

### 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 <u>Inventor YouTube</u>, everything. During my time as Inventor Community Forum Specialist I created a <u>Collection of Screencasts</u> showing different modeling techniques that might be interesting. Here is a <u>great link repository</u>, 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 <u>Inventor Forum</u>, ask and answer questions. You will gain invaluable knowledge, confidence, and insights.

When you get ready to move to the next level, download this <u>Collection of iLogic Models for Beginners</u>. 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 <u>weldment environment</u> and add our welds. For good measure we'll go through a few design accelerators to get you familiar with things like <u>Bolted Connection</u> and <u>Frame Generator</u>.

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.

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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.

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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.

File       Nesting       Factory       Get Started       Tools       CAM       Vault       Collaborate       Image: Collaborate         Image: Collaboration options       Image: Collaboration option option       Image: Collaboration option       Image
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Constraint Settings   Settings   Settings   General Inference Relax Mode   General Inference Relax Mode   Constraint Settings   Spline fit method   Standard   AutoCAD   Minimum Energy - Default Ters   Display Coincident constraints in Sketch   Dimension   Edit dimension when created
Snap to grid         Snap to grid         Autoproject edges during curve of         Autoproject edges for sketch creat         Autoproject part origin on sketch         Project objects as construction ge         Look at sketch plane on sketch creati

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.

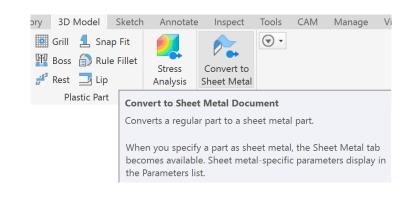
Time to get started with a New > Sheet Metal Part.ipt.

- 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.

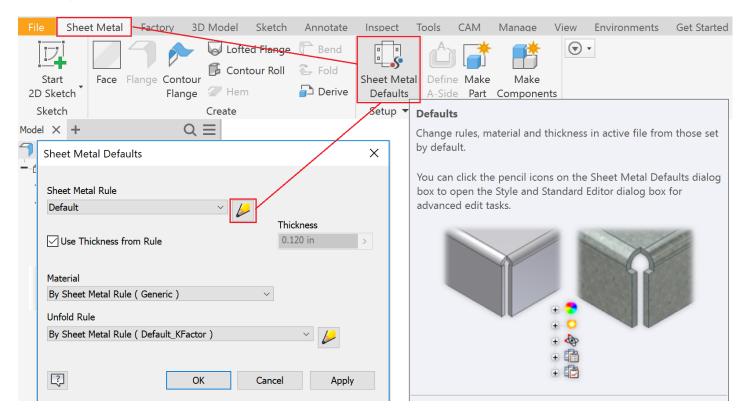
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If you create a regular **Part** by mistake, no problem, you can **Convert To Sheet Metal**, and vise versa.



#### 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**.

🍫 Style and Standard Editor [Library - Read/W	/rite]				×
	Back     New       Sheet Metal [ 1/8 Stainless ]       Sheet Bend	Save Re	eset	₩ All Styles	~
<ul> <li>1/8 Mild</li> <li>1/8 Stainless</li> <li>3/16</li> <li>Default</li> <li>Default_mm</li> <li>Sheet Metal Unfold</li> <li>BendCompensation</li> <li>Default_KFactor</li> </ul>	Bend Relief Relief Shape Straight	Relief Width (A) Thickness Relief Depth (B) Thickness * 0.5 ul Minimum Remnant	>		
⊕ A Text	Bend Radius Thickness	Thickness * 2.0 ul Remnant (C) Bend Transition	>		

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

🍫 Style and Standard Editor [Library - Read	d/Write]	×
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	$\begin{array}{ c c c c c c c c } \hline Custom Equation & Bounding Condition \\ \hline v & n * ((180 deg - \beta)/180 deg) * (p + (\mu/2ul) * (0.65ul + 0.5ul * log(p/\mu))) & 0 deg \le \beta \le 90 deg \\ \hline & - 2ul * (p + \mu) & 0 deg \le \beta \le 165 deg \\ \hline & - 2ul * (p + \mu) * tan((180 deg - \beta)/2ul) & 0 deg \le \beta \le 165 deg \\ \hline & v & 0 in & 165 deg < \beta \le 180 deg \\ \hline & \\ \hline \hline & \\ \hline \hline & \\ \hline \hline & \\ \hline & \\ \hline \hline & \\ \hline \hline \\ \hline & \\ \hline \hline \hline \\ \hline \hline \\ \hline \hline \hline \\ \hline \hline \hline \hline \\ \hline \hline \hline \hline \hline \\ \hline \hline \hline \hline \hline \hline \\ \hline \\ \hline \hline$	
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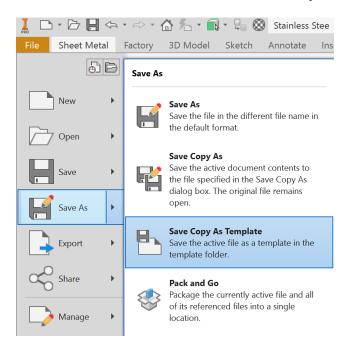
**Corner** will change the bend intersection relief. Depending on the Cutter and Press Break this will vary with the project scope.

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E Sheet Metal Rule 1/2 1/4 1/8	Sheet Metal [ 1/8 Stainle Sheet Bend Corne				
<ul> <li>1/8 Mild</li> <li>1/8 Stainless</li> <li>3/16</li> <li>Default</li> <li>Default_mm</li> <li>Sheet Metal Unfold</li> <li>A Text</li> </ul>	2 Bend Intersection Relief Shape Trim to Bend Round Square Tear Trim to Bend Linear Weld	Relief Roun Relief	Intersection Shape d with Radius × Radius IRadius >		
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Save and Close.

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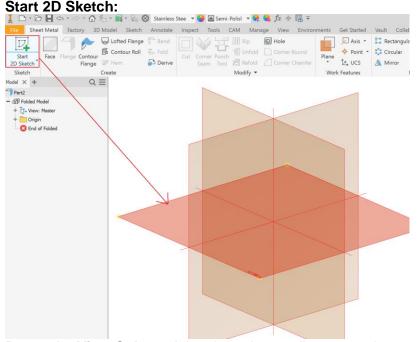
Now that we have written the Sheet Metal Style to the library it can be used in the future.

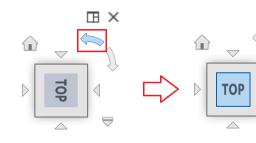


We have the settings for **Stainless Steel Material** and **Thickness** at 1/8. We can **Save Copy As Template** to ensure any new Sheet Metal part we create is set to this every time we start.

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

Time to start designing.



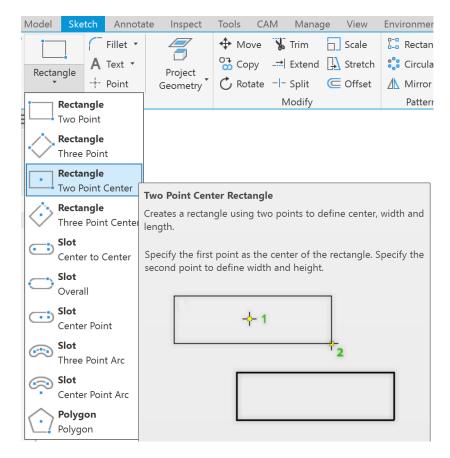


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Rotate the View Cube so it is oriented normally to your view.



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=

Edit Dimension : Re Length=60 in	eplaceMe = 60 in ReplaceMe × - Edit Dimension : Length × 60 in 60 in × •
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Exits the active environment	ive environment and returns to the general working t.
Face: File Sheet Metal Start Sketch Sketch Model X + Field.ipt Field.ipt Folded Model Corigin Face1 Sketch1	Image Contour   Flange

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:

File     Sheet Metal     Factory     3D Model     Sketch     Annotate       Start     Sheet Metal     Factory     3D Model     Sketch     Annotate       Start     Sheet Metal     Face     Flange     Contour     Contour Roll     Seton       2D Sketch     Sketch     Face     Flange     Contour     Plem     Derive       Sketch     Sketch     Create     Create     Create     Create	Inspect Tools CAM Manage View Environ	nments Get Started Vault Collaborate	entor Professional 2020 Field.ipt ren Sheet Metal Defaults Setup • Flat Pattern Part Components Flat Pattern
Flange : Flange1       X         Shape       Unfold Options       Bend       Corner         Edges       Flange Angle       90.0 deg       >         Olick to add       Bend Radius       >       Bend Radius       >         Height Extents       Height Datum       Image: Constraint of the standard of the stan	B		

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Height Exten

Distance

Flip Direction

#### Set FLANGE\_HEIGHT=1.5

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

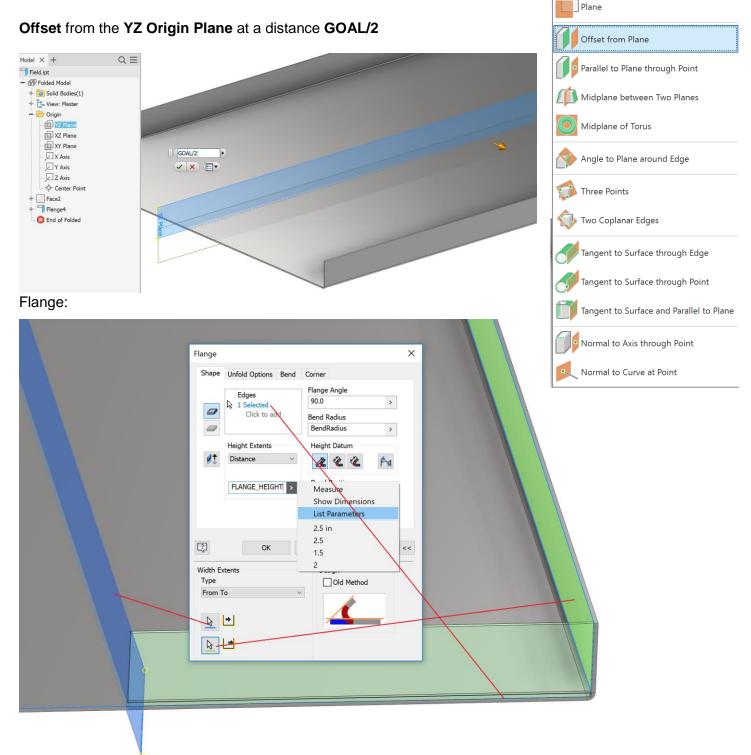
- Use the **View Cube** to look at the **Front View** and zoom in on the bend.
- Change the **Bend Position** and **Height Datum** to see what they control.
- The other tabs will follow the **Sheet Metal Defaults** but you can override them here if needed.

Access Parameters and	Add Numeric User	Parameter cal	lled GOAL set to 8in
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arameter Name	Consumed by	Unit/Type	Equation	Nominal Value	Driving Rule	Tol.	Model Value	Key	Export	Comment
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BendRadius	d82, JacobiRadiusSize, T	in	Thickness	0.125000		0	0.125000			
BendReliefWidth	d76	in	Thickness	0.125000		0	0.125000		<b>V</b>	
BendReliefDepth	d77	in	Thickness * 0.5 ul	0.062500		0	0.062500		1	
CornerReliefSize	d83	in	Thickness * 4 ul	0.500000		0	0.500000		2	
- MinimumRemnant	d78	in	Thickness * 2.0 ul	0.250000		0	0.250000		V	
TransitionRadius	d79	in	BendRadius	0.125000		0	0.125000		<b>v</b>	
JacobiRadiusSize	d85	in	BendRadius	0.125000		0	0.125000		2	
GapSize	d84	in	Thickness	0.125000		0	0.125000		V	
Model Parameters										
WIDTH	Sketch8	in	36 in	36.000000		0	36.000000			
LENGTH	Sketch8	in	60 in	60.000000		0	60.000000			
d76		in	BendReliefWidth	0.125000		0	0.125000			
- d77		in	BendReliefDepth	0.062500		0	0.062500			
d78		in	MinimumRemnant	0.250000		0	0.250000			
d79		in	TransitionRadius	0.125000		0	0.125000			
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d83		in	CornerReliefSize	0.500000		0	0.500000			
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d85		in	JacobiRadiusSize	0.125000		0	0.125000			
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	Flange4, Face2	in	Thickness	0.125000			0.125000			

#### SAVE

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



🔎 Axis 🔹

🔶 Point 🔻

🛃 UCS

Plane

Rectangular Sk

🚦 Circular

/ Mirror

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.

Flange: Flange5     X       Shape     Unfold Options     Bend       Comer     Flange Angle       90.0 deg     X       Bend Radius     Bend Radius       Bend Radius     X	Image: Corner Seam       Image: Fill and Fi	Plane Work Features	Rectangular Circular Mirror Pat
Height Extents Distance Height Datum PLANGE_HEIGHT Bend Position Bend Positi	Modify ▼ Corner Seam Shape Bend Corner Shape Seam Maximum Gap Distance Face/Edge Distance Face/Edge Distance Gap Gap GapSize >	×	Pat
	Cancel A	pply >>	

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.

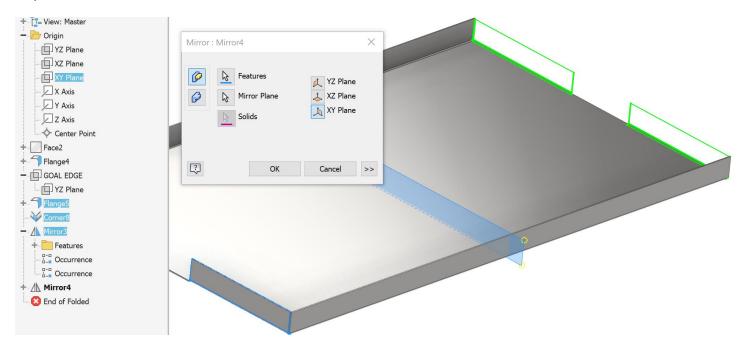
Turn the **Visibility** of the **Plane** Off from the **Browser** or by RMB on the plane.

Mirror:		
File         Sheet Metal         Factory         3D Model         Sketch           Start         20 Sketch         Sketch         Image: Contour Roll         Image: Contour         Image: Contour Roll	<ul> <li>Pend</li> <li>Pold</li> <li>Derive</li> <li>Cut Corner Punch</li> <li>Seam Tool</li> <li>R</li> </ul>	Infold Corner Round
iLogic X +  Rules Forms Global Forms External Rules Field.jpt	Mirror	Features X VZ Plane
Model X + Q ≡ → Field.ipt → @ Folded Model + @ Solid Bodies(1) + "t-view: Mater - @ Origin		Solids J XY Plane
C2 Plane     C2 Plane     C2 Plane     C2 Y Plane     C2 X Axis     C Y Axis	00	on Method ptimized entical ijust
C Z Axis		



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

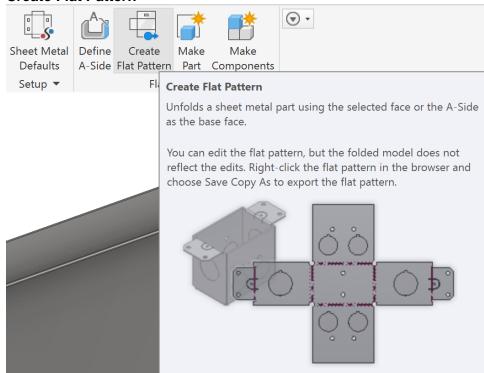
#### Repeat but this time use the **XY Plane**.



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...** 

×

~ 🔍

Configuration Configuration Not Saved

File Version AutoCAD R12/LT 2 DXF AutoCAD 2018 DXF AutoCAD 2013 DXF AutoCAD 2010 DXF AutoCAD 2001 DXF AutoCAD 2004 DXF AutoCAD 2000/LT 2000 DXF

Next >

Save as type: AutoCAD DXF (\*.dxf)

Finish

Options...

)\Design Data\DWG-DXF\FlatPattern.xml

Cancel

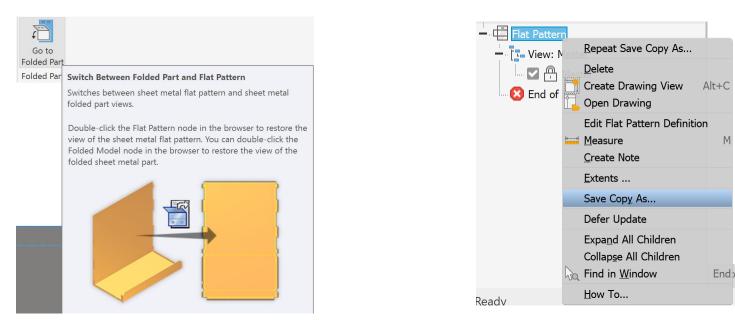
Save

Cancel

	DAF File Export C
 Go to Folded Part	
Cosmetic Centerline 😥 🕞 👘	
Undo 🤝 🙀 Select Other	
Cut 🔲 🐺 PunchTool	
New Sketch	
Repeat Bend Order	
Select Tangencies	
Measure M	
Greate Note	
Export Face As	
Look At Page Up	
Previous View F5     Home View F6	
Home View F6 Properties	
Help Topics	
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**************************************	

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

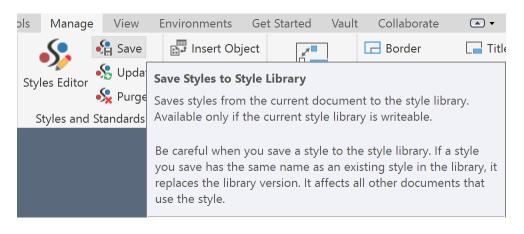


Now if we want to reference this part or use it for the **Nesting Utility** we can easily get to the **Flat Pattern**. **Section 4: How to document and dimension the part for shop fabrication**.

#### 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 <u>other tutorials</u> for that and should be decided what your company **Styles and Standards** are going to be.

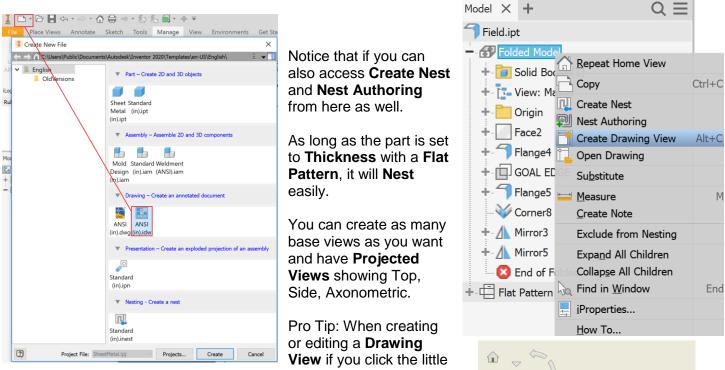
Set everything like you want through Style Editor, then Save to the Style Library.



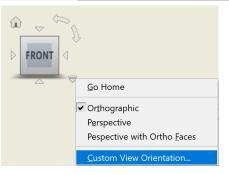
After that you can Save As > Save Copy As Template.

There are multiple ways to Create a New Drawings

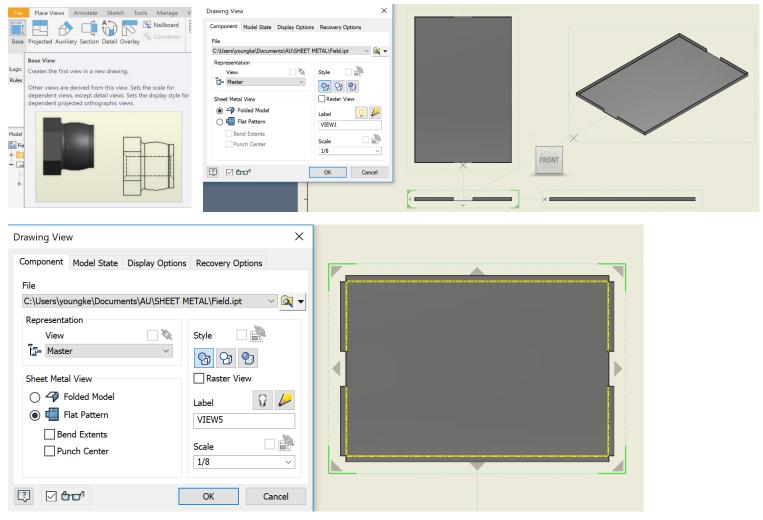
- From the New icon
- Ctrl+N
- From the **Browser** RMB on the part icon > **Create Drawing View** which automatically loads the model and starts a **Base View** saving the step of creating one.



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.

Annotate Sket	ch Tools Mana	ge View Enviro	nments Get Star	ted Vault	Collaborate	•								
ne • 📑 Arrange ite •	Hole and Thread	Text Leader Text	Insert Sketch Symbol	Surface V	Velding Impor	)))))) Caterp	▲ / /// ↓ + +↓+	Retrieve Model Annotations	Start Sketch	Parts	Hole • Revision •	alloon	Edit Layers	B  ►●  By St
sion The External Rules Q =	Feature Notes		note associated wi	te is above ti I text to the r n Offset valu	he selected midpoint of	90° R1/8 UP 90° R1/8		Retrieve	Sketch	90° R1/8	Table			90° R1/8 UP 90° R1/8
		Press F1 for more	help						UPS	90° R1/8				16 dN

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.

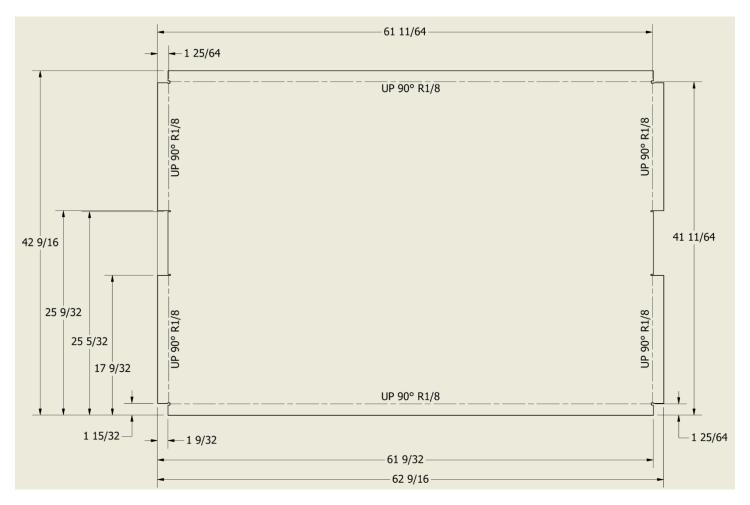
File Place Views	Annotate Sketch Tools Manage View Environments	Get Started Vault Collaborate 📼 🕶		
Dimension	Creates one or more individual baseline dimensions in a single process.	sert Symbol Surface Welding Impor Caterp =	Retrieve Mod Retrieve	
iLogic × +	A baseline dimension displays the orthogonal distance between	1699518	Car	ncel (ESC) 🗙 🔜 🖕 Make Origin 🗋 💷
Rules Forms Global Forms	the origin (base line) and selected edge or point. You specify an origin from which to calculate dimensions, and select the geometry to dimension.			± <sub>o</sub> Zoom
Model × +	10.10 - 28.00	international and a second and		Pan Previous View F5
Drawing Resources     Drawing Resources     Sheet:1     Default Border     ANSL Large				How To

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.

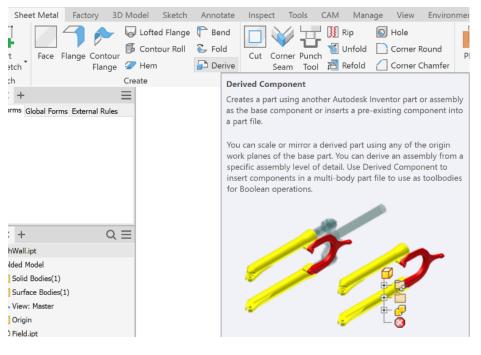
Time for some advanced skills and helpful tricks.

01. . . . . . .

. . .

Create New >	Sneet Meta	al Part.
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Use color override from so	ource component	
Link sheet metal styles		
Reduced Memory Mode		
E Design View:		
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2	OK	Cancel

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60.000 x 40.000 < Add Parameter
~
< >
OK         Cancel



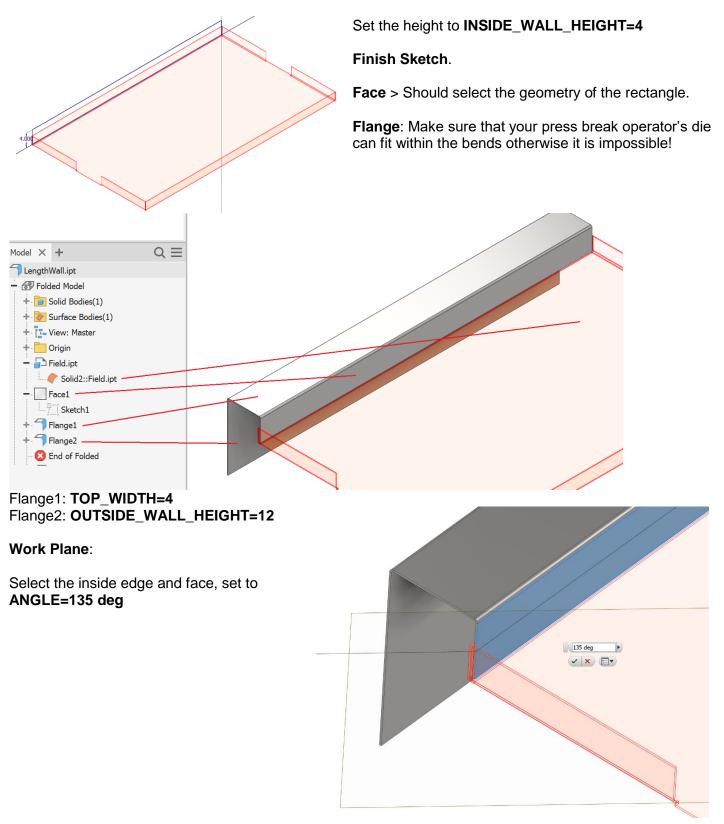
#### Derive

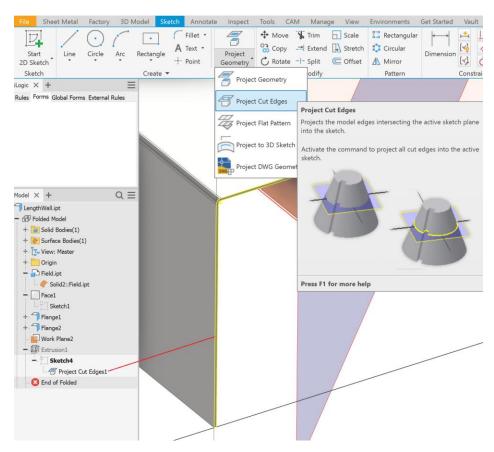
Select **Body as Work Surface** to use as a reference within this part, you will see it transparent in the preview.

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.

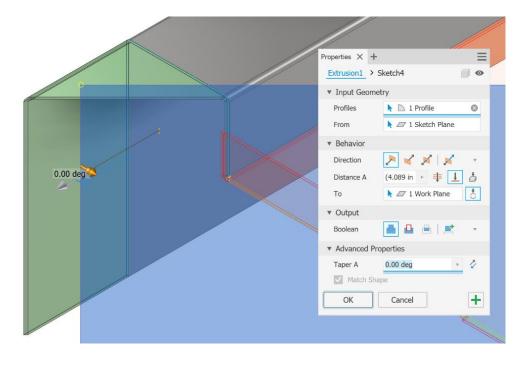
Project Geometry of the edge and Constrain the rectangle to the ends.



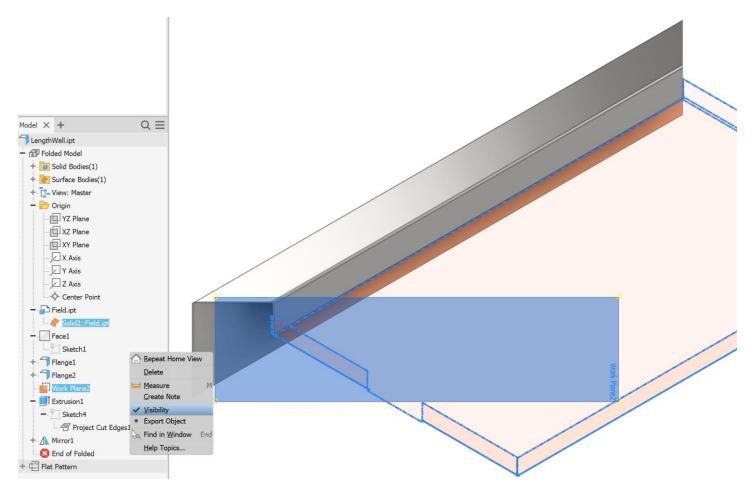


### 3D Model > Start 2D Sketch > Project Cut Edges > Finish Sketch

#### Extrude to the Plane.

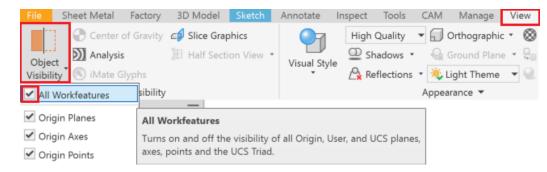


Mirror across the XY Origin Plane.



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.

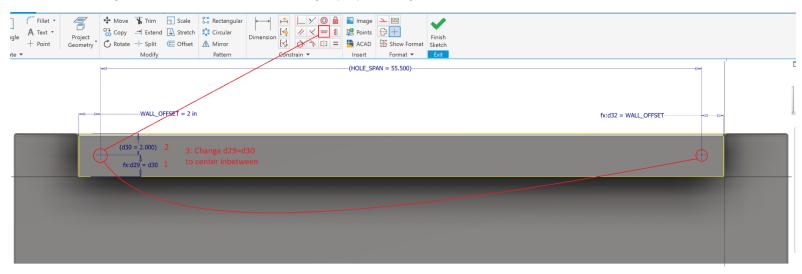
**SAVE** – Keep saving more often, get in the habit.

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.

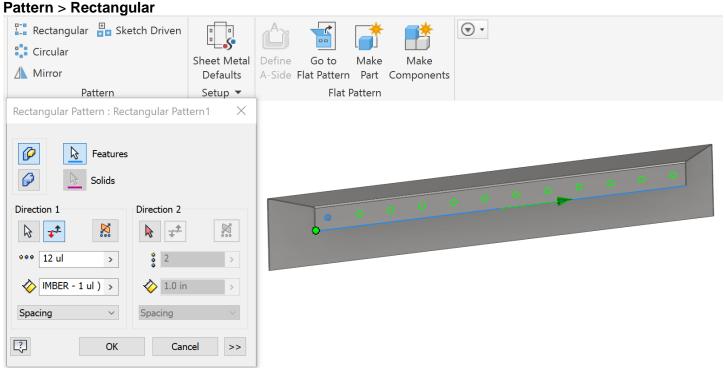
Explore the **Options**, there are many. You can now save presets you use often in the top right **+** menu.

Type: Through All.

**OK. SAVE** 

Hole Corner R Corner C		Plane • Work	Axis • Point • L. UCS Features	•	Circular Mirror	Sketch Driven	Sheet Metal Defaults Setup
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Access Parameters and rename the driven dimension (from between the points) to HOLE\_SPAN.



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!

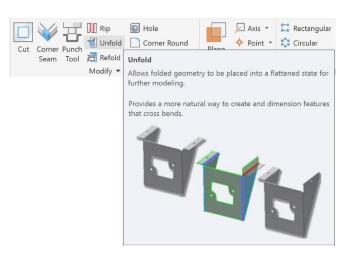
#### Create Flat Pattern.

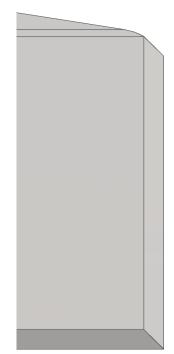
Turn the ViewCube to Bottom and you will notice that there is an edge that is at an angle!

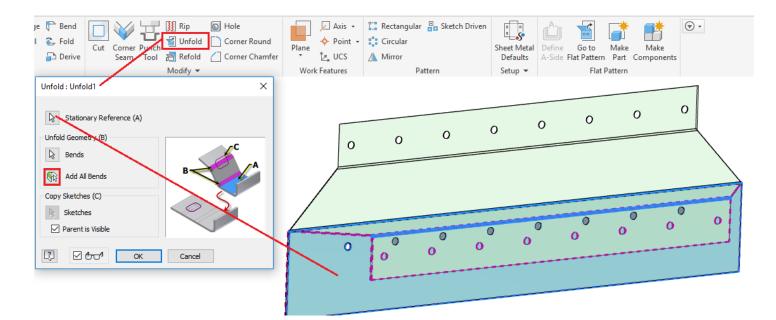
We have illustrated this for things to beware of. You can easily make square, straight flanges all day, but when you want to get advanced shapes, you will run into this issue. Better to learn how to fix it now.

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.

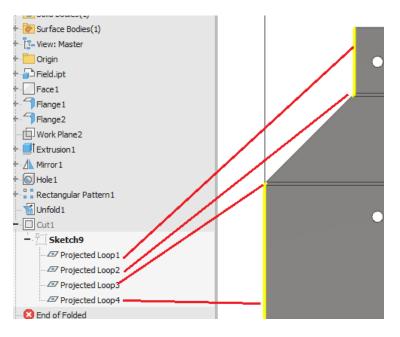
This is an example of using clean up techniques after you get the basic shape.

#### 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.



(X)

Construction

÷

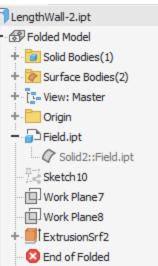
# Changes selected sketch geometry to construction geometry, or creates new geometry as sketch construction geometry. Select sketch objects and activate the command, or activate the command and create new sketch construction geometry.

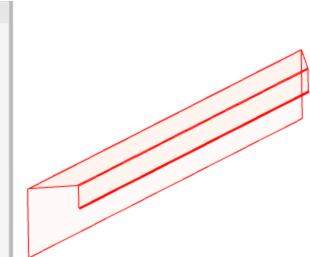
If the Field Surface **Visibility** is turned **Off**, we now have the Flange but as a thin **Surface Body**.

3D Model > Modify > Thicken/Offset

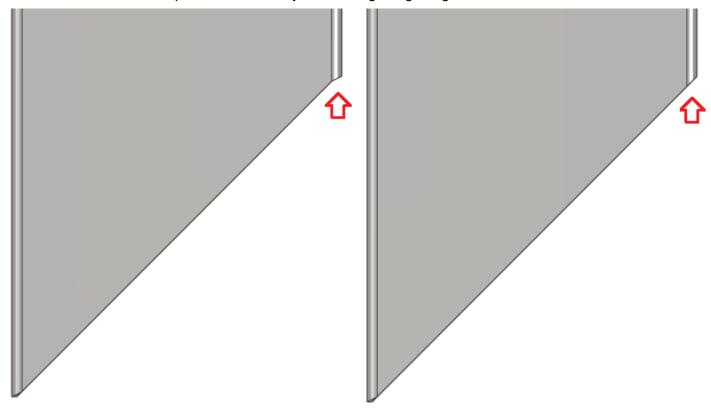
Select the **Surface Body** with **Quilt** at distance of **Thickness**.

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.

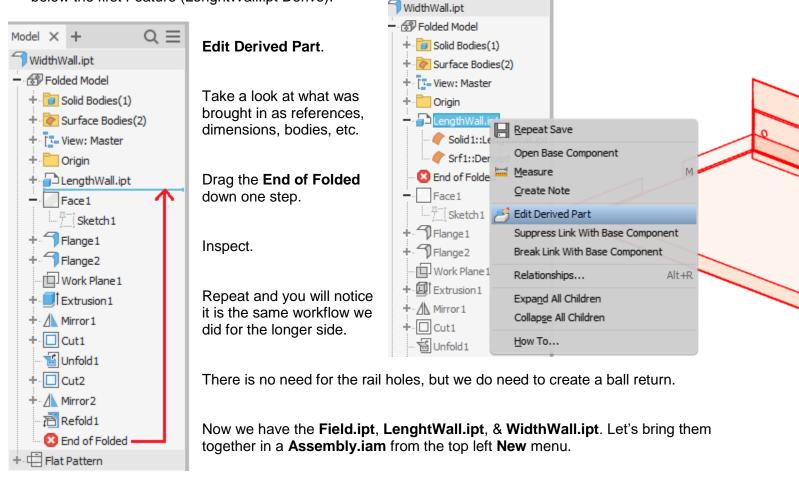


Pro Tip: Using the **Surface Thicken** at **Thickness Parameter** will get you out of a tough situation when you can't use the **Sheet Metal Tools** to do what you want.

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).



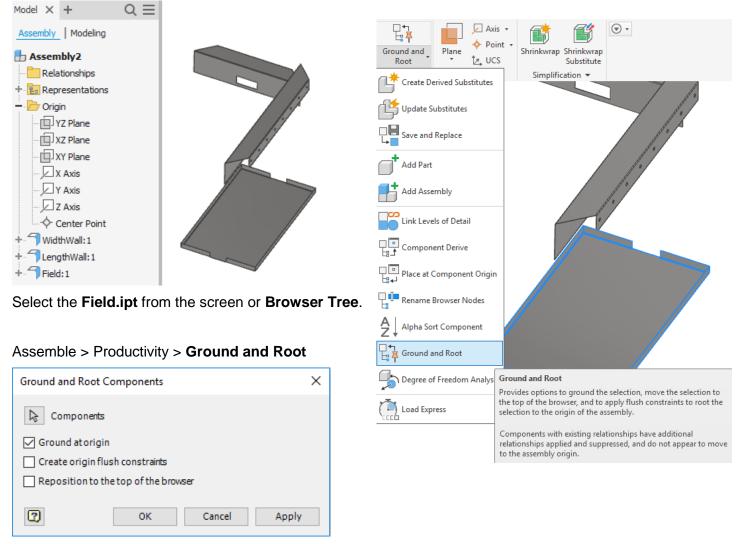
For this exercise we will be using the premade parts for ease of use.

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

	🕽 Material 🔹 🔻 🜔 Appearance 🛛 👻 🤮 f 🗴 🗮
File Factory Ass	
Model X +	Look in: SHEET METAL Name Screenshots OldVersions FoosballTable FAB322732-L - Inventor Sheet Metal from Start To SheetMetal.ipj WidthWall.ipt LengthWall.2.ipt LengthWall.ipt Leg.ipt Handle.ipt Field.ipt

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

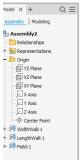
You should have something that looks like this:



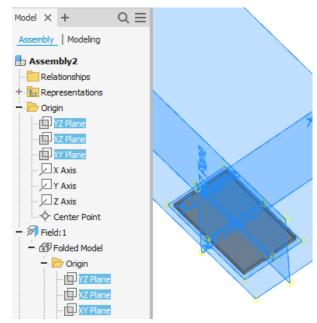
There is an option for Create origin flush constraints, but we will leave that off, just know it is an option.

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**.

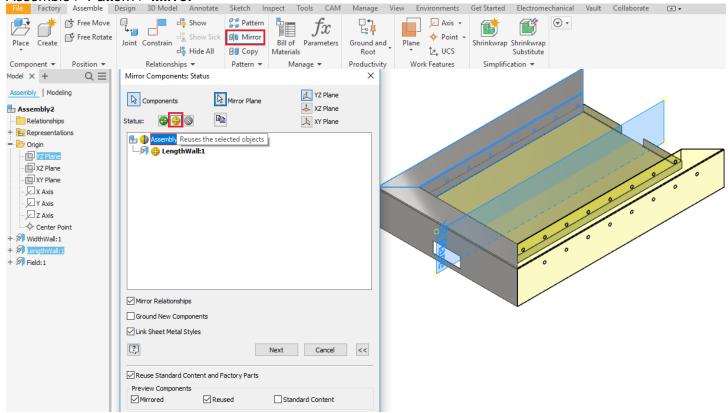
Do the same thing with the other two parts. They should match up in the correct places, no constraints needed!







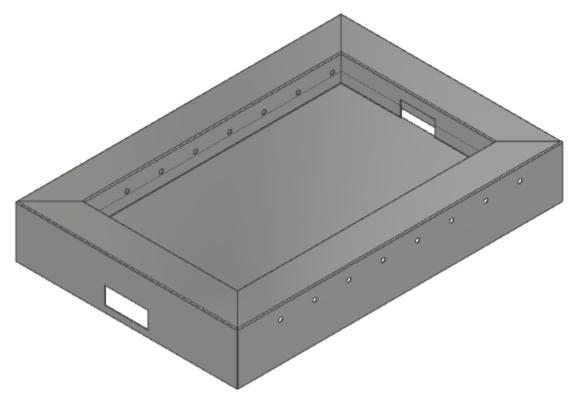
#### Assemble > Pattern > Mirror



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.

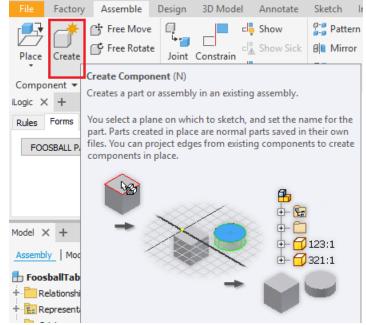


#### **Open > FoosballTable.iam**

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.

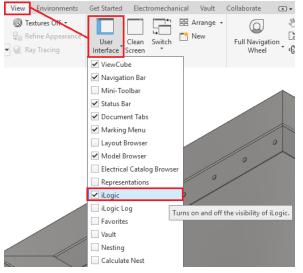
Pro Tip: Become more familiar with iLogic. Inspect all models at Collection of iLogic Examples For Beginners.

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.

	,	ound type	Equation	Nominal Value	Driving Rule	Tol.	Model Value	Key	Export Parameter	Comment	
User Parameters											
BALL_RETURN_HEIGHT		in	4 in	4.000000		0	4.000000				
BALL_RETURN_WIDTH		in	10 in	10.000000		0	10.000000				
BRACE_HEIGHT		in	10 in	10.000000		0	10.000000				
FLANGE_HEIGHT		in	2.5 in	2.500000		0	2.500000				
GOAL_HEIGHT		in	3.5 in	3.500000		0	3.500000				
GOAL_WIDTH		in	8 in	8.000000		0	8.000000				
INSIDE_WALL_HEIGHT		in	6 in	6.000000		0	6.000000				
LEG		in	36 in	36.000000		0	36.000000				
- LENGTH		in	60 in	60.000000		0	60.000000				
OUTSIDE_WALL_HEIGHT		in	12 in	12.000000		0	12.000000				
- RAIL_NUMBER		ul	8 ul	8.000000		0	8.000000				
TOP_WIDTH		in	8 in	8.000000		0	8.000000				
WIDTH		in	36 in	36.000000		$\left  \bigcirc \right $	36.000000		Г		

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

iLogic × +	$\equiv$								
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ᡖ FoosballTable.iam									
😰 Foosball Parameter									
	Run Rule								
	Edit Rule								
	Regenerate Rule								
Model × +	Suppress Rule								
Assembly   Modeling	Delete								

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.

#### ? × Model File Tree Files Options Search and Replace Wizards Snippets п System Custom ✓ ♣ FoosballTable.iam Parameters Names $f_x$ Model Parameters te to Parameter Equation ~ $f_x$ User Parameters WIDTH 36 in > 📄 Relationships . Features I ENGTH 60 in Origin . Components (classic) d76 BendReliefWidth ✓ 分 Field:1 . ... iLogic Assemblies/Components d77 BendReliefDepth ∫n Mod . • Positioning d78 MinimumRemnant $f_x$ Sheet Metal Parameters d79 TransitionRadius . Excel Data Links $f_x$ Reference Parameters FLANGE\_HEIGHT 2.5 in i Parts $f_x$ User Parameters d81 90.0 dea . iFeatures > 😚 Folded Model d82 BendRadius > 🛱 Flat Pattern d83 CornerReliefSize . . Relationships (Add) Flush:1 d84 GanSize . . . Measure Flush:2 d85 JacobiRadiusSize . • Forms Flush:3 d86 GOAL / 2 ul > 🔊 LengthWall:1 d87 BendReliefWidth . - Log Messages Document > 🚿 WidthWall:1 d88 BendReliefDepth Run Other > 🔊 WidthWall:2 d89 MinimumRemnant . ∎ · BOM d90 TransitionRadius > 🔊 BallReturn:1 . ⊕ • Math < > 🔊 BallReturn:2 5 . • Strings • Variables 1 🗋 👗 🖹 🗳 🔍 If...Then...End If ▼ Keywords ▼ Operators ▼ Header... Ξ 0 4 Parameter("Field:1", "WIDTH") = WIDTH Drawing Parameter("Field:1", "LENGTH") =LENGTH Parameter("Field:1", "FLANGE\_HEIGHT") = FLANGE\_HEIGHT · Advanced Drawing API Advanced API Parameter("LengthWall:1", "TOP\_WIDTH") =TOP\_WIDTH Parameter("LengthWall:1", "INSIDE\_WALL\_HEIGHT") =INSIDE\_WALL\_HEIGHT Parameter ("LengthWall:1", "OUTSIDE WALL HEIGHT") = OUTSIDE WALL HEIGHT Parameter ("WidthWall:1", "GOAL HEIGHT") =GOAL HEIGHT Parameter ("Field:1", "GOAL") = GOAL WIDTH Parameter("WidthWall:1", "BALL RETURN\_HEIGHT") =BALL RETURN\_HEIGHT Parameter("WidthWall:1", "BALL\_RETURN\_WIDTH") = BALL\_RETURN\_WIDTH Parameter("BraceAdaptive:1", "BRACE HEIGHT") = BRACE HEIGHT Parameter("LengthWall:1", "RAIL NUMBER") = RAIL NUMBER Parameter("Leg:1", "LEG")=LEG Ln 15 Col 1 Log Level Info ✓ ✓ Detailed Trace Save & Run Close

#### Edit Rule: Foosball Parameters

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.

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	f <sub>x</sub> width	WIDTH	
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🖻 User	$- f_x$ INSIDE_WALL_HEIGHT	INSIDE_WALL_HEIGHT	
··· GOAL_WIDTH	$-f_x$ OUTSIDE_WALL_HEIGHT	OUTSIDE_WALL_HEIGHT	
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	Label	BRACE_HEIGHT	
	Appearance     Text Location	Left	
	Tooltip	Leit	
	Font	Tahoma, 8pt	
Toolbox	> Size Limits	0,0, 0,0	
Group	✓ Behavior		
Tab Group	Enabling Parameter Name	(none)	
Row	Edit Control Type	Slider	
Picture	Slider Properties     Minimum Value	1; 15; 1	
Picture Folder	Maximum Value	15	
Empty Space	Step Size	1	
A Label	V Misc		
♦ Splitter	ReadOnly	False	
	Slider Properties Minimum, maximum and step size values for	the slider.	
2		Preview OK Cano	

Click on the FOOSBALL PARAMETERS Form Button

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FOOSBALL PARAMETERS		

Play with the Values	FOOSBALL PARAMETE	RS ×
Do not go too crazy big or small or it will break the model. Make	WIDTH	36 in
small adjustments and see the updates.	LENGTH	60 in
	TOP_WIDTH	8 in
You can setup a Slider with limits and increment values for easy visualization.	INSIDE_WALL_HEIGHT	6 in
	OUTSIDE_WALL_HEIGHT	12 in
You now have a Foosball table parametric to any size.	GOAL_WIDTH	8 in
Toking it one stop further, you could upload your model for	GOAL_HEIGHT	3.5 in
Taking it one step further, you could upload your model for <u>Configurator360</u> website use and have customers customize	BALL_RETURN_WIDTH	10 in
on the fly, make a .pdf, image, or get a quote.	BALL_RETURN_HEIGHT	4 in
See this example site:	RAIL_NUMBER	8 ul
C360: FoosballTable AU2019	LEG	36 in
		10 in
Here is a preview of the interface, it is easy to use. From the Assembly File > Save As > Pack & Go as .zip.	BRACE_HEIGHT	•••••
Entry the name of the top-level assembly.		
		Done

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File to upload	P&G.zip		
Format	Inventor Part Inventor Assembly Skeleton		
Name	Foosball AU2019		
Top-level assembly	FoosballTable	0	
You can include IDW o	or DWG files in the zip file and they will be available to	the end-user.	
	Upload		

C Designs × +	- 🗆 X
$\leftarrow$ $\rightarrow$ C $\triangleq$ configurator360.autodesk.com/Dashboard/Designs	☆ • • ▶   🎂 :
C AUTODESK <sup>®</sup> CONFIGURATOR 360 <sup>™</sup> Administrator	Kelly Young 🔻 Newsgroup Feedback Help
Designs Accessories Options Users Log	Analytics RFQs
🞧 New Design 📃 New Group	
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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.

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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