AUTODESK UNIVERSITY

CP500829

How to Design Plastic Parts More Effectively with Autodesk Fusion 360

Jim Swain
Synergis Technologies LLC

Learning Objectives

- Learn how to configure and apply plastic rules to a design, then how to interpret the design advice for a part.
- Learn how to add common plastic features to a part, such as snaps and bosses.
- Learn how to analyze a snap design for performance, including deformation during engagement.
- Learn how to determine the mold-filling characteristics for a part.

Description

Designing parts for the injection molding process often relies on experience and generic guidelines. Determining the needed geometry for design features can be very time consuming. In this session, we'll look at how we can streamline this process using Autodesk Fusion 360 software and its extensions. Autodesk Fusion 360 Product Design Extension includes tools for the quick addition of common features, such as snaps and bosses, to Autodesk Fusion 360 parts. Will these features work? Will the snap fail during initial assembly? Will there be sink marks if a boss is molded as designed? Autodesk Fusion 360 Simulation Extension tools for stress analysis and mold-filling simulation help answer these key questions. Now we can catch and correct potential issues early in the design cycle, when there's still time to react and cost of changes are relatively low.

Speaker(s)

I am an Applications Consultant with Synergis Technologies LLC, specializing in mechanical design and analysis. I have worked with CAD/CAE tools for almost 40 years as a design engineer, CAD manager, trainer and consultant. My primary focus has been in the mechanical design and manufacturing areas.
Prior to joining Synergis my work often included designing injection molded parts. I was part of a group that developed an in-house program for teaching the design methods for injection molded parts to new hires.
I have taught design classes at Autodesk University, our own Synergis University and at the college level.

General Notes About This Class

- This class was developed with the assumption that a designer will be using Fusion 360 with the Product Design extension.
- I will use the phrase “plastic part(s)”, and “injection molded plastic part(s)” interchangeably in this document. While there are many other manufacturing techniques available for creating plastic parts besides injection molding, this class will focus only on injection molding.
- This class will focus on parts made from a single material. Other processes, such as over-molding and dual-material injection are beyond the scope of this session.
- The term “resin” will refer to the plastic raw material that is injected into the mold to create the part.
- I won’t go into the history of the manufacturing process here, other than to say that the methodology of designing mass produced, injection molded parts has been evolving for over seventy years.

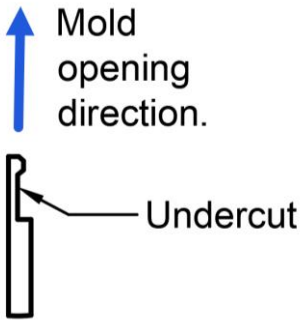
Challenges Designing Injection Molded Plastic Parts

Typical resins used for plastic parts shrink as they cool during the injection molding process. This shrinking will happen until the resin solidifies. Portions of a part that are thicker than others may cool slower than other portions, so may have the opportunity to shrink more.
This creates several challenges for the part designer:

- Unequal shrinkage due to different cross-sectional wall thicknesses.
- Surface irregularities due to unequal shrinkage. These are known as “sink marks”.
- Stresses within the part, caused by resin trying to shrink away from already frozen portions of the part. This is known as “molded-in stress”.
- The part shrinking onto the core of the mold, making part ejection more difficult.
- Part deformation due to unequal shrinkage. This is known in general as “warpage”.

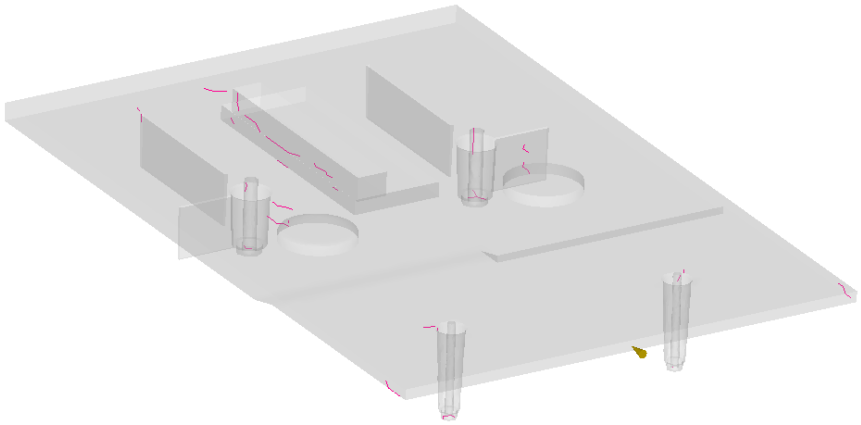
Other challenges for the part designer include:

- Portions of the design being “shadowed” by the rest of the part. This is also called an undercut, and can lead to more expensive mold design techniques to allow the part to be ejected from the mold at the end of the cycle.



Example of an Undercut

- Locations of weaknesses in molded part due to the molding process. These include areas when the flow of the molten material has separated and has to rejoin to finish filling the cavity. These are known as “weld lines”.



Predicted Weld Lines (shown in magenta)

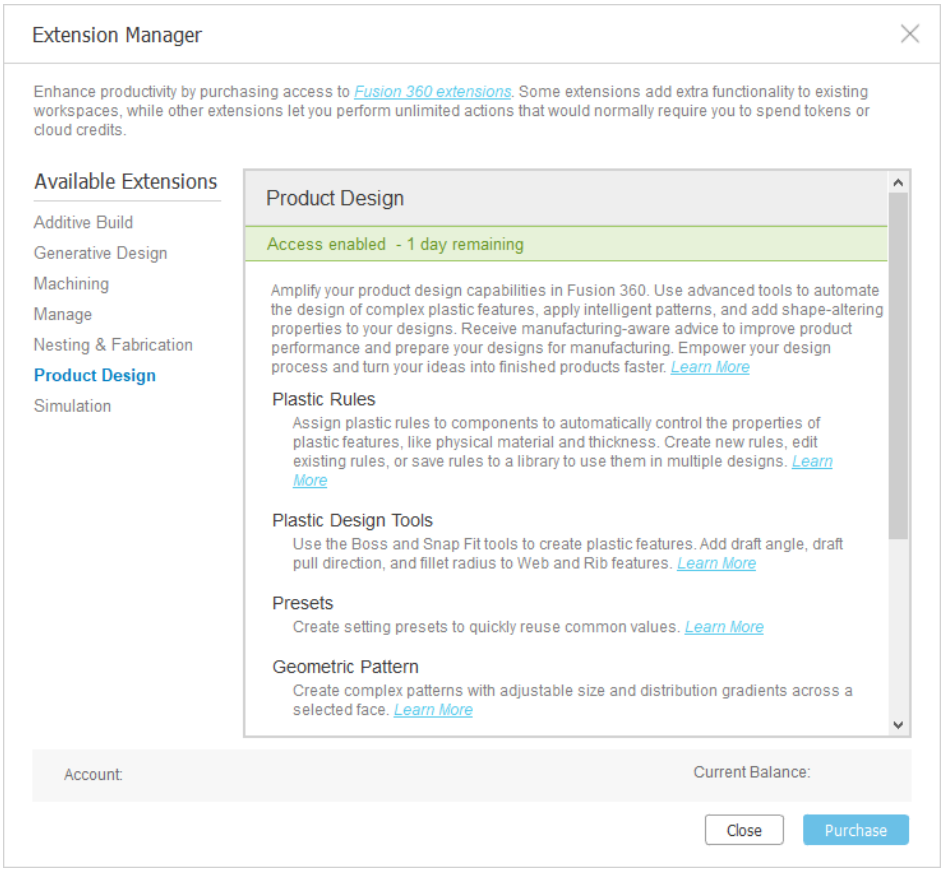
Actual Weld Line Example

By addressing these issues during the design process their effects can be accounted for, and maybe even avoided. That helps the designer take advantage to the benefits of injection molded parts, including:

- The large assortment of materials available for injection molding. These offer a large range of properties, including:
  - Operating temperature ranges
  - Stiffness/flexibility
  - Thermal insulation
  - Electrical insulation
  - Molded-in coloring
- A wide range of surface appearances are possible.
- Useful for Design for Assembly (DFM). A single part can perform many functions.

The “nominal wall” is the main consideration when designing plastic parts. It can be defined as the typical thickness of a part. It is often the value used when using the “shell” command in CAD.

Basic Fusion 360 is certainly capable of designing plastic parts. It has the SHELL command to help develop the nominal wall. It has a tool for inspecting wall thickness for variation. Unfortunately it is still easy to miss portions of a design that may cause problems, both in terms of part cost and performance. The addition of the tools in the Product Design Extension greatly reduces that risk.



Product Design Extension Tools

FUSION 360 CORE DESIGN TOOLS

CREATE

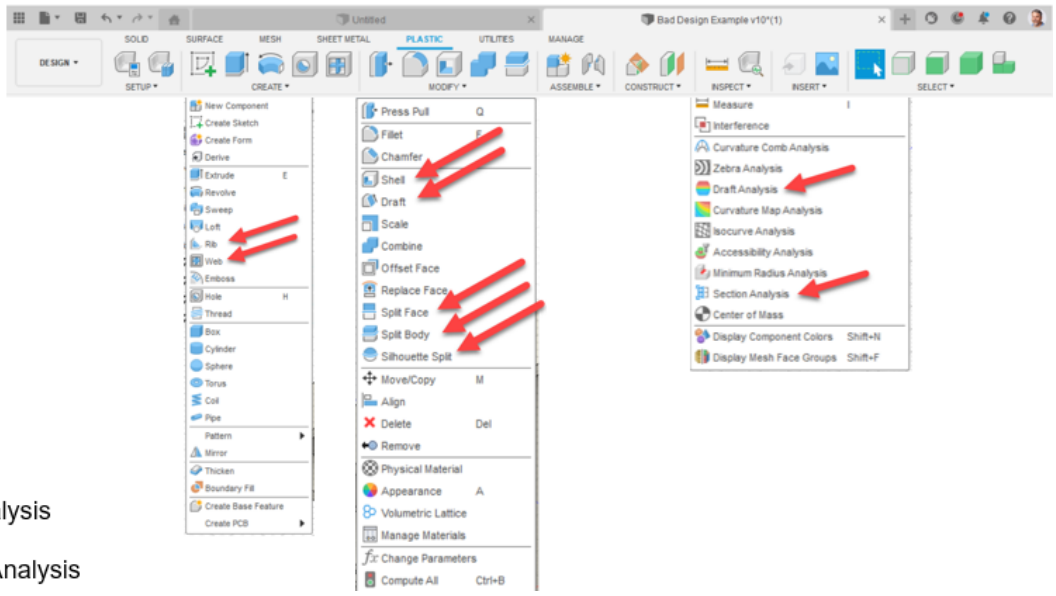
- Ribs
- Webs

MODIFY

- Shell
- Draft
- Splits

INSPECT

- Draft Analysis
- Section Analysis



Core Fusion 360 Design Tools Useful for Plastic Parts

PRODUCT DESIGN EXTENSION - PLASTIC

SETUP

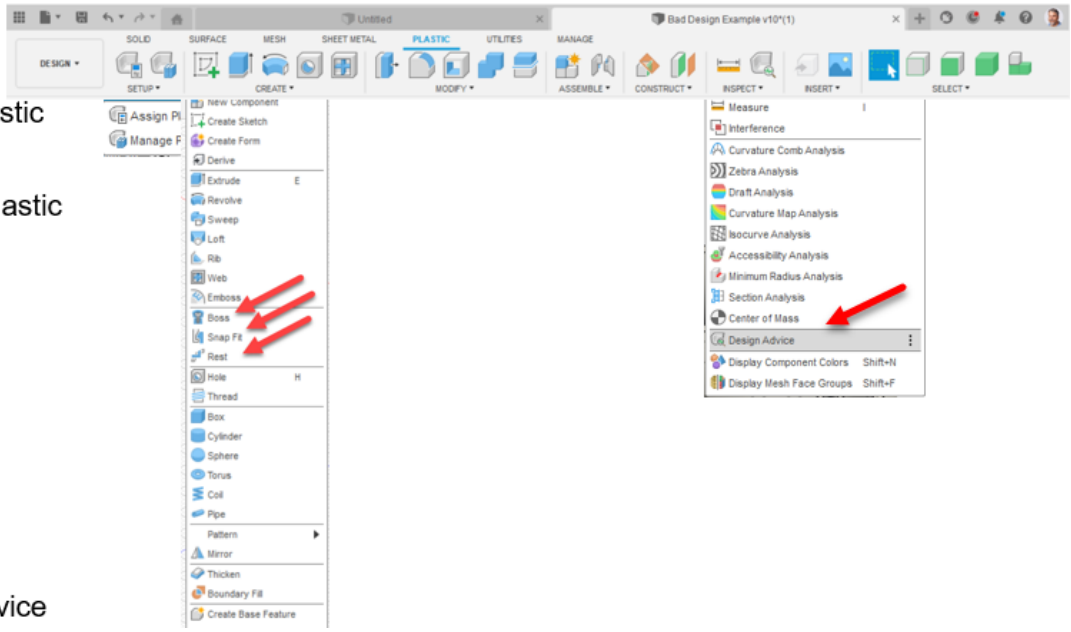
- Assign Plastic Rule
- Manage Plastic Rules

CREATE

- Boss
- Snap Fit
- Rest

Inspect

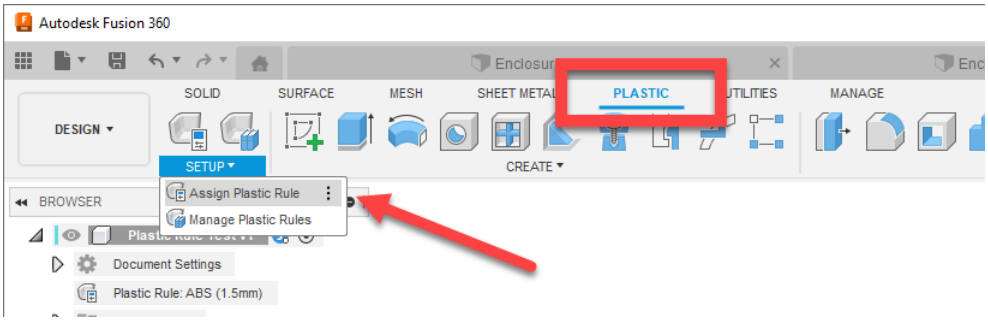
- Design Advice



Fusion 360 Product Design Tools Useful for Plastic Parts

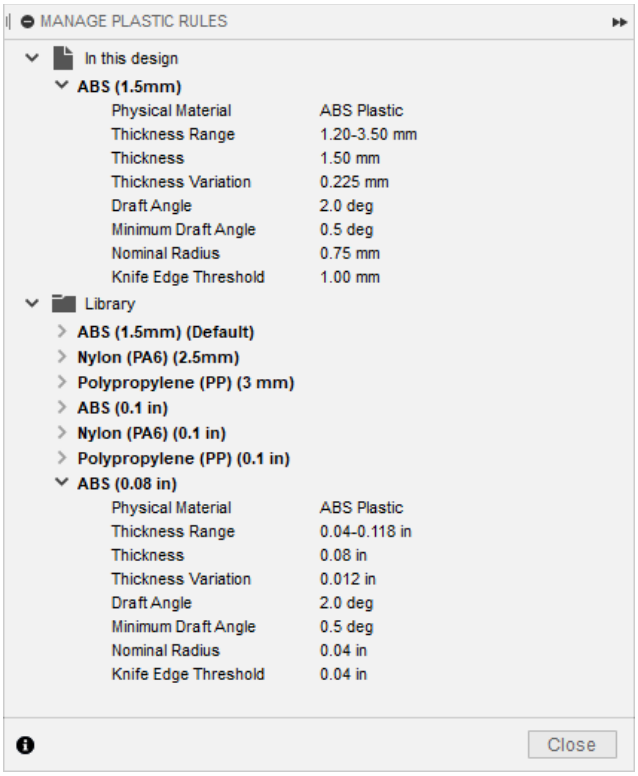
Configuring And Appling Plastic Rules

When the decision is made to design a part for injection molding the first step should be to go to the PLASTIC toolbar and assign a design rule to the part.

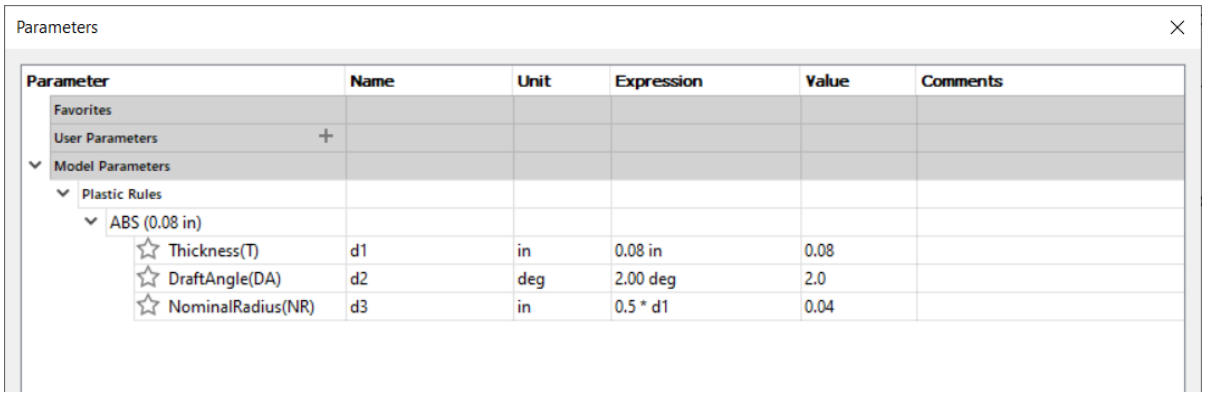


Applying a Plastic Rule.

This will set the Physical Material of the part. It will also specify various parameters that will be used by tools such as the wall thickness in the Shell tool.

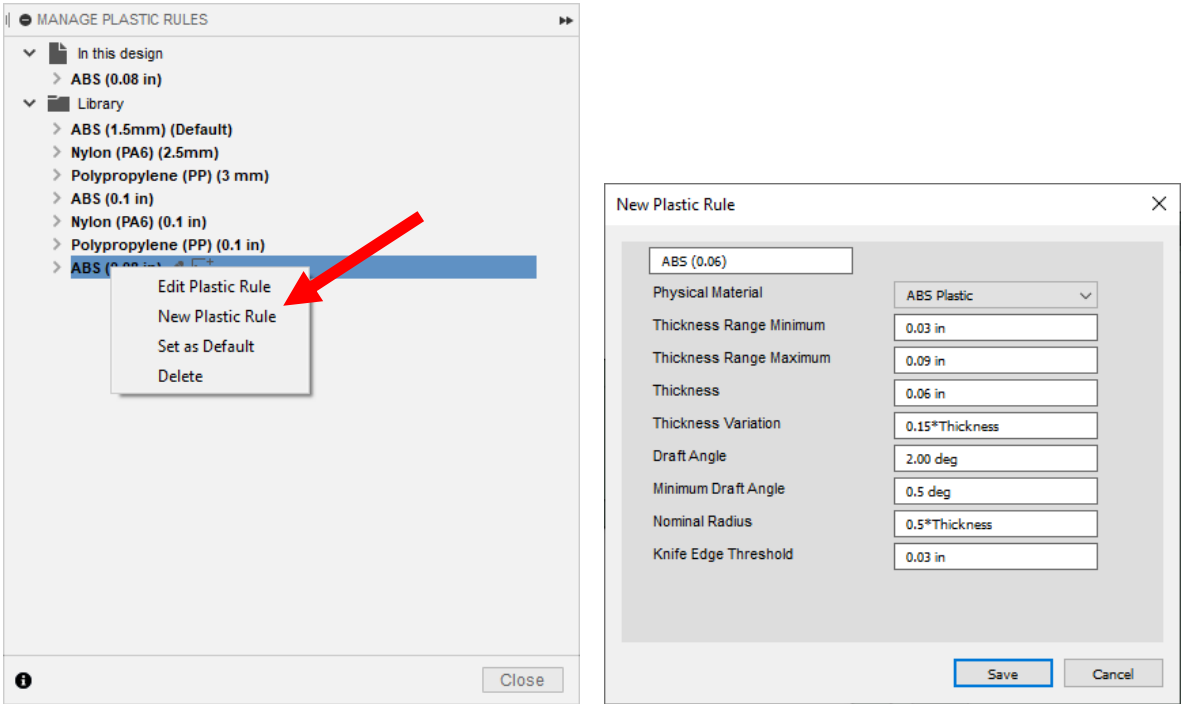


Plastic Rules.
ABS (1.5mm) has been applied to the part.



Parameters Created via Plastic Rules.

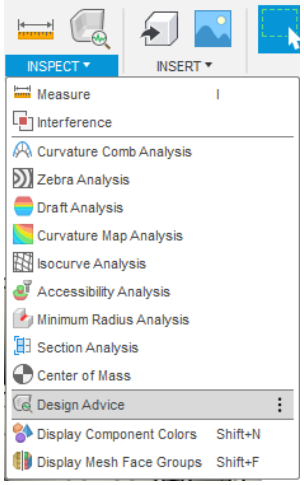
Several common engineering grade resins are included in the Library. Other materials can be added to the Library as needed.



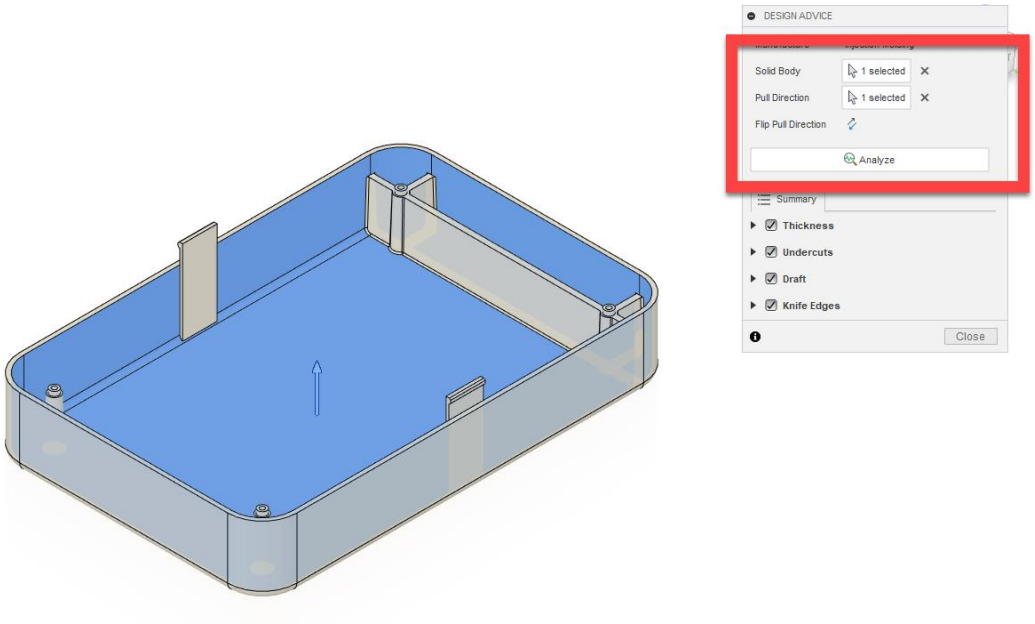
Creating a Plastic Rule

Interpreting Design Advice

The Design Advice tool set gives feedback while the design is being developed. This advice is based on both part geometry and the indicated mold opening direction. This allows the tool to interpret inclines as draft angles and shadowed geometry as possible undercuts on the part.



Activating Design Advice

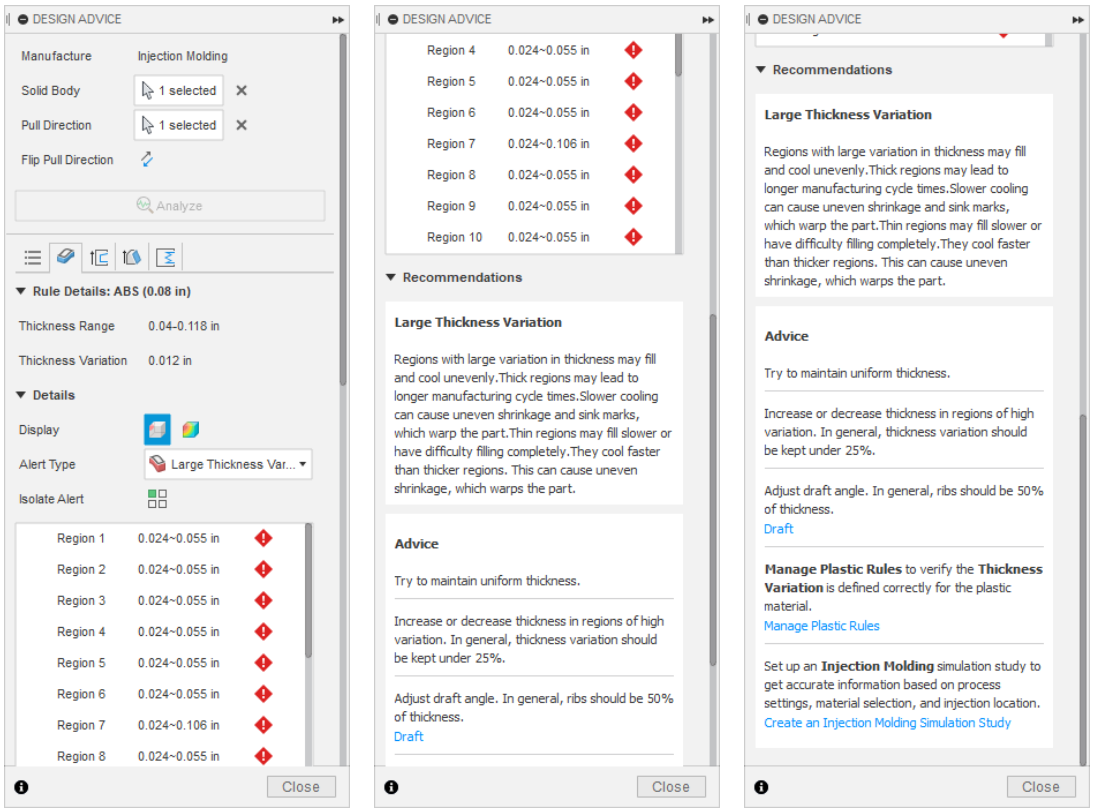


Select Body and Pull Direction, then Analyze

Types of advice include:

- Thickness and Thickness Variation
- Undercuts
- Draft angles
- Knife Edges

Initial Results



Detailed Results and Recommendations

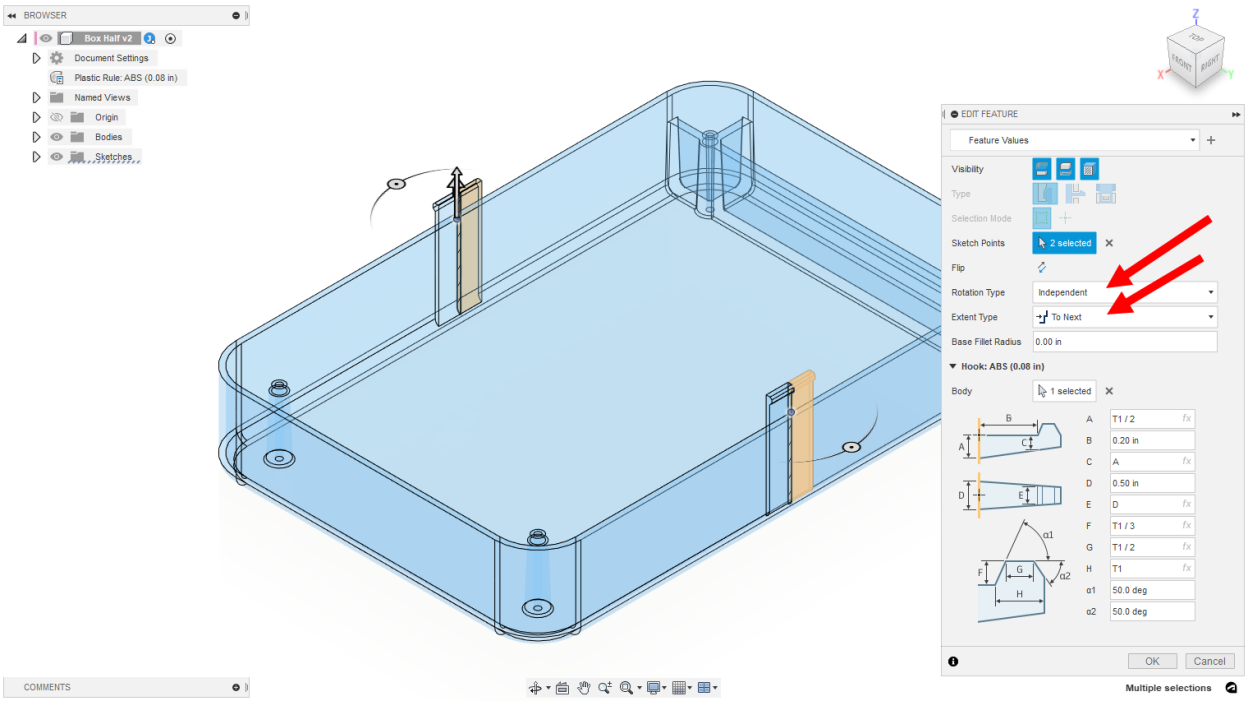
Each of the Design Advice categories can be investigated separately. Areas of concern are highlighted, and recommendations are offered.

After a design change is made, re-run the analysis.

Adding Snaps and Bosses

Snaps

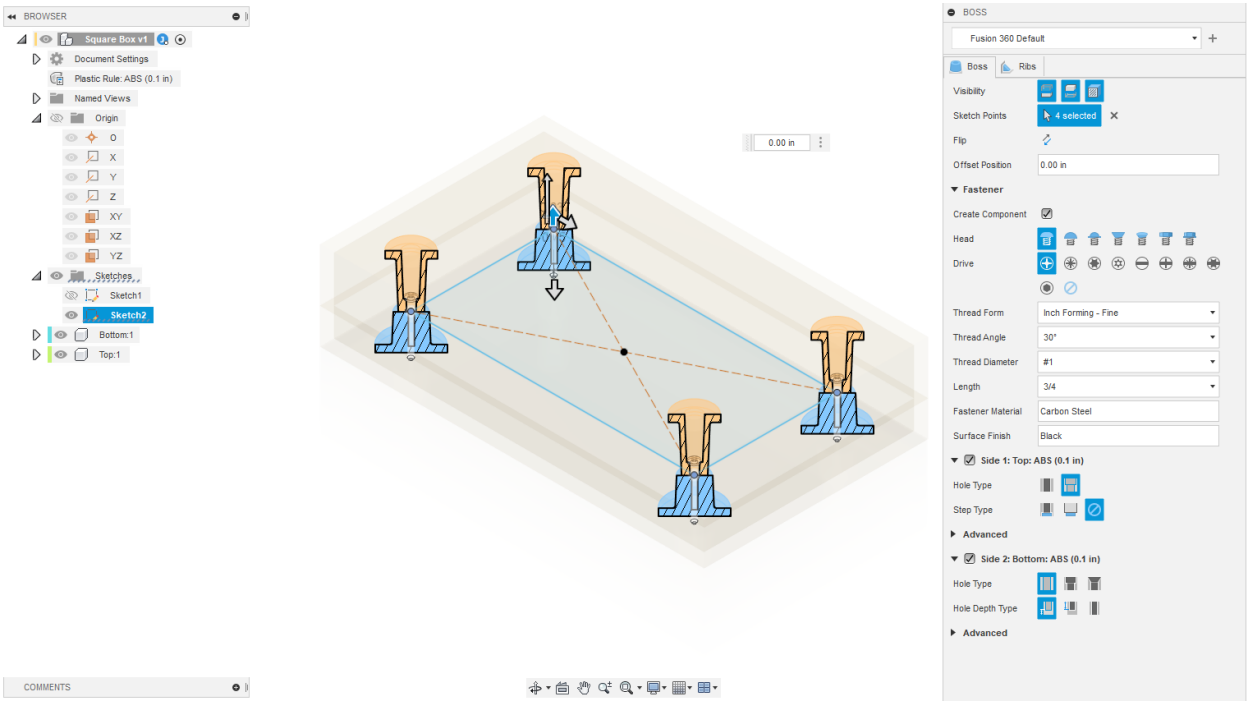
Placing multiple, individual snaps allows each snap to have a different orientation, while helping to keep the Design History small.
Adjust the shared geometry features, including draft angles, to develop the desired snap shape.



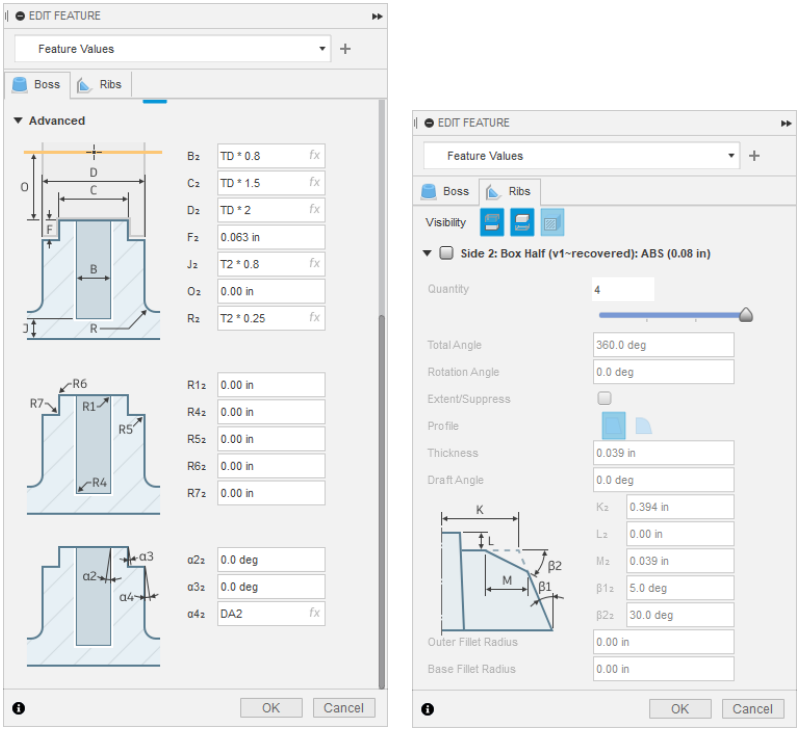
Adding Independent Snaps to Sketch Points

Bosses

Bosses are also placed on sketch points. The tool can be used to place mating bosses, and can include hole features for assembly screws. If holes are included



Creating Bosses

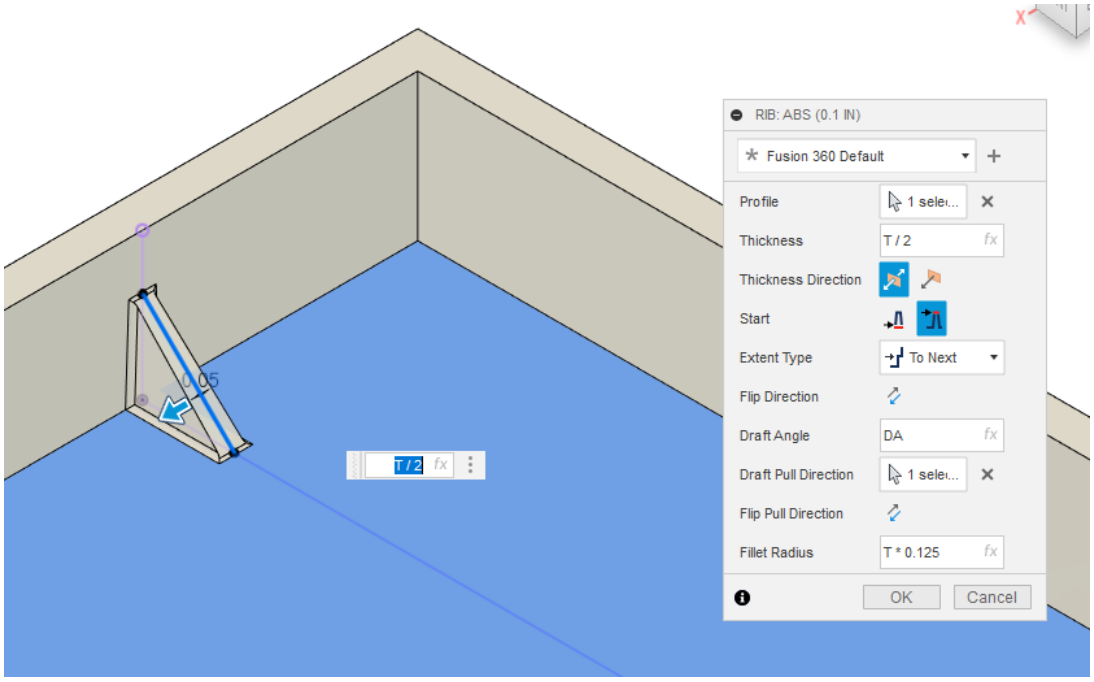


Boss Tool Advanced and Rib Tabs

Bosses can be created on one or both parts of an enclosure. Holes can be included, and fasteners automatically added within the command.
The Advanced option within the tool allows the adjusting of the default shape values, such as draft angles on holes and wall thicknesses.
The Ribs palette allows stiffening ribs to be directly added to the bosses and includes the ability to adjust their parameters.

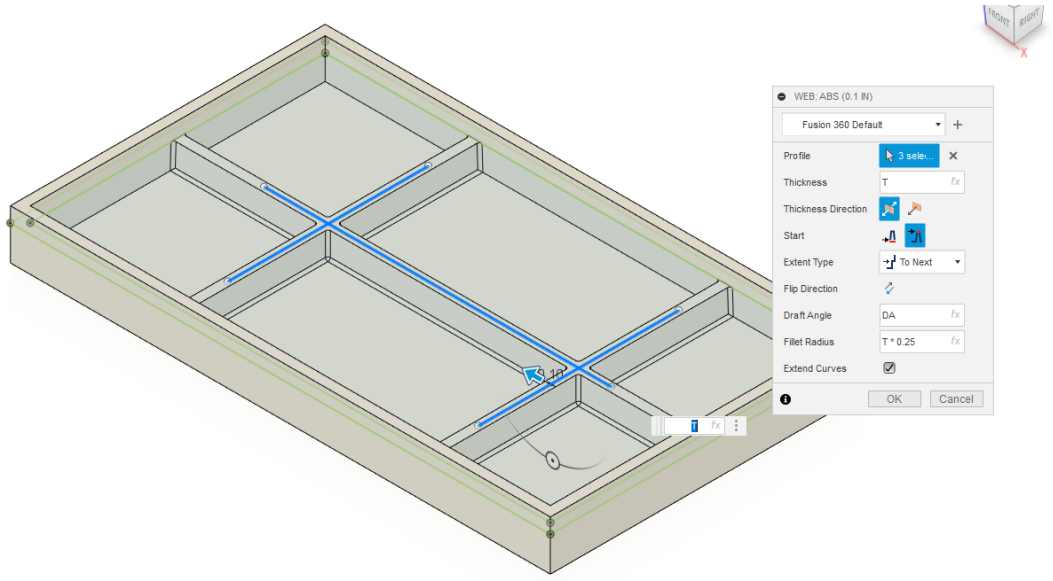
Adding Ribs and Webs

Ribs can be added to stiffen sections of a design, or to help channel resin flow during injection molding. They are created parallel to the sketch plane.
Ribs for stiffening bosses should typically be added with the Boss tool.



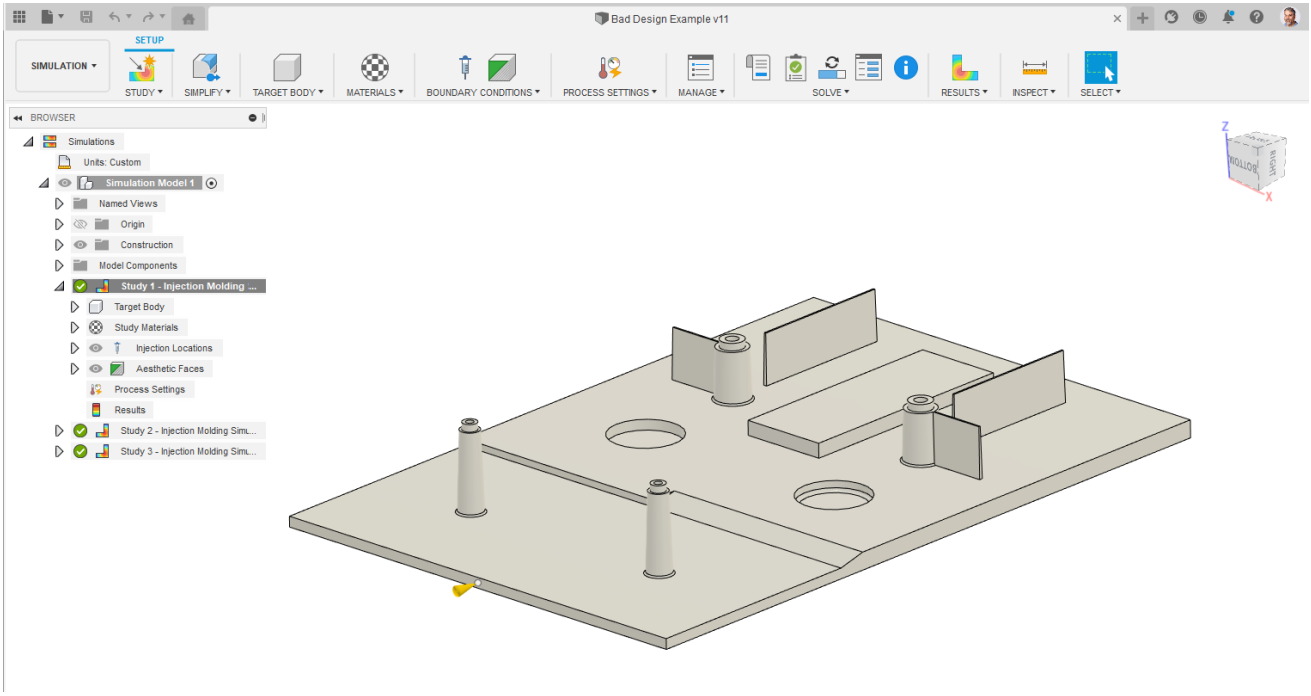
Creating a Rib

Webs are networks of ribs that are created perpendicular to the sketch plane. The sketch lines don’t have to extend all the way to the bounding walls, as there is an Extend Curve option.



Creating a Web (Sketch lines are in blue.)

Analyzing Mold Filling of a Design



Injection Molding Simulation Environment

The Injection Molding Simulation follows the basic pattern of the other Fusion 360 simulation studies: start at the left of the ribbon and work towards the right.

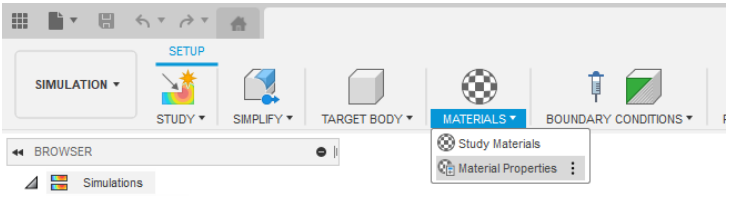
Analysis Setup

The STUDY is automatically created when the analysis type is selected.

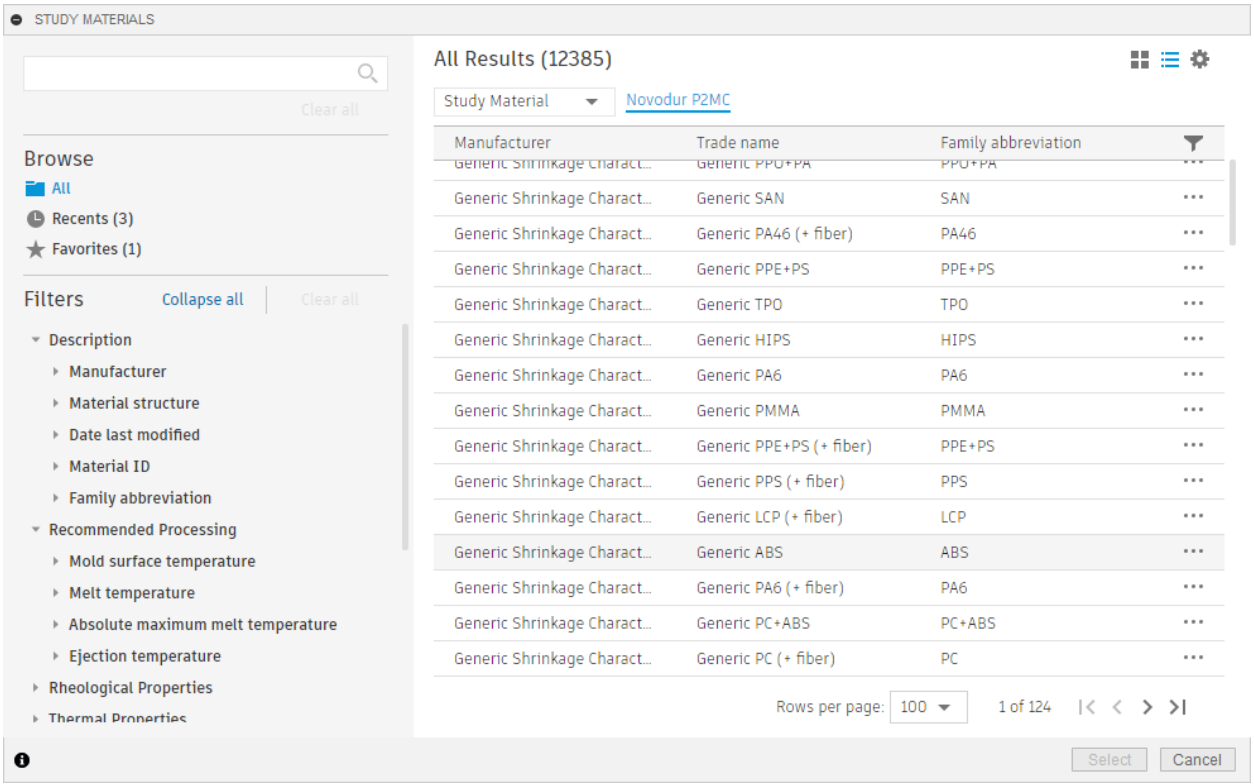
The SIMPLIFY tool can be used to remove small faces or features, but isn't used as often in mold filling simulations as it is in stress analysis. If the design included molded-in lettering or similar features that were very shallow when compared to the thickness of the part, then consider removing those faces, but otherwise analyze the part as it is modeled.

The TARGET BODY is identifying the volume that will be analyzed.

STUDY MATERIALS brings up a list of over 12,000 materials to select for the study. For general design work I typically stay with choices that start with GENERIC, as they aren't a specific manufacturer's resin. Of course, if you know the specific resin that will be used that should be selected.



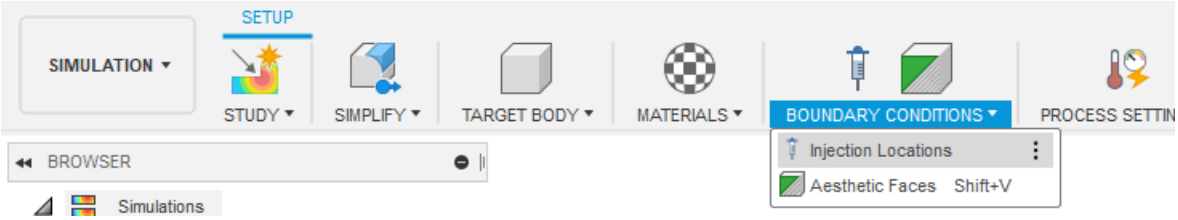
Materials



Study Materials (Generic Shrinkage Examples)

Note: The detailed set of material properties can be reviewed with the Material Properties tool.

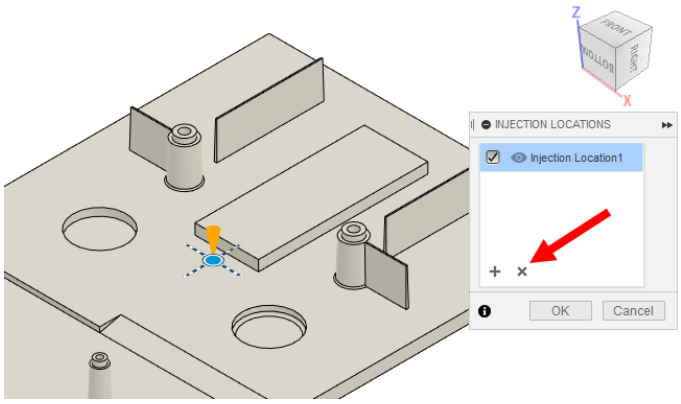
BOUNDARY CONDITIONS tools are used to add injection locations and specify aesthetic faces for the analysis.



Boundary Conditions for Injection Molding Simulations

The Injection Locations tool is used to specify where resin will be begin filling the part. These can be thought of as ideal gates. Resin will enter from each gate at the same time and with the same pressure.

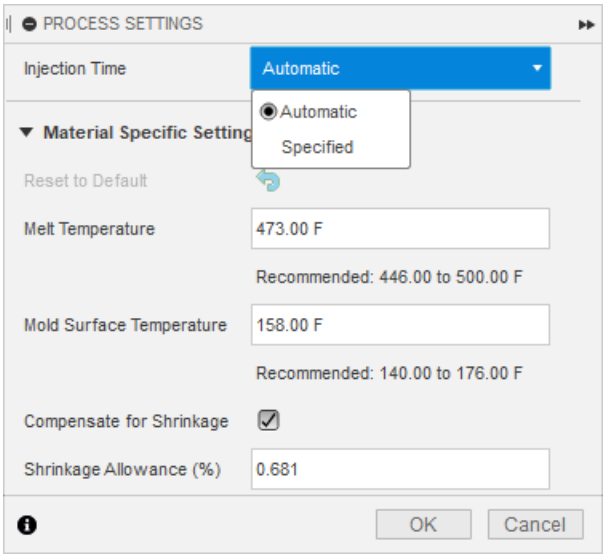
Injection locations can be moved by placing the cursor on the tip of the yellow cone, then dragging the cone as needed. Note that the cone won't snap to work points, but work points can be used to visually position an injection location. Injection locations can also be deleted if desired.



Adding Injection Locations

Selecting faces to be Aesthetic Faces will have those faces considered when looking for sink mark in the Results.

The PROCESS SETTINGS can be used to tweak temperatures and injection speed if desired.



Process Settings

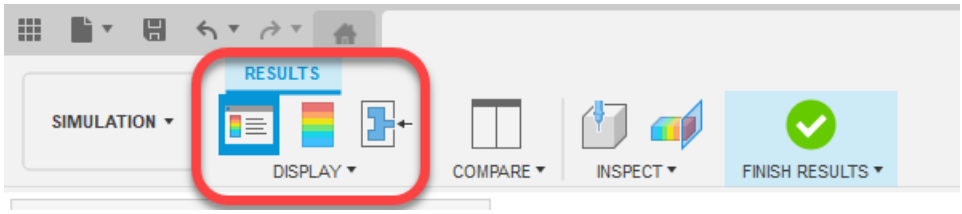
The MANAGE tool can be used to change the name of the simulation study.

The Setup Summary tool gives a good last check of the study before running the analysis. The analysis itself runs on the cloud solver, and results are downloaded into the design, if it is open, or when it is next opened.

Results

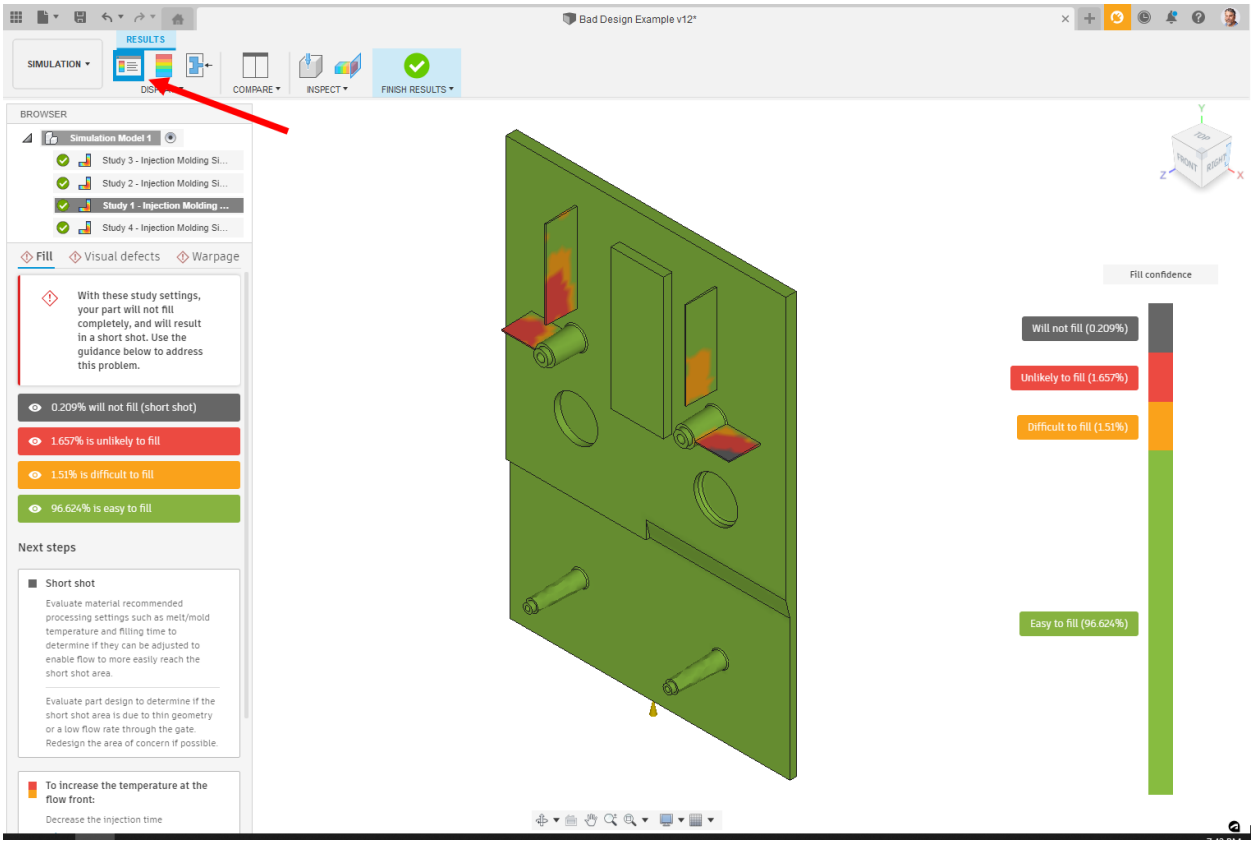
The analysis results are available in three general formats: Guided Results, Results, and Molding Process.

Note: The part used for the images in this section was deliberately created to show design problems for injection molding.



Results Tools

Guided Results

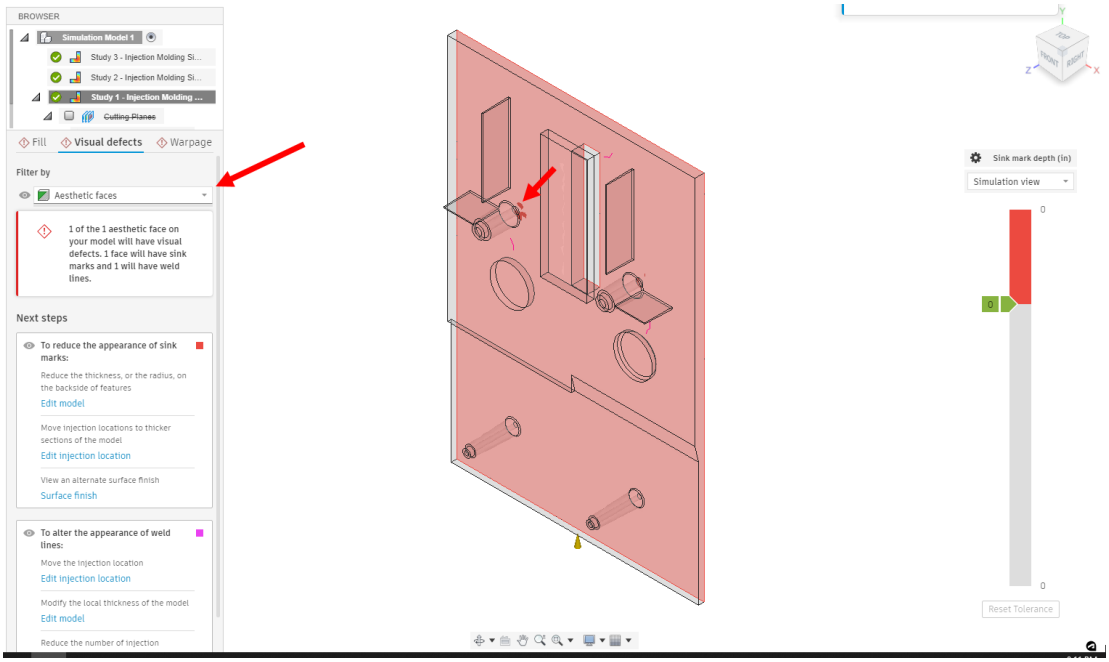


Guided Results for Fill

The Guided Results tool gives overall information on the final molded part, indicates areas that have challenges, and suggests possible ways to improve the results. The guided results are separated into three general areas: Fill, Visual Defects, and Warpage.

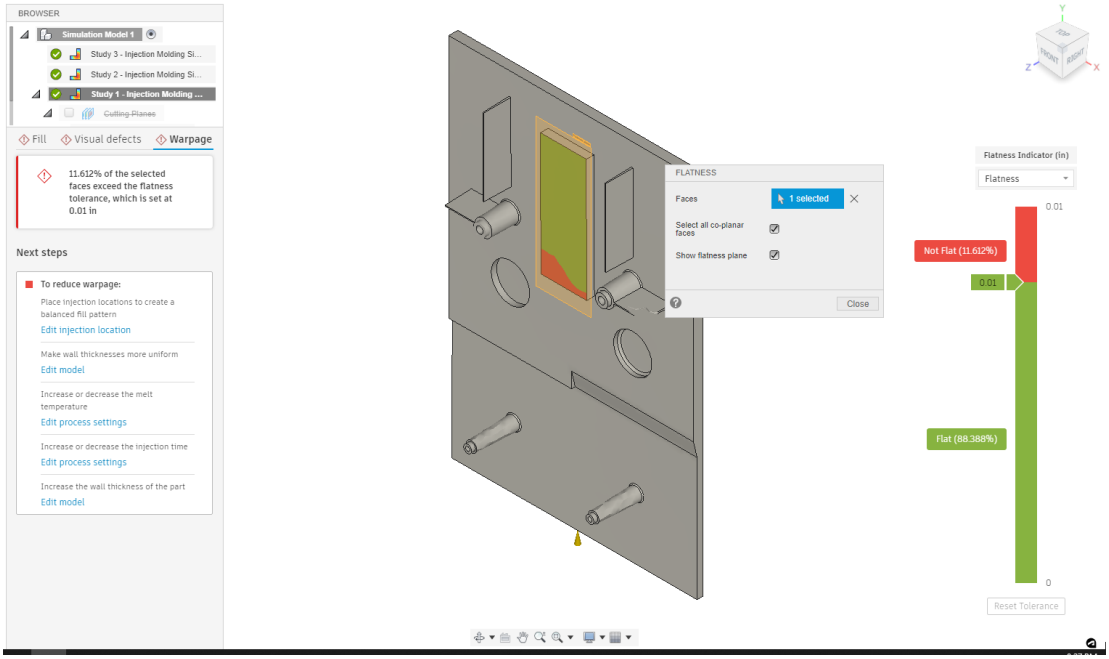
In the above image the very thin sections, which are far from the injection location, likely won't fill. Possible improvements include thickening the geometry or changing the process settings to help the molten resin to fill the entire mold.

Similarly, the aesthetic face selected in the analysis setup will likely have sink marks around the base of the screw bosses and several weld lines.



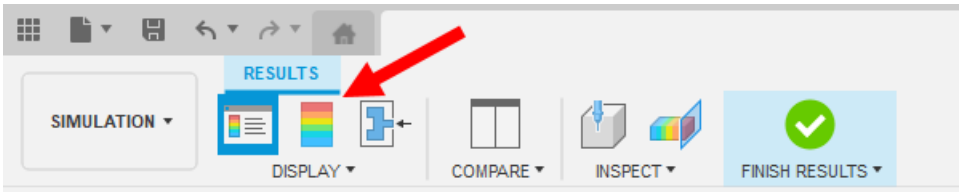
Guided Results for Visual Defects

The Warpage results highlight areas where the final molded part is expected to not be flat. The information can be shown as a warpage value, or a flatness value. The tolerance slider can be adjusted to see the extent of the warping.



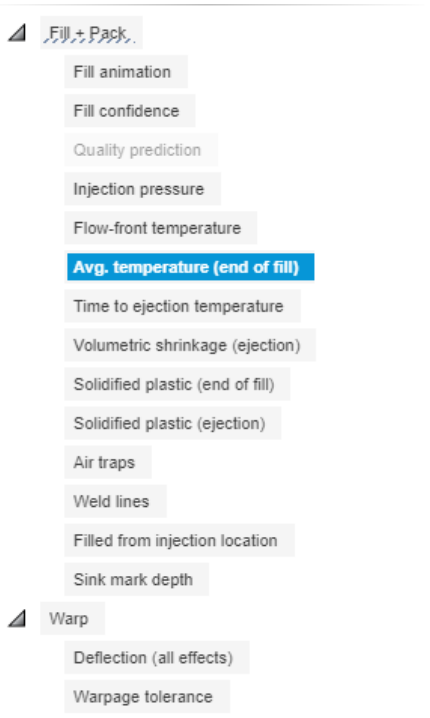
Guided Results for Warpage (Flatness)

Fill and Pack Results

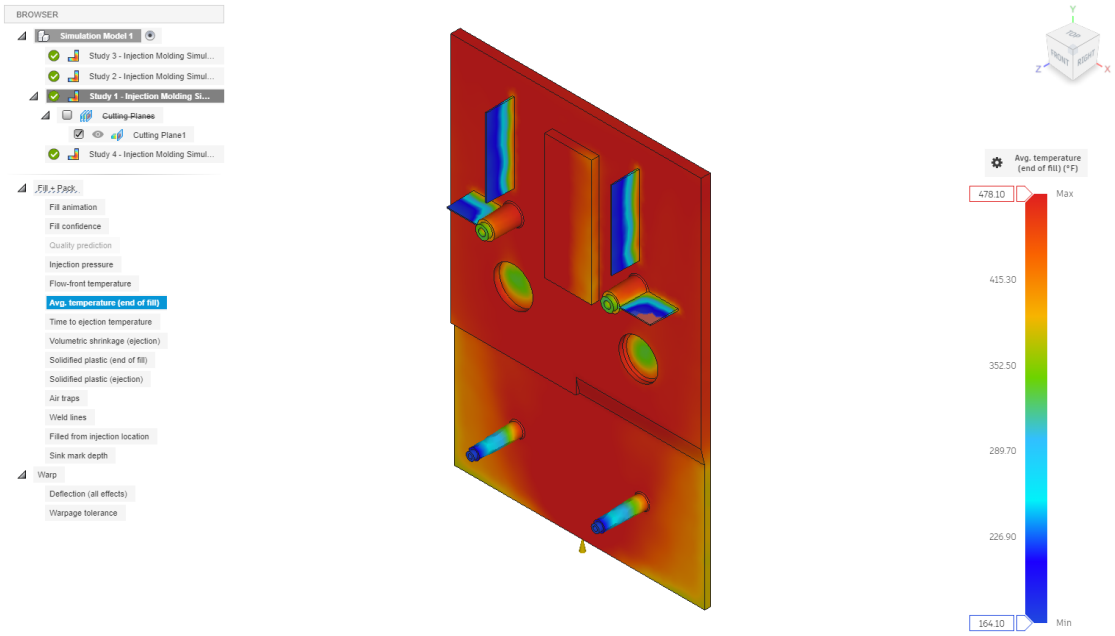


Fill and Pack Results

The full filling results, including animations of the filling process, are available. These can provide valuable insight as to how the mold is filling, and what the conditions of the resin are during the molding cycle.



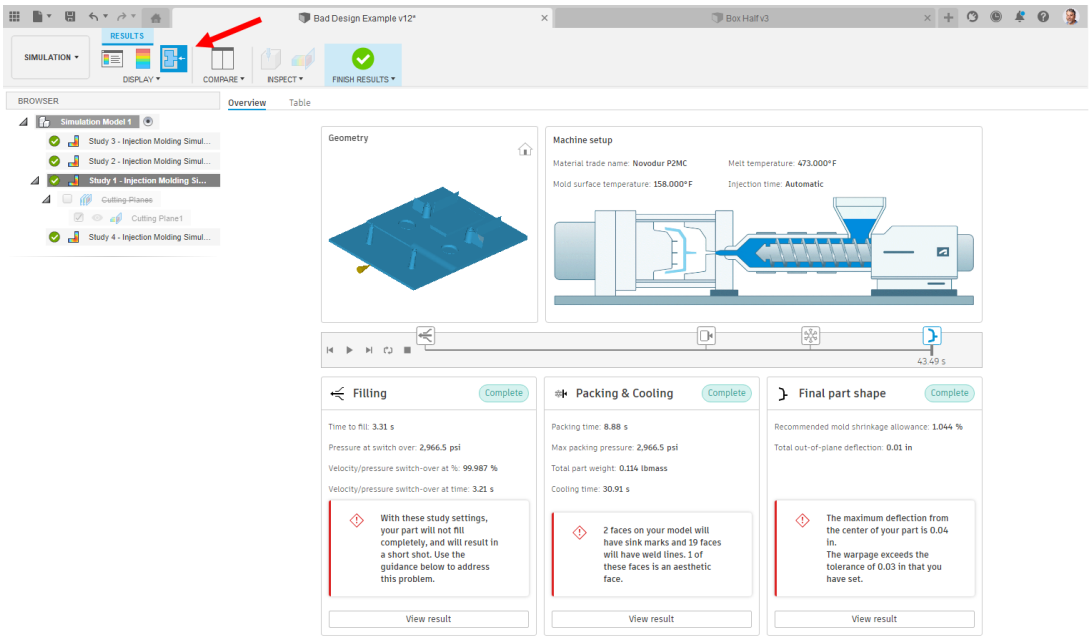
Available Fill and Pack Information



Example Fill and Pack Results (Average temperature at the end of fill)

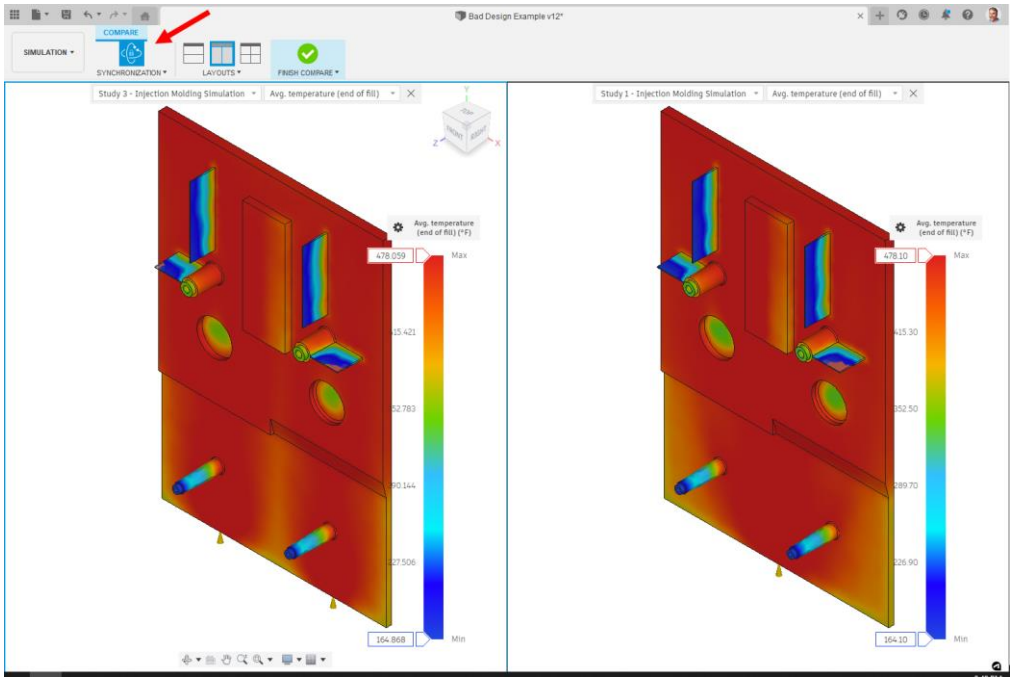
Molding Process

The Molding Process results include an overview of the molding cycle for the design, and links back to the Guided Results. There is also a Table view that shows the detailed numbers used in the simulation.



Molding Process

Comparing Results



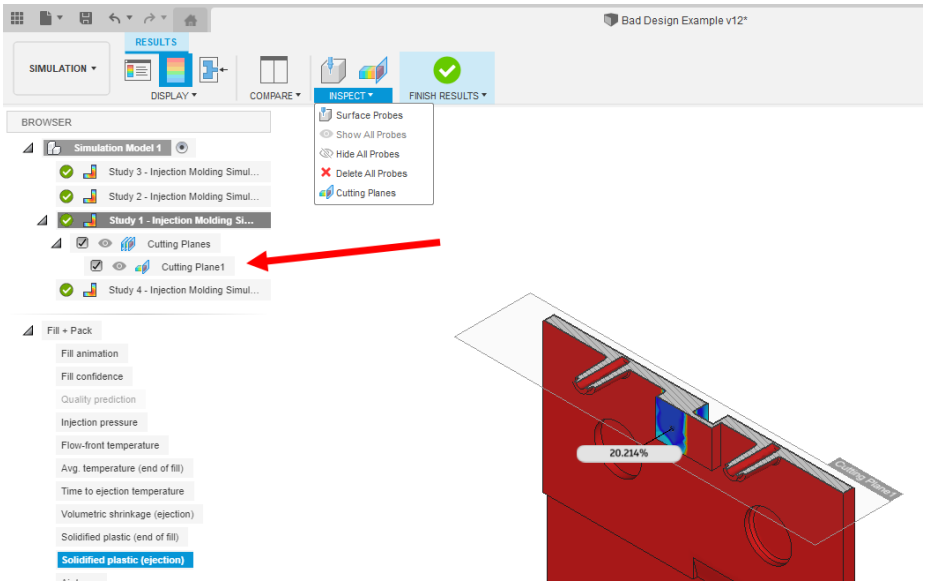
Dual and Single Injection Location Results

In this example the results are shown for one versus two injection locations. In this case both locations are still in the thin portions of the design and adding a second location did little to improve the overall filling of the mold.

The displays are synchronized, so changes in the view in one window are shown in the other window.

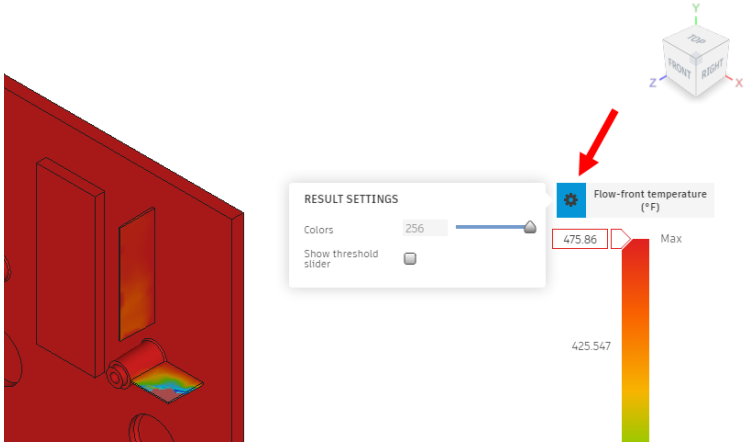
Additional Results Tools

Surface probes and cutting planes can be used to understand results in a specific area.



Surface Probe and Cutting Plane Tools

Many of the results have settings options available by clicking on the gear at the top of the legend.



Results Settings

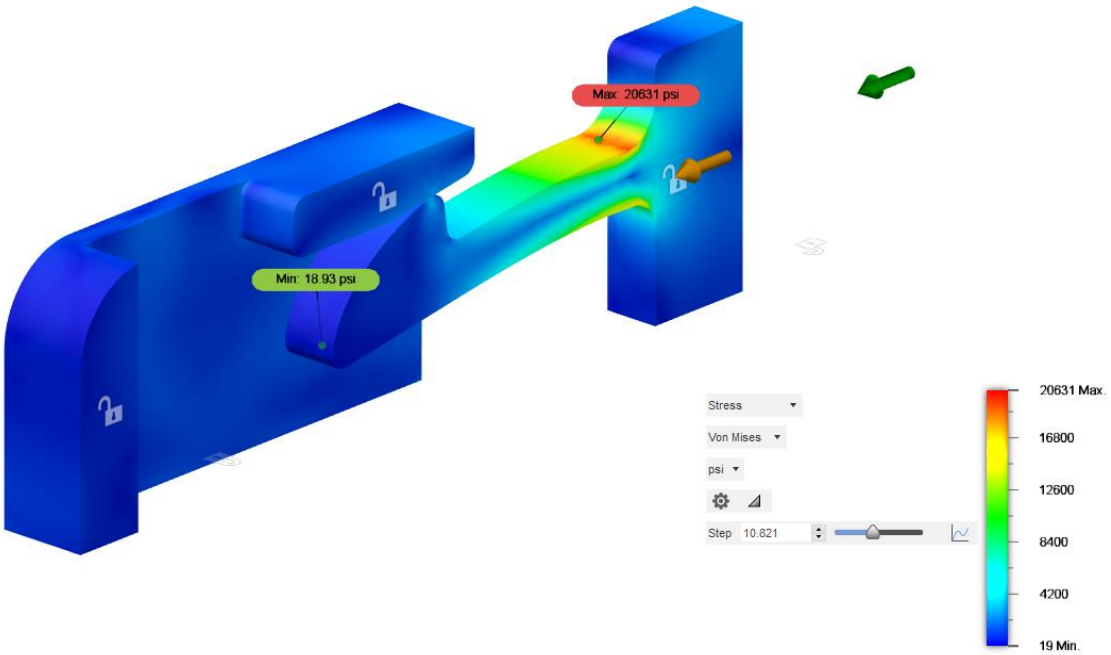
Snap Performance

The ability of a snap to perform its design task will depend on several questions:

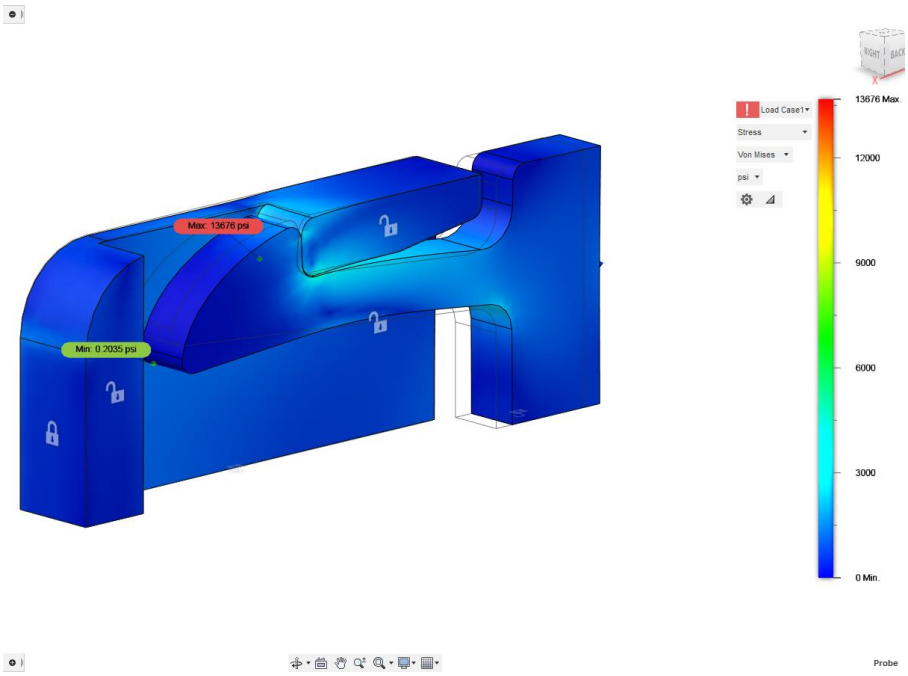
- Can the designed shape be molded?
- Will it survive assembly?
- Will it provide sufficient assembly strength?

The first question is answered by the mold filling studies. The answers to the second and third questions can be estimated by additional simulations:

- A quasi-static analysis can look at the motion of the snap during assembly. This is like a dynamic event simulation except that the event is broken into a series of snapshots, and time itself isn't a factor. This means that the inertia of the bodies isn't considered, and neither is the rate of deformation.
  A challenge with this analysis is that it is a non-linear analysis due to the large deflections involved, and accurate non-linear material properties can be challenging to find. Linear material properties can be used with caution.
- A linear static stress analysis can be run on the assembled design, with holding strength and snap pull-apart strength being evaluated from the results.



Quasi-Static Stress Results Example



Static Stress Results Example

In the example shown here the Nylon 6 initially chosen for the buckle should be replaced with a stronger material to withstand the design load when buckled and the deflection while the buckle is being snapped together.

Conclusion

Core Fusion 360 design tools can be used to develop parts that will be manufactured by injection molding, but the design productivity, overall manufacturability, and expected quality of the final pieces are greatly improved using the Product Design Extension.