

MFG225530

Sheet Metal Like a Pro with Fusion 360

Robert Savage

Advanced Solutions

Ron Jones

Advanced Solutions

Learning Objectives

- Learn how to use the flange tool to create faces and flanges
- Learn how to create a cutout across multiple faces
- Learn how to create, modify, and apply Sheet Metal Rules in Fusion 360
- Learn how to build a flat pattern and use it in a drawing

Description

When it comes to making sheet metal parts, there are distinct and complex issues that you run into. Solving those issues are unique to the software that you are using. Last year, Fusion 360 software added sheet metal tools to its already large assortment of tools (such as CAM simulation).

In this class, we will explore some complex sheet metal components and how the Fusion 360 sheet metal tools can be used to address these complex features like extended flanges, punch tools, and complex flanges. From there, we will complete the process by creating a flat pattern and using it to make a drawing.

Speaker(s)

Robert Savage

Education Specialist

Advanced Solutions Inc.

I have been doing Mechanical Design since 1984 and I have used several different products to do my designs, during that time I have had the good fortune to work on many interesting projects. In early 1999 I was using Mechanical Desktop to do Casting Molds when I was approached by the local reseller about testing a new product that was coming on to the market later that year called Rubicon (Autodesk Inventor) that changed things for me and the way I did design. I have used in some way every release since that time.



Ron Jones

Engagement Manager

ADVANCED SOLUTIONS

As a kid, I was known for taking apart; toys, electronics, and even my first computer, all to understand what makes them work. That kid is now an accomplished adult who specializes in helping companies use Autodesk software to its full ability. I still take things apart, only now it's called reverse engineering... I currently reside in Carmel, Indiana with my fiancé and Labrador Retriever "Juice". Previously I've worked as a product engineer in the automotive industry, mechanical designer in the automation industry, and an applications engineer in the compressed air industry. My educational background is in engineering and drafting from Purdue University. I am professionally certified in Autodesk Inventor Professional, Autodesk Fusion 360 and an Autodesk Certified Instructor. Recently I was honored with the rank of Autodesk Expert Elite.



Introduction to Fusion Sheet Metal

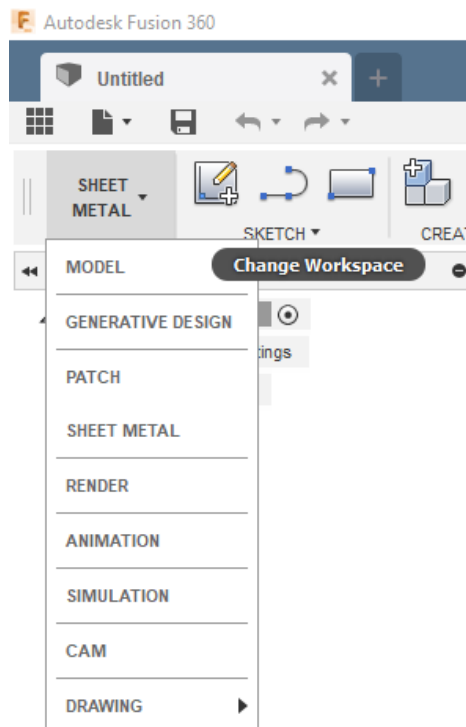
Sheet Metal parts are created to define components that are rules based. These rules are made to define things like a consistent thickness, consistent bends, and consistent reliefs. When a sheet metal part is made in software it is made opposite the way the part is built in the shop.

When it is designed it is designed as a finished part as a folded model the flat pattern is developed from the folded part. When a part is made in the shop it starts with a flat piece of metal that is cut into the shape of the flat pattern then that is folded into the finished part.

Sheet Metal Workspace

When starting a new file in Fusion 360 you start with a base file meaning that it will be modified to what you need it to be, that includes changing it from a part to an assembly and using the workspaces to define whether the part is a modeled part, T-Spline part, or a Sheet metal file. You can change that part to an assembly by right clicking in the browser and adding a New Component.

To change the workspace you select the model drop down and select the desired workspace. Fusion 360 uses these workspaces to define the environment and tools that you are using to create your part.

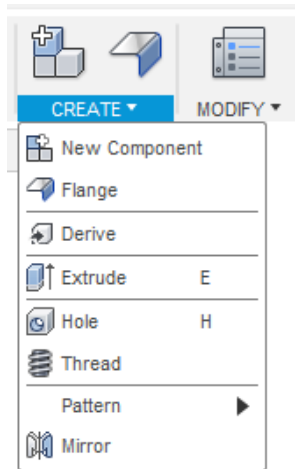


[Change Workspace](#)

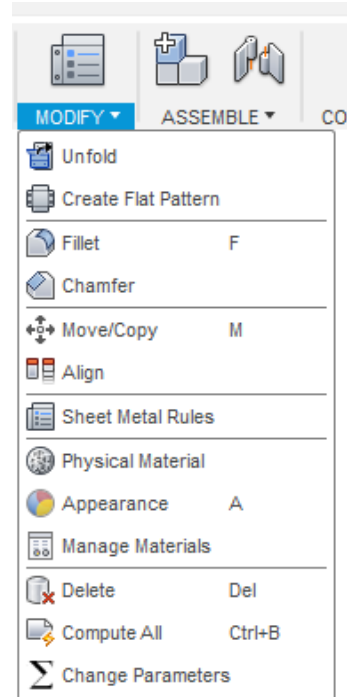
Sheet Metal Ribbon

The Sheet Metal ribbon has several sections to it for this class we will be focusing on 2 main sections the create section, and the modify section. The headers are self-explanatory to what the tools they contain do.

Create Section



Modify Section



Using the tools shown here we are going to create a sheet metal part and a flat pattern that can be used to machine the part or create a drawing from.

Let's get started.

Faces and Flanges

When creating faces or flanges you are using the same tool it is called the Flange tool and it can be used to create faces, edge flanges, and contoured flanges.



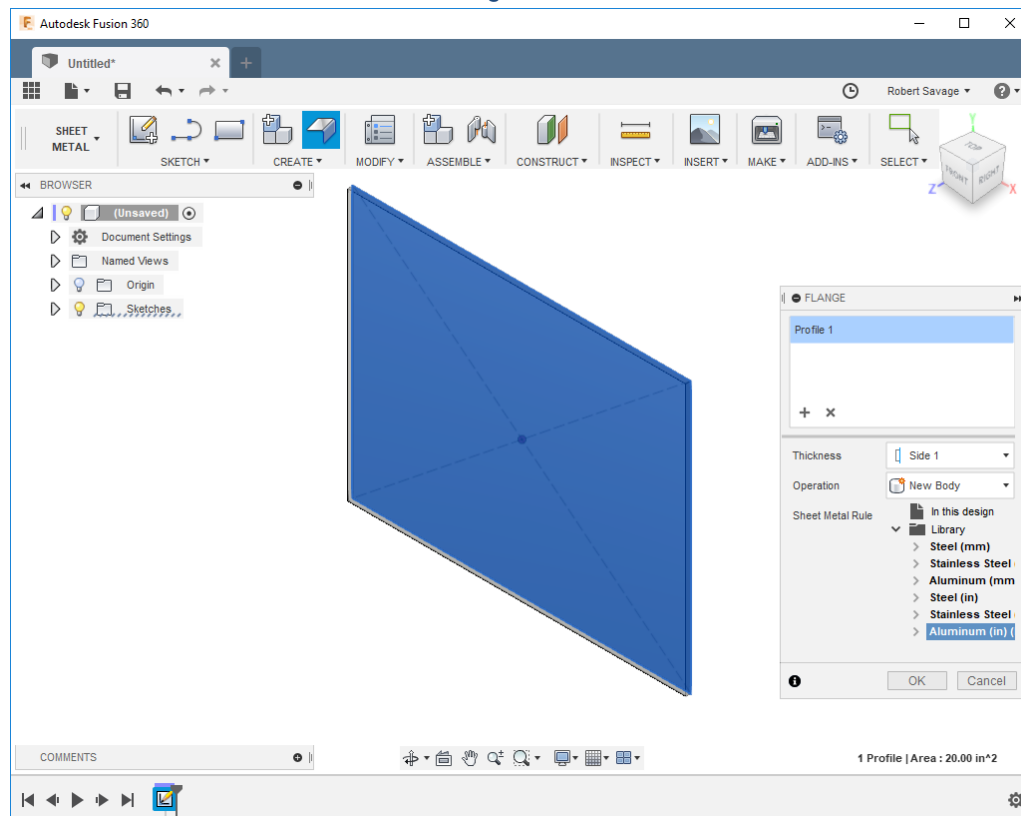
Face

A face is generally defined as a closed loop sketch that defines a flat face of the part, they are considered a base feature.

The process is:

1. Start your sketch.
2. Select the plane or face you want to start on.
3. Sketch the desired closed profile.
4. Select the flange tool from the create section.
5. Select the closed loop.
6. Define the side you are adding material to.
7. Select OK

Creating a Face



Edge Flange

The edge flange is created as a secondary feature because it is applied to edges of a face. It allows the user to select a single edge or multiple edges of a face and apply flanges off the selected edges.

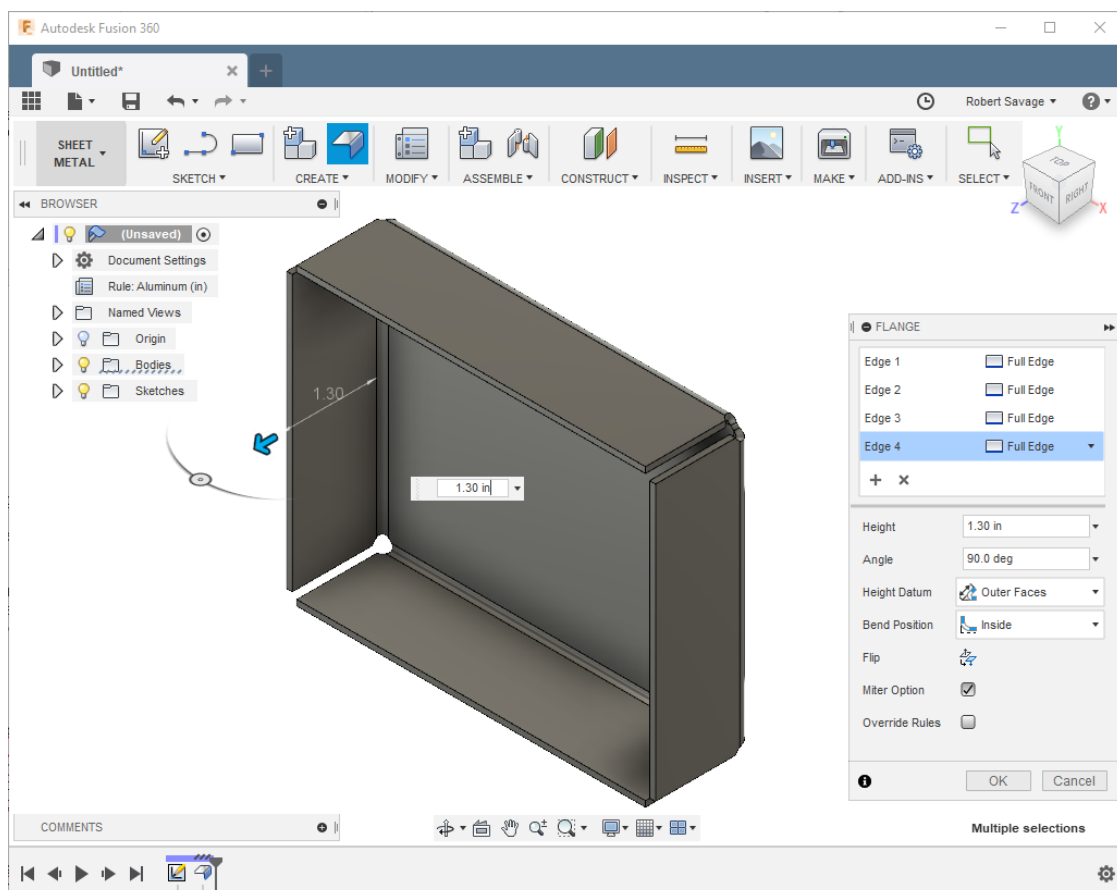
In the Flange dialog box, you can define the height of the flange, the flange datum or how the height is measured, the angle of the flange, the bend position, and the direction of the flange.

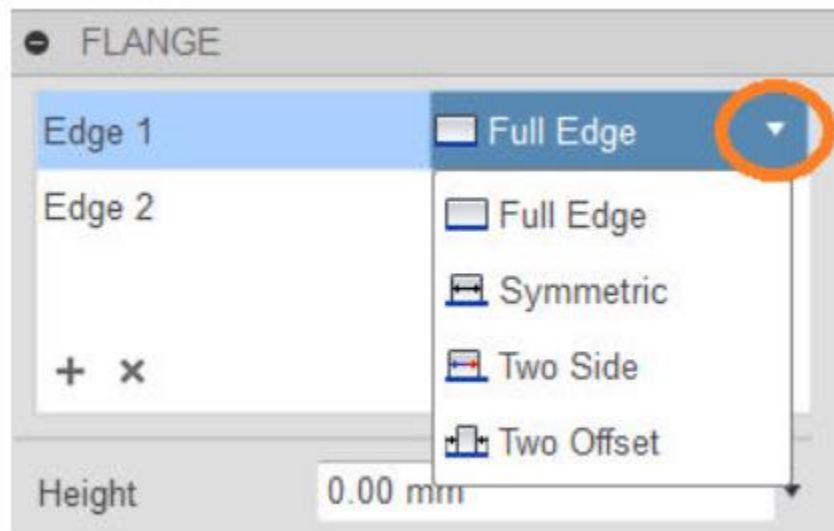
The flange extent option allows the user to say if the length of the flange is covering the full length or a section of the length.

The process is:

1. Select the flange tool from the create section.
2. Select the edges of the face.
3. Define the distance and angle of the flange.
4. Define the Extent of the flange.
5. Define the direction and side of the flange.
6. Select OK

Edge Flange





Edge Extents

Edge Extents

Edge extents can be defined on a per-edge basis and the options are Full Edge, Symmetric, Two Sides or Two Offsets.

The full edge option allows you to define based off the length of the edge.

The symmetric option allows to place a flange centered on the edge and define a length of the flange.

The two side option places a flange centered but allows the user to define the distance from the center of each side.

The two offset allows you to define the distance of the flange from each end of the edge selected.

Contour Flange

A contour flange can be a base feature or a secondary feature depending on whether it is the first feature or is created after a base feature.

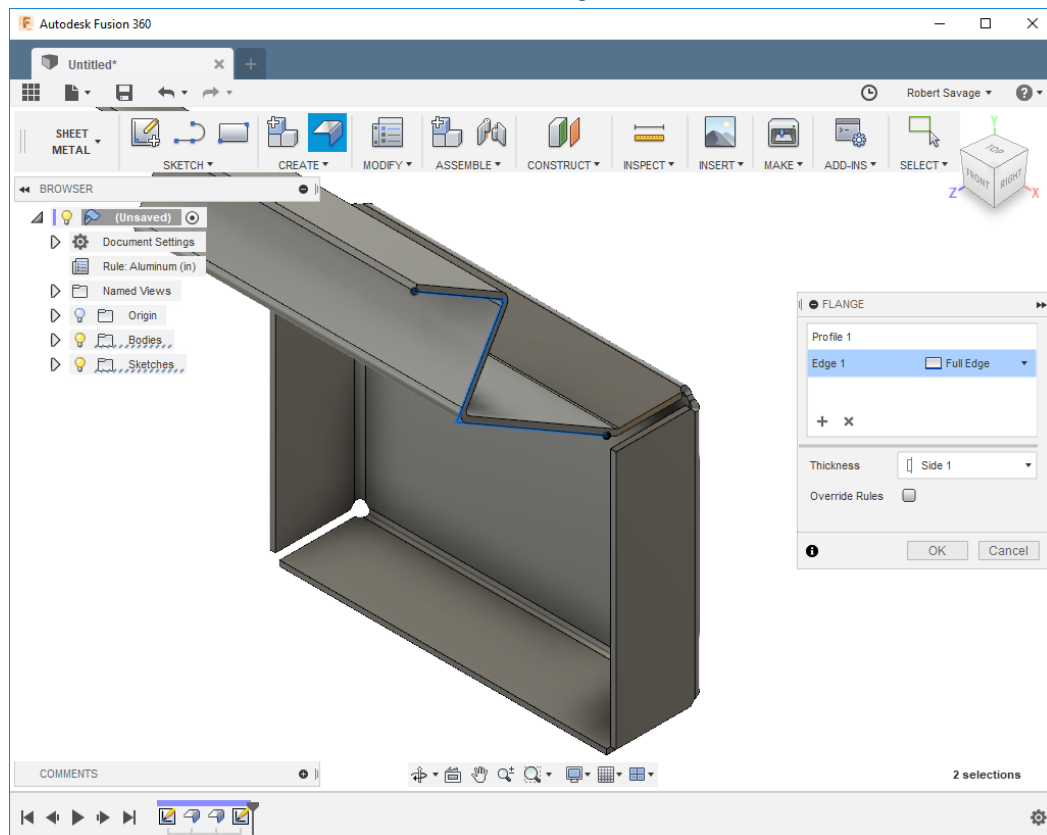
It allows the user to select an open loop sketch or profile and define a depth either in the dialog box or by selecting an associated edge.

You can also set which side of the sketch you are applying the material to.

The process is:

1. Start your sketch.
2. Select the plane or face you want to start on.
3. Sketch the desired open profile.
4. Select the flange tool from the create section.
5. Select the sketched open loop or the profile.
6. Define the extent with a dimension or select the edge to associate to.
7. Define the side you are adding material to.
8. Select OK

Contour Flange



Cuts

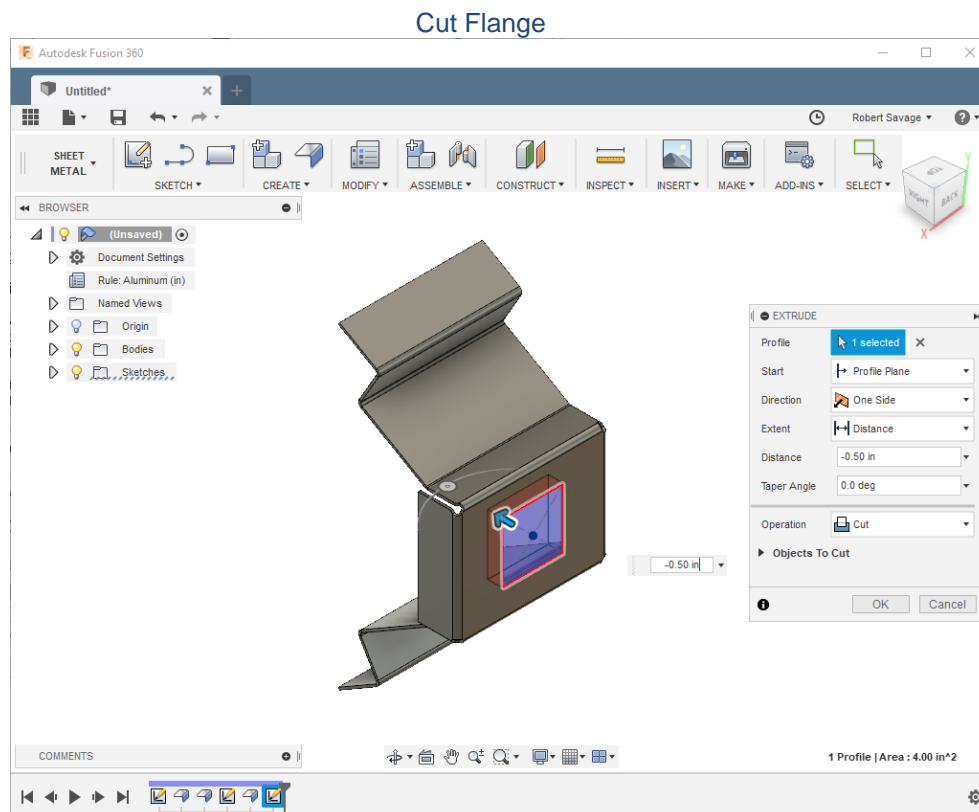
In this section we are going to talk about tools that are used to make openings in the sheet metal part. They fall into two categories sketch based or pick and place features. These tend to fall into two groups cuts and holes.

Cuts

Cuts are sketch based features this is where you define a closed loop sketch and then use that sketch to define an opening on one or more faces of the sheet metal part.

The process is:

1. Start your sketch.
2. Select the plane or face you want to start on.
3. Sketch the desired closed profile.
4. Select the extrude create section.
5. Select the closed loop.
6. Define the side you are adding material to.
7. Define the depth of the extrusion.
8. Select OK



Cuts Across Multiple Faces

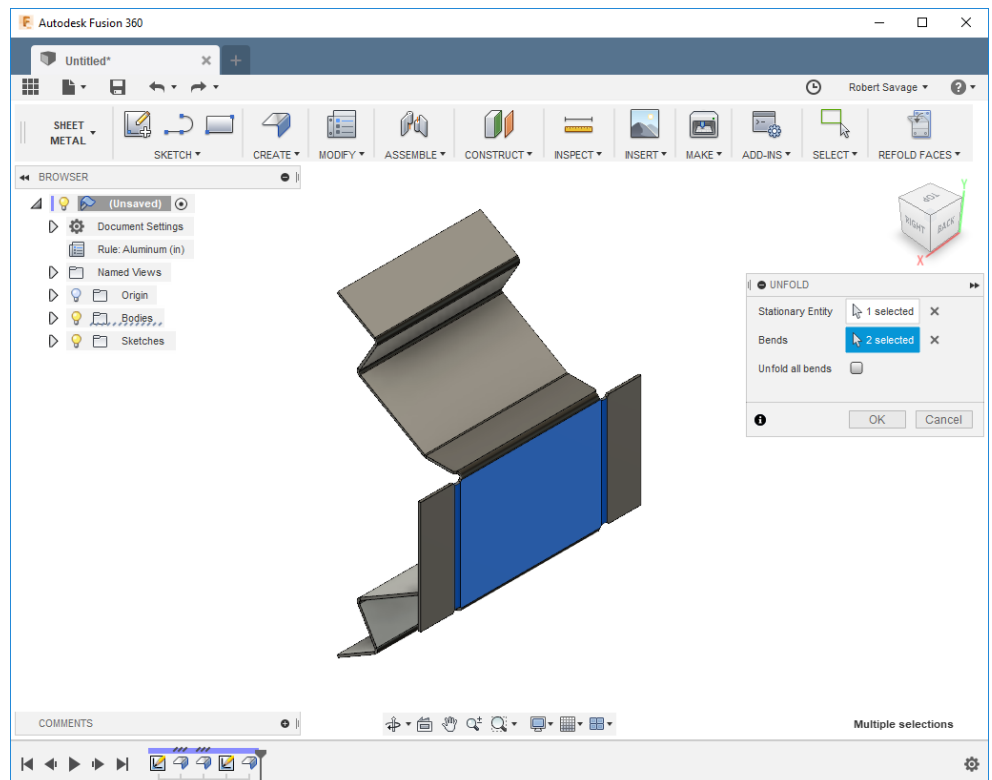
If your cut crosses multiple faces or flanges the you will add in the Unfold tool which is in the modify tools section.



The unfold tool allows the user to flatten selected bends on a part so that the sketch can extend across multiple sections of the part. The after doing your cut you use the Refold tool to return it to the folded model.

The process is:

1. Select the Unfold tool
2. Select the face that will be stationary
3. Select the bends that you want to flatten.
4. Start your sketch.
5. Select the plane or face you want to start on.
6. Sketch the desired closed profile.
7. Select the extrude create section.
8. Select the closed loop.
9. Define the side you are adding material to.
10. Define the depth of the extrusion.
11. Select OK
12. Use the refold tool to return it to its folded state.

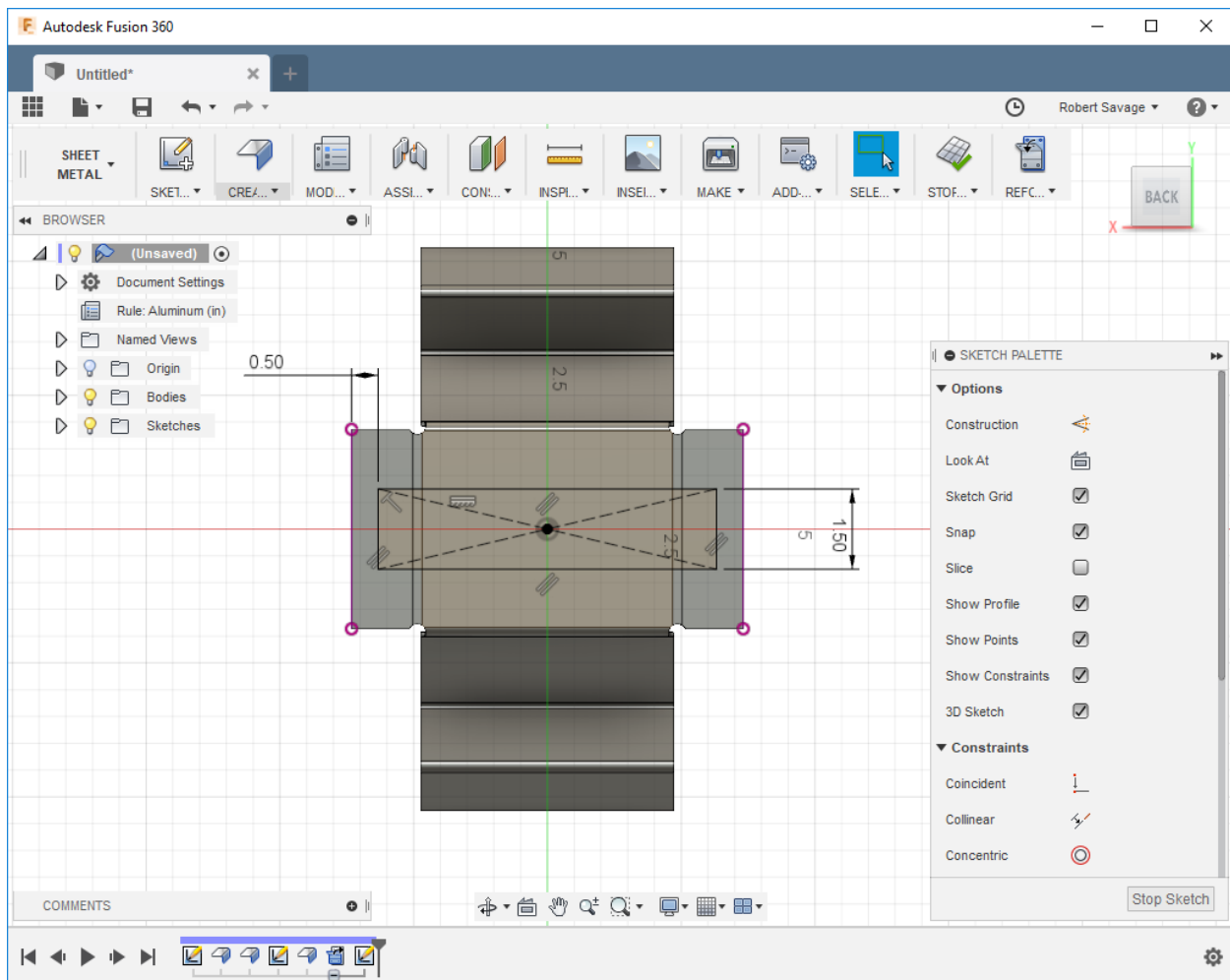


Unfold / Refold Tool

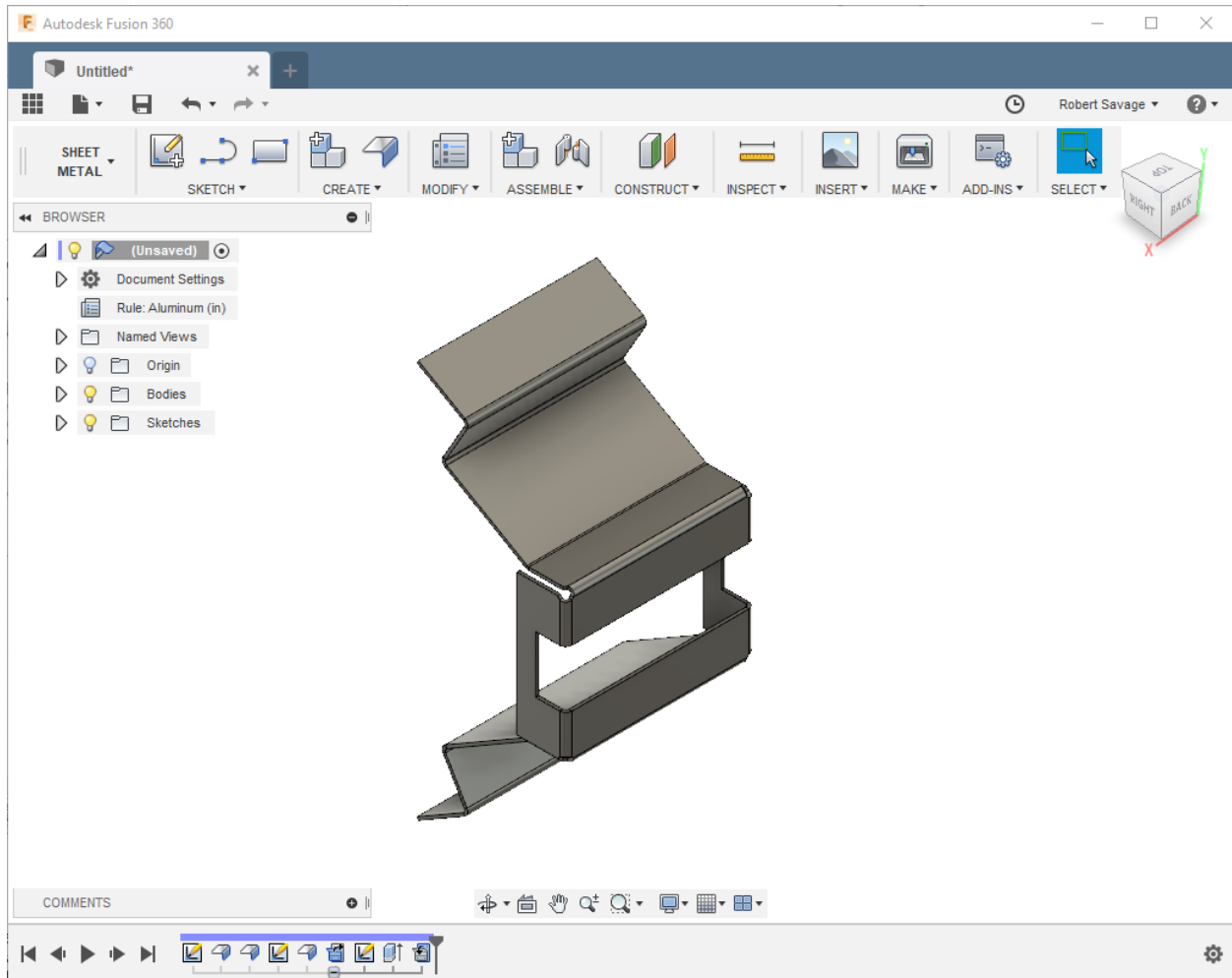
Unfold Refold Tools

The unfold tool is not the same as the flat pattern tool. It is used to unfold selected edges of the folded model so that either verify that the model will unfold or so that you can get an extended flat area to sketch on.

The refold tool becomes available on the main ribbon once you enter the unfold environment. It is used to return the part to the original folded state.



Sketched Opening



Refolded Opening

Sheet Metal Rules

A sheet metal rule sets predefined parameters that allow us to define defaults for material, thickness, bends, bend reliefs, corner reliefs, gaps, and K Factor.

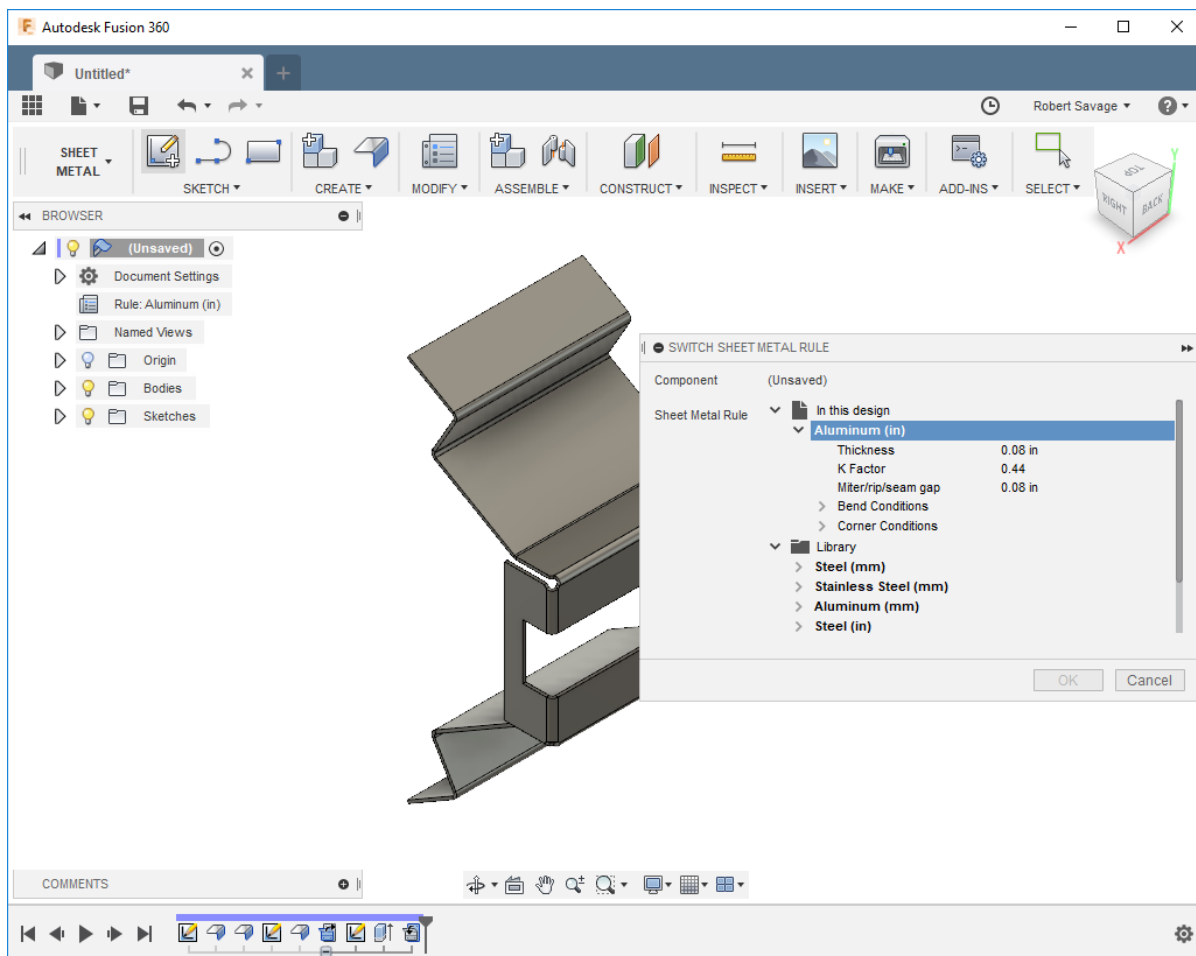
There are default rules that are already defined in Fusion, but you also can go into a part and create your own custom rules. The user can also change which rule is being used to set the defaults of a part.

Rules

Rules are your configuration settings that define the parameters of a part. The current rule for the part is in the browser.


If you select on it you can open the rules dialog box where you can modify the current rule, create a new rule, or select a different predefined rule.



To change the rule, select from the Library the desired rule and select ok to apply it.

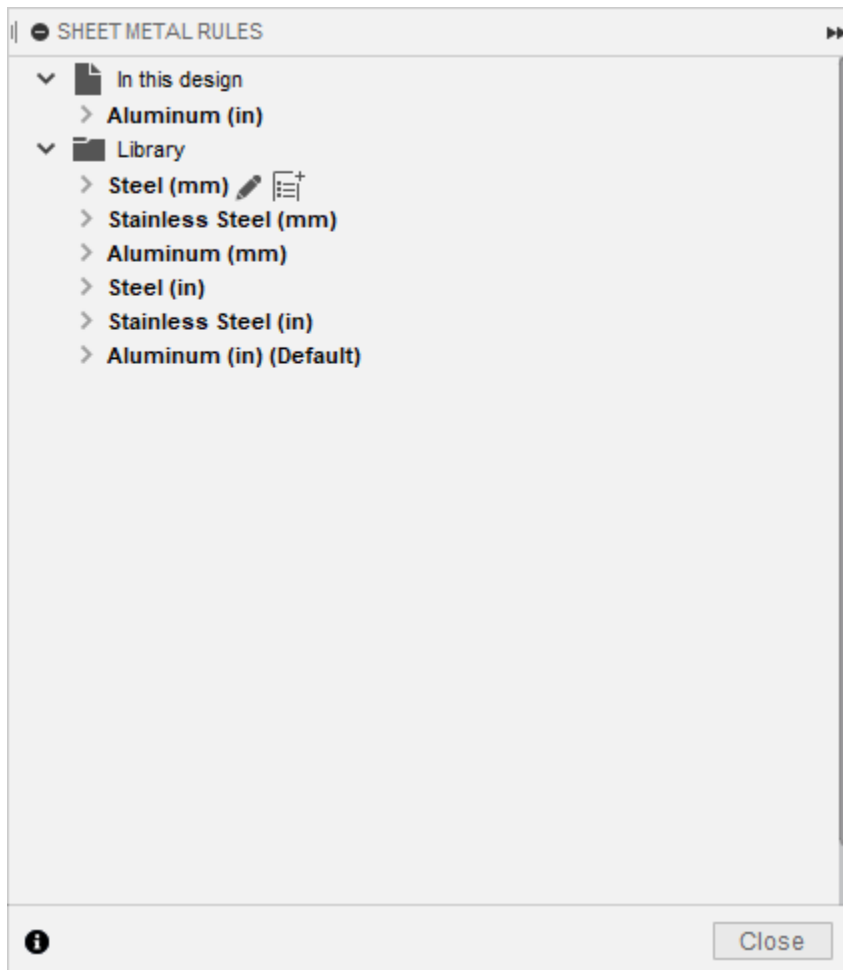


Switch Sheet Metal Rule

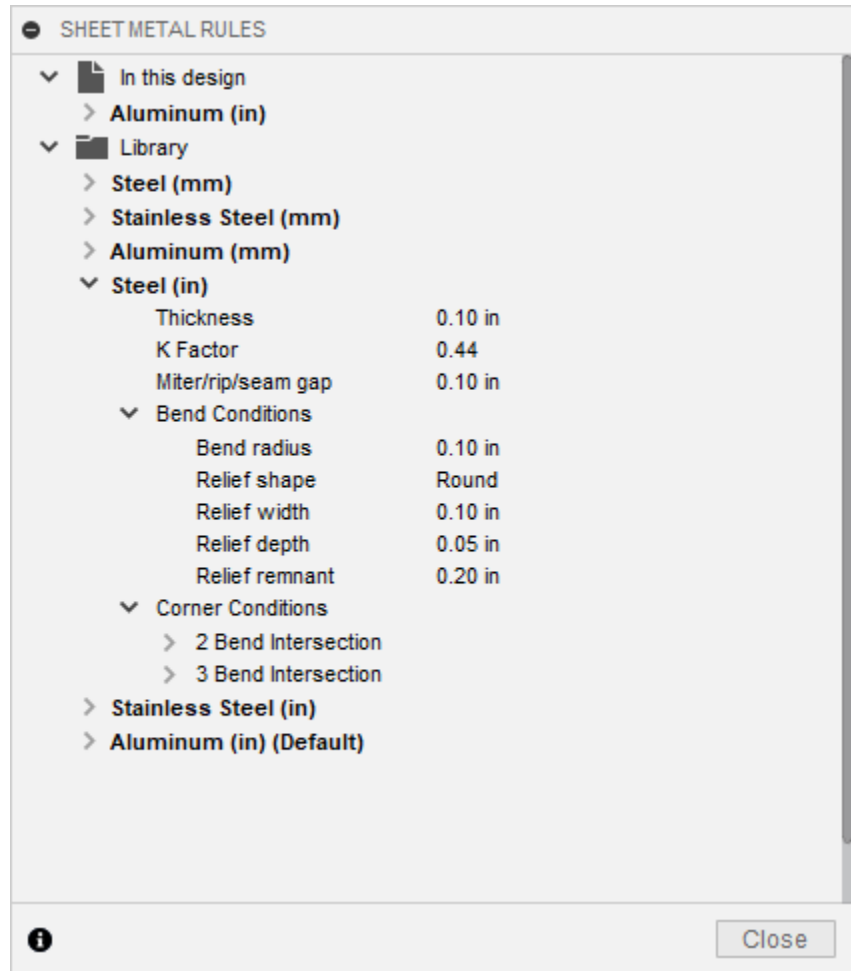
New Rules

To create a new rule, you would select the Sheet Metal Rules  tool from the Modify section. In the dialog you will find it broken up into two areas in this design, and library.

In the dialog box you can modify an existing rule  or create a new rule  based off the selected rule.



Sheet Metal Rules



Sheet Metal Rules

When defining a rule there are many things to take into consideration let's explore them next.

Material and Units

Material is set by selecting the material and units of an existing Rule and creating a new rule based off that rule. If the material is not in the list, you would set the material in the material dialog box.

Thickness

Material thickness for a sheet metal part is defined as a constant value because the part is created out of a single sheet of material.

K Factor

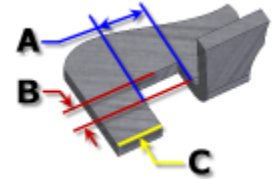
K Factor defines how much the material will deform as it is bent the default is .44 which means that the centerline of the material will move from .5 percent of the thickness to .44 percent of the thickness to the inside face.

Miter, Rip/Seam Gap

This is the distance between edges of a where a miter gap, rip gap or seam gap exist in a part. The default value for this is equal to the thickness of the material but the value can be changed to a constant value or a variable of the thickness.

Bend Conditions

Bend Conditions allow the user to define how the bend and bend relief rules are defined in the part. These options include Bend Radius, Relief Shape, Relief Width, Relief Depth and Relief Remnant and except for relief shape the default values are variables of thickness.



Corner Conditions

Corner Conditions allow the user to define how the intersection of bends will be defined in the rules. There are two categories for these and they are 2 Bend Intersections, and 3 Bend Intersections each allows the user to define Relief Shape, Relief Size and Relief Placement.

New Rule

Steel (in)(1)

Thickness0.10 in

K Factor0.44

Miter/rip/seam gapThickness

Bend Conditions

Bend radiusThickness

Relief shapeRound

Relief widthThickness

Relief depthThickness * 0.5

Relief remnantThickness * 2.0

Corner Conditions

2 Bend Intersection

Relief shapeRound

Relief sizeThickness * 4.0

Relief placementIntersection

3 Bend Intersection

Relief shapeRound with Radius

Relief radiusThickness

Save

Cancel


New Rule

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Flat Patterns and Documentation

The flat pattern and documentation are together because they are both created as outputs of the folded models.

Flat Pattern

The flat pattern is created with a single click of the Create Flat Pattern tool  located on the modify section of the ribbon.

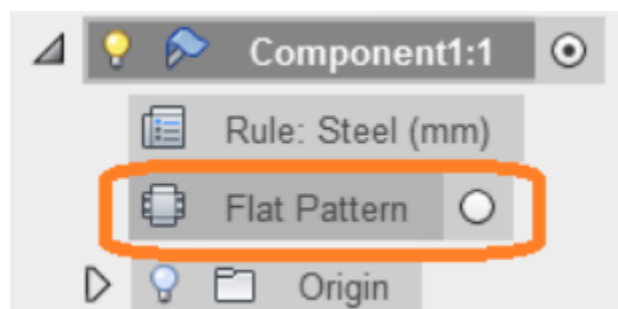
The flat pattern is a result of the folded model with bend deformations applied so that it shows an accurate representation of what the pre-bent size and shape of the part should be to create the accurate folded model when completed.

Once a flat pattern is created it becomes a subsection of the part in the browser. The bend lines are also displayed on the flat pattern and represent where the part should be bent.

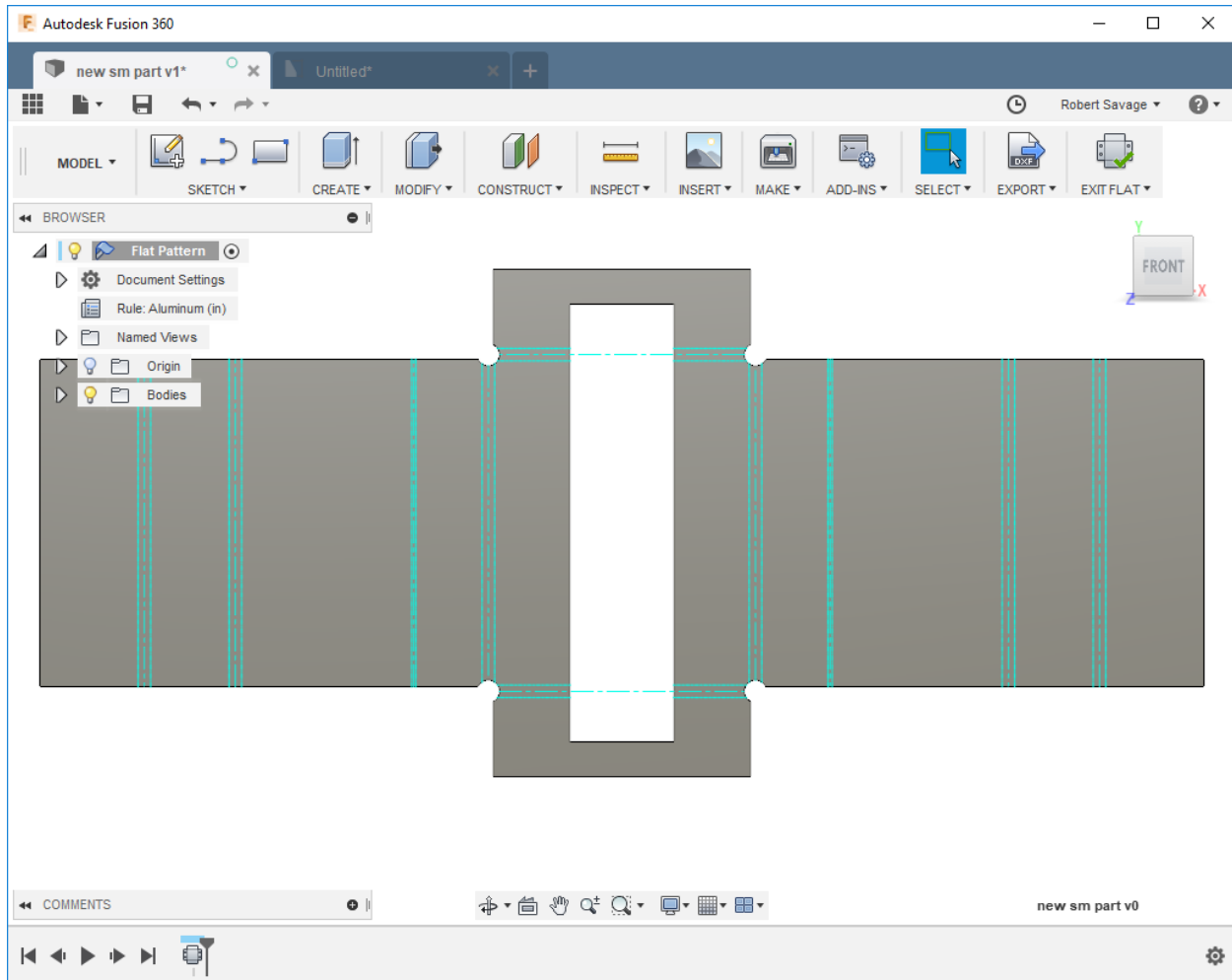
To exit the flat pattern, you select the Exit Flat that becomes available in the ribbon once a flat pattern is created.

To return to the flat pattern select the radial button beside the Flat Pattern node in the browser.

The flat pattern is also used in the machining side for defining the laser or water jet cut path or can be exported to a DXF file to send to a machine shop for importing into a machine.



Flat Pattern Node



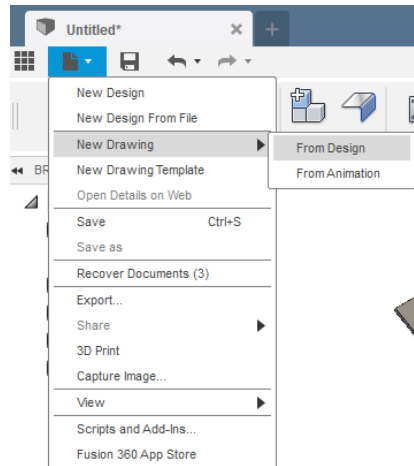
Flat Pattern View

Drawing

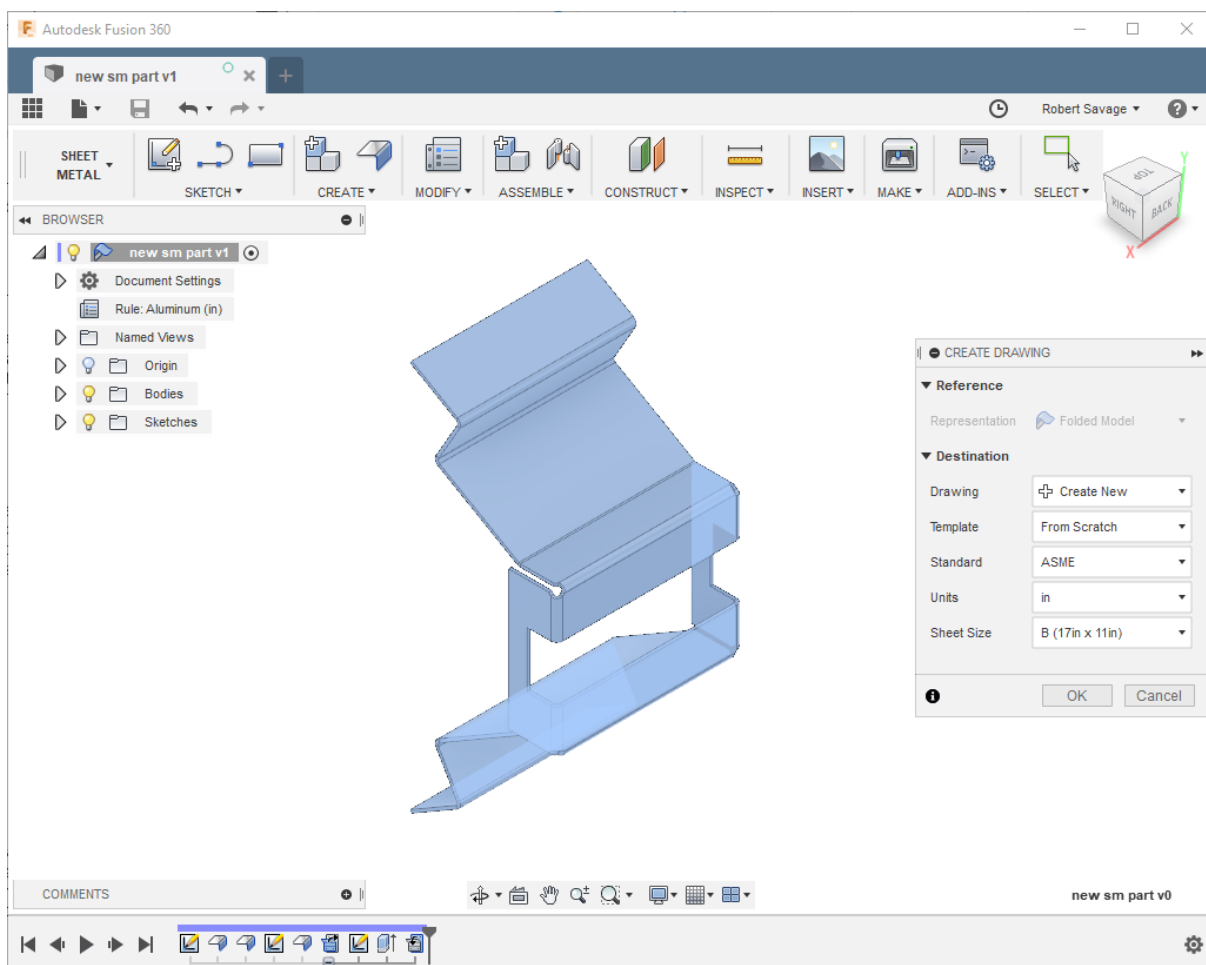
The drawing is used to document the folded part, flat pattern or both. It is made up of views and annotations that tell the person making the part how and where it should be cut and folded to create the completed part.

Creating a drawing is done by selecting the file drop down going to New Drawing then the From Design Option enter the file name and location.

In the Create Drawing Dialog box define drawing standard, units and Sheet size. Once you select ok you will be put in the drawing environment to begin defining your views and annotations,



New Drawing



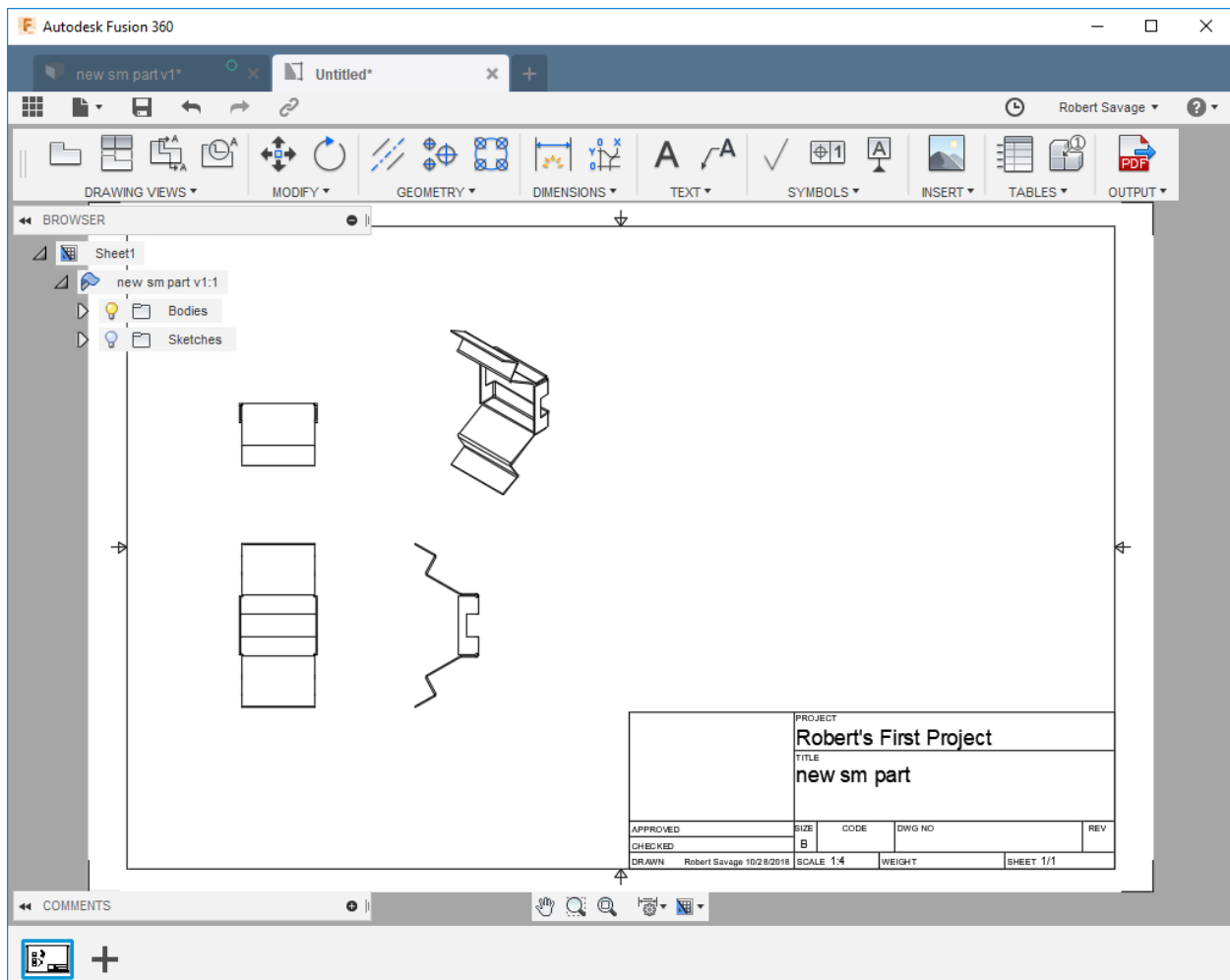
Drawing Settings

Views

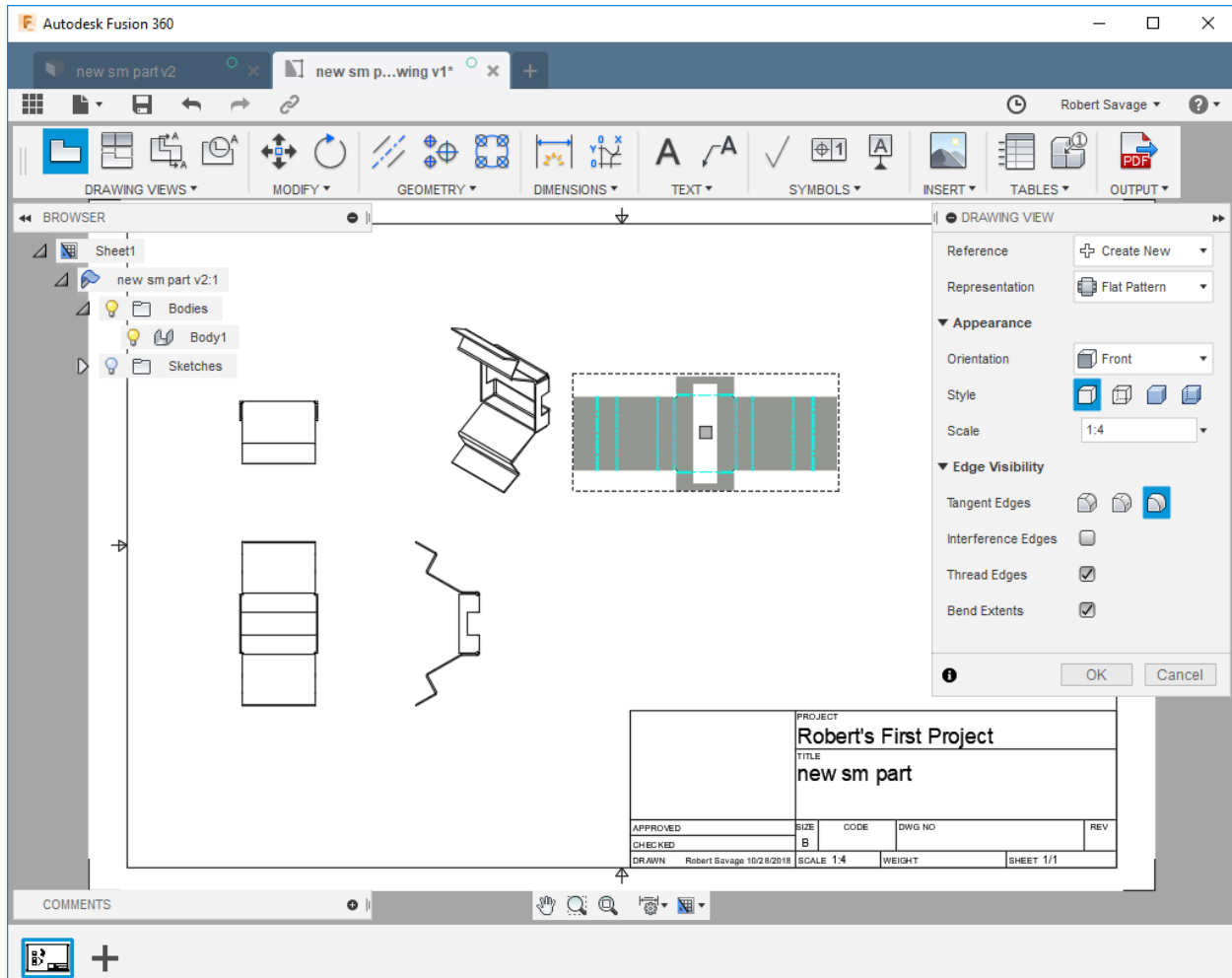
There are many different view tools in this section. I want to focus on the ones that are used for defining sheet metal views.

The Base view is placed of the folded model with the desired options and the projected view tool is used to create additional views.

To add a flat pattern view, you will need to go back to the base view tool and in the options change folded model to flat pattern the place the view where desired.



Folded Views.

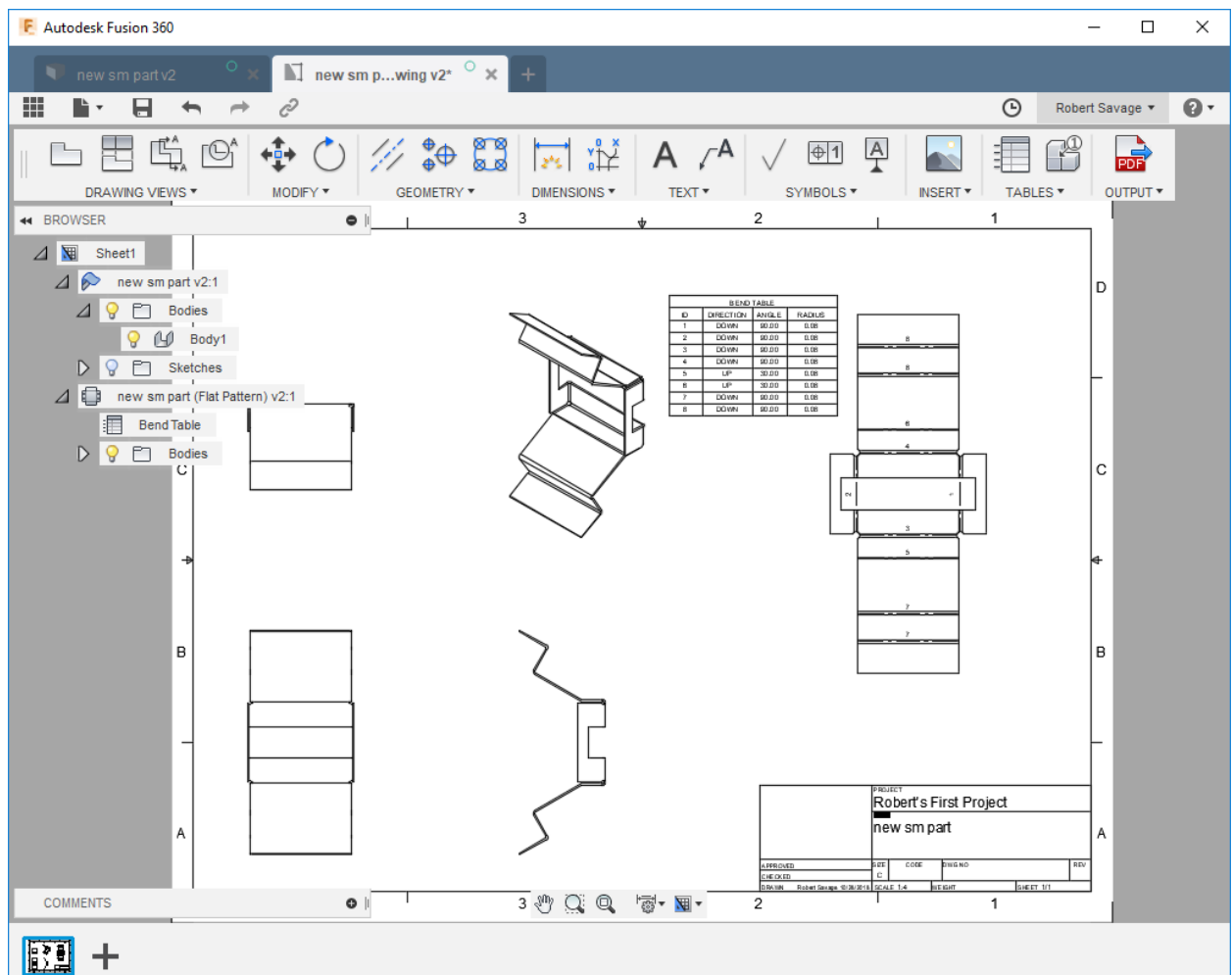


Folded and Flat Pattern Views.

Annotations

There are many different view tools in this section. I want to focus on the ones that are used for defining sheet metal views.

In addition to being able to add dimensions to the drawing you can also add a Bend table by selecting the flat pattern view and selecting the Table button on the ribbon then selecting where on the drawing you want the table placed.



Bend Table.

Closing

There are many ways of creating sheet metal content. In this class I have tried to focus on the tools and procedures that will help you create effective and usable sheet metal parts.

If you have any questions or comments, please feel free to reach out to me and I will do my best to help.

Thanks for your time and I hope you enjoy Autodesk University.

Robert Savage

rsavage@advancedsolutions.com

For More Information on Fusion and Fusion Training here are a couple of links that might help.

<http://au.autodesk.com/>

<https://www.autodesk.com/products/fusion-360/overview>

www.advancedsolutions.com

<https://graitec.co.uk/blog/entry/autodesk-inventor-sheet-metal-flat-pattern-success-every-time>

<https://www.youtube.com/channel/UCiMwMz3RMbW5mbx0iDcRQ2g>

<https://www.youtube.com/user/cadcamstuff>

<https://www.youtube.com/user/saunixcomp>