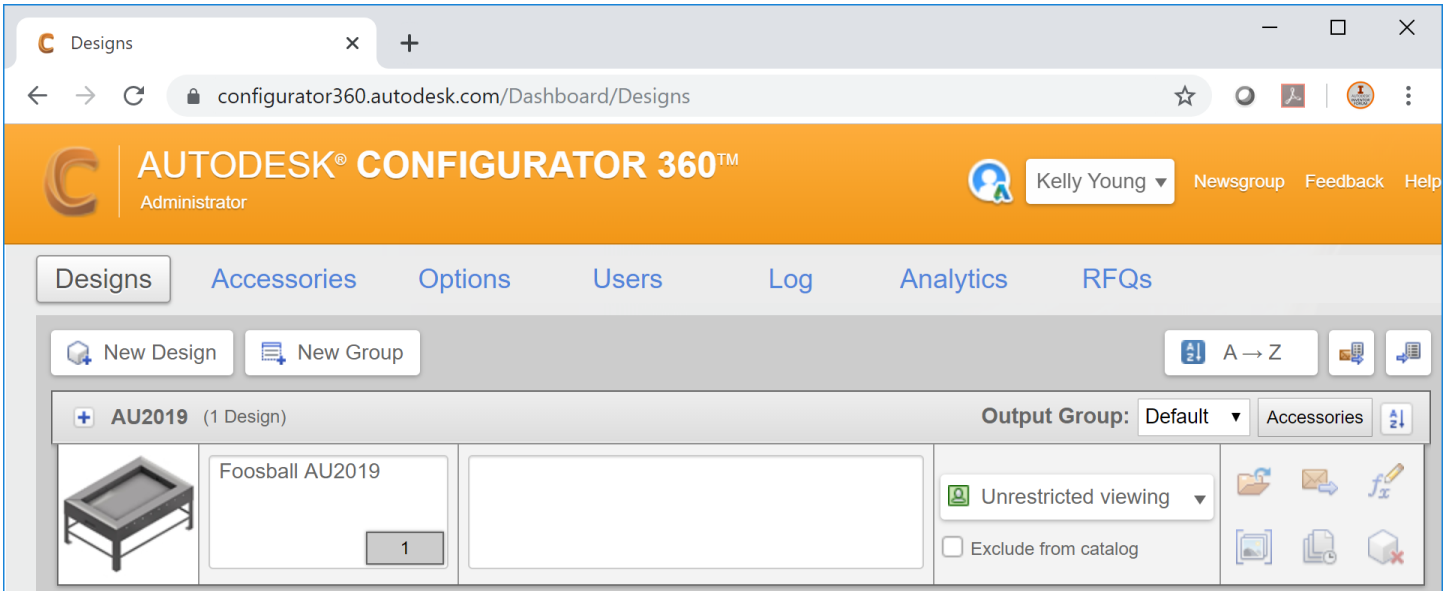
C360: FoosballTable AU2019

Here is a preview of the interface, it is easy to use. From the Assembly File > Save As > Pack & Go as .zip.

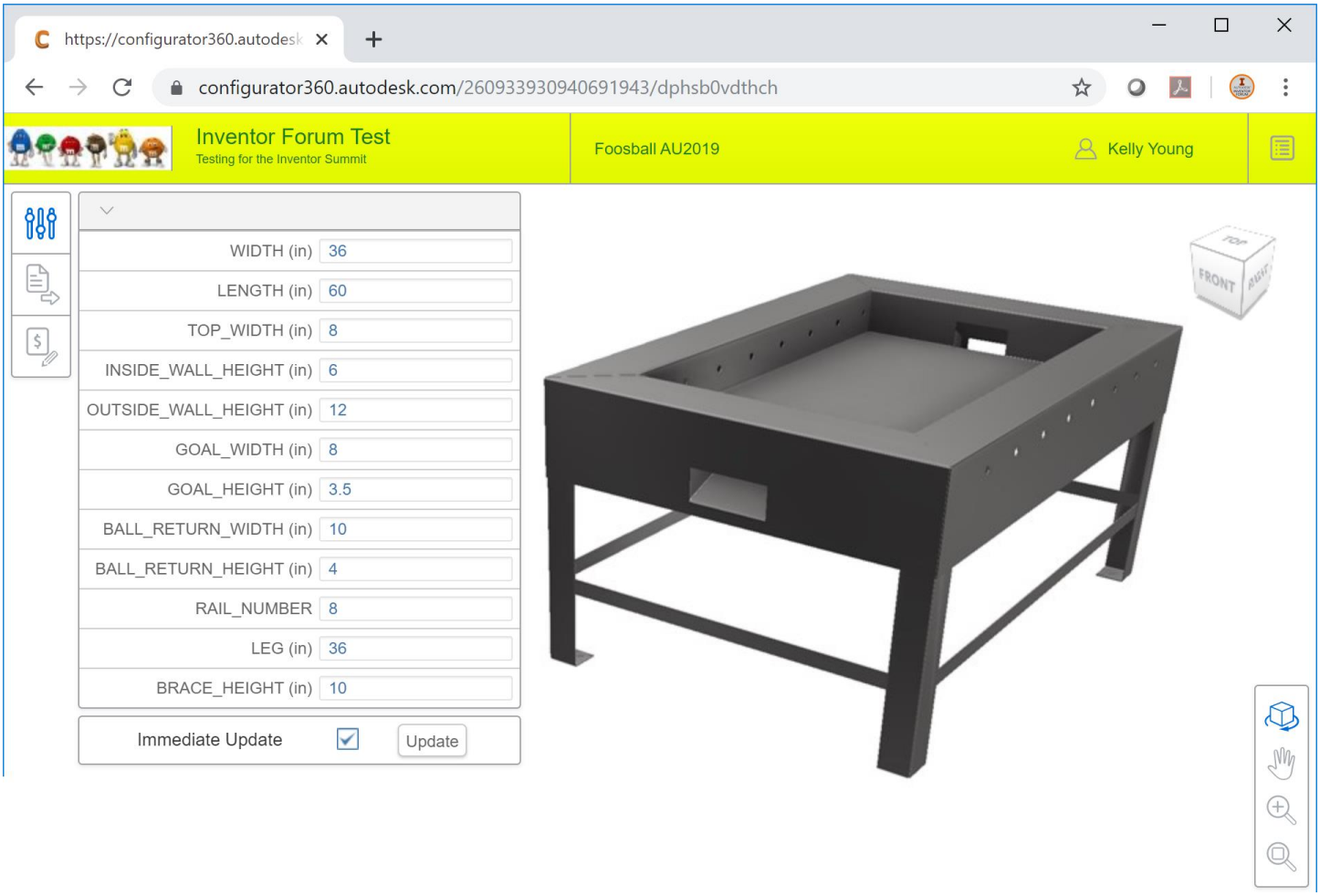
Entry the name of the top-level assembly.





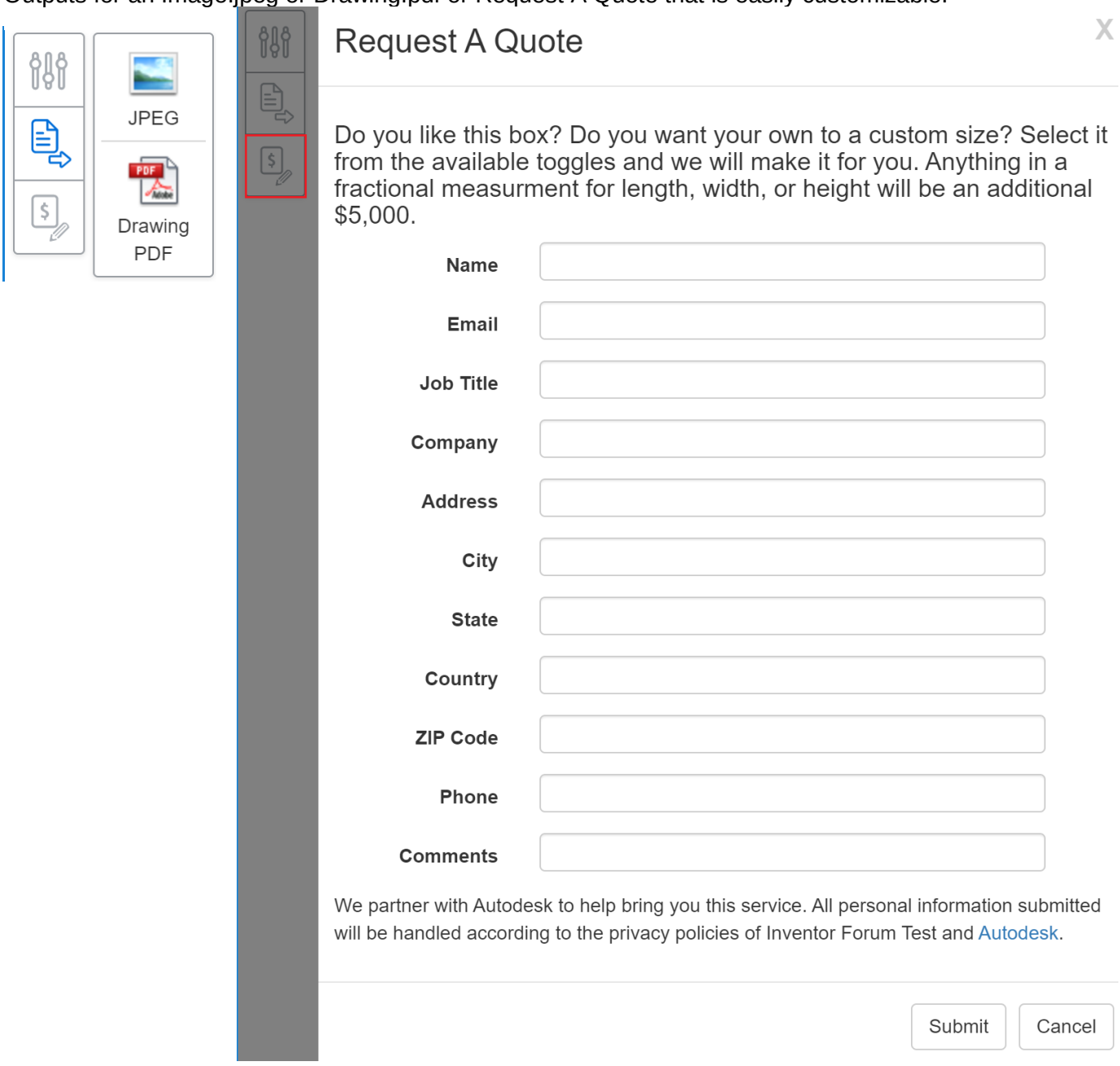


Go To Design:



The Parameters we setup with our simple iLogic code show up and we can drive the Assembly through the web browser! Play around, and see how it works.

Outputs for an Image.jpeg or Drawing.pdf or Request A Quote that is easily customizable.



Thank you for taking the time to go through this course and I hope that it helps you understand the Sheet Metal Environment, learned a few advanced tips, and gets you moving forward in your career. Save often, scour the Internet for tutorials, get involved in the Inventor Forum, and keep inventing.

Kelly Young

Autodesk Inventor Technical Support