CP500672

Optimize Plastic Part Designs: Injection Molding Simulation in Autodesk Fusion 360

Tim VanAst
Autodesk

Mason Myers
Autodesk

Learning Objectives

- Simulate the injection molding process to improve manufacturability
- Learn about potential manufacurating issues within plastic part design
- Understand plastic part design changes that can reduce cost and time to market
- Validate that your design is ready to be injection molded

Description

Just because you designed a plastic part, it doesn't mean that it can be manufactured at scale. Product design has a significant impact on the cost and timing to manufacture your plastic parts. You will learn how to identify and address costly manufacturing issues with Injection Molding Simulation in Fusion 360.

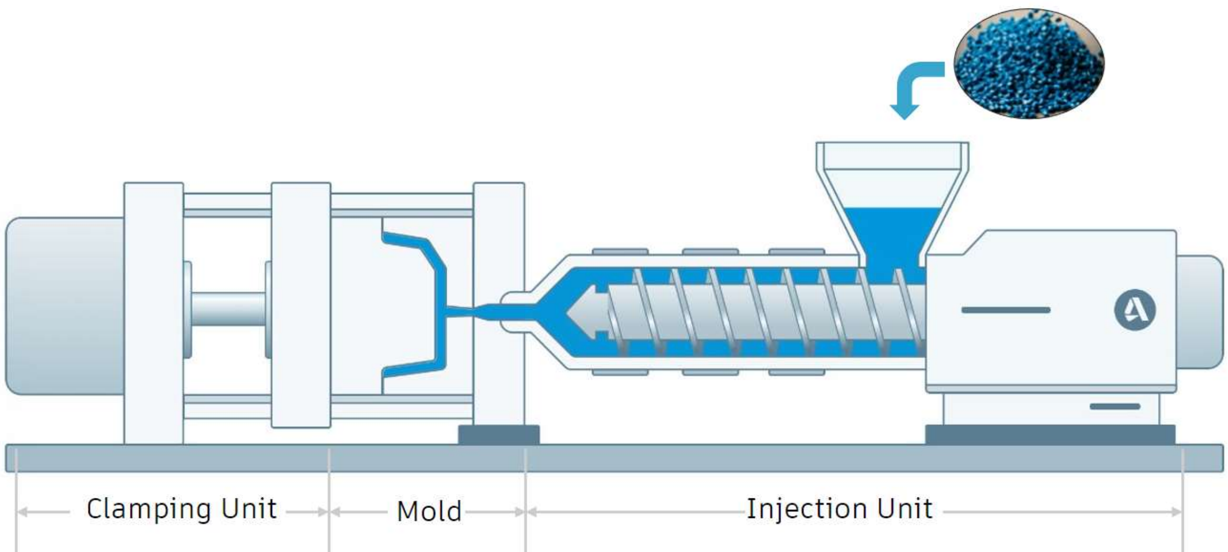
Speaker(s)

Tim VanAst has been with Autodesk for over 6 years as a Principal Implementation Consultant. He performs training and consulting to help customers to learn, understand and implement Autodesk's Moldflow injection molding simulation software. Tim has over 30 years' experience in the injection molding industry and has presented many times at local and international Moldflow user meetings, including multiple appearances at Autodesk University. Tim also actively uses Fusion Generative Design for projects for home and work. He is an Autodesk Moldflow Certified Expert. In his spare time, he like to spend time with his wife and 2 kids, read, travel, and play soccer to stay active.

Mason Myers is a Principal Implementation Consultant and has been with Autodesk over 6 years. He is a Certified Moldflow Expert and currently works with Autodesk customers to implement and adopt simulation software. Mason is a degreed Plastics Engineer from Penn State University has experience in both the thermoplastic and thermoset industries. When Mason is not running Moldflow, he enjoys spending time with his family, archery hunting, and fly fishing

Review of Injection Molding Process

Injection molding is an economical method of manufacturing many plastic parts within a short cycle. The injection molding process starts with an injection molding machine. Plastic pellets are placed into the hooper within the injection unit where they pass through to the heated barrel and screw. As plastic materials are heated, they expand slightly. Plastic materials, in turn, shrink slightly while being cooled. Various molds can be placed within the clamping unit to manufacture different plastic parts.



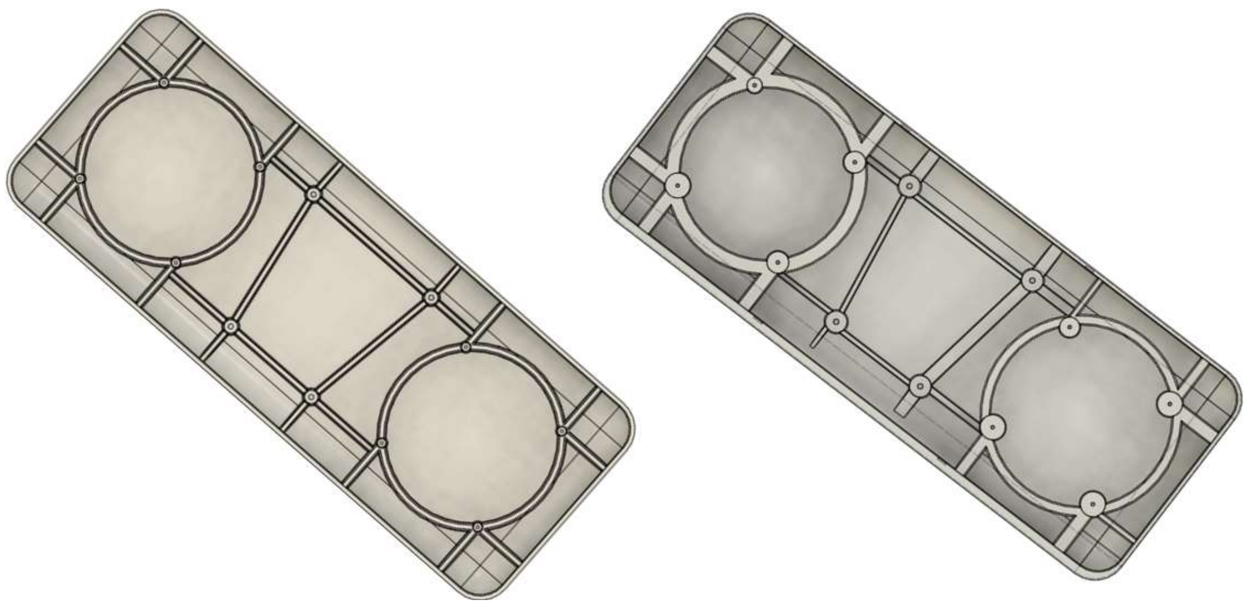
Inejction Molding Machine

The injection molding cycle can be segmented into 4 sections: filling phase, packing phase, cooling phase, and mold open phase. Typically the cooling and packing phases are the longest duration.

Rules for Good Plastic Part Design

1. Uniform wall thickness
2. Radii
3. Draft
4. Undercuts

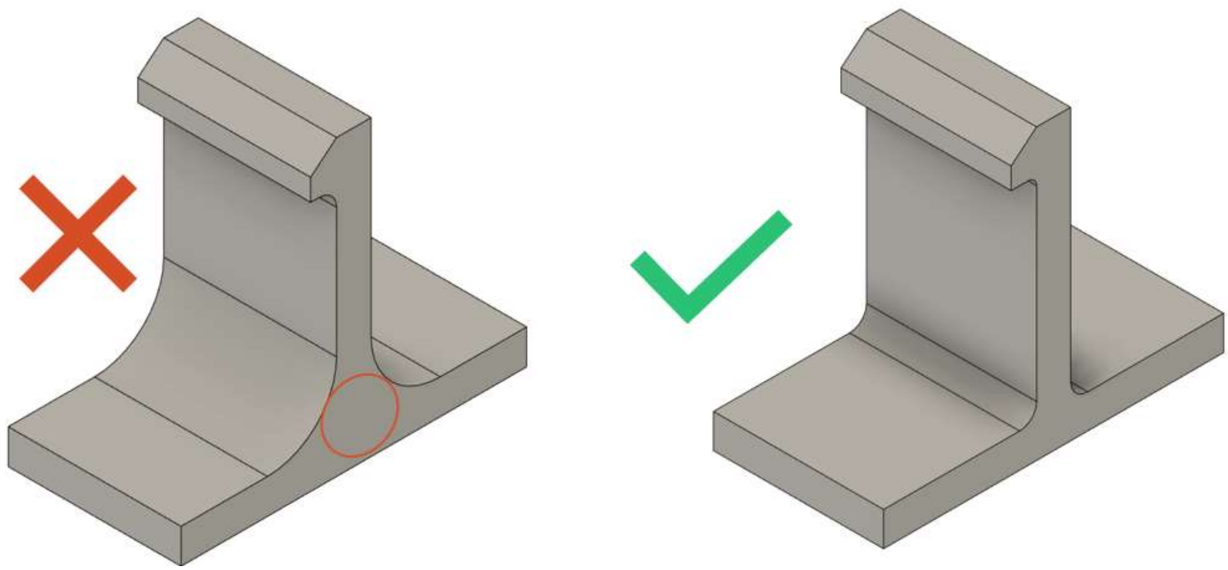
The number one rule when designing plastic parts is uniform wall thickness. This promotes even filling patterns, uniform temperatures and pressures, uniform cooling, and uniform shrinkage, which reduced the chances of warpage.



Uniform wall thickness vs non uniform wall thickness

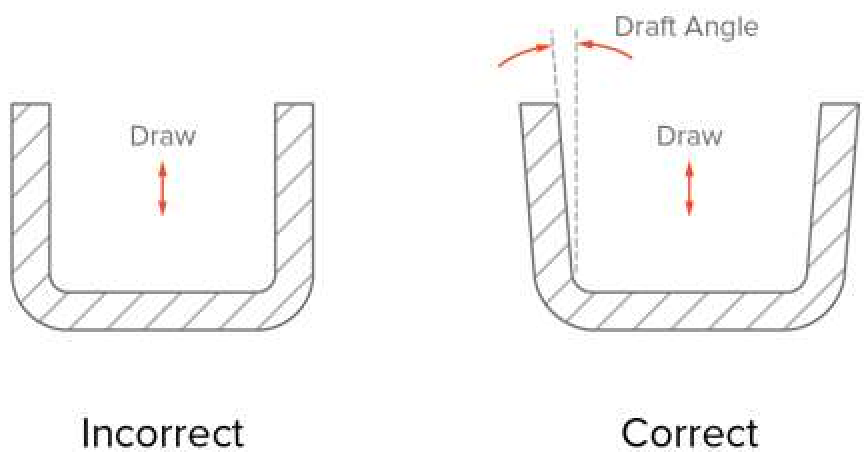
While Uniform Wall Thickness is the first rule of good plastic design, it's also the first rule that we break. If designing ribs into you part, they are normall 50-75% of the uniform wall thickness to reduce the likelihood of a visual defect called a sink mark. See, we've already broken the first design rule.

The second rule for designing plastics parts is to add Radii to all interior courners. Sharp corners are a stress concentration which significantly reduce the load that a part can withstand. However, too much of a good thing can turn into a bad thing. If you add too large of radii you can create thick sections at the rib and wall intersections, thus breaking rule #1.



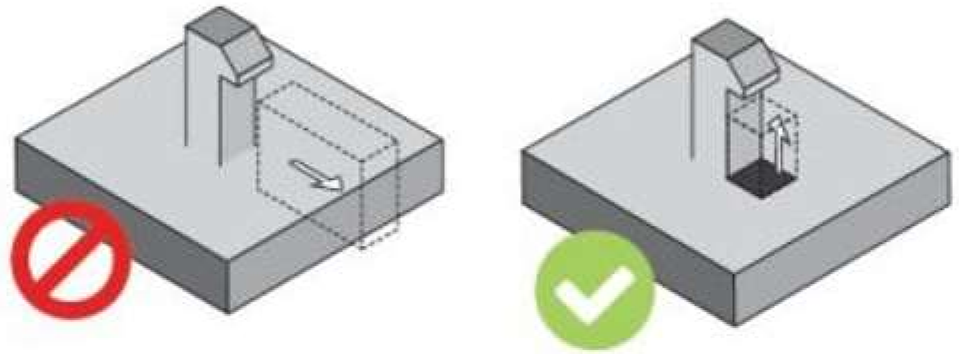
Radii

The third rule is to include Draft on all walls that are in die draw direction. Die draw is the direction that the mold will open and close. Draft allows the part to release from the tool during ejection. Improper draft can cause variation in wall thickness and may cause the part to stick in the tool.



Draft Angle

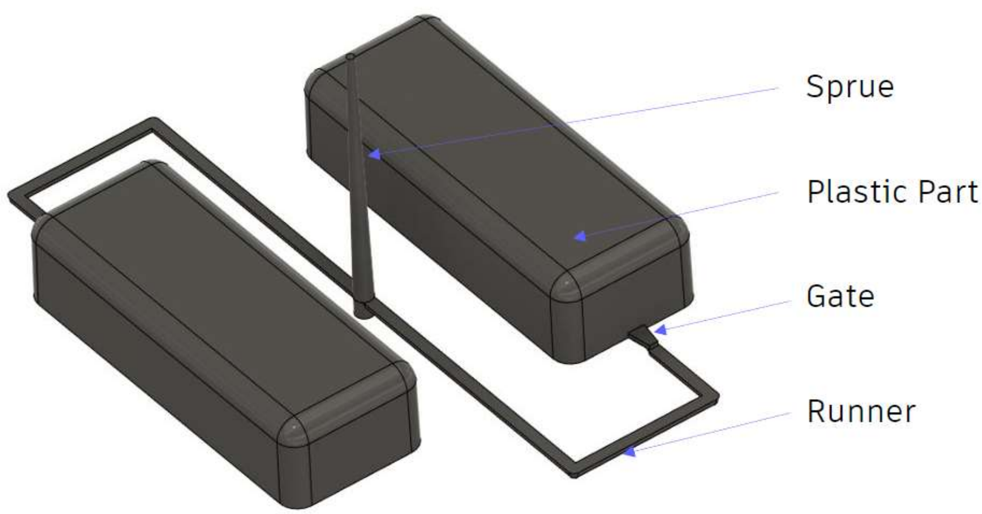
The fourth rule is to avoid undercuts when necessary. Undercuts are features in the design that will not allow the tool to open once a part is produced, thus creating a “locked” tool. If accounted for, the tool can be designed to accommodate undercuts, but it will increase the cost of the tool. This “action” in the tool to allow for undercuts is common in the industry, but some forthought can sometimes allow you to design the part without undercuts, thus avoiding additional cost.



Undercuts

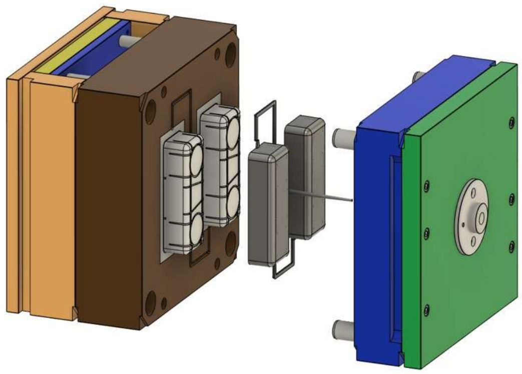
Review of an Injection Molding Tool

Injection molded components usually have some form of runner system that connects the injection unit to the molded part. Sprues, runners, and gates allow the molten plastic material to flow from the injection unit into the plastic part cavity.

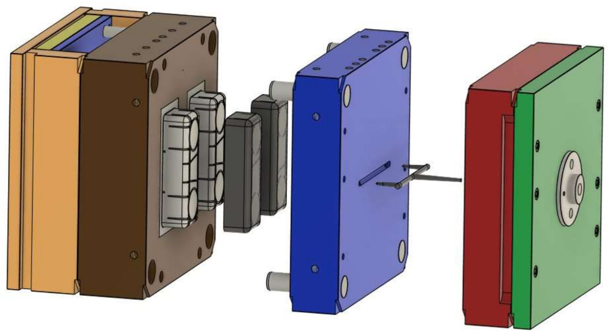


Plastic Part with Gate, Runner, and Sprue

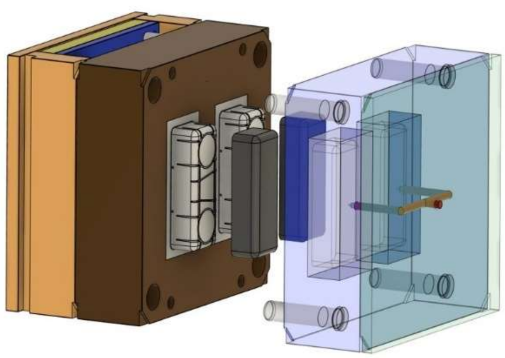
Injection molds can be designed and manufactured in different forms. 2 Plate and 3 Plate mold constructions are common with cold runner designs. 2 Plate cold runner molds limit the gate to the outer edge of the plastic part. 3 Plate cold runner molds have an additional plate to contain a different runner configuration that allows for additional gate locations. Hot runner molds can also be used to reduce the amount of molded scrap produced by cold runner molds, but they add to the cost of the tool.



2 Plate Cold Runner Mold

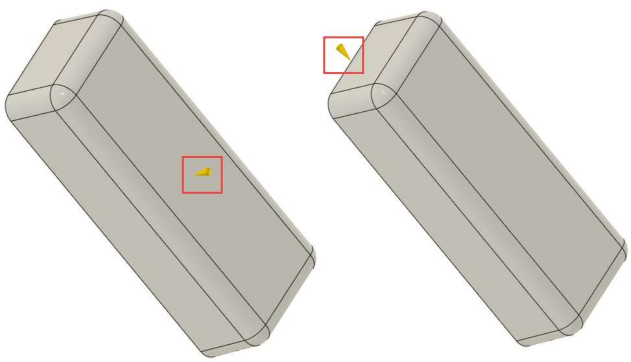


3 Plate Cold Runner Mold



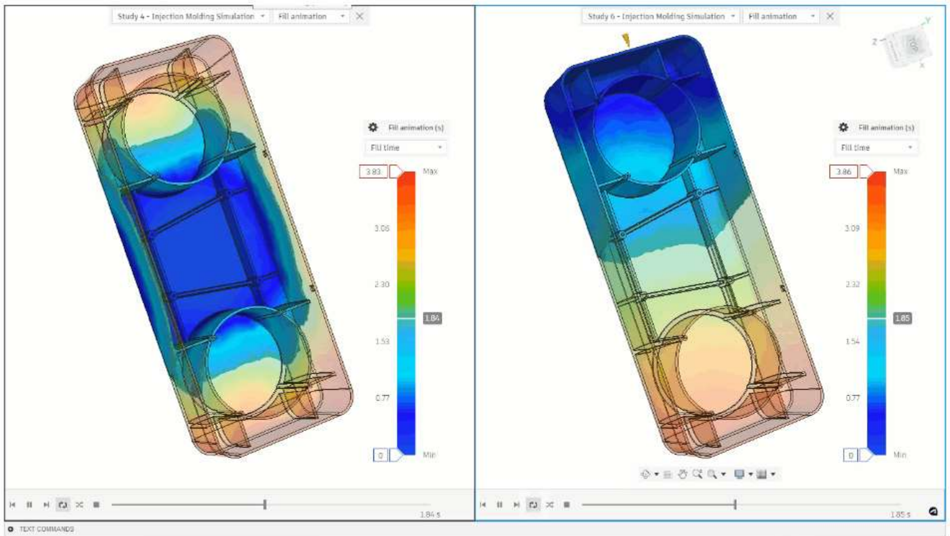
Hot Runner Mold

Injection molding simulation in Fusion 360 allows you to place an injection location to simulate the injection molding process. Placing the injection location around the perimeter would represent a 2 plate mold while gating on the top surface would represent either a 3 plate or hot runner design.

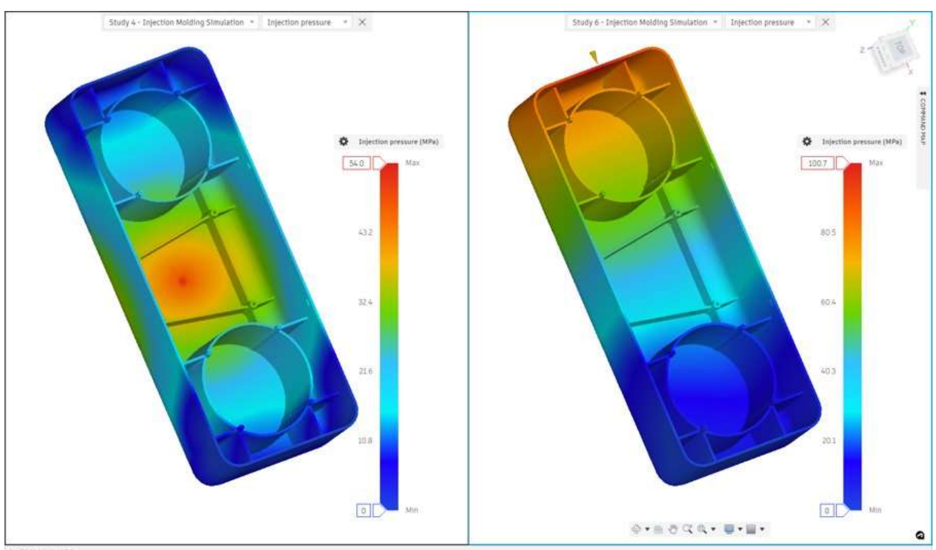


Two Different Injection Locations in Fusion 360

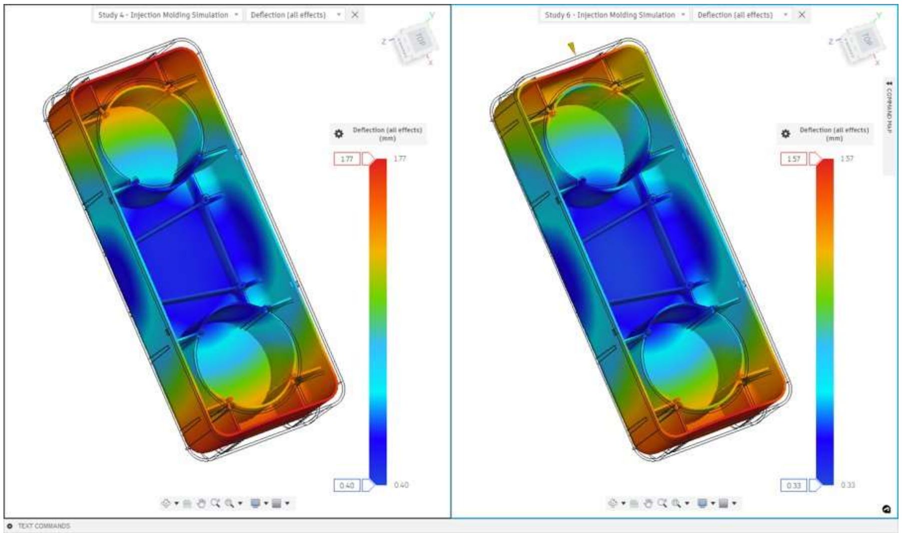
Placing the injection location at different areas can impact the filling, required injection pressure, and overall deflection after the part has been molded.



Fill time comparison with two different injection locations



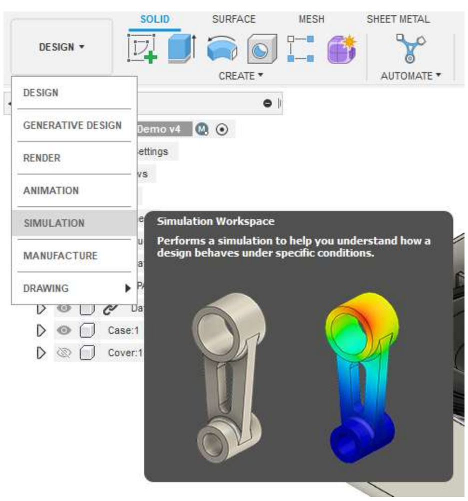
Injection pressure comparison with two different injection locations

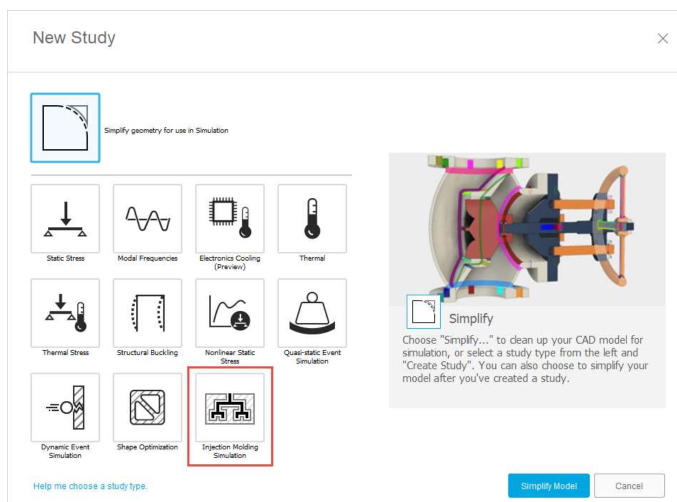


Deflection comparison with two different injection locations

Injection Molding Simulation in Fusion 360

Switching from the Design environment in Fusion 360 to the Simulation environment will allow the use of an Injection Molding Simulation study.

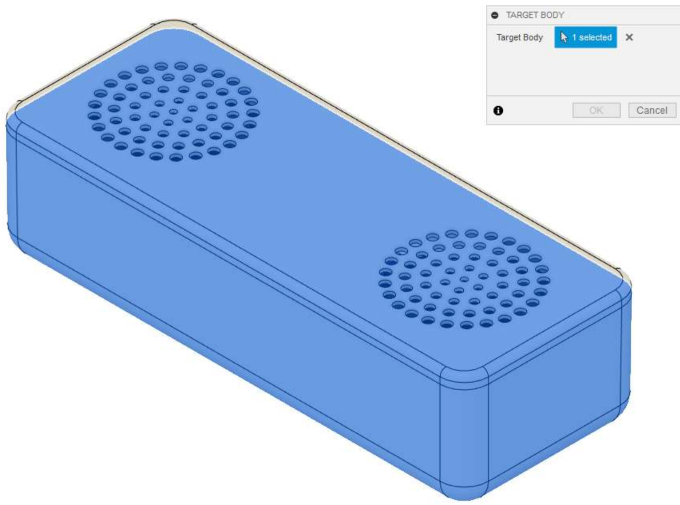




Simulation Workspace

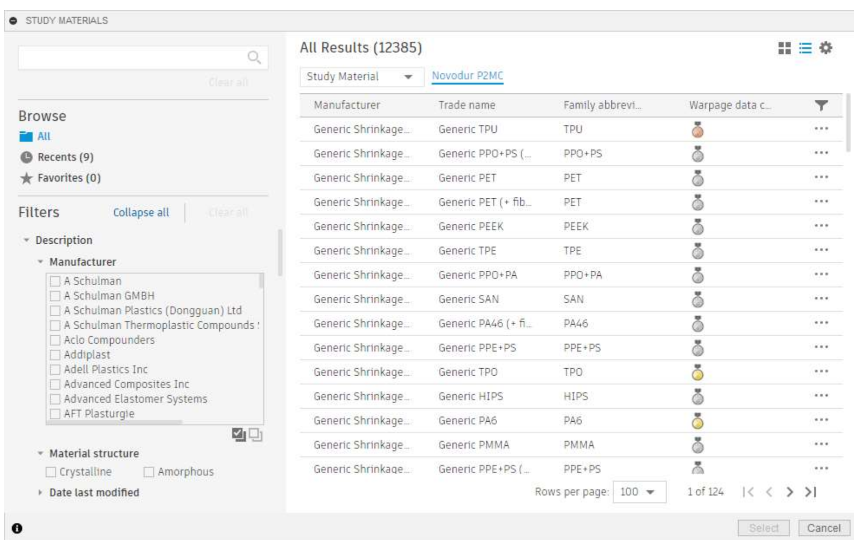
Injection Molding Simulation Study

If simulating an assembly, a target body can be selected to simulate the injection molding process.

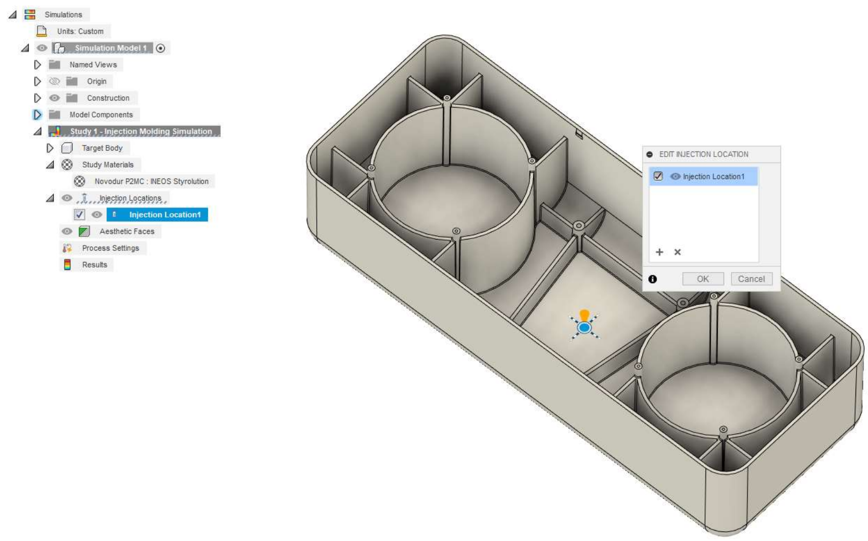


Target Body Selection

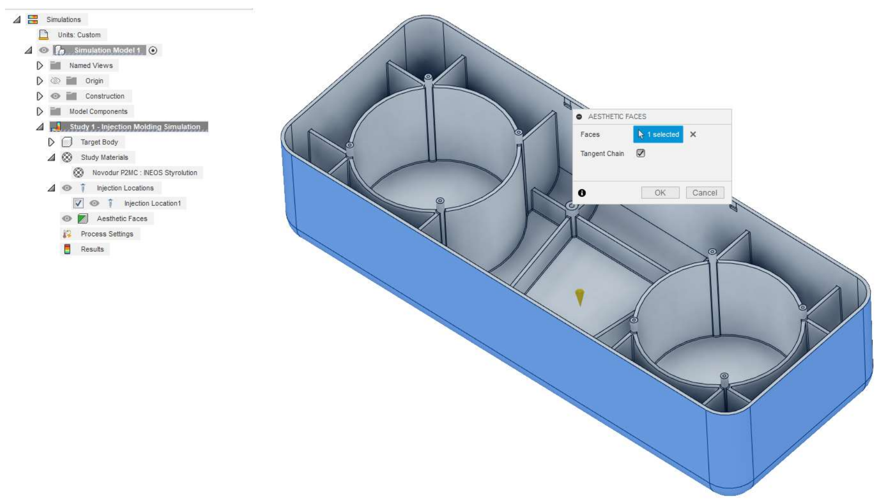
Study materials, injection location(s), aesthetic faces, and process settings can all be included or adjusted prior to solving the simulation.



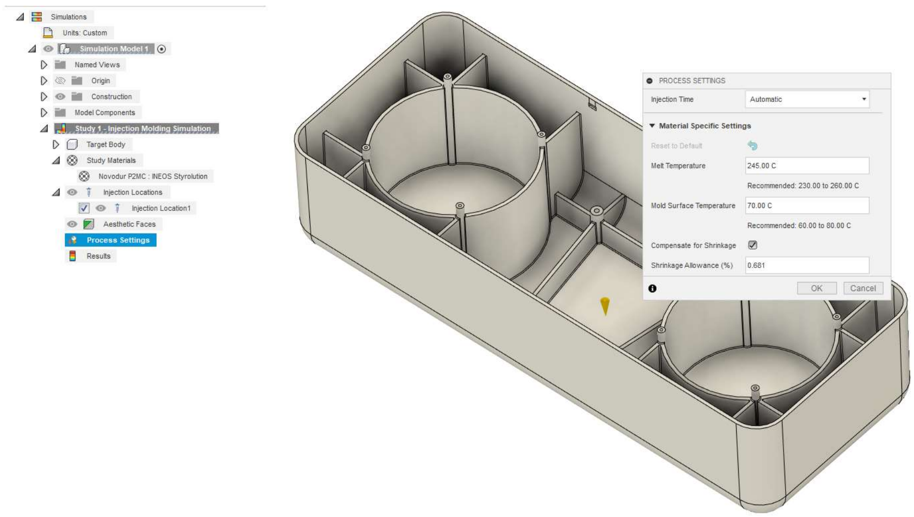
Study Materials list



Injection locations can be added or moved



Aesthetic Faces can be selected

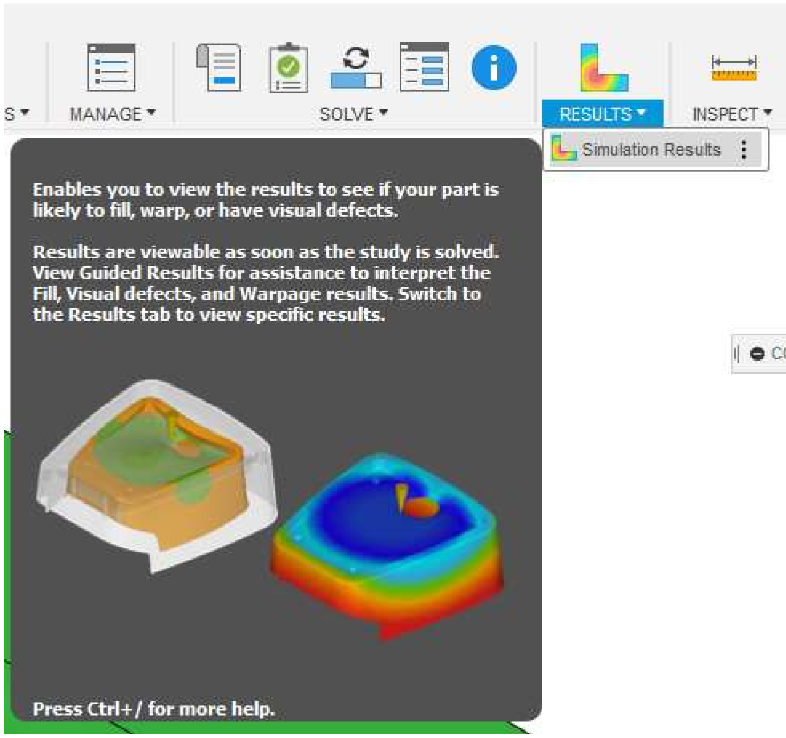


Process Settings

Once you are ready to solve the simulation you can review the Setup Summary, complete the Pre-Check, and/or Solve the Simulation.

Location of Setup Summary, Pre-Check and Solve

Once the Simulation has completed, results can be reviewed under the Results – Simulation Results dropdown.

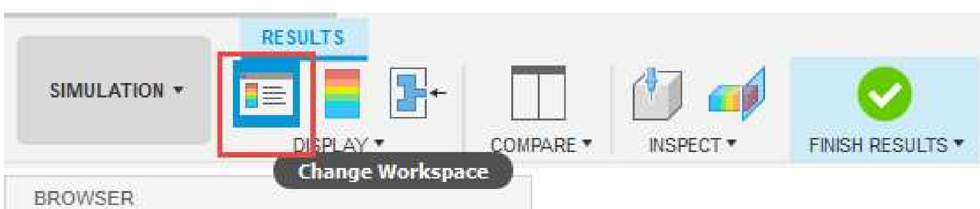


Results dropdown

There are multiple ways to review injection molding simulation results in Fusion 360 ranging from Guided Results, Results, and Molding Process.

Guideded Results help to answer these three primary questions:

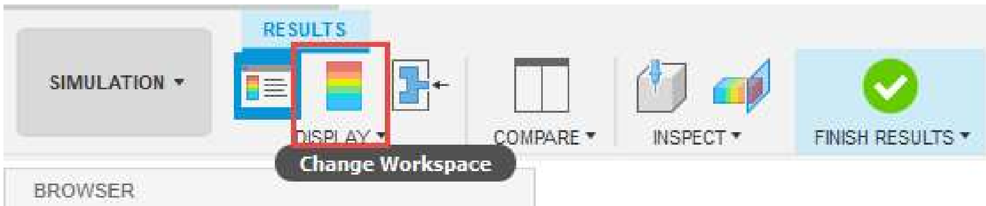
Will my part fill?
Will my part have visual defects?
Will my part warp?



Guided Results

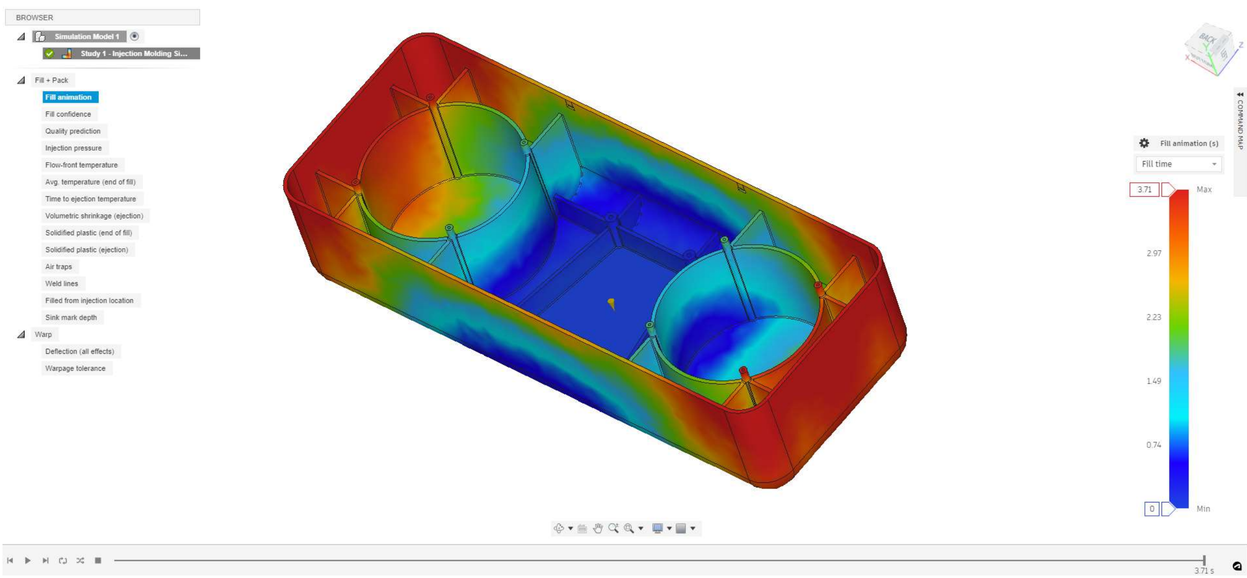
For each question, a plot can can be viewed that helps to highlight potential problems. These problems can range from difficulty filling the part, cosmetic issues, or deflection problems. Next Step information is also provided on changes that can be made to reduce or eliminate the potential problems.

Results display show various plots grouped by Fill+Pack and Warpage. Each plot has a unique scale that should be reviewed while evaluating the individual plots.



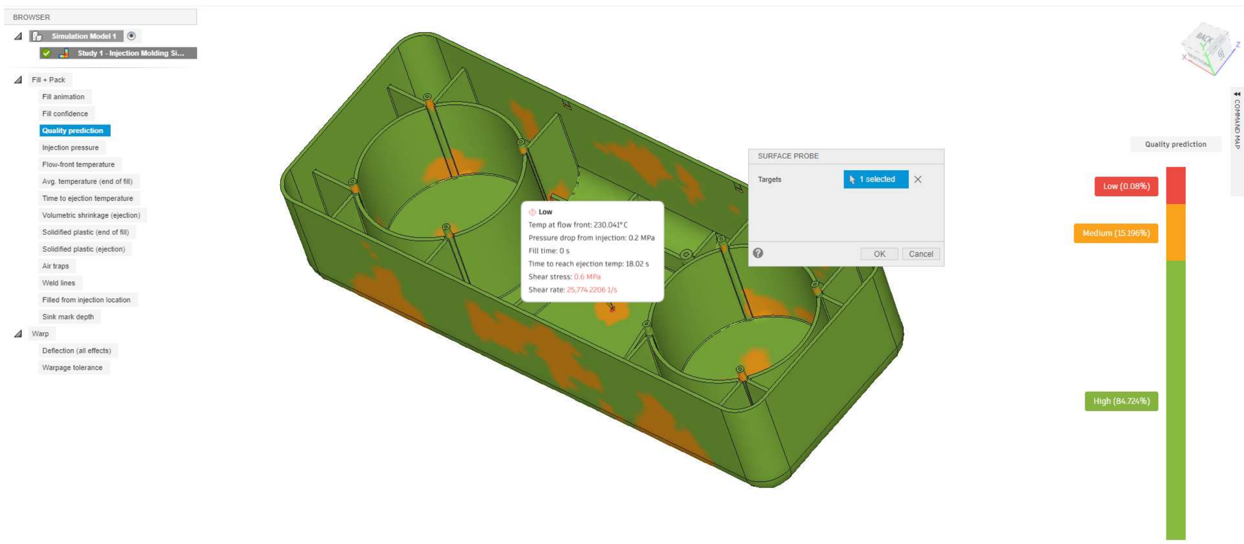
Results

Some plots like Fill animation can be animated over time to show how the plastic flows within the mold.



Fill animation

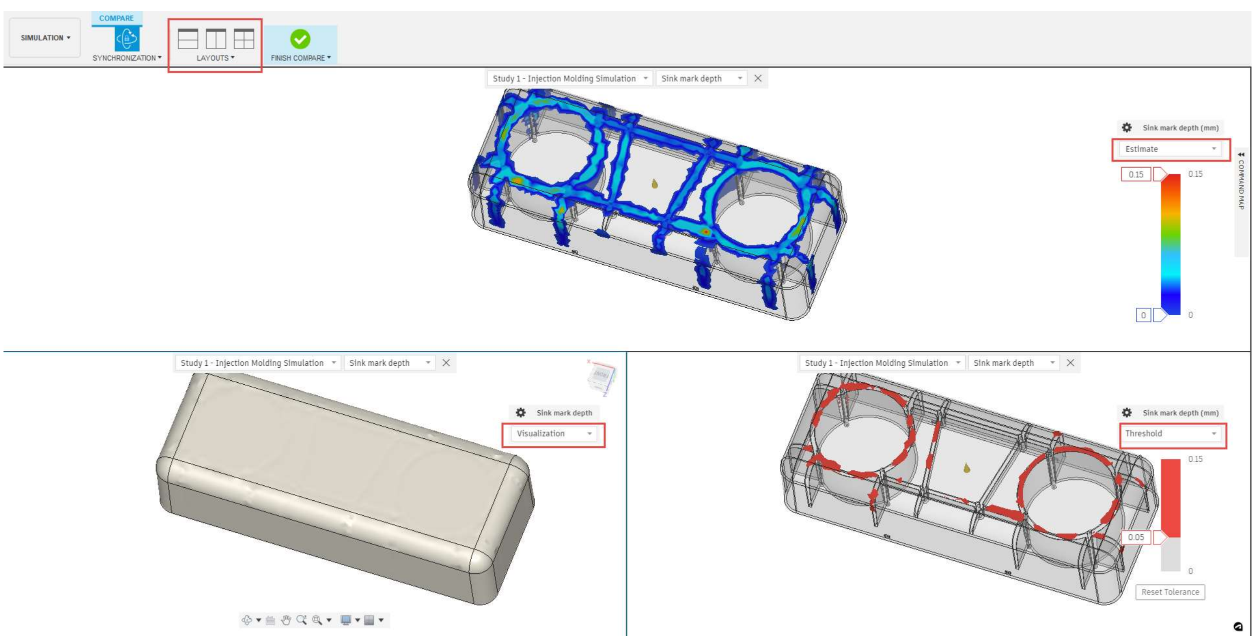
Plots like Quality Prediction and Fill Confidence use a stop light color scheme of red, yellow, and green. Using Surface Probes can additional information to aread of concern.



Quality Prediction with Surface Probes

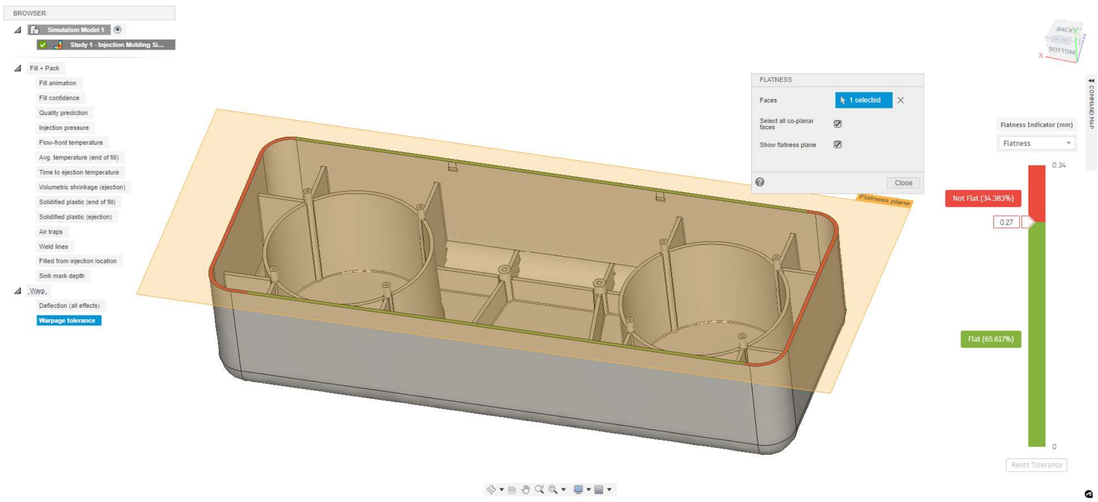
Molding Process results can provide information that occurs during the cycle in either an Overview or Table format. The Overview area can be animated throughout the cycle to show key aspects of the Filling, Packing & Cooling, and Ejection stages.

Results like Sink Mark Depth can be viewed in multiple ways by using the dropdown on the scale. Estimate, Threshold, and Visulation can all be viewed at the same time by using the Compare and Layouts view.

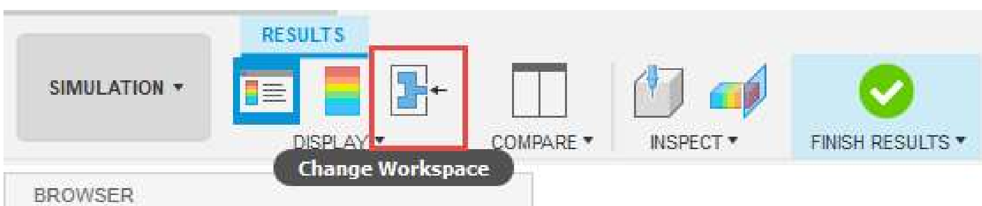


Sink Mark Depth Estimate, Visulatization, and Threshold

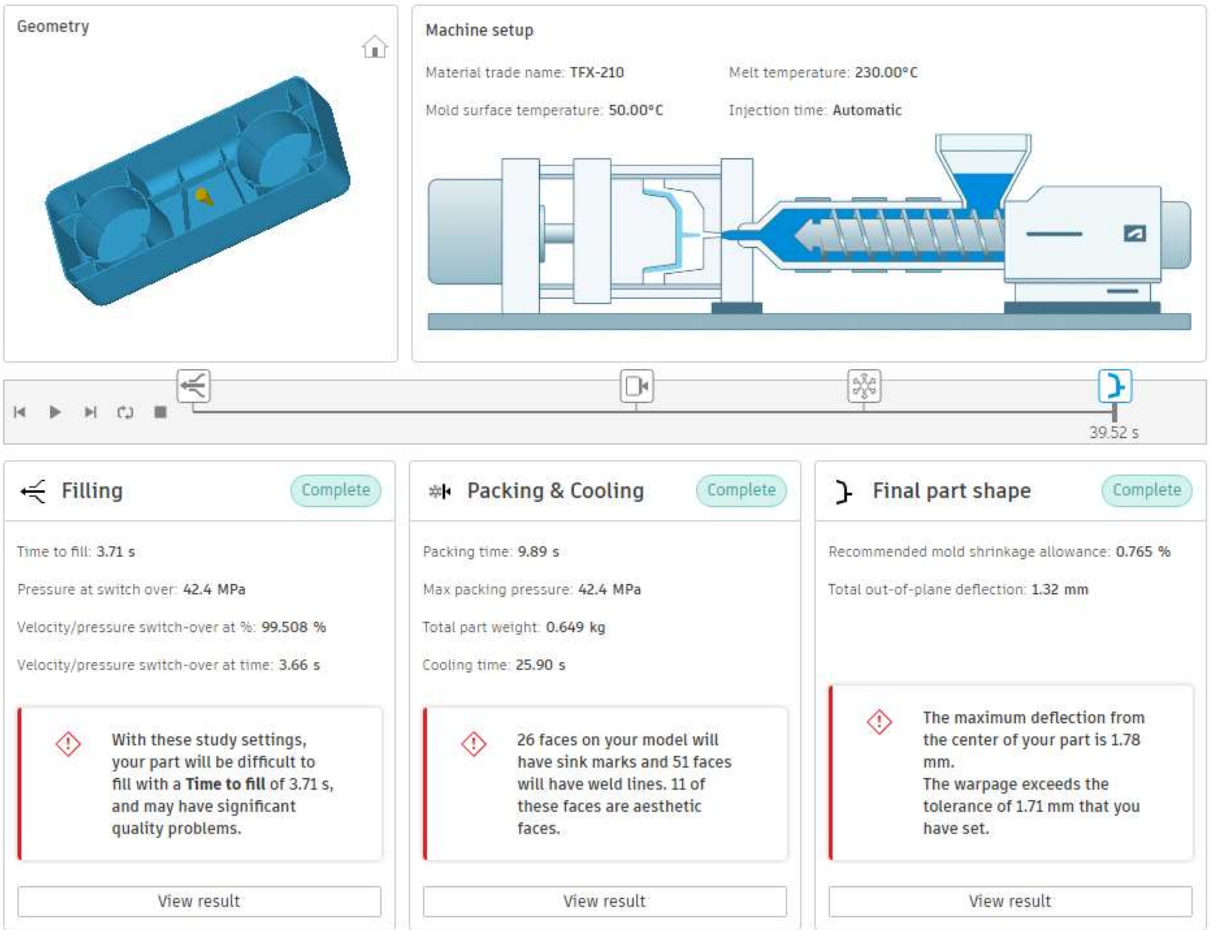
Deflection (all effects) and Warpage Tolerance can be view to understand how the part will shrink and warp post molding. Warpage Tolerance can also be adjusted to show Flatness compared to a reference plane.



Flatness



Molding Process



Molding Process Overview

Time (s)	Volume (%)	Pressure (MPa)	Flow rate (m^3/hr)	Frozen vol (%)	Status
Filling Phase Complete					
0	0.002	0.5	0.016	0	Velocity
0	0.03	0.8	0.011	0	Velocity
0.01	0.061	3.4	0.025	0	Velocity
0.03	0.198	6.5	0.106	0	Velocity
0.06	0.607	10.2	0.259	3.331e-4	Velocity
0.10	1.407	13.6	0.397	0.071	Velocity
0.19	3.771	17.6	0.498	0.57	Velocity
0.32	6.854	20.0	0.58	1.057	Velocity
0.42	9.779	21.7	0.608	1.393	Velocity
0.50	11.97	22.9	0.614	1.74	Velocity
0.59	14.572	24.1	0.619	2.03	Velocity
0.71	17.857	25.6	0.622	2.294	Velocity
0.83	21.202	26.7	0.626	2.56	Velocity
0.95	24.504	27.5	0.632	2.763	Velocity
1.08	28.333	28.4	0.634	2.974	Velocity
1.20	31.514	29.2	0.635	3.161	Velocity
1.32	35.097	30.0	0.636	3.37	Velocity
1.45	38.679	30.9	0.636	3.568	Velocity
1.58	42.291	31.9	0.635	3.779	Velocity
1.70	45.467	32.8	0.635	3.976	Velocity
1.82	49.061	33.9	0.635	4.168	Velocity
1.96	52.684	35.2	0.635	4.376	Velocity
2.08	56.161	36.2	0.636	4.57	Velocity
2.21	59.746	37.3	0.637	4.753	Velocity
2.34	63.379	38.3	0.637	4.947	Velocity
2.45	66.618	39.1	0.639	5.094	Velocity

Molding Process Table

Additional self paced learning on Injection Molding Simulation can be found here on our Fusion 360 webiste.

Summary

When designing parts to be manufactured in plastics there are some basics rules that should be followed. Autodesk Fusion 360 can help you verify if these rules were followed and can help you understand if your design is manufacturable. The Simulation Extension now includes injection molding simulation. Ensuring that your part is manufacturable will save time and money.