MFG468245

The Essential Skills for Sheet Metal Modeling in Fusion 360

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Learning Objectives

- Understand the importance and role of Sheet Metal Rules
- Apply the Flange command to make faces, flanges, and contour flanges
- Explain why a sheet metal part may not generate a flat pattern
- Document formed parts and flat patterns with a 2D drawing

Description

Fusion 360 has four unique design workflows, Solid, Surface, Sculpt (T-Spline), and Sheet Metal. Each workflow has unique tools that are specialized for the different types of modeling. The Sheet Metal workflow can be simple, yet confusing. Certain tools have multiple functions and use. There are Sheet Metal Rules and it is important to understand the role they play. Join Steve Olson for this class where he will explain the proper workflow for modeling sheet metal components in Fusion 360. He will explain how to leverage the Sheet Metal Rules and the multiple applications of the Flange command. During this demonstration, he will recreate a component and show how to document the formed part and flat pattern with a 2D drawing.

Speaker

Steve Olson is the Manager of Training Services at MESA Inc., an Autodesk Partner serving Western Pennsylvania, Ohio, and beyond. He has been working with Autodesk software since 2005. He has experience with Inventor, Fusion 360, Vault, AutoCAD, AutoCAD Civil 3D, InfraWorks, ReCap, and more. Steve gained industry experience during his 5 years as a draftsman and Vault Administrator for Fleetwood Folding Trailers, an RV manufacturer. At MESA, Steve teaches classes, develops and implements training programs, supports Autodesk products, and consults with customers regarding their use of Autodesk products. He is an Autodesk Certified Instructor and holds Professional Certification in several Autodesk products. Steve has spoken at AU and MESA sponsored Events several times, and he contributes content to the It's A CAD World blog and YouTube Channel.

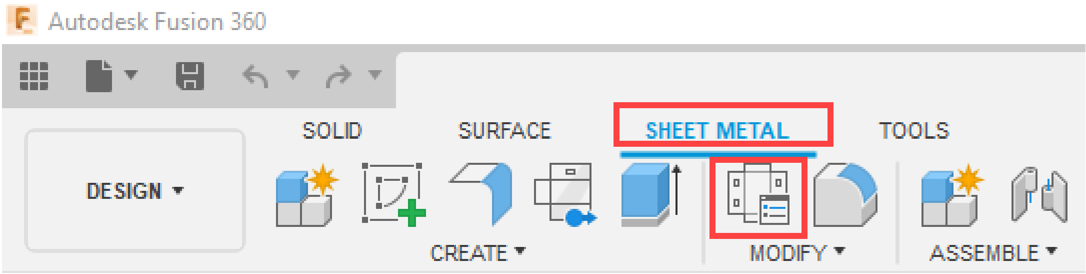
If you are transitioning from another CAD program, or just adding sheet metal modeling to your solid modeling skillset, Fusion 360's Sheet Metal modeling workflow can be a bit confusing at first. If you have never modeled sheet metal before, understanding the role of the Sheet Metal Rules is critical to creating an accurate design. Personally, using Fusion 360 after using Inventor for years, I was confused as to where to find the Fusion 360 tools that were equivalent to the tools I used in Inventor. Once I understood that the Flange command is essentially the combination of several commands I mastered in Inventor, the workflow made perfect sense and I appreciated the simplicity of the command set.

Having made the transition and gained that knowledge, I wanted to document what I have learned and pass it on to other users that are looking to add sheet metal modeling to their Fusion 360 knowledge base.

Sheet Metal Rules

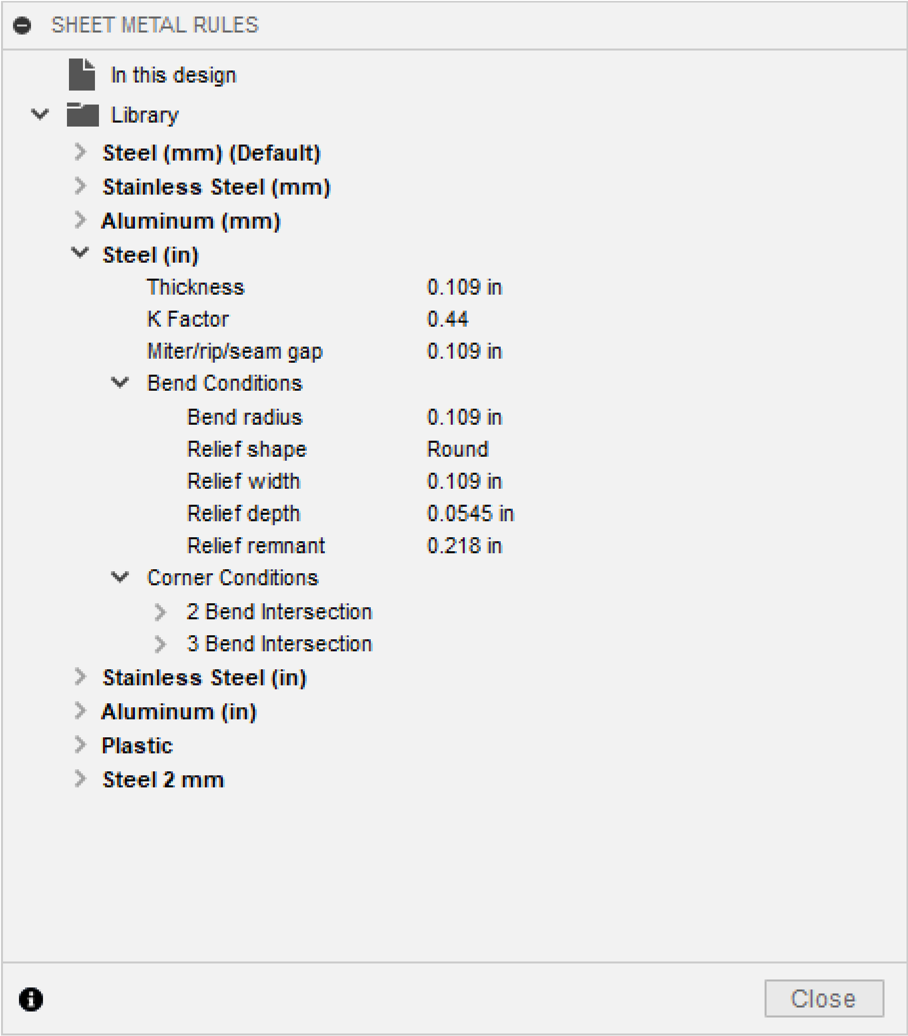
Sheet Metal Rules are your way of defining the material that will be used to fabricate the part. You will have to include attributes like material thickness, bend radius, reliefs, K Factor, and more. This is critical so the part has the right design attributes. It also gives us a centralized location to change these attributes. As you begin building your model, you will be adding faces, flanges, and bends. All of these features will follow the active style of this model. It is possible to over-ride parameters, like bend radius, when creating features. However, by default features will use the attributes in the active style.

Selecting and Editing a Sheet Metal Rule can be done through the Sheet Metal Rules dialog.

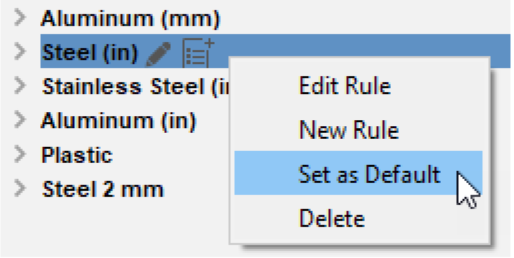


The Sheet Metal Rules dialog will have a list of rules that exist in your Library, as well as rules that have been saved in the active file because it was once the active style.

Expanding a rule will show the attributes for that rule.

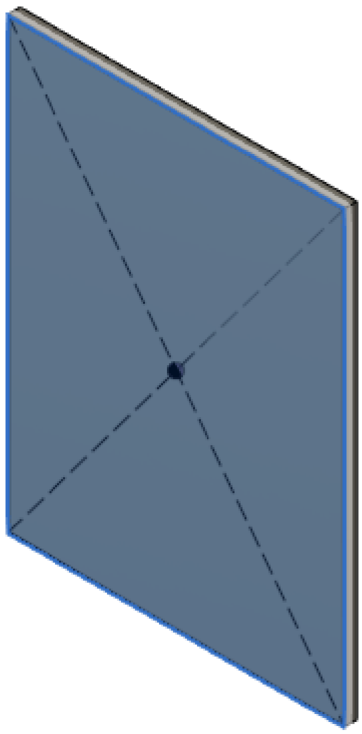


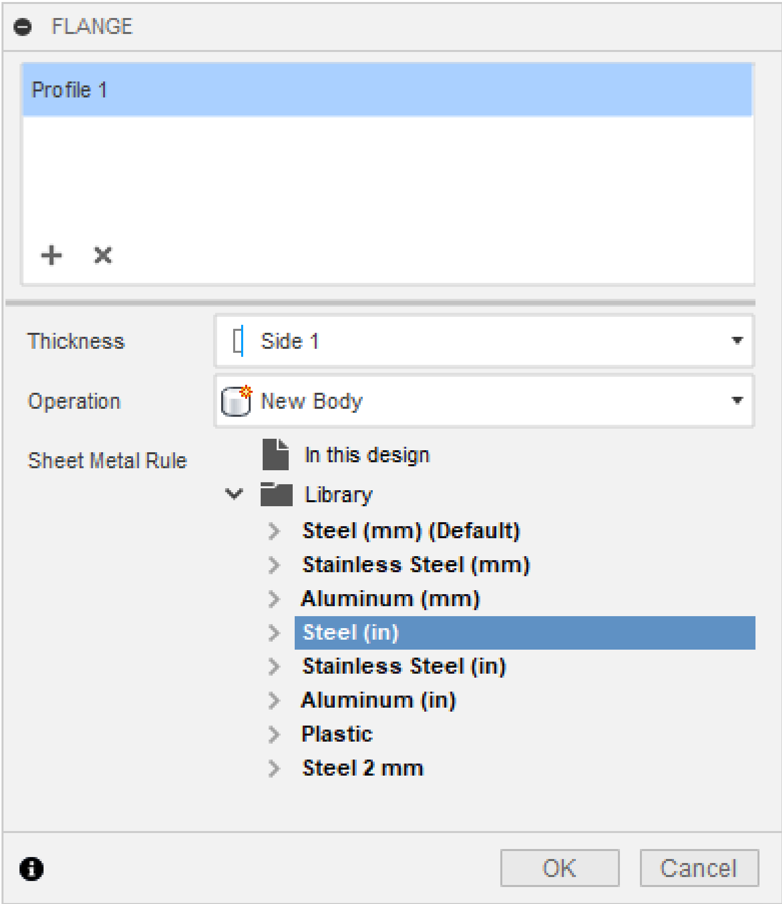
These can be copied, edited, or deleted to match your company's standards. You can also set which rule to use by default, by right-clicking on the rule.



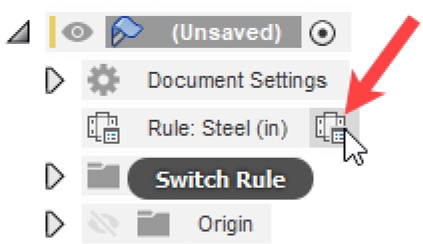
At any point in the design process, the active rule can be edited or replaced by another rule. Doing so will update all features in the model to the new active rule.

You will be able to set the active rule when creating the first Flange feature. The Flange command will give you the ability to set the active Sheet Metal Rule.

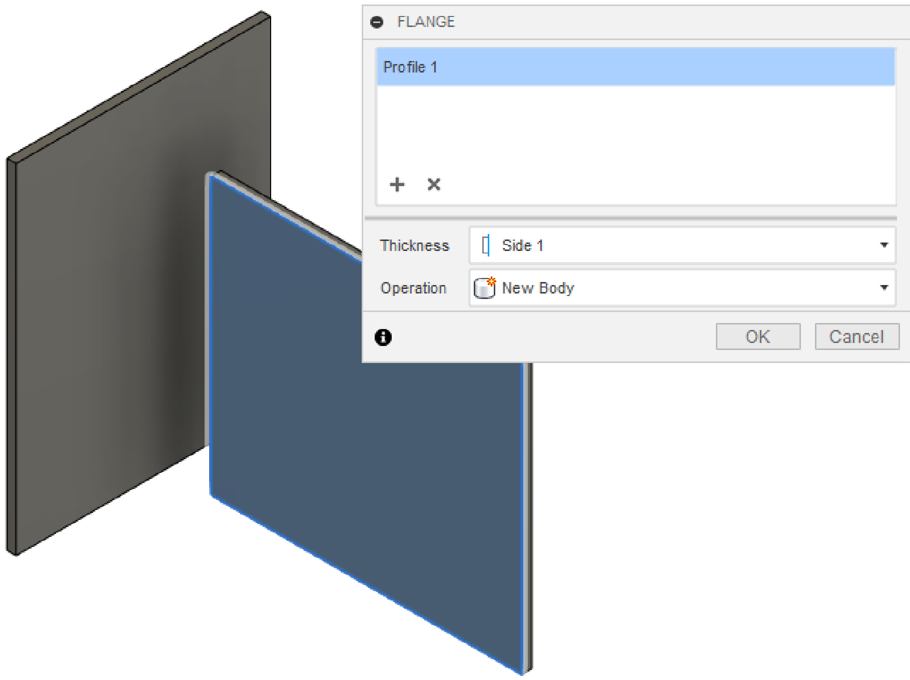




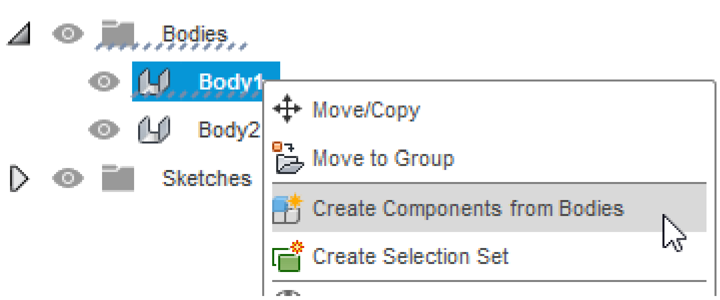
Once the feature is created, editing the feature will not allow for changing the active rule. You will have to make that change through the Rule option in the Browser.

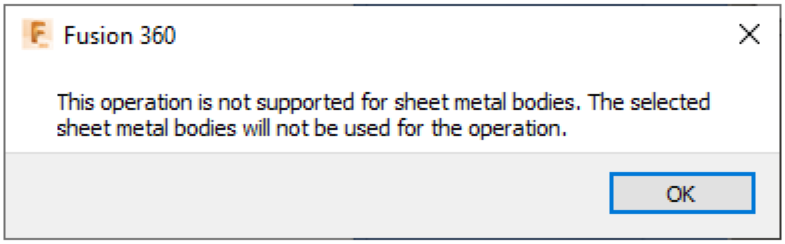


Even as subsequent flanges are added, the Sheet Metal Rule portion of the dialog will not be shown.



I do want to point out, that if you are working with more than one Sheet Metal Body, the only way to assign them different Sheet Metal Rules is to put them in different Components. The first time I attempted this, I was also surprised to find out that the Create Components from Bodies command and dragging a sheet metal body into an existing component are not supported for sheet metal bodies. Just be certain to create your components first before creating any sheet metal bodies.



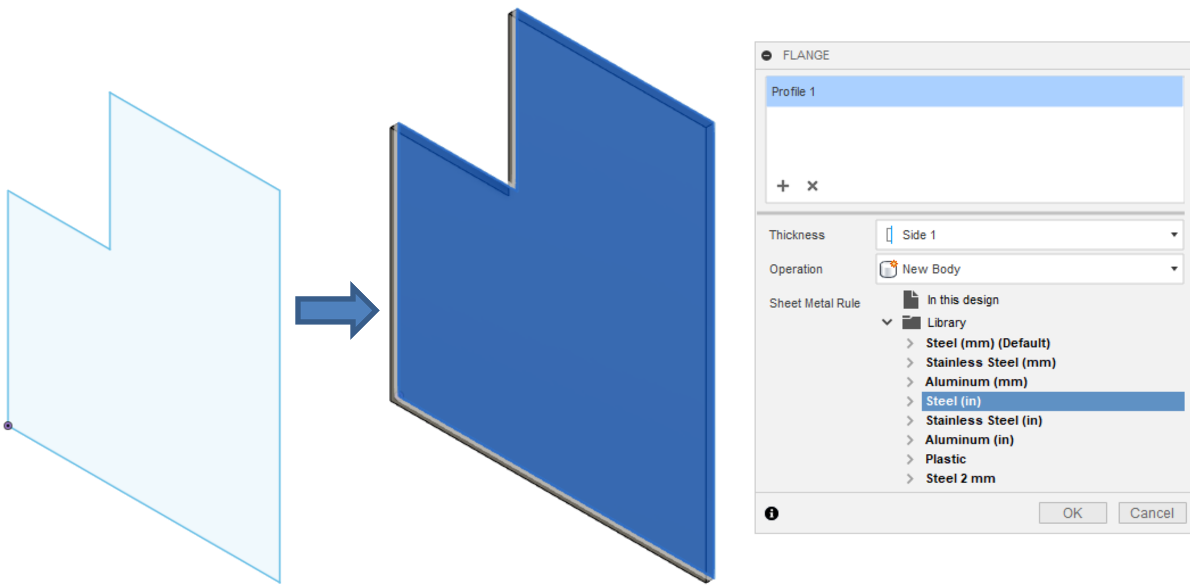


The Many Uses of the Flange Command

The Flange command in Fusion 360 can be used to create several different types of geometry. Having gotten my start in Inventor, I am used to commands like Face, Flange, and Contour Flange. It took me a bit to recognize all the different ways that the Flange command can be used. To help me explain the different uses, I will rely on the Inventor terminology to help explain the different ways the Flange command can be utilized.

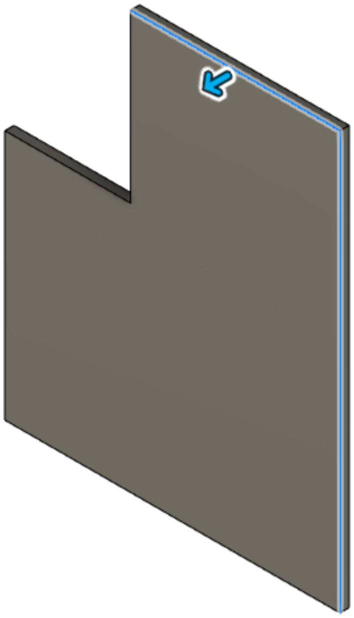
Face

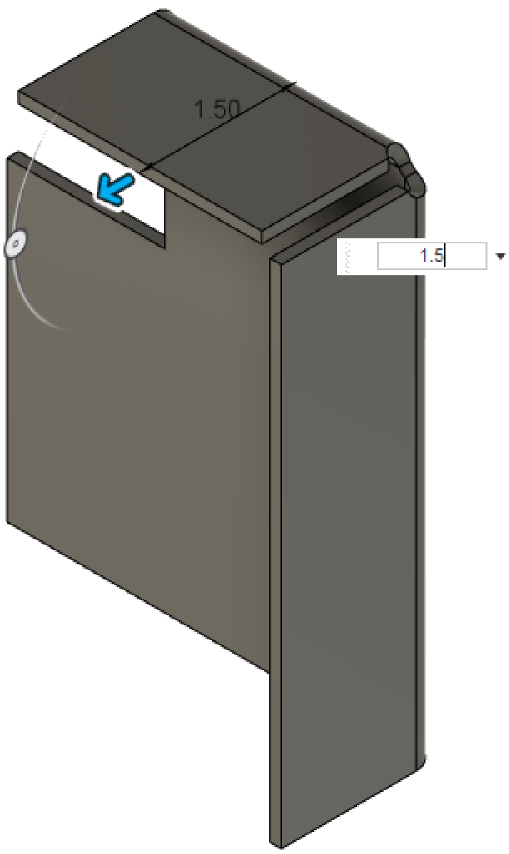
Fusion's Flange command can be used to add thickness to a closed 2D sketch profile. This is identical to Inventor's Face command. During the creation of this type of Flange, the user will be able to determine which side of the profile to add the material. If this is the first feature in the model, the user will also be able to select the appropriate Sheet Metal Rule.

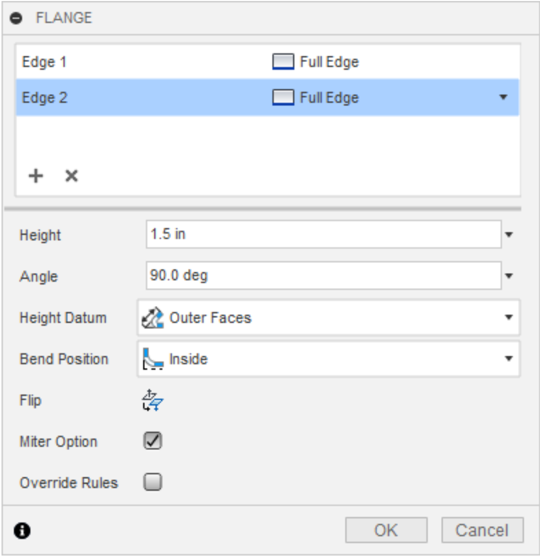


Flange

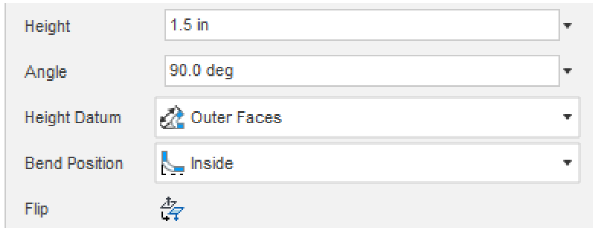
Fusion's Flange command can be used to add a bent flange to an existing body, this is equivalent to Inventor's Flange command. This type can only be used to add a feature to an existing sheet metal body. To create a Flange using this method, you just need to select an edge or edges where the flange will be created, then define the height and angle for the flange.



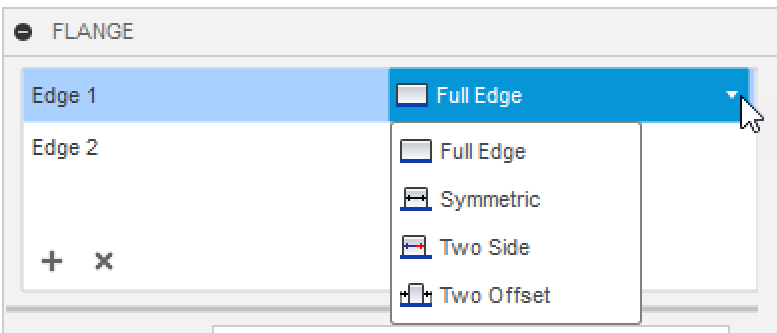

1.50
1.5


FLANGE
Edge 1 Full Edge
Edge 2 Full Edge
Height 1.5 in
Angle 90.0 deg
Height Datum Outer Faces
Bend Position Inside
Flip
Miter Option
Override Rules
OK
Cancel

Additional options are available to control where the bend starts in relation to the selected edge, how to measure the height of the flange, and flip the direction of the flange.

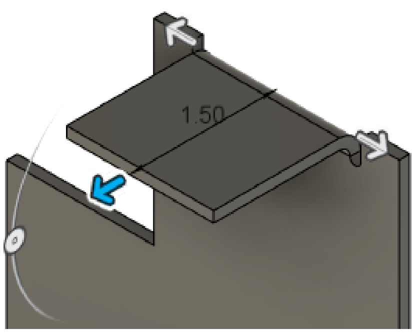


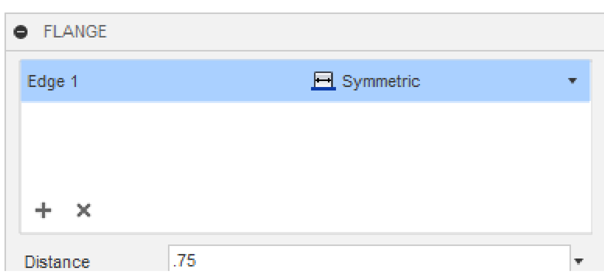
If two or more adjacent edges are selected, Fusion 360 will automatically add a corner relief and miter the corners according to the Sheet Metal Rule. Also by default, the flange will match the length of the selected edge, however, it is possible to change this by using the drop-down menu next to each selected edge.



<u>Symmetric</u>

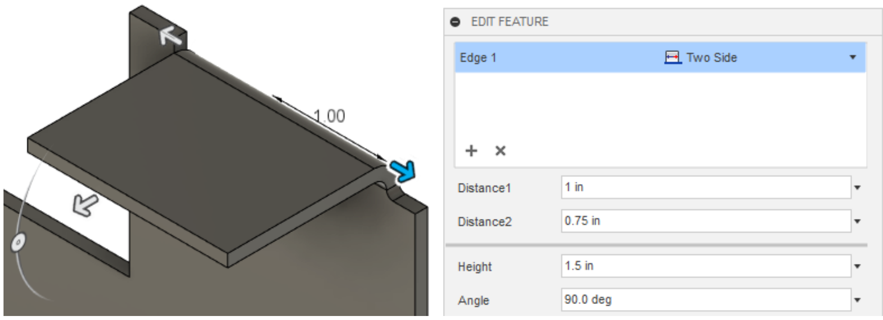
Fusion 360 will ask for a distance for the flange width and center the flange on the edge.





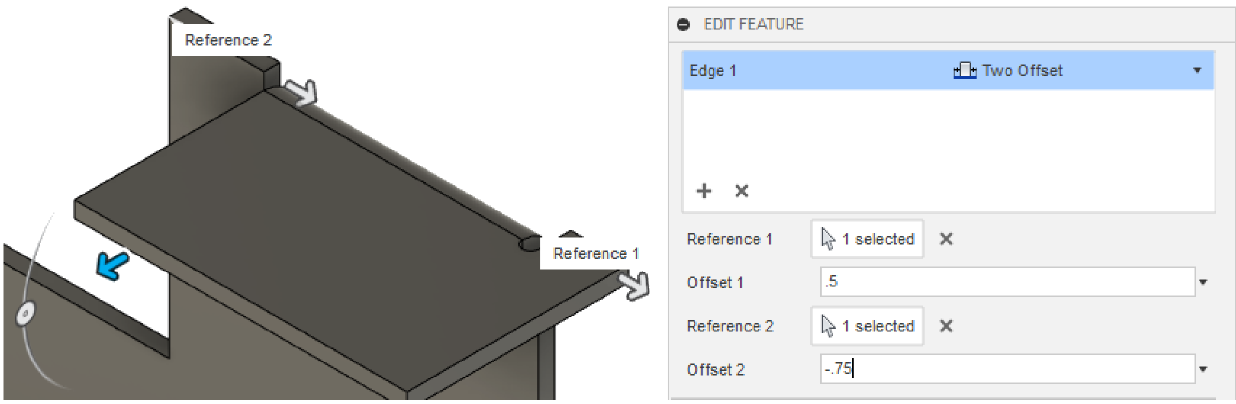
Two Side

Fusion 360 will ask for two distances, which will be the distance from the center of the selected edge to the start/end of the flange.



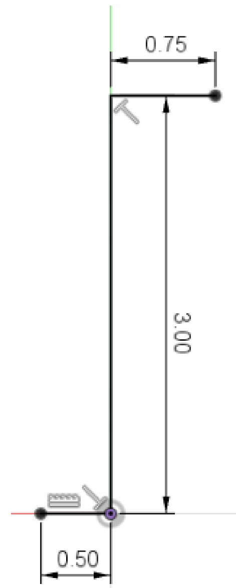
Two Offset

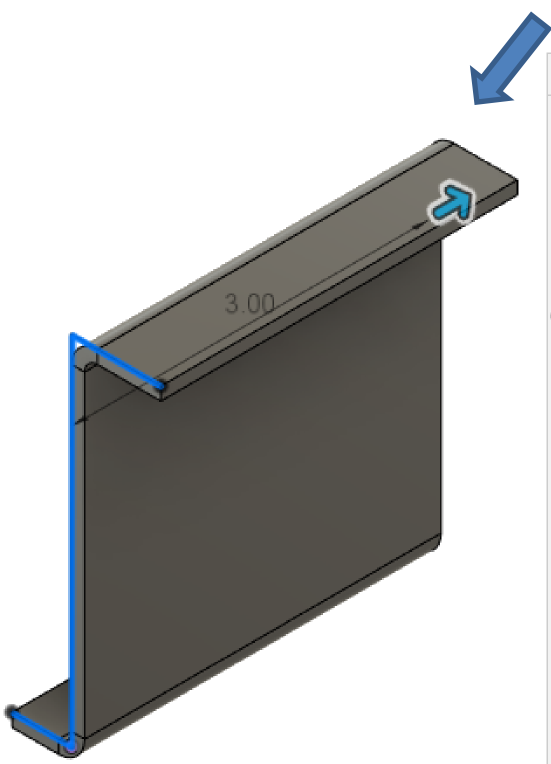
Fusion 360 will ask for two references and the distances from those references. By default, the references will be either end of the selected edge, but these can be changed as well. It is also important to note that positive offset values will put the flange past the end of the selected edge, while a negative value will move the end of the flange towards the middle of the selected edge.

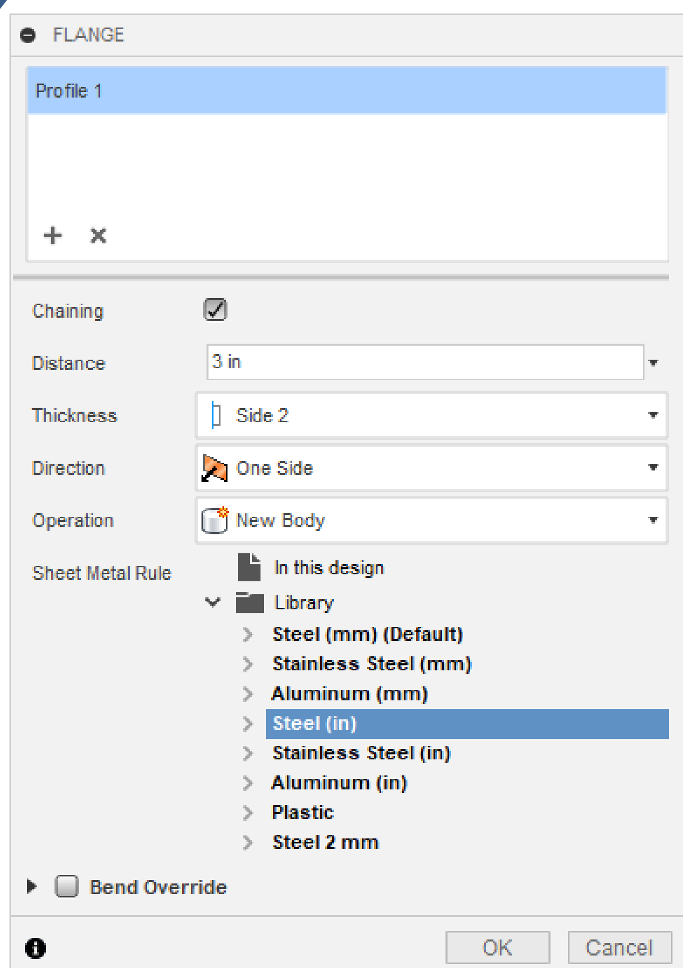


Contour Flange

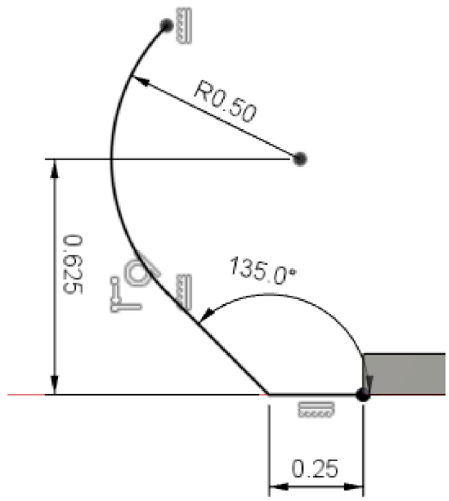
Fusion's Flange command can also be used on an open 2D sketch profile. You will not have to sketch the bend radii because Fusion 360 will use the default Bend Radius from the Sheet Metal Rule on any sharp corners in the sketch profile. In this case, you will be able to determine which side of the sketch profile to add the material thickness and you will also be able to specify the length of the flange being made.

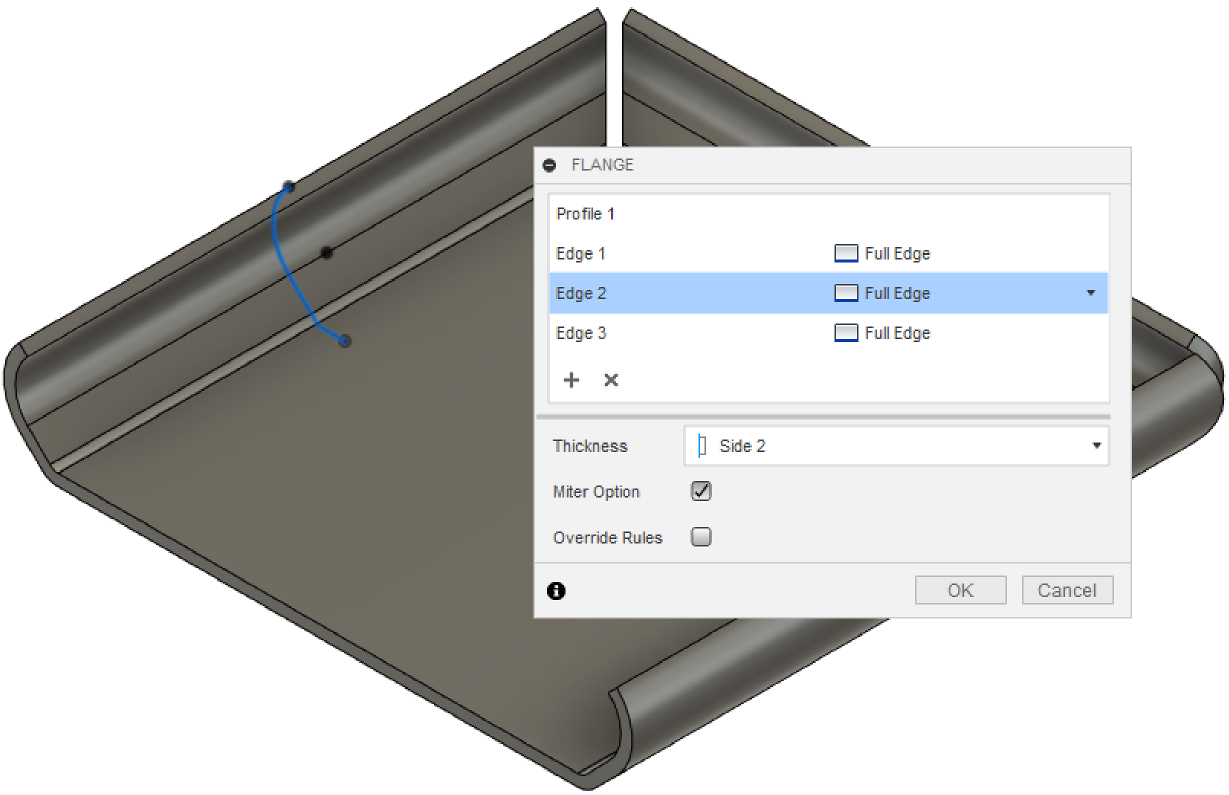

0.75
3.00
0.50


3.00


FLANGE
Profile 1
Chaining
Distance
3 in
Thickness
Side 2
Direction
One Side
Operation
New Body
Sheet Metal Rule
In this design
Library
Steel (mm) (Default)
Stainless Steel (mm)
Aluminum (mm)
Steel (in)
Stainless Steel (in)
Aluminum (in)
Plastic
Steel 2 mm
Bend Override
OK
Cancel

This method can be used to create a base feature or add a secondary feature. In the case of the secondary feature, the options are very similar to the flange type feature that was discussed in the last section.





When it comes to the Flange command, there are many different types of shapes that it can create. The software is smart enough to recognize your combination of selections and make the right type of flange based on those inputs. It took me a while to get used to it, but now I appreciate the simplicity of it. When I am describing it to other users, I just say, “Everything is a flange.” Well, not really, but you probably get what I mean.

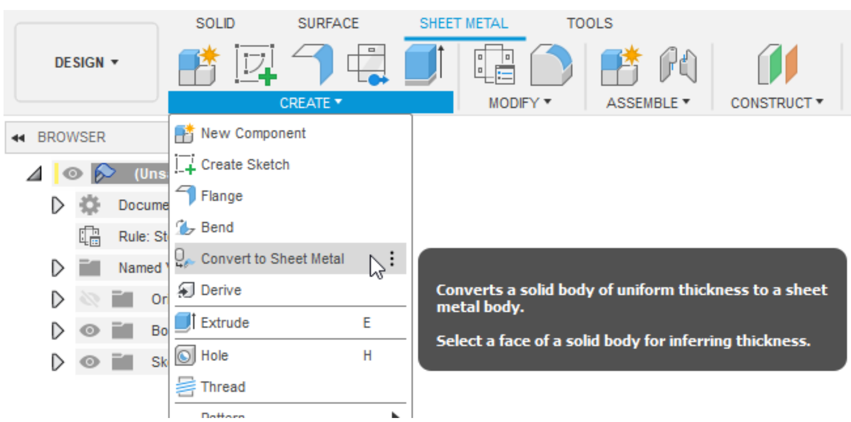
Reasons Why a Part Will Not Generate a Flat Pattern

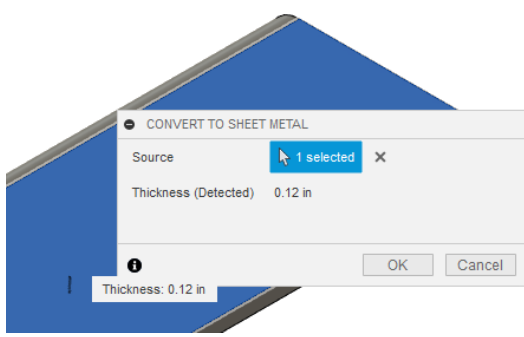
It does not happen often, in fact, Fusion 360’s workflow is pretty good at helping you not make the following mistakes. However, you might find yourself in a situation where your sheet metal part will not generate a flat pattern. There are a few scenarios where this could happen, so the following are the reasons why a part might not flatten and how to correct or avoid that situation.

The Part Was Not Created With Sheet Metal Tools

It is possible to correctly model a sheet metal component using Fusion’s Solid Modeling tools, however, it will not flatten. This is because Fusion 360 assigns a Sheet Metal Rule when the first Sheet Metal feature/body is created. Since that had not happened, Fusion doesn’t know what thickness to look for in the model.

In these cases, Fusion has a command to convert Solid Models to Sheet Metal models. During this process, the user can identify a face in the model and pick a Sheet Metal Rule. Then Fusion will use the thickness of that face as the material thickness for the selected Sheet Metal rule.





The Part is Not a Uniform Thickness

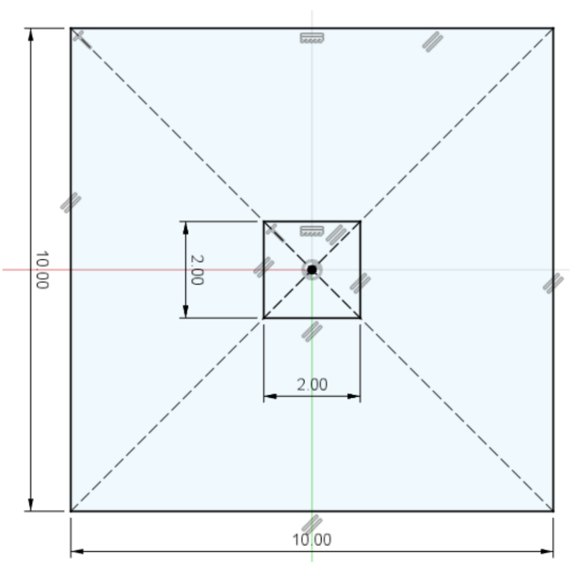
This can occur in one of two scenarios, on parts modeled with Solid Modeling tools or on imported components. For parts modeled with solid modeling tools, it would be necessary to find the sketch(s) and/or feature(s) that created the differing wall thickness and correct them. For imported components, you will have to use the measure command to determine which faces differ from the rest of the model, then use Move/Copy to move the faces to correct those issues.

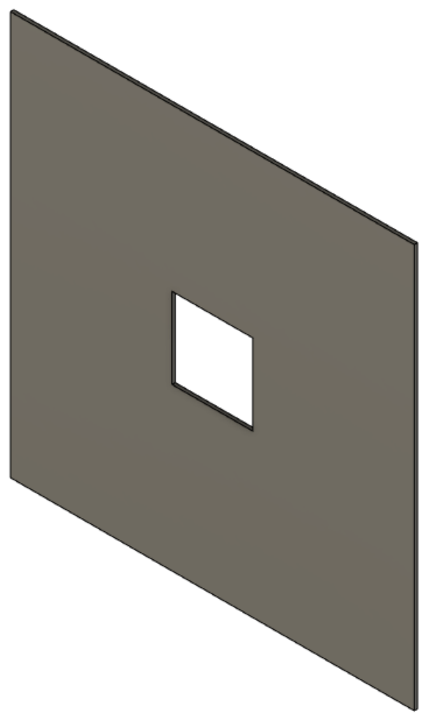
An alternative solution would be to use a surface and Boundary Fill to cut the part to the uniform thickness, then convert the part to a Sheet Metal part.

Flattened Flange/Bend Intersects or Overlaps Another

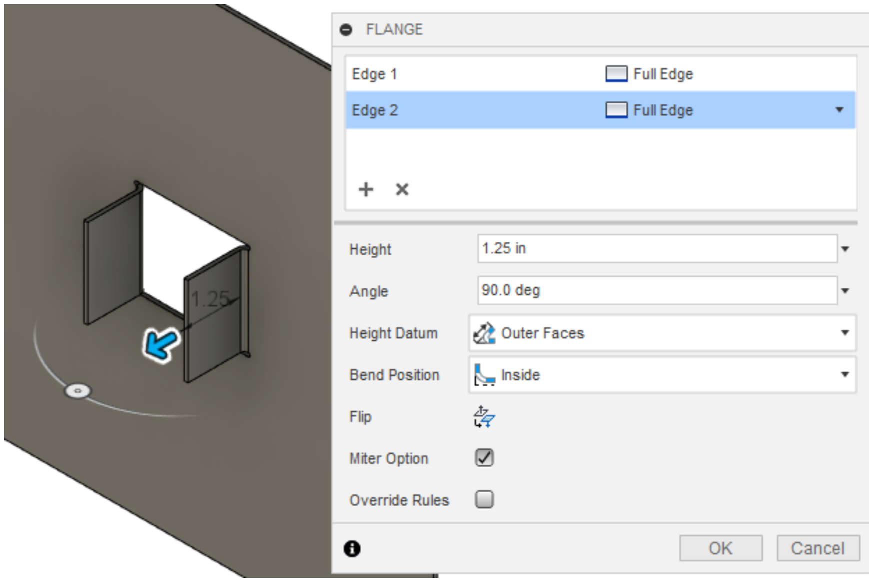
It is possible to create flanges that would interfere with one another when in a flattened state. If you have a knockout or any notches that get flanges, you need to be careful with the height value on those flanges. The Flange feature will not fail if the flanges will interfere in the flat pattern. You will not see that error until you generate a flat pattern.

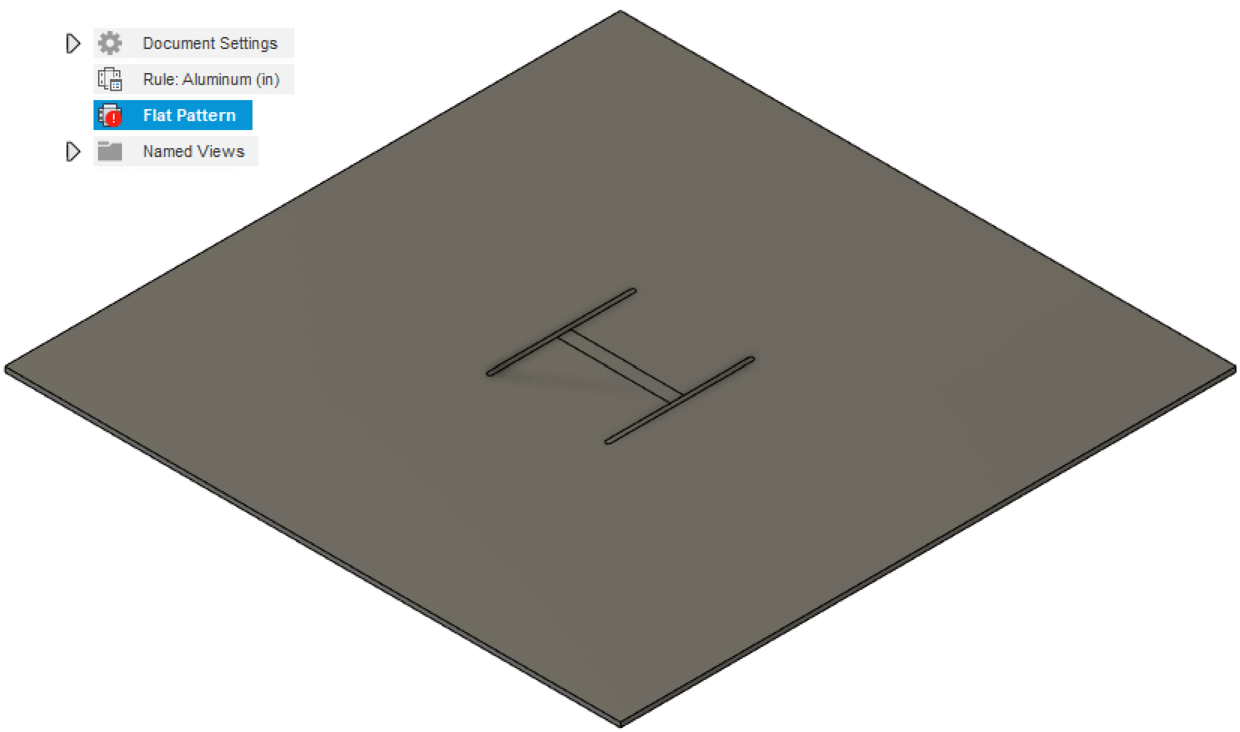
In this example, I created a flange from the following sketch.



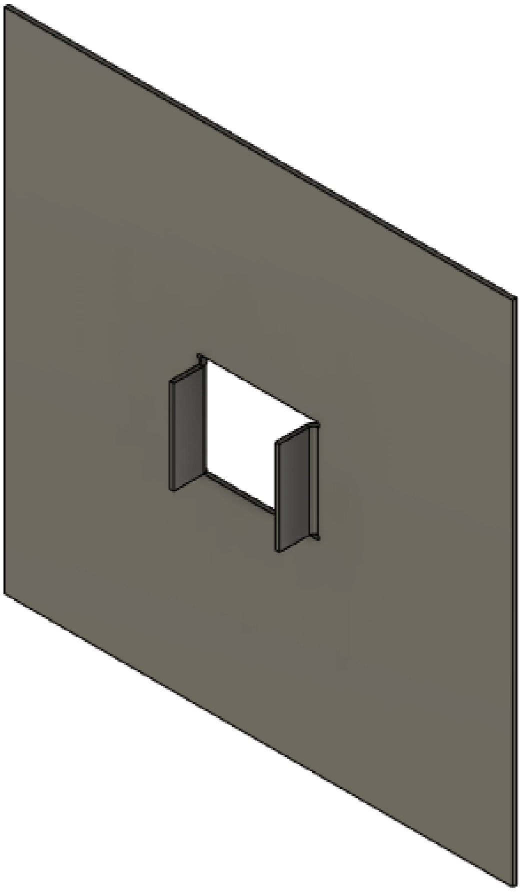


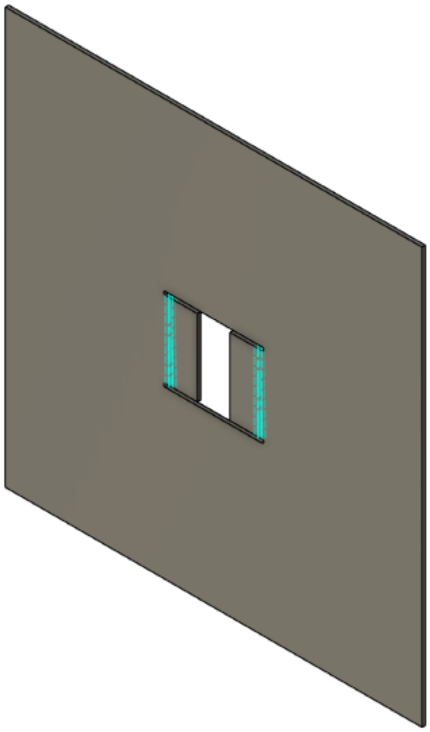
If I create flanges on the knockout that are 1.25” in height, there will not be enough space for the flanges to flatten. I will not receive an error on the feature, but the flat pattern will have an error.





Then I can shorten the height of those flanges to eliminate the error from the flat pattern.

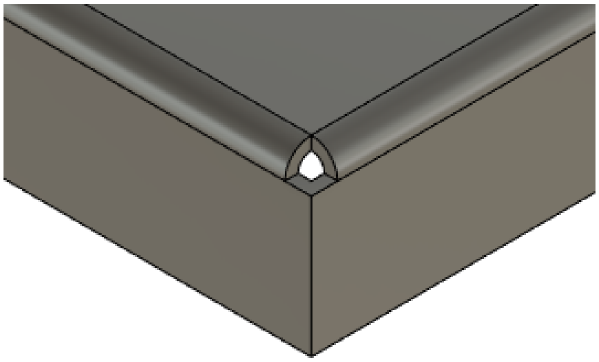




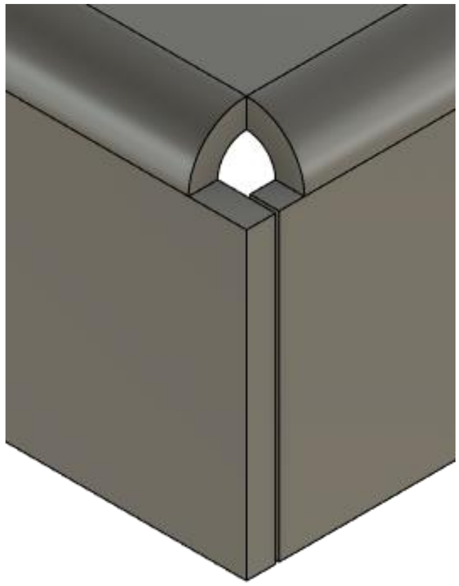
Zero Distance Gap between Flanges

Often in Sheet Metal, we want two edges to touch and close off a region. When Fusion sees a zero distance between two faces, it blends them into one face. My recommendation in these cases is to use a distance of .001. That will get the two faces extremely close together, but leave enough of a gap that Fusion doesn't blend the faces.

This is an example of two flanges that don't have a gap, this part will not flatten.



Here is the same part with a gap of .001, this part will flatten.

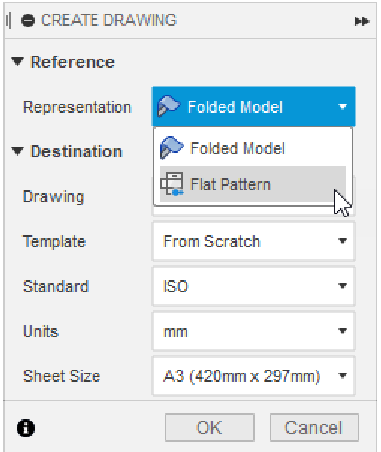


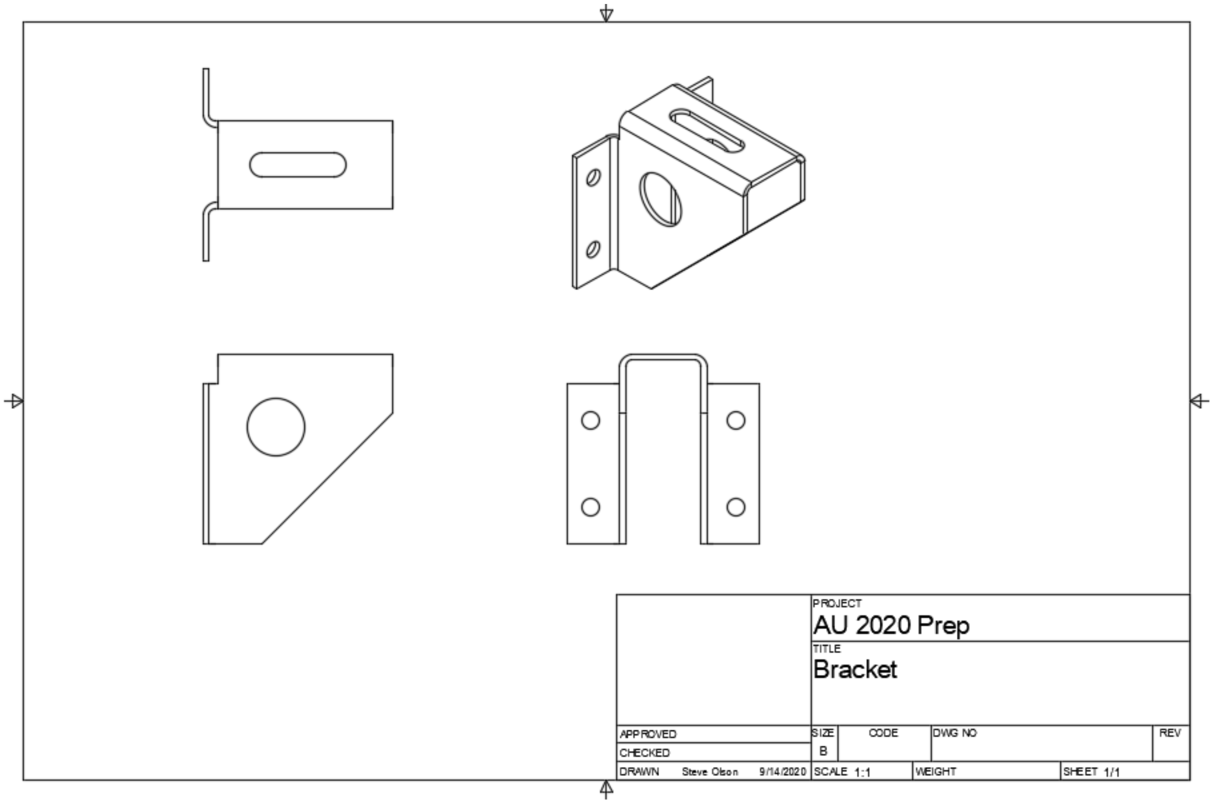
Documenting Formed Parts and Flat Patterns

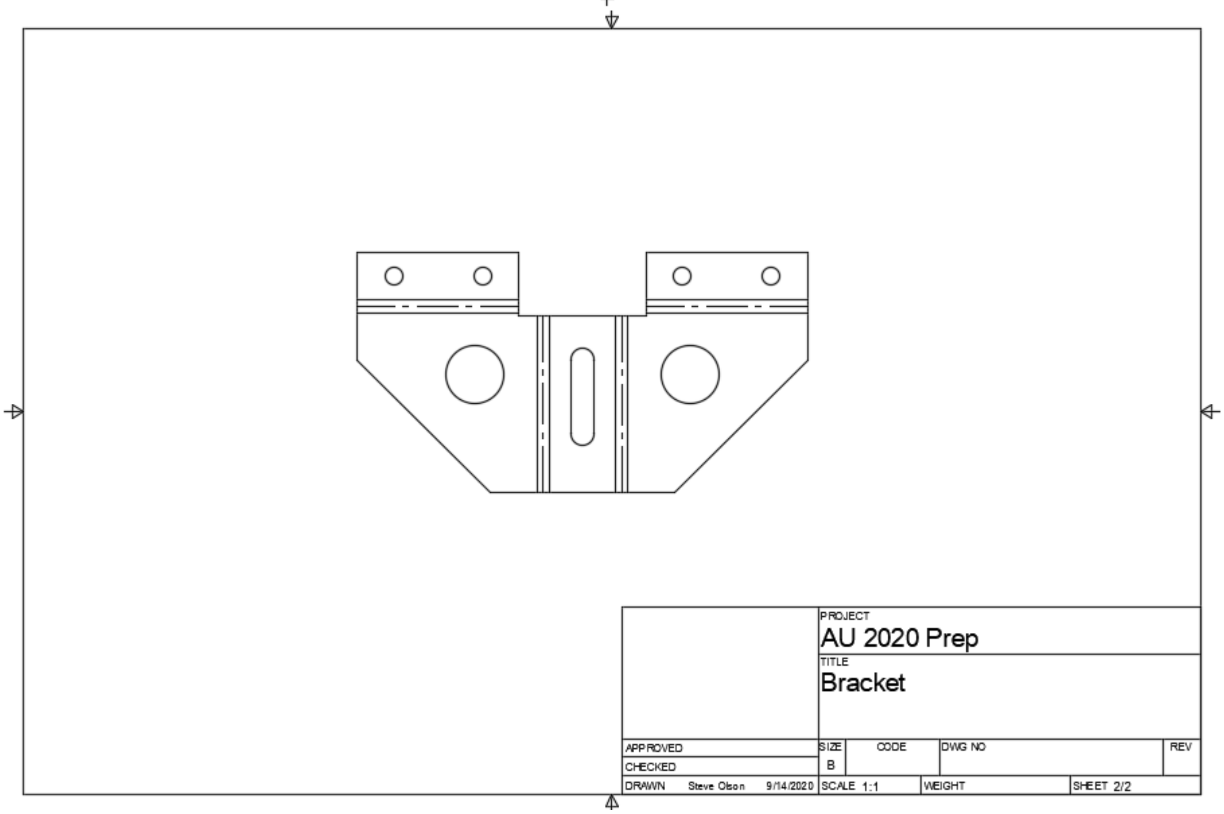
To properly document a sheet metal component, it is necessary to show the formed and flattened versions of the component. I like to say that Fusion 360 sheet metal components are like duplex parts. They contain both for the formed and flattened states.

Creating Drawing Views

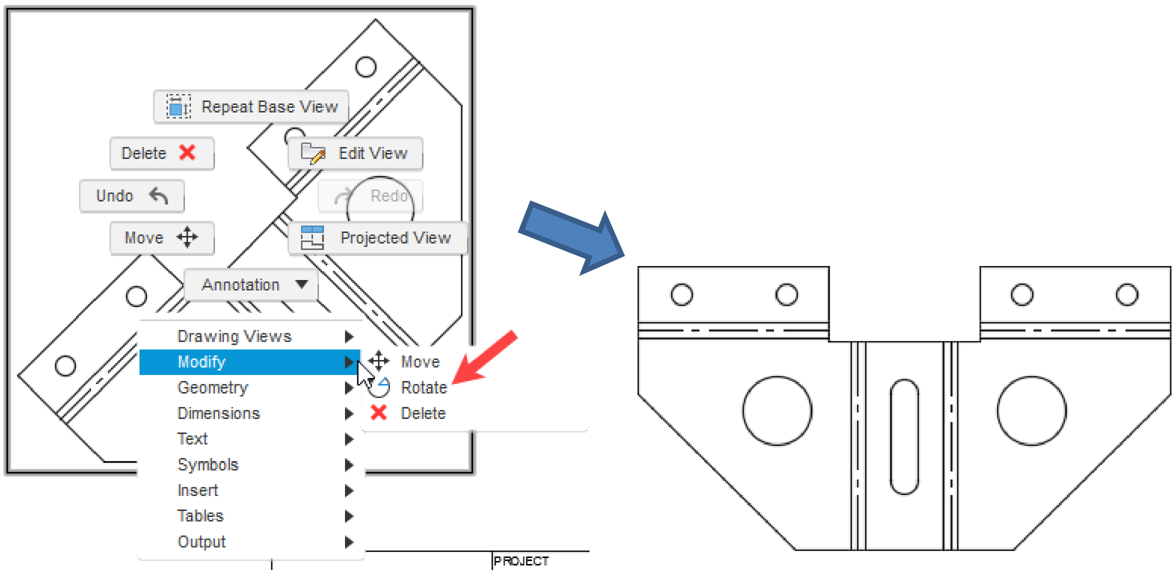
When creating your drawing views, the software will ask you which version of the part the view should be based on. As long as the flat pattern has been generated, it will be offered as an option for the drawing view.




PROJECT
AU 2020 Prep
TITLE
Bracket
APPROVED
CHECKED
DRAWN Steve Olson 9/14/2020
SIZE B
CODE
DWG NO
REV
SCALE 1:1
WEIGHT
SHEET 1/1


PROJECT
AU 2020 Prep
TITLE
Bracket
APPROVED
CHECKED
DRAWN Steve Olson 9/14/2020
SIZE B
CODE
DWG NO
REV
SCALE 1:1
WEIGHT
SHEET 2/2

One common issue when working with sheet metal flat pattern views is making sure the view is rotated to the desired position. If the flat pattern view is not oriented correctly, it is just a matter of using Rotate to rotate the view to the correct angle.

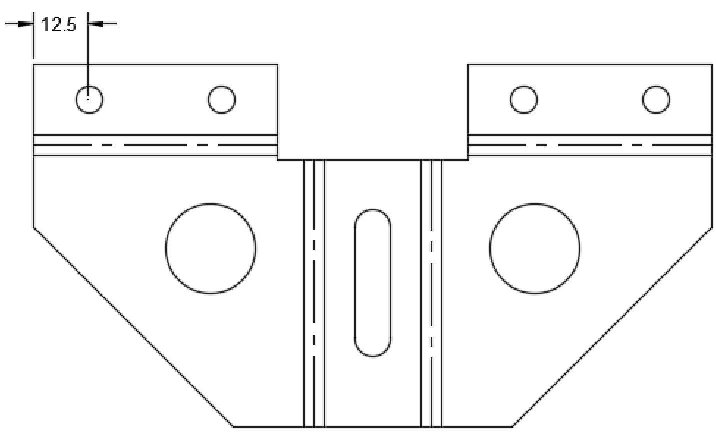


Creating Annotations

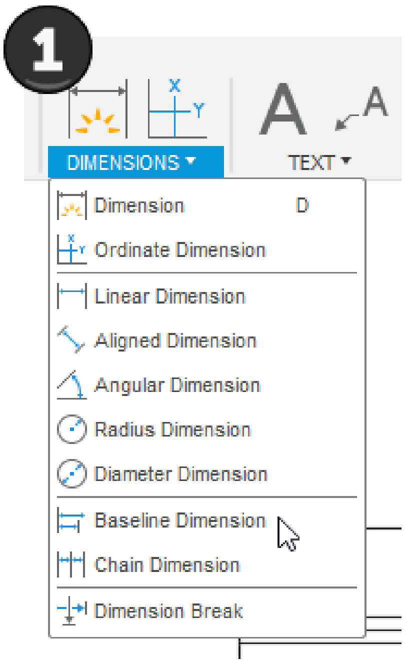
Every company has different standards for how to document flat patterns. At my previous job, for a time we used Baseline dimensions for all of our flat patterns. Then we changed from Pro-E to Inventor. We found that some of our drawings got crowded by the baseline dimensions, so we transitioned to Ordinate dimensions.

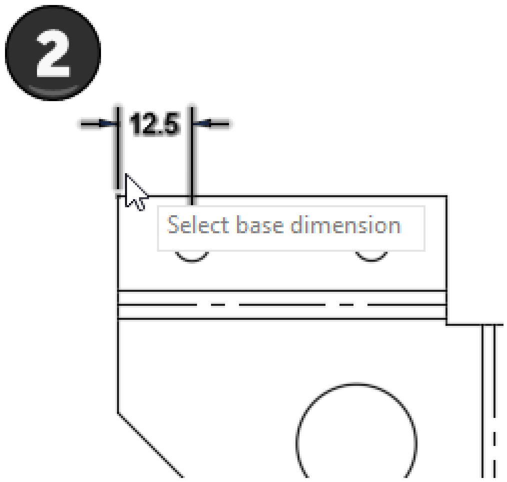
<u>Baseline Dimensions</u>

To create baseline dimensions in Fusion 360, you first need to have a linear dimension that Fusion 360 will use to interpret the origin.

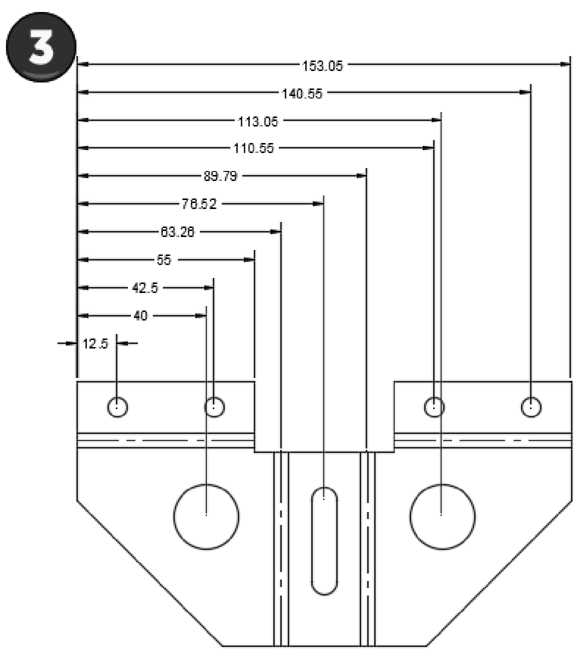


After you launch the Baseline Dimension command, you will be asked to select a reference dimension. You will want to select close to the extension line that will be the origin of the baseline dimensions.

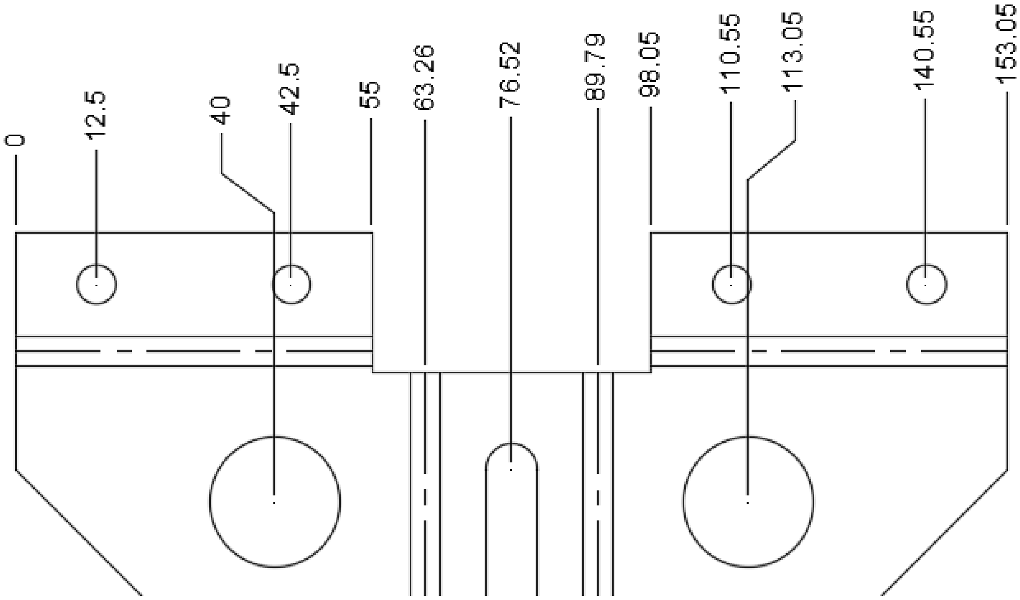




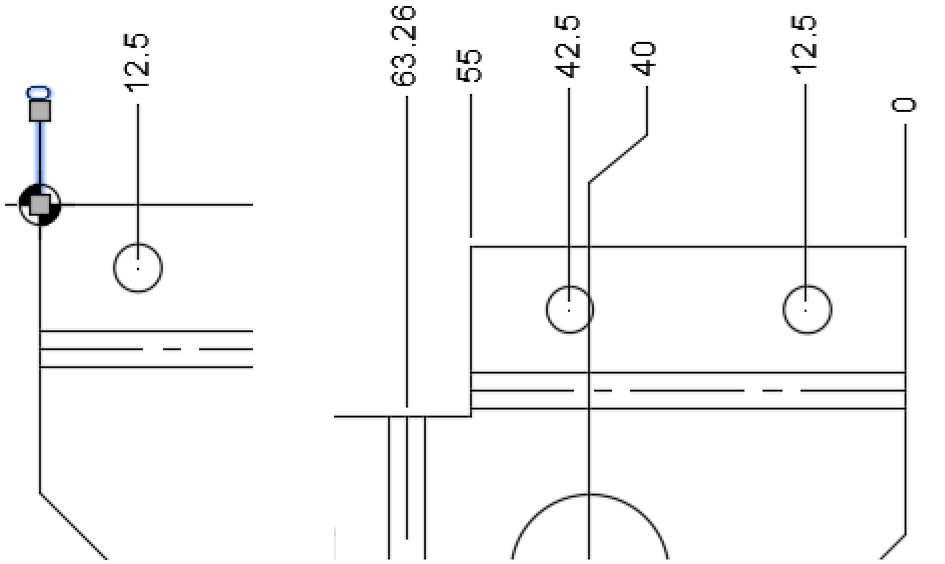
Then you can begin selecting references and placing dimensions. The command will loop until you right-click and select Ok.



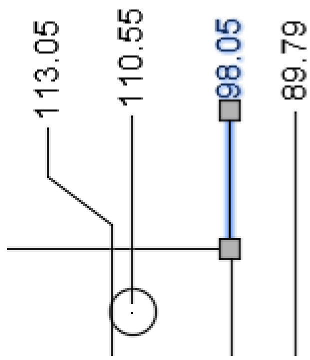
Ordinate Dimensions
The main element to be aware of when creating Ordinate dimensions is that the first dimension in the view is going to determine the location of the origin for the view. Every other ordinate dimension in the view will be measured back to that point.



If the origin needs to be moved, selecting the 0 dimension will also display the Origin Indicator, which the user can select and reposition.



Likewise, after placement, all ordinate dimensions will have grips for the reference attachment and text location.



Conclusion

Sheet Metal modeling and documentation is a very in-depth topic, however, these skills are the foundation of the proper techniques and workflows. Sheet Metal Rules drive many attributes of the design and give a single point of access to change the default characteristics of the design. When modeling, just remember, “Everything is a flange.” Even though it is possible to create components that will not flatten if you stick to the Sheet Metal commands and standard workflows you should have success when generating your flat patterns. When it comes time to create your drawing views, remember that sheet metal components are duplex parts and contain both the formed and flattened state of the part. Hopefully, using the information in this document as a guide, you will get a jump-start toward mastery of modeling and documentation of your sheet metal designs.