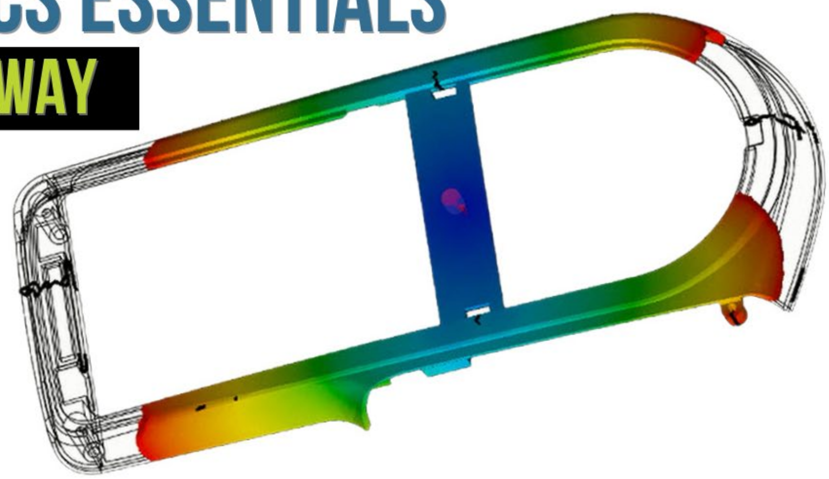
SOLIDWORKS PLASTICS ESSENTIALS TAKE-AWAY



Lessons

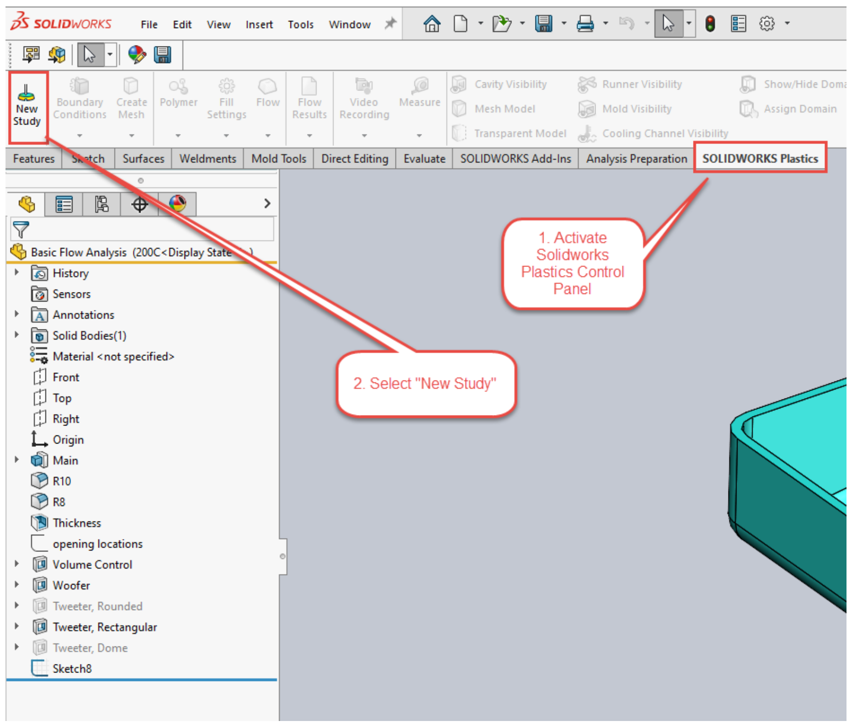
Disclaimer: This document is a comprehensive summary of critical key takeaways from lessons within SOLIDWORKS Plastics Essentials offered by GoEngineer. This document should not be considered a substitute for an official SOLIDWORKS training course.

LESSON 1: BASIC FLOW ANALYSIS

Starting a New Study

SOLIDWORKS Plastics is an add-in in the SOLIDWORKS CAD program. Once activated, the SOLIDWORKS Plastics CommandManager is available.

New Study: A simulation always begins by defining a new study. With a part file open, click on the Plastics CommandManager tab and select New Study.



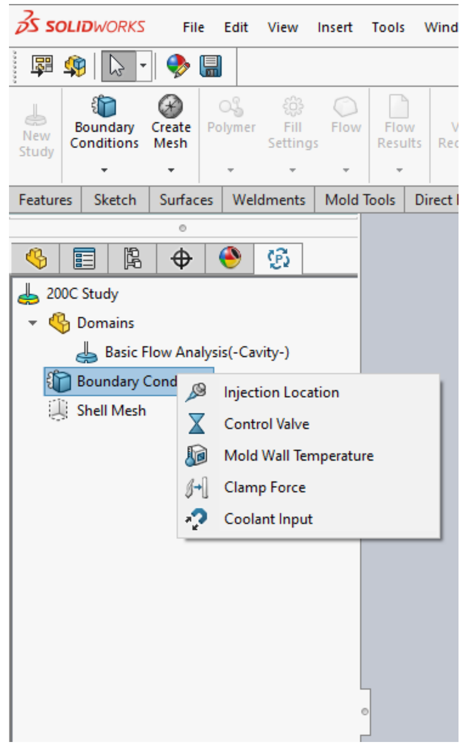
Injection Process: The injection process settings define the injection method into the mold.

Single Material is most common.

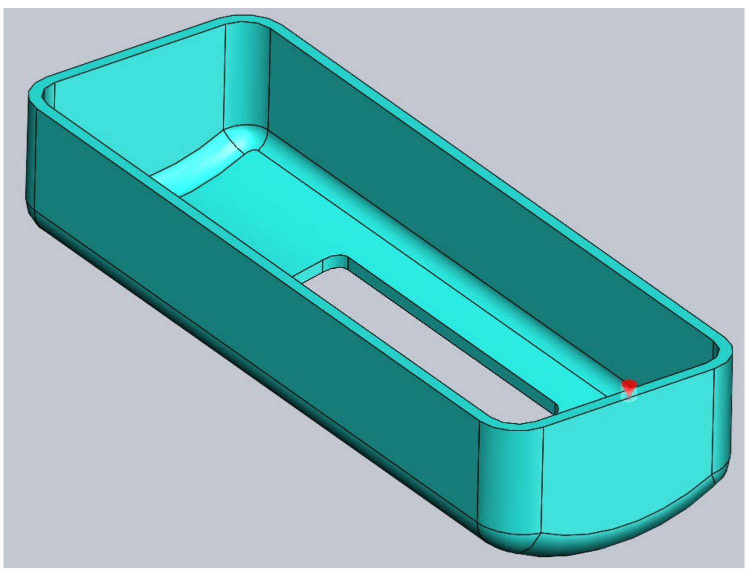
Injection Process	Mesh Support	Description
Single Material	Shell/Solid	Only a single polymer material is considered in the injection process.
Co-Injection	Solid	Two different polymer materials are injected at the same location.
Bi-Injection	Solid	Two different polymer materials are injected (possibly simultaneously) at different locations, with each injection process being independently controlled.
Gas-Assist	Solid	A bubble of gas, typically nitrogen, is injected into the melt stream to create a hollow core in the molded part.
Water-Assist	Solid	Water is injected into the melted polymer stream to create a hollow part.

Boundary Conditions

Boundary conditions are used to define what happens to the outer elements of the physical region where the simulation occurs. Boundary conditions can be used to define the injection location, clamp force direction, mold wall temperatures, and several other parameters.



Injection Location



The injection location is where the melted plastic flows from the runner system into the part cavity. Multiple locations can be used but at least one is required to run an analysis.

Injection locations can be placed on vertices, sketch points, or faces.

The mesh is made denser at the injection location for accuracy purposes.

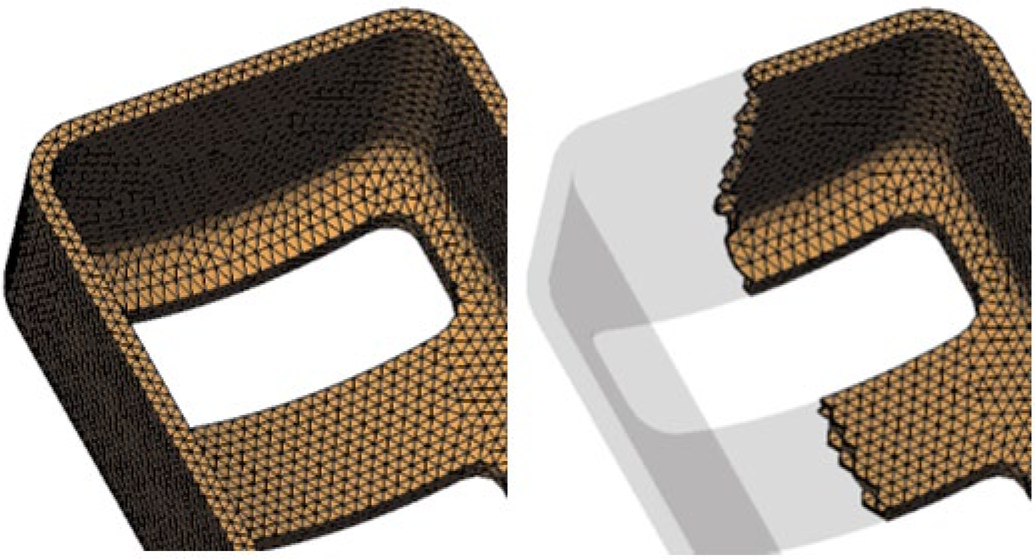
A solid mesh is required to apply an injection location to a face.

Mesh Elements

The plastic flow is simulated through a cavity by representing the cavity geometry with a collection of simple shapes. These simple shapes are idealized so that flow can be calculated through them. Each shape is called an element and the collection of elements is called a mesh.

Element Types

The elements are either **Shell (2D)** or **Solid (3D).** Shell elements cover just the surface of the body while solid elements fill up the volume of the body.



Shell Elements

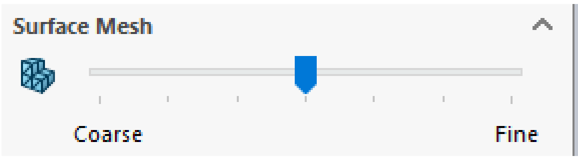
A shell mesh is used on thin-walled parts for obtaining quick results in the early stages of the analysis process.

Solid Elements

A solid mesh can provide accurate results for any type of model, thin or thick. A solid mesh will provide greater accuracy for models with complex and intricate geometry.

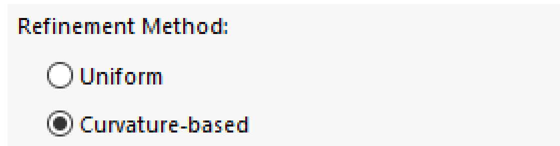
Create Mesh

The Create Mesh command sizes the elements. Element size is important because the mesh must accurately represent the geometry and flow conditions.

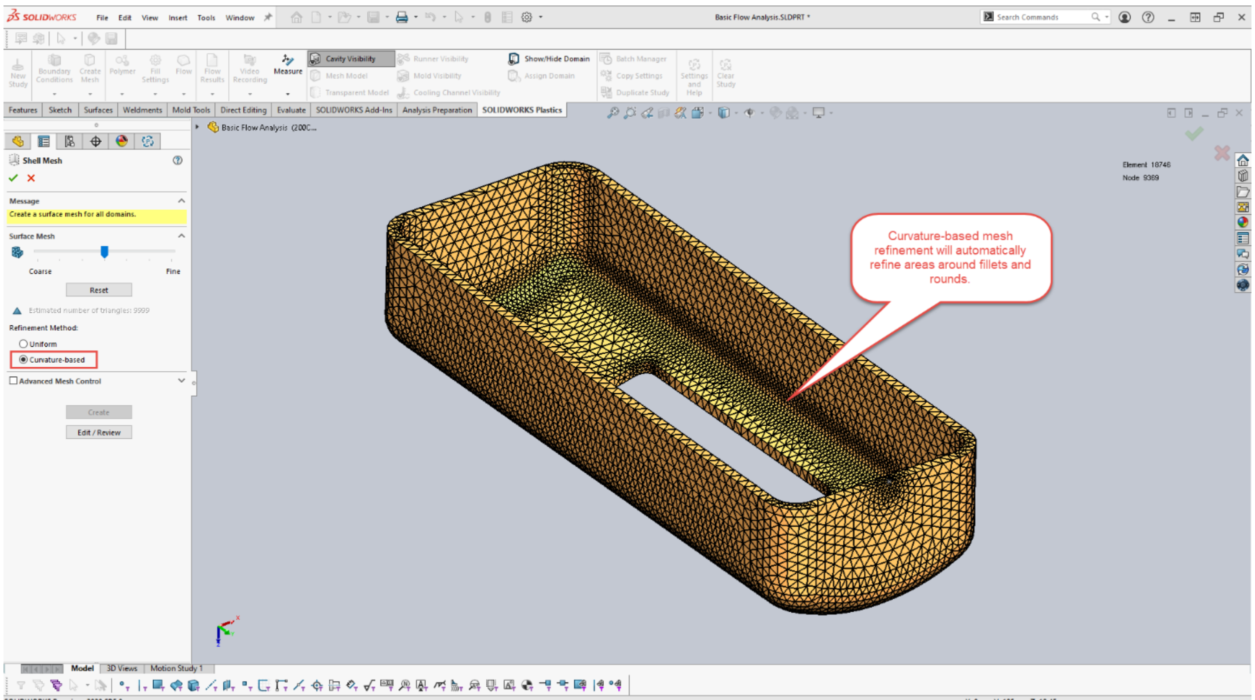


The slider within the Create Mesh command controls the global size of the mesh.

The uniform refinement method will apply the global mesh value to all geometry.



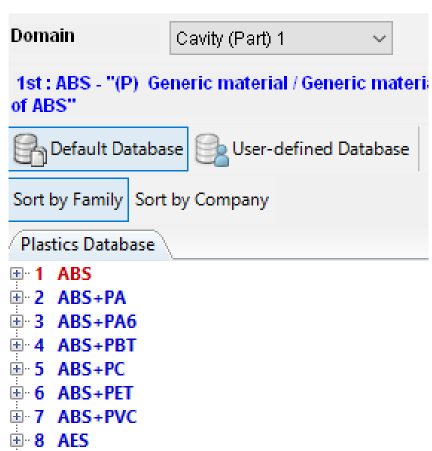
The Curvature-based refinement method will create a denser mesh in areas of curvature such are radii and rounds.



Material

The material database provides access to thousands of materials. Materials can be applied to cavities, mold bodies, cooling fluids, and inserts.

Polymers are organized in the material database by family and company. Polymers can be applied to cavities and inserts.



Running a Flow Analysis

The process of creating an analysis in SOLIDWORKS Plastics is broken down into three distinct phases: preprocessing, processing, and post-processing. The stages of creating a mesh, applying material, and specifying fill settings are preprocessing operations. The next phase, the processing or RUN phase, is where the simulation is calculated. There are several processing options available.

Flow is used to analyze the fill stage.

Pack is used to calculate the packing stage and the pure cooling stage of the analysis.

Warp is used to analyze the shrinkage and warpage of the part due to residual stresses during the injection molding cycle.

When using **Flow+Pack+Warp,** all three stages are performed sequentially during the analysis.

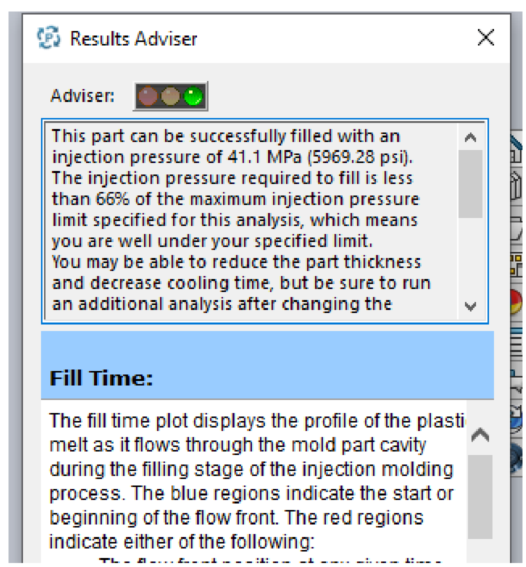
Cool is used to analyze how the part cools throughout all stages of the injection molding process.

Note: **Flow+Pack** is only available with SOLIDWORKS Plastics Professional and **Flow+Pack+Warp** and **Cool** are available with SOLIDWORKS Plastics Premium.

Results Advisor

At the end of the Flow analysis RUN, the Results Advisor will pop-up.

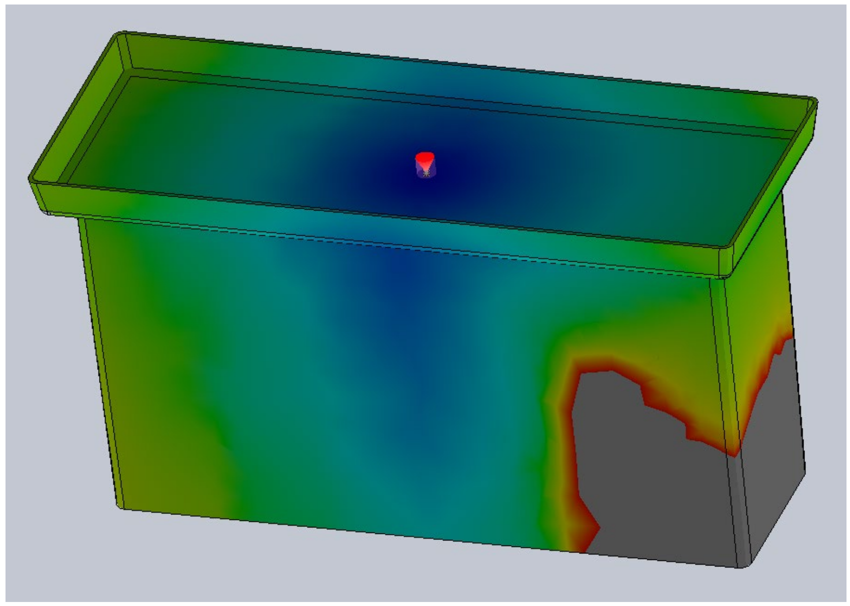
The advisor defines various result plots available in post-processing. This is a very informative tool when learning SOLIDWORKS Plastics simulation. It will describe what is being shown in each of the result plots and what settings may have an effect on the results.



LESSON 2: DETECTING A SHORT SHOT

Detecting Short Shots

Short shots occur when molten plastic cools and solidifies before the cavity fills all the way.



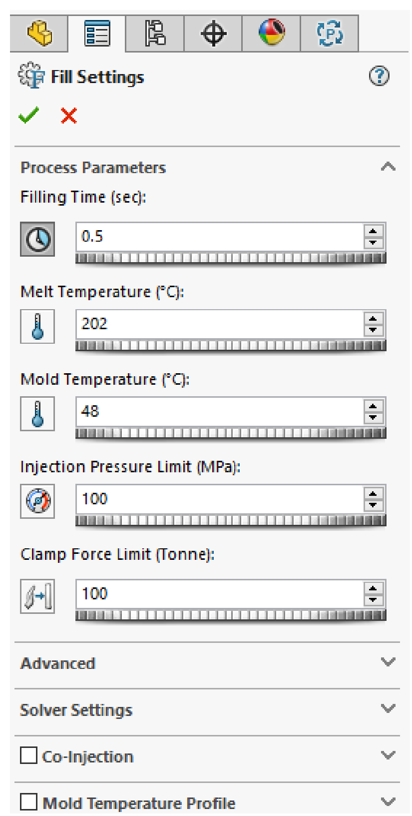
The system will indicate that the filling analysis has failed when a short shot occurs. This will happen if the maximum injection pressure exceeds the limit in the **Fill Settings** and the flow rate drops to a point that material quits flowing and solidifies.

** Warning C1002 ** Pressure has reached the max. inject pressure 100.0 MPa
!! Error C2005 !! A Short-Shot has occurred during the Flow Process
The flow rate is less than 0.010000 of initial flow rate.
Inlet Gate Pres.= 100.00 Mpa Is too Small

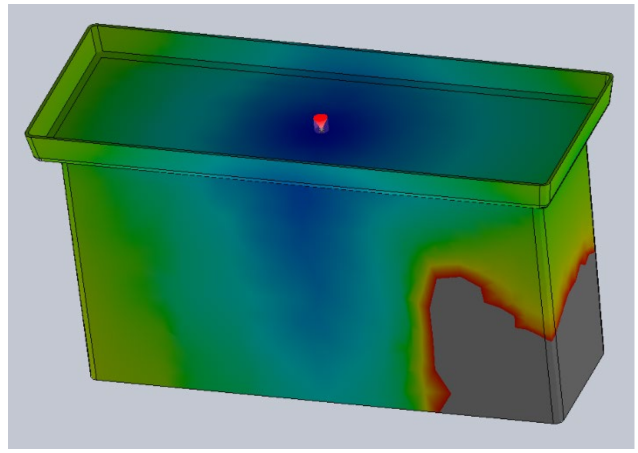
Fill Settings

The **Fill Settings** control the parameters of the machine throughout the fill stage. These parameters include filling time, melt temperature, and mold temperature. Changing these parameters might be able to fix the short shot issue but may cause other problems to arise. For example, a hotter mold and a shorter filling time may fix the short shot issue but changing these parameters will also increase cooling time and molded-in stresses.

The default values used in the **Fill Settings** dialog are estimated by the software based on the part volume and material manufacture recommendations for melt and mold temperatures.

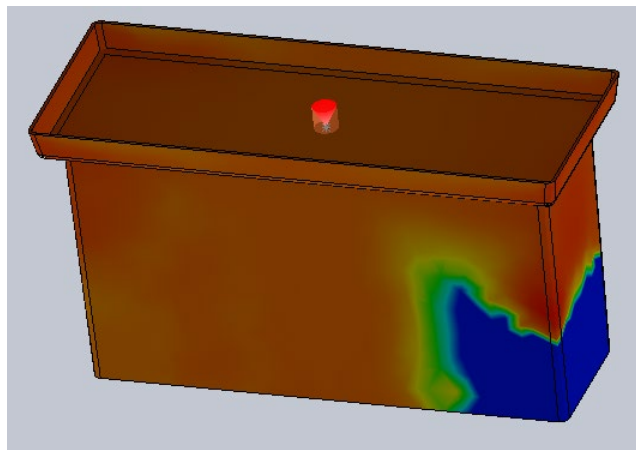


Results Plots with Short Shots



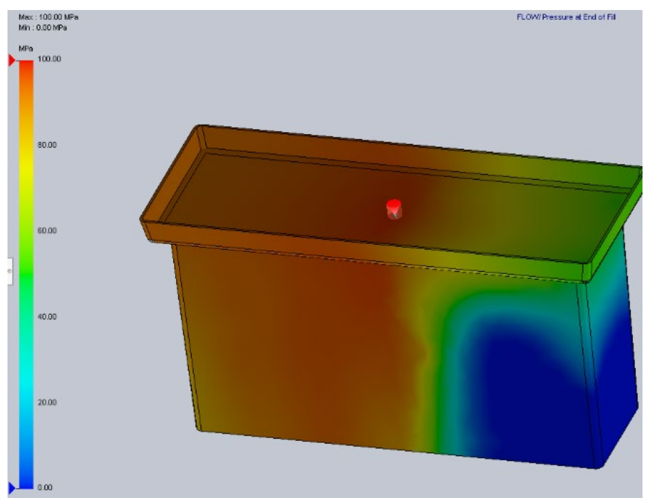
Fill Time Plot

The Fill Time result shows that the flow stops well short of filling the mold.



Flow Front Central Temperature

The Flow Front Central Temperature measures the temperature of the plastic flow front as it flows through the mold. This plot will show that the plastic cools before filling the mold.



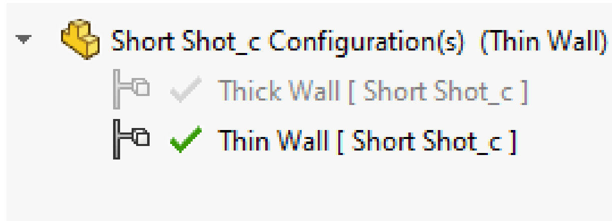
Pressure at End of Fill

The Pressure at End of Fill result shows the pressure needed to fill the cavity. This result can predict the required pressure for the entire mold including the sprue, runner, gate, and cavity is all the geometry that is represented in the mesh.

This plot indicates that the injection pressure limit of 100 Mpa has been reached.

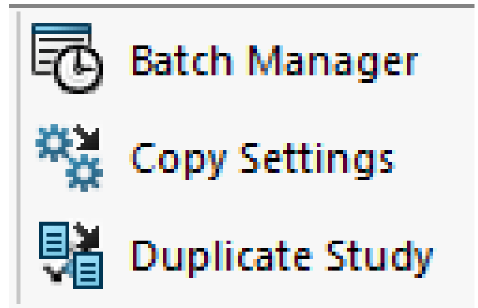
Configurations

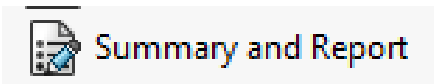
In SOLIDWORKS Plastics, configurations are assigned for each study. A configuration cannot contain more than one project. If the geometry of a study changes, the mesh must be recreated, the boundary conditions may need to be reconstructed, the simulation must be rerun, and the results from the original study are lost. Configurations help solve the issue of overwriting previous analyses.



LESSON 3: AUTOMATION TOOLS

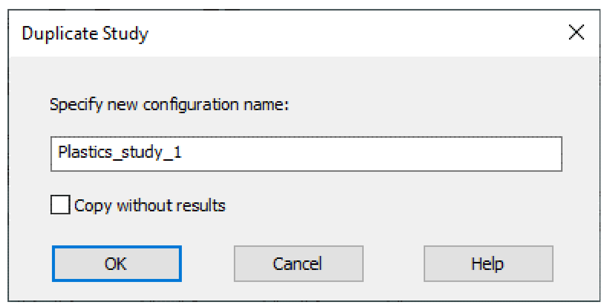
The automation tools in SOLIDWORKS Plastics are the **Batch Manager**, **Copy Settings**, and **Duplicate Study** commands. These are used in the preprocessing and processing stages. The **Summary and Report** generator is also an automation tool used in post-processing.





Duplicate Study

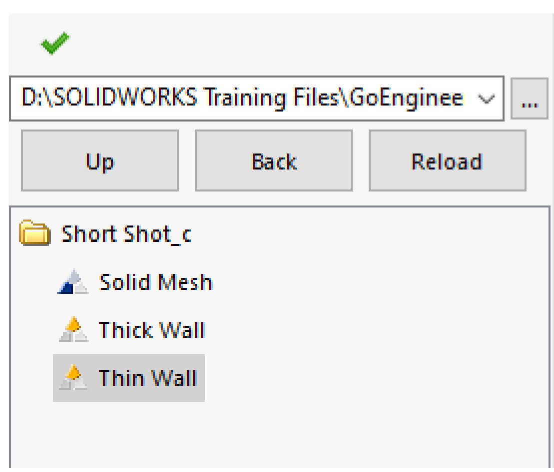
The **Duplicate Study** command is used to copy the settings of a simulation to a new configuration. Using configurations allows for multiple studies to be run on a single part file without overwriting existing studies.



The **Duplicate Study** command always copies the preprocessing information and can be used to copy the results as well.

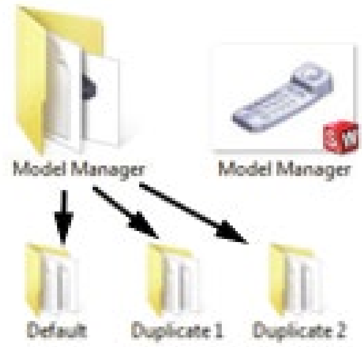
Copy Settings

The **Copy Settings** command is used to copy preprocessing data from one simulation to another. This includes parameters such as material and fill settings, but not information pertaining to the mesh or boundary conditions.



The settings can be copied from studies of another part file completely.

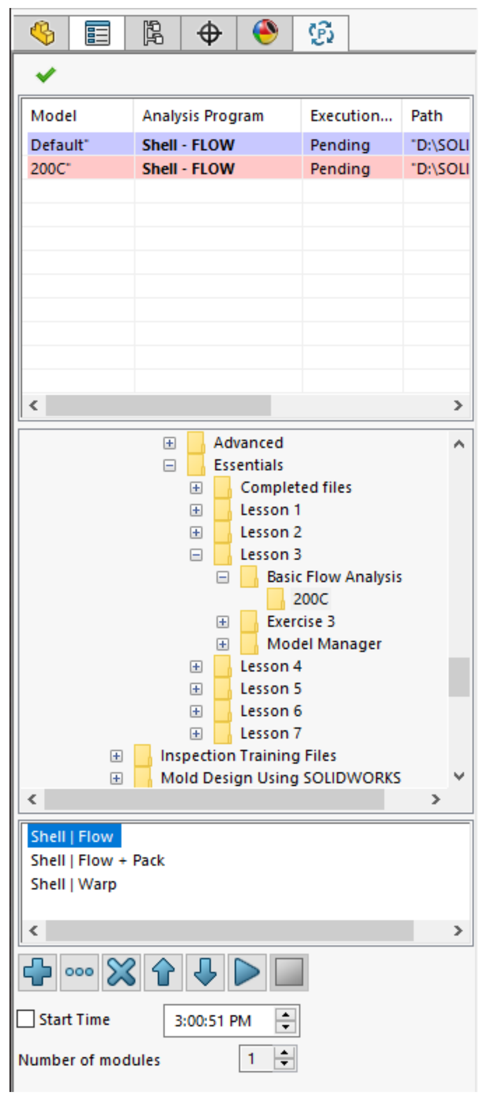
Plastics File Management



For each simulation that is created, there are data folders created to store the analysis data. The folder structure has a top-level folder with the same name as the part file. Under the top-level folder is another set of folders that contain the simulation files. These folders are named after the configurations the studies are attached to.

The **Duplicate Study** command is used to create the configurations and folders while information from existing folders is transferred through the **Copy Settings** command.

Batch Manger



The **Batch Manager** allows multiple studies to be run immediately or at a later time.

Batch Controls

The control panel at the bottom of the dialog can be used to add, change, or delete the batch analysis jobs.

Add Analysis: Add a batch analysis.

Change Analysis: Used to change the type of analysis, such as **Shell-FLOW** to **Shell-PACK**.

Delete Analysis: Delete a batch analysis from the list.

Move Down: Move a batch analysis down the list.

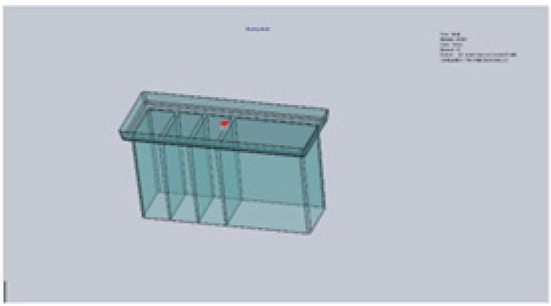
Move Up: Move a batch analysis up the list.

Start: Start the batch analysis.

Stop: Stop the batch analysis.

Summary and Report

The Summary and Report tool is used to automatically gather preprocessing and post-processing information into a PowerPoint or Word document. This document can include the plastic used, gate locations, and results graphs. The report is customizable and can be edited after creation.



Injection Molding of Short Shot_c

Date:
Designer:
Analysis: Shell/ Solid Based Surface Model

Table of Contents

Introduction

Process Parameters

Comments:

Fill Settings

Filling Time	0.5 sec
Main Material Melt Temperature	202 °C
Mold Wall Temperature	48 °C
Injection Pressure Limit	100 MPa
Flow Rate Limit	194 cc/s
Flow/Pack Switch Point (% Filled Volume)	100 %
Pressure Holding Time	1.54 sec
Total Time in Pack Stage	2.11 sec
Auto Filling Time (1: Yes, 0: No)	0
Auto Packing Time (1: Yes, 0: No)	1
Venting Analysis (1: Yes, 0: No)	0
Cavity Initial Air Pressure	0.101 MPa
Cavity Initial Air Temperature	30 °C
Temperature Criteria for Short Shots (1: Yes, 0: No)	1
Temperature Criteria for Short Shots	122 °C
Clamp Force Limit	100 Tonne

LESSON 4: INJECTION LOCATIONS AND SINK MARKS

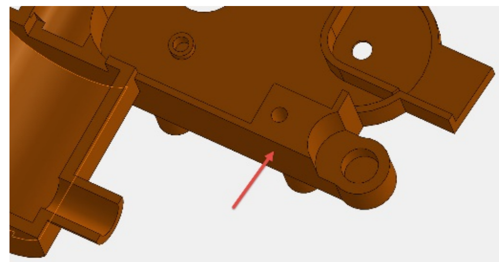
Positioning an Injection Location

Single or multiple injection locations can be created within the cavity.

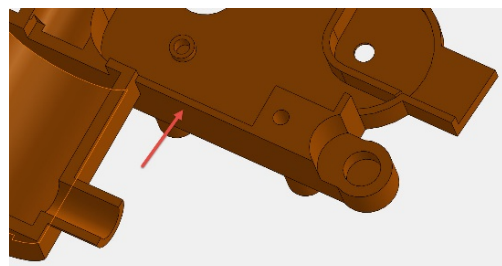
Injection Location Rules

- Avoid placing the injection location on highly visible areas of the part. This may cause a visible defect on that surface.

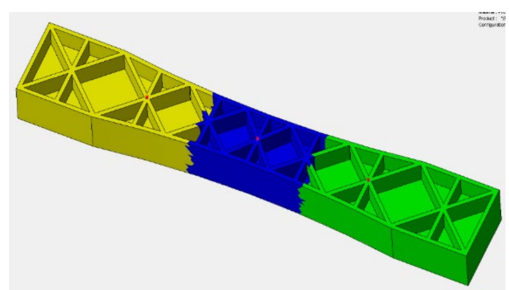
- When possible, always inject into a thick section of the part geometry.



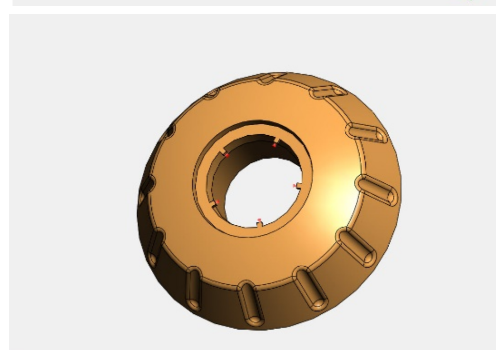
- Avoid injection into these areas of the part. The pressure drop through a thin section will require higher injection pressures to fill and pack out the part. The gate may completely freeze before the part is packed out, resulting in larger sink marks.



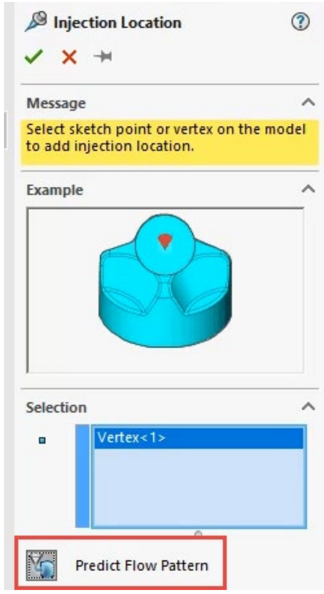
- If possible, place the injection location towards the middle of the part geometry to minimize flow lengths within the mold cavity.
- For large parts, use several injection locations spaced so that each injection location fills equal volumes with uniform pressure distributions.



- Place 3 or 5 injection locations positioned radially outward from the center in revolved parts. This results in uniform packing and minimizes oval deformation as the part cools.



Predict Flow Pattern



The **Predict Flow Pattern** command displays a prediction of the fill pattern without running a complete simulation. The geometry must be meshed first before this command will function.

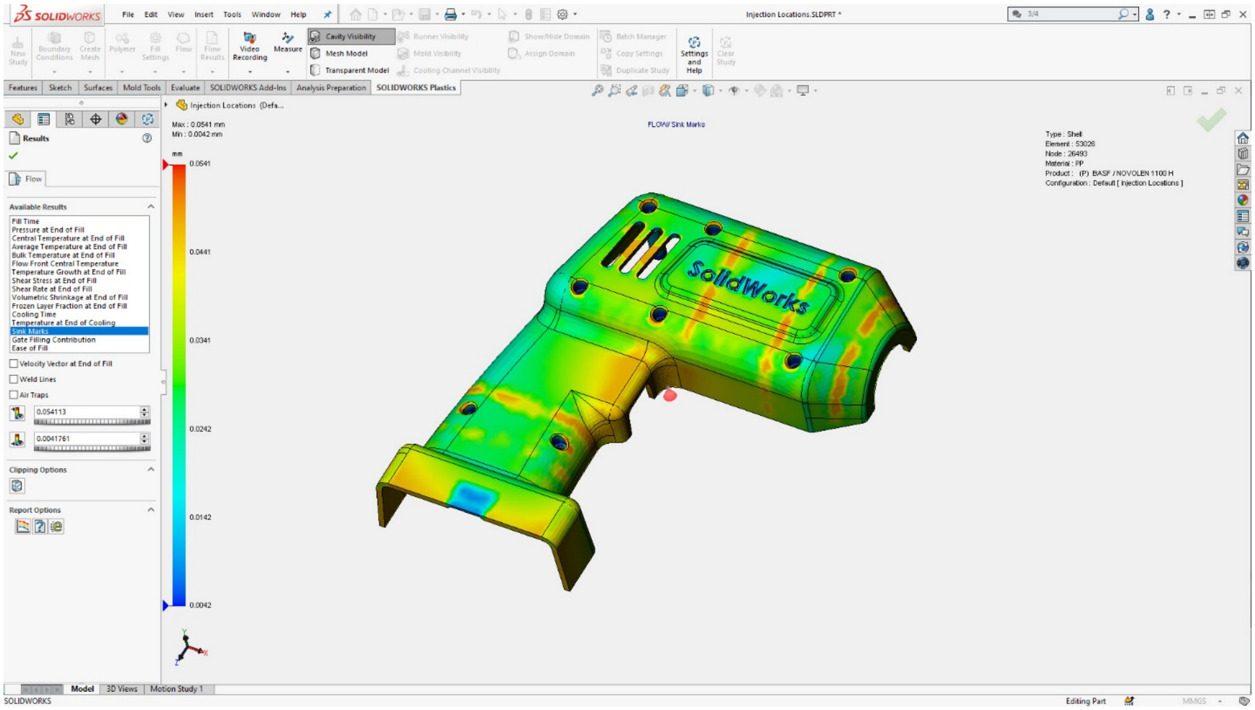
The **Predict Flow Pattern** command is found in the **Injection Location** boundary condition.

This command is helpful when determining the initial injection location on a part.

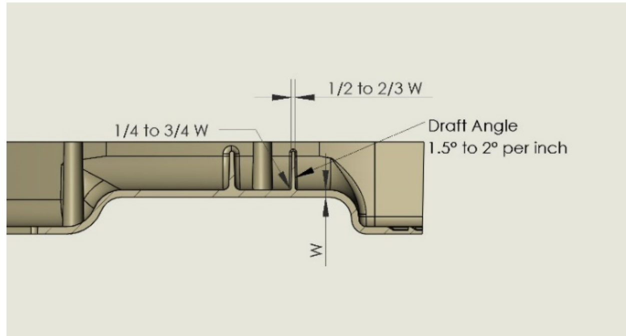
Predict and Minimize Sink Marks

The severity of sink marks can be identified through analysis and reduced by following some general design rules.

Sink marks occur during the cooling process and appear as depressions on the surface of the molded part. The **Sink Mark** result plot in the flow analysis will predict areas where they would occur.



Minimizing Sink Marks in Ribs

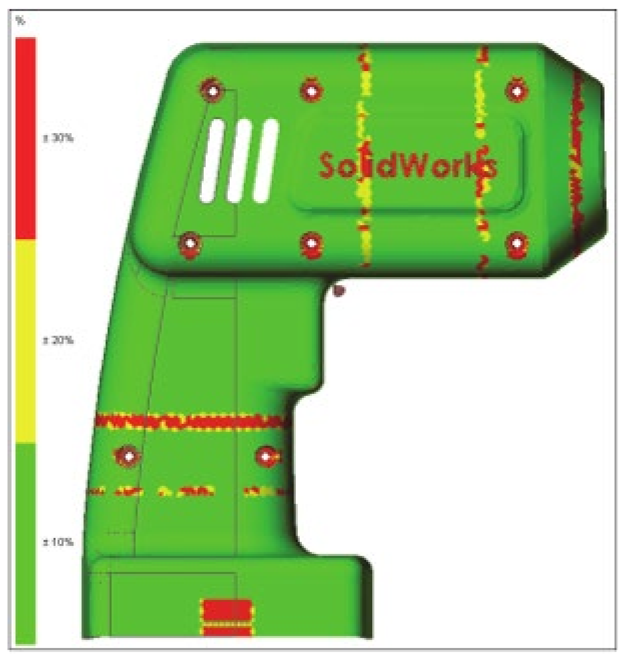


Using design rules for ribs and fillet will minimize possible sink marks in plastic parts.

- **Rib Thickness:** The base thickness of the rib should fall in the range of **1/2** to **2/3** of the attached parent wall thickness.
- **Rib Fillet:** The rib fillet radius should fall in the range of **1/4** to **3/4** of the part thickness.
- **Draft Angle:** The draft angle of ribs should fall in the range of **1.5°** to **2.0 °** per inch of rib length, to facilitate part ejection.

Nominal Wall Thickness Advisor

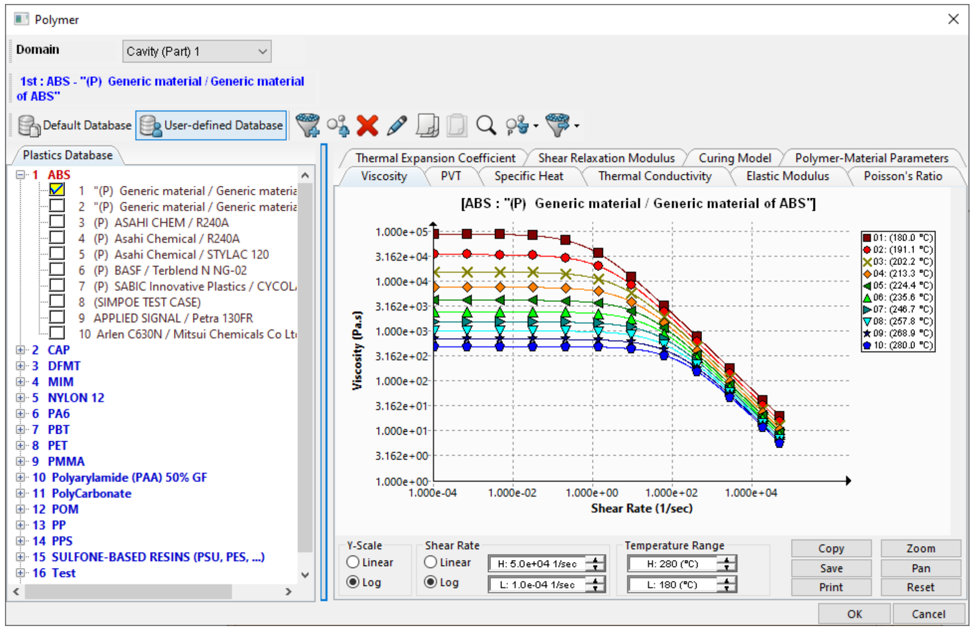
The primary design rule for an injection molded plastic part is to maintain a uniform wall thickness. The **Nominal Wall Thickness Advisor** calculates the overall nominal wall thickness in the model and the deviation from this nominal value.



LESSON 5: MATERIALS

Material Database

The SOLIDWORKS material default database contains thousands of materials including plastic resins, metals, and fluids. If the material being used is not in the default database, a custom material can be created in the User-defined Database.



The first step to creating a custom material would be to obtain the proper data. Sometimes the material manufacturers will supply the data upon request. Another method would be to send the material samples to a lab and have them tested for the data.

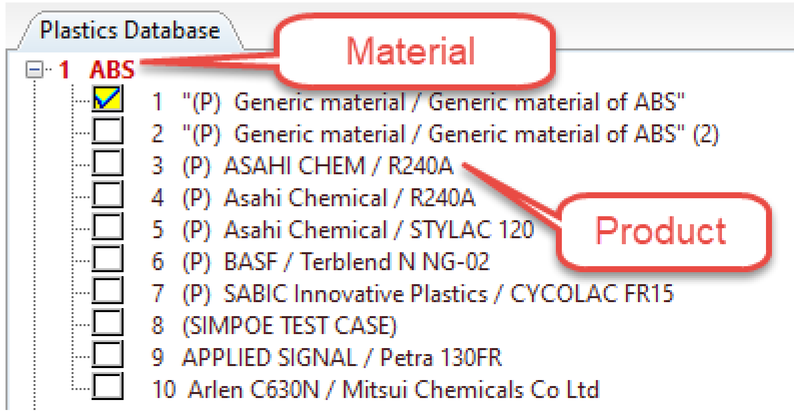
User-defined Database

The **User-defined Database** is used to create and store custom materials. Custom materials can be created in one of two ways.

The first and easiest way is to copy and paste an existing material from the default database into the User-defined Database then modify the properties from there.

The second is to create a new material from scratch.

The term **Material** is used to define a grouping of resins, metals, or cooling fluids. The term **Product** is used to define a particular resin, metal, or cooling fluid.

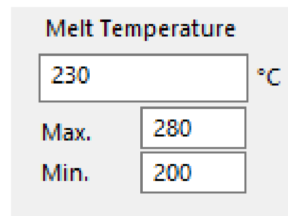


Products can also be imported into the database using an Excel document or a bin file.

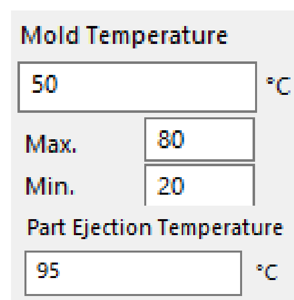
Resin Properties

The following resin properties are required to run a simulation.

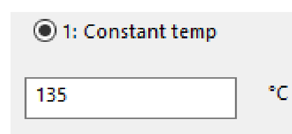
- **Melt Temperature** - Temperature the resin should be heated to while inside the barrel.



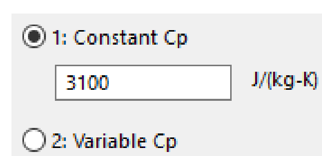
- **Mold Temperature** - Temperature the mold maintains through the injection molding process.
- **Part Ejection Temperature** - Temperature the part must reach before it is ejected.



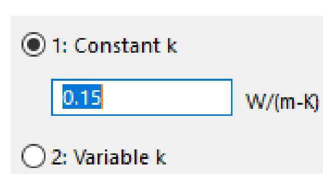
- **Glass Transition Temperature** - Temperature at which a resin is considered solid.



- **Specific Heat** - Measure of how much thermal energy it takes to heat one Kg of material one degree Kelvin. Variable Cp would give more accurate results.



- **Thermal Conductivity** - Characterizes how thermal energy can transfer through a material. Variable Cp would give more accurate results.



- **Viscosity** - Measure of how easily fluid can flow. High viscosity resists flow, low viscosity flows easier.

◉ 7: 7 parameters Modified Cross model

D1	4.444890e+14	Pa-s	A2bar	51.6	K
D2	263.15	K	tau	26260	Pa
D3	0	K/Pa	n	0.272	
A1	32.7				

- **PVT Data** - **P**ressure, **V**olume, and **T**emperature determines how much the plastic will shrink as it cools during the injection molding process.

◉ 13: Modified Tait Equation (b1l-b4l, b1s-b4s, b5-b9)

b1l	1.202300e-03	m3/Kg	b4s	0.004745	1/K
b2l	9.200000e-07	m3/(Kg-K)	b5	423	K
b3l	8.800000e+07	Pa	b6	1.200000e-07	K/Pa
b4l	0.00482	1/K	b7	0.000096	m3/Kg
b1s	0.001107	m3/Kg	b8	0.15	1/K
b2s	5.100000e-07	m3/(Kg-K)	b9	2.060000e-08	1/Pa
b3s	1.630000e+08	Pa			

- **Thermal Expansion Coefficient** - How a material expands and contracts with regards to changes in temperature.

○ 1: Constant Coefficient

◉ 2: Constant modulus (Parallel, Normal)

Parallel	9.050000e-05	1/deg C
Normal	9.050000e-05	1/deg C

- **Elastic Modulus (Young's Modulus)** - Describes the stiffness of the material. Relationship between stress and strain.

○ 1: Constant modulus

◉ 2: Constant modulus (Parallel, Normal)

Parallel	1350	MPa
Normal	1350	MPa

- **Poisson's Ratio** - Describes how much a material deforms normal to a load.

○ 1: Constant modulus

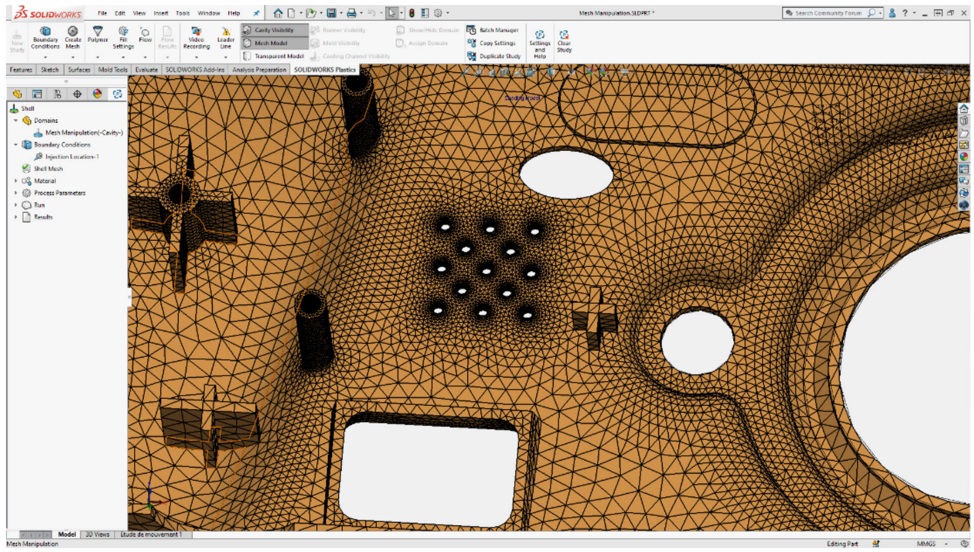
◉ 2: Constant modulus (Parallel, Normal)

Parallel	0.4
Normal	0.4

LESSON 6: MESH MANIPULATION

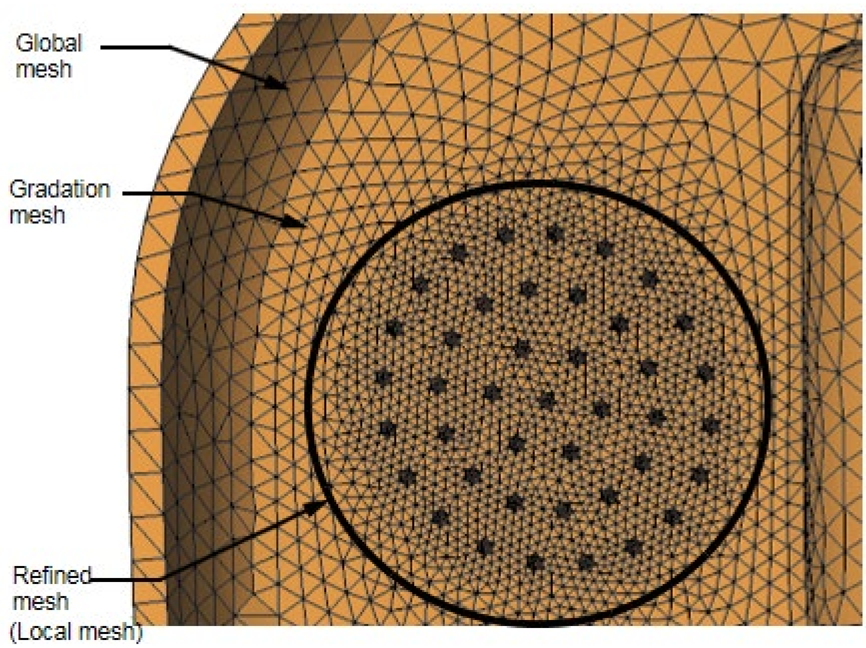
Mesh Manipulation

Mesh is a mathematical representation of the original geometry which is used in the simulation process. Creating an accurate mesh may sometimes require manual editing.



Mesh Refinement

Mesh density does not have to be uniform throughout the model. A finer mesh in more critical areas, as well as locations with greater detail, should be used.



Global Element Size – Overall size of the mesh throughout.

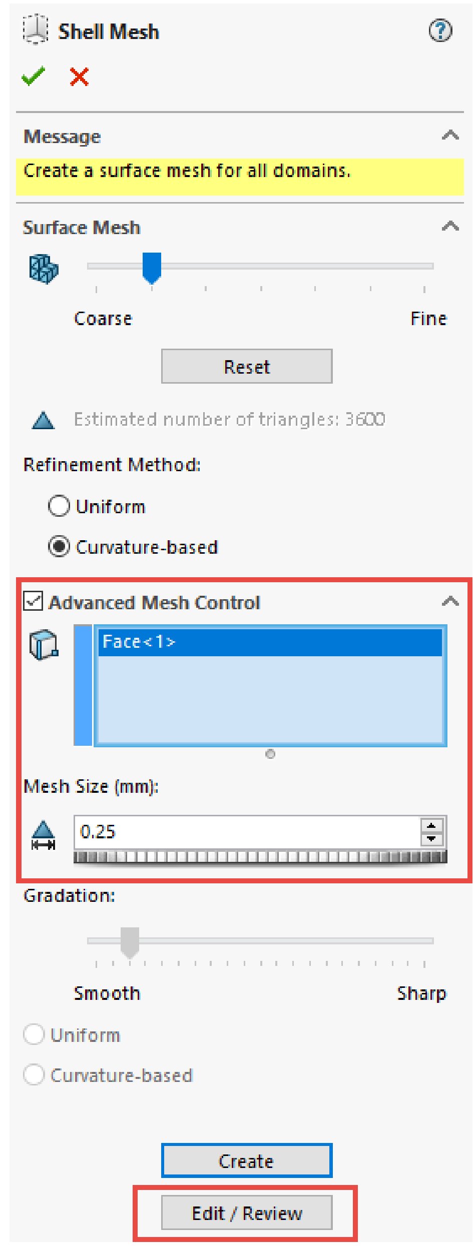
Local Element Size – Size of the mesh in specific areas.

Gradation mesh – Transition between Global and Local mesh.

Local mesh refinement is applied using the **Advanced Mesh Control.**

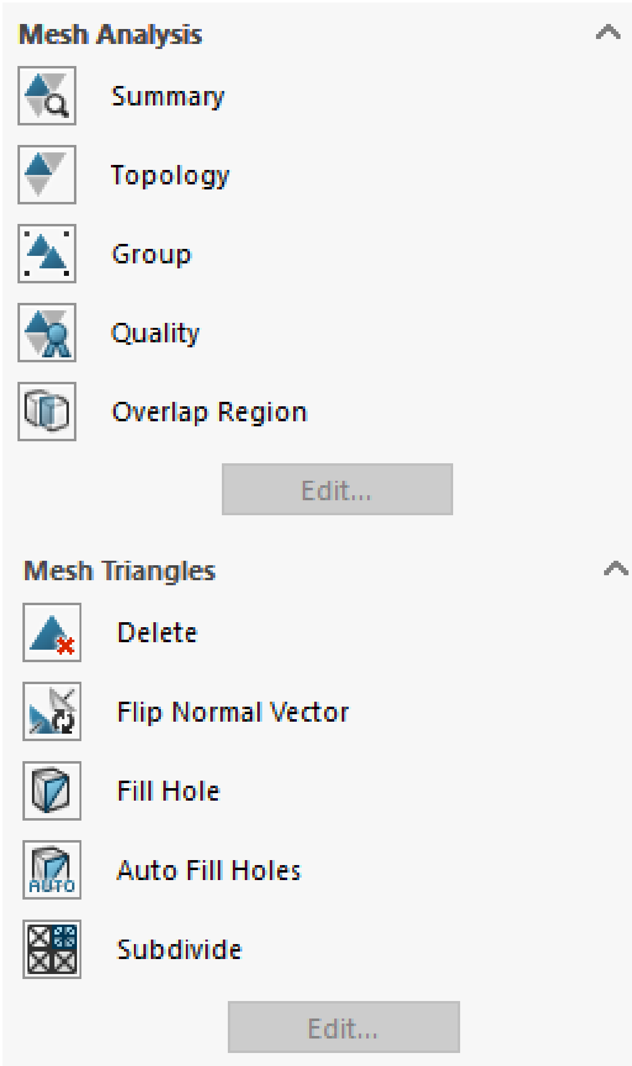
Local refinement can be applied to faces, edges, or vertices.

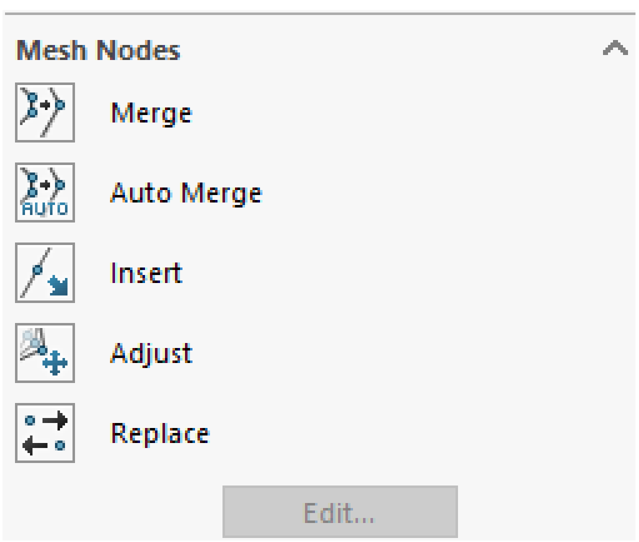
Each face, edge, or vertices selected in the selection pane can have a different mesh size applied to it.



Edit / Review

The Edit / Review dialog contains information and editing options that can be used to make precise changes to the mesh.





Summary identifies element issues.

Topology identifies missing faces which cause waterproof errors.

Group identifies which elements are in each domain.

Quality identifies elements with large aspect ratios.

Overlap Region identifies contacting faces between two bodies where the mesh is incompatible.

Delete is used to delete triangle elements.

Flip Normal Vector is used to reverse the direction of a group of triangles in a mesh.

Fill Hole is used to repair elements that are not waterproof.

Auto Fill Holes is used to fill all the holes detected in the mesh automatically.

Subdivide is used to divide selected elements into smaller elements.

Merge is used to merge nodes that are separated by a distance.

Auto Merge is used to find nodes which are separated by a distance and merge them.

Insert is used to insert a new node at a selected location.

Adjust is used to adjust the location of a specific node.

Replace is used to delete and merge selected nodes precisely.

Solid Mesh

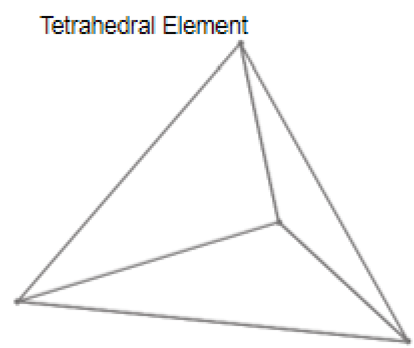
The **Solid Mesh** is made from 3D elements and explicitly represents the inside geometry of a part. In SOLIDWORKS Plastics, a 3D solid mesh is created by referencing a typical 2D shell mesh.

A **Shell Mesh** works by interpolating the flow profile between the shell walls. This can be a fair assumption especially in the early stages of analysis and for thin-walled parts.

A **Solid Mesh**, on the other hand, can calculate the flow profile through the thickness of a cavity without interpolating the results. Therefore, a solid mesh should almost always be used in the later stages of the analysis process.

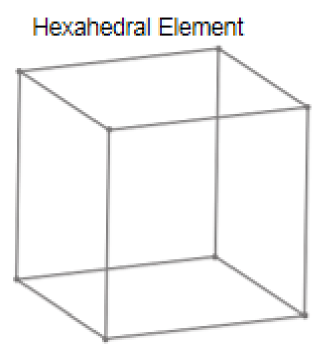
Tetrahedral Solid Mesh

Tetrahedral elements are the most used element type because they can represent complicated shapes much more reliably than a hexahedral element. Despite this advantage, tetrahedral elements are more prone to numerical diffusion and are generally slower to solve than hexahedral elements if the number of elements is the same.



Hexahedral Solid Mesh

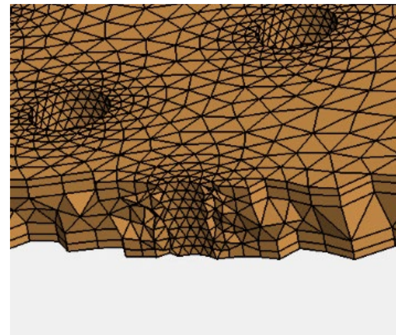
Hexahedral elements are superior in many ways to tetrahedral elements. Hexahedral elements solve faster and are less prone to numerical diffusion. However, when working with complicated geometry, hexahedrals are much more difficult to use and often require many more elements than typical tetrahedral mesh.



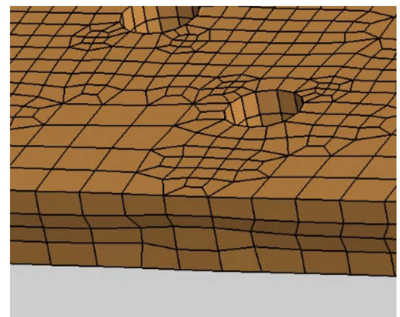
Solid Mesh Options

There are three solid mesh options available when creating a solid mesh: tetrahedral hybrid, hexahedral, and automatic.

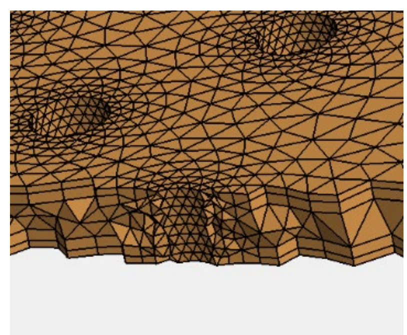
Tetrahedral hybrid mesh uses a combination of tetrahedral elements on the inside in extruded triangular elements, often called boundary layers, to represent the part geometry. This makes the hybrid approach great for thin parts because more elements are forced across the thickness.

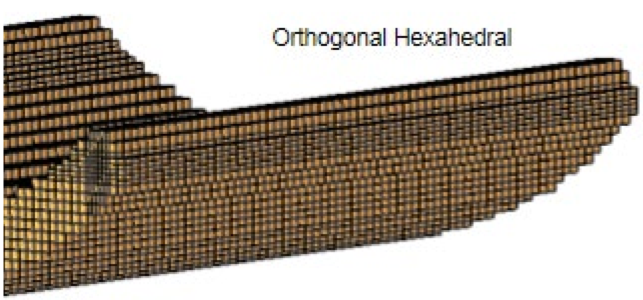


Hexahedral mesh elements can curve with the part geometry. Some bodies cannot be matched exclusively with hexahedral elements. In regions where this occurs, tetrahedral elements are used. The software uses a surface mesh with triangular elements as the initial shape to form the hexahedral mesh. This extra step introduces a potential source of error in terms of approximation to the original geometry.



Automatic mesh type is the fastest approach and does not require any user input. First, the software creates a curvature-based surface mesh with the size of the elements determined by part dimensions. Next, the software creates a solid mesh from this shell mesh using tetrahedral hybrid elements. If the tetrahedral hybrid approach fails, the software will attempt to create a mesh using orthogonal hexahedral elements. The elements in an orthogonal hexahedral mesh do not curve.

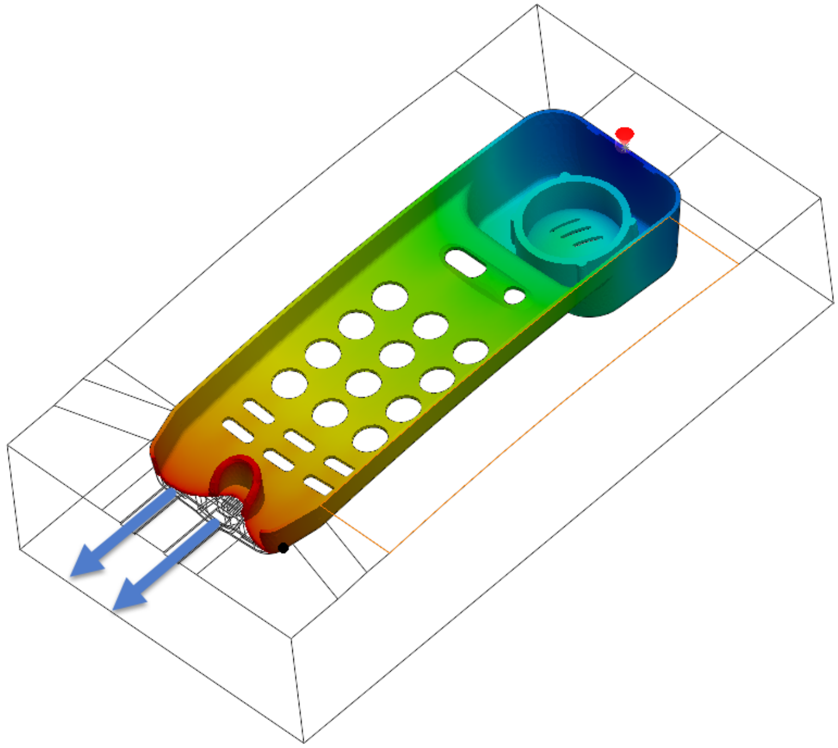




LESSON 7: DETECTING AIR TRAPS

Air Traps

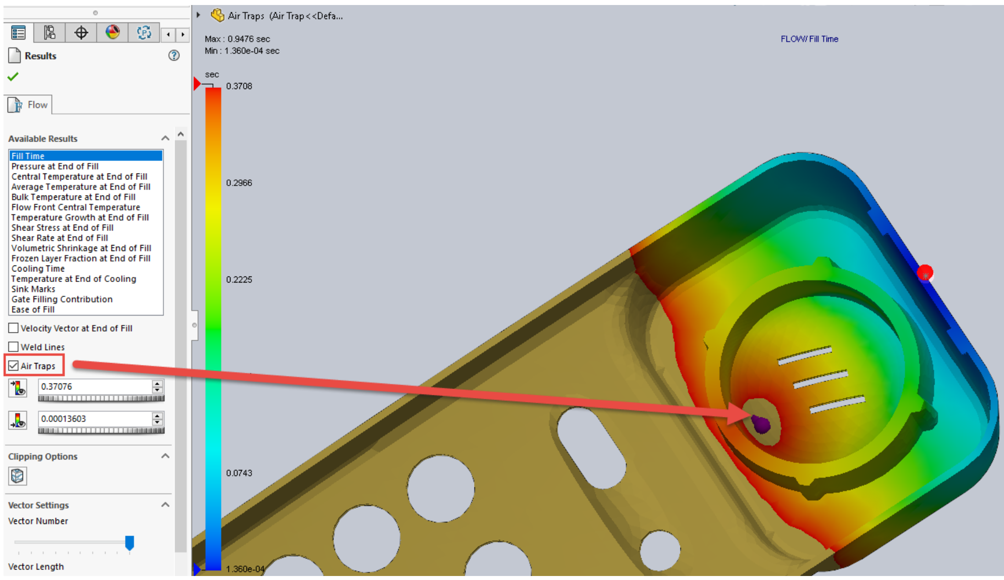
As the cavity fills with melted plastic, the air inside the cavity is displaced. Ideally, this air is vented through machined channels at the parting line or ejector pins. However, when air is pushed to an area of the cavity where it cannot easily be vented, it is referred to as an air trap.



Air traps usually cause molded part defects that range from bubbles in the plastic to burn marks caused by the combustion of the trapped air.

Detecting Air Traps

The Air Trap option in the Flow Results will show locations of air traps superimposed over the Fill Time plot or other results. They are caused when converging plastic melt flow fronts create an air pocket.



Race-Tracking

When melted plastic is injected into a mold, it will follow the path of least resistance, usually the thickest areas of the cavity. The phenomenon known as race-tracking occurs when a uniform melt flow front encounters wall sections of varying thicknesses and flows into the thickest area first.