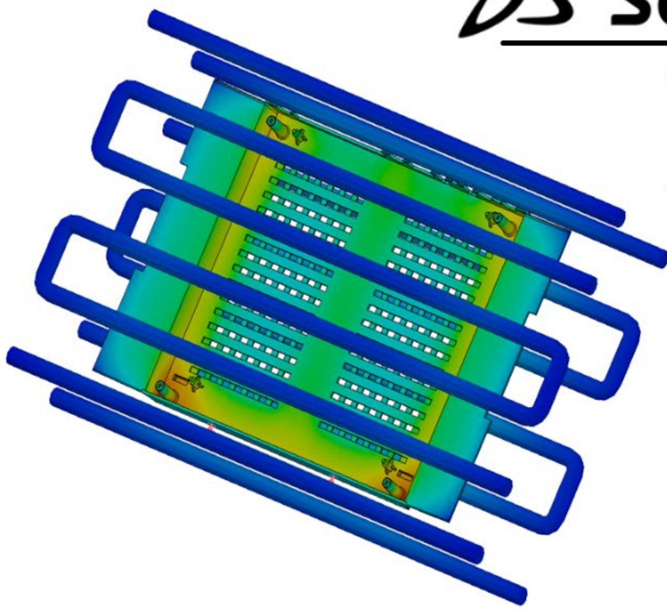
SOLIDWORKS

# PLASTICS ADVANCED



## TAKE-AWAY

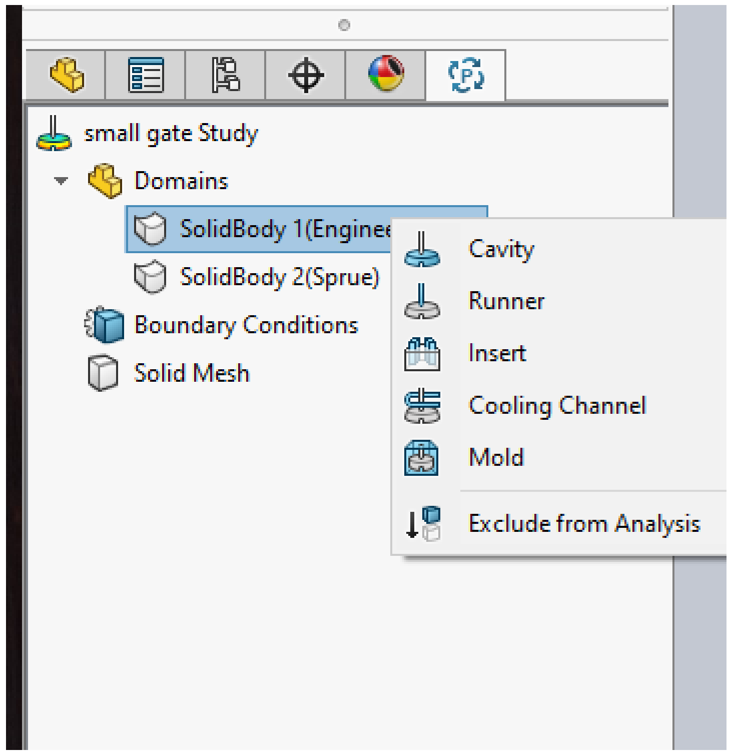
## Lessons



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

# LESSON 1: GATE BLUSH AND VENTING ANALYSIS

## Runner Elements




In the injection molding process of a standard two-plate mold, resin is injected into a runner system before it is injected into the cavity. The runner system and cavity are ejected from the mold at the same time and the runner system is cut off later.

In SOLIDWORKS Plastics, the term **domain** is used to designate different components in the injection molding process. Domains can exist for runners, inserts, cooling channels, and the mold itself. The purpose of identifying and simulating multiple domains is to create a more realistic simulation by specifying the unique properties of each domain.

## Gate Blush

After ejection, the runner system is cut from the cavity at the gate. So, it makes sense that a small gate produces a small visible mark on the part, however, this is not always the case. Resin is composed of long carbon molecules that can be damaged if the shear rate between the molecules reaches a critical value. These high shear rates are commonly experienced at gate locations due to the amount of plastic flowing through the small opening of the gate. This is known as **gate blush** and it is characterized by a visible mark at the gate location.

There is a maximum recommended shear stress and shear rate for every resin. If the polymer exceeds the Max shear stress or the Max shear rate it will be damaged.

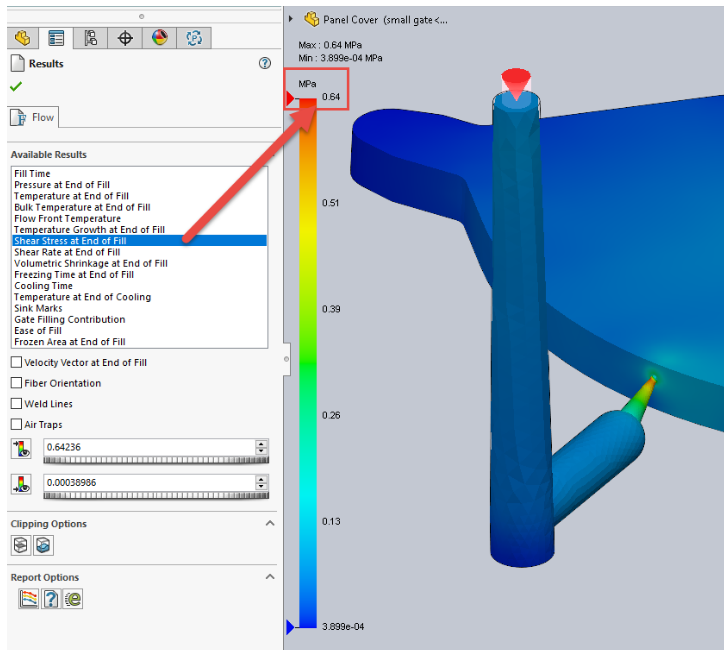
| | |
|---|---|
| Max. Shear Rate | 60600 1/s |
| Max. Shear Stress | 0.5 MPa |

There are several methods for reducing gate blush:

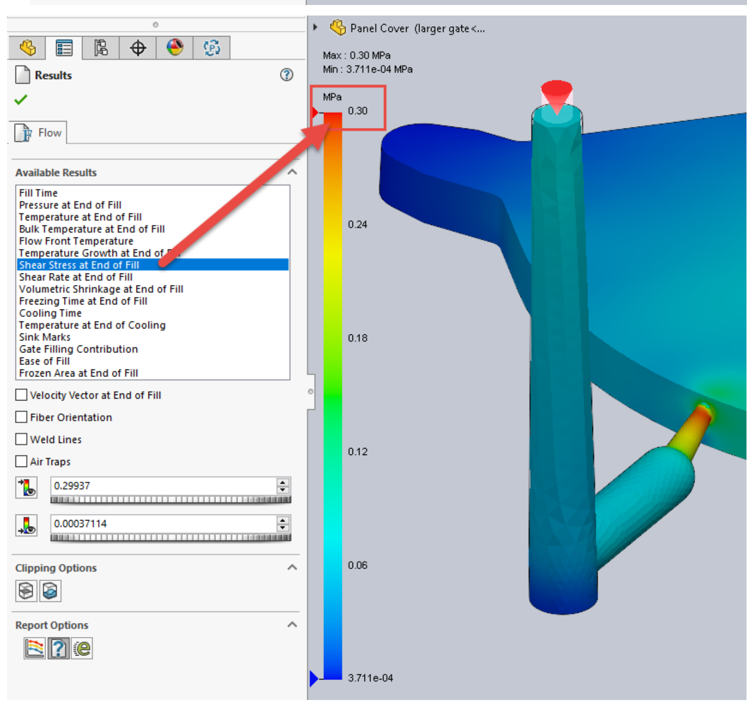
- Changing the fill time
- Change the melt temperature
- Change the mold temperature
- Change the fill rate profile
- Increase the size of the gate. (**The easiest solution**)

## Shear Stress at End of Fill

The results from Shear stress at end of fill will show the maximum shear stress for the filling analysis. The result in this plot is .64 MPa which is above the 0.5 MPa maximum recommended by the material manufacturer.




By enlarging the gate diameter and rerunning the analysis, the maximum shear stress is reduced to 0.30Mpa which is within the recommendations by the manufacturer. This part will be less likely to have gate blush.



## Venting Analysis

Another defect that can happen in the injection molding process is burn marks. Burn marks occur when trapped air in the mold becomes superheated under pressure and the temperature of the hot resin and combusts. This is also known as the dieseling effect and can be prevented with proper mold venting. In SOLIDWORKS Plastics Standard, we can see where air traps may occur. In SOLIDWORKS Plastics Professional, we can run a venting analysis to help determine mold venting locations and quantity.

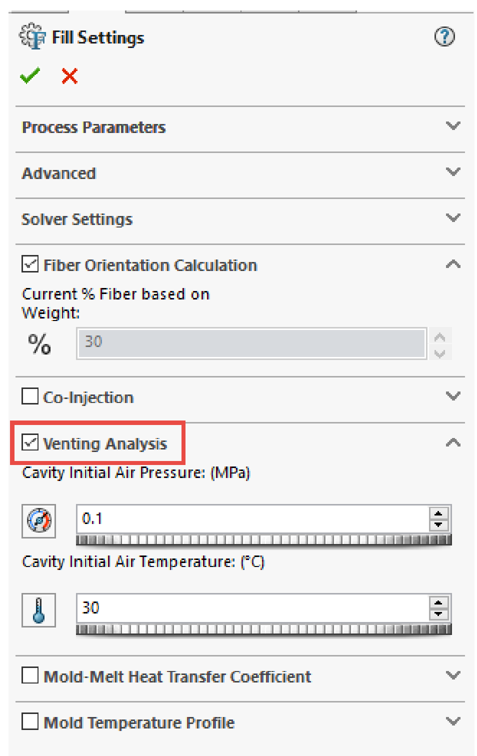
A **Venting analysis** is a multi-step process.

## Step One

First, run a fill analysis using the venting analysis option selected and no vent boundaries defined.

The **Venting Analysis** option is found in the **Fill Settings**.

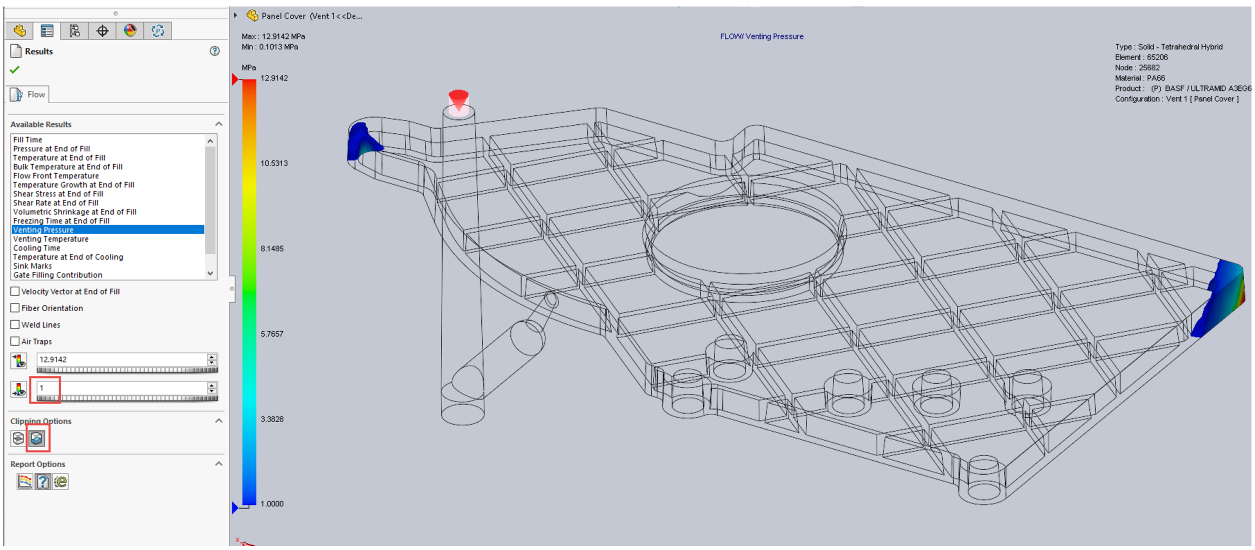
Activating the vent analysis without defined vent locations will reveal the major areas that require venting.



## Vent Analysis Results

The **Venting Pressure** results will show areas where the venting pressures are much higher than the rest of the model. When analyzing venting pressure, we want the Max pressure value to be less than one megapascal.

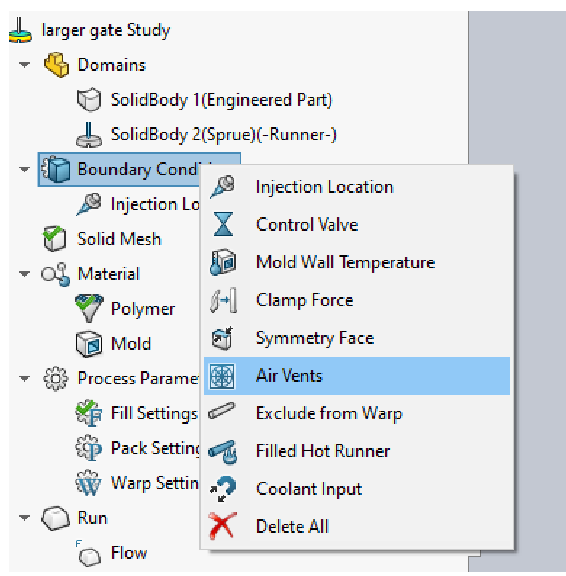
Set the minimum result to one and select **Isosurface clipping** mode to show only the areas with pressures above 1 MPa.




## Step 2

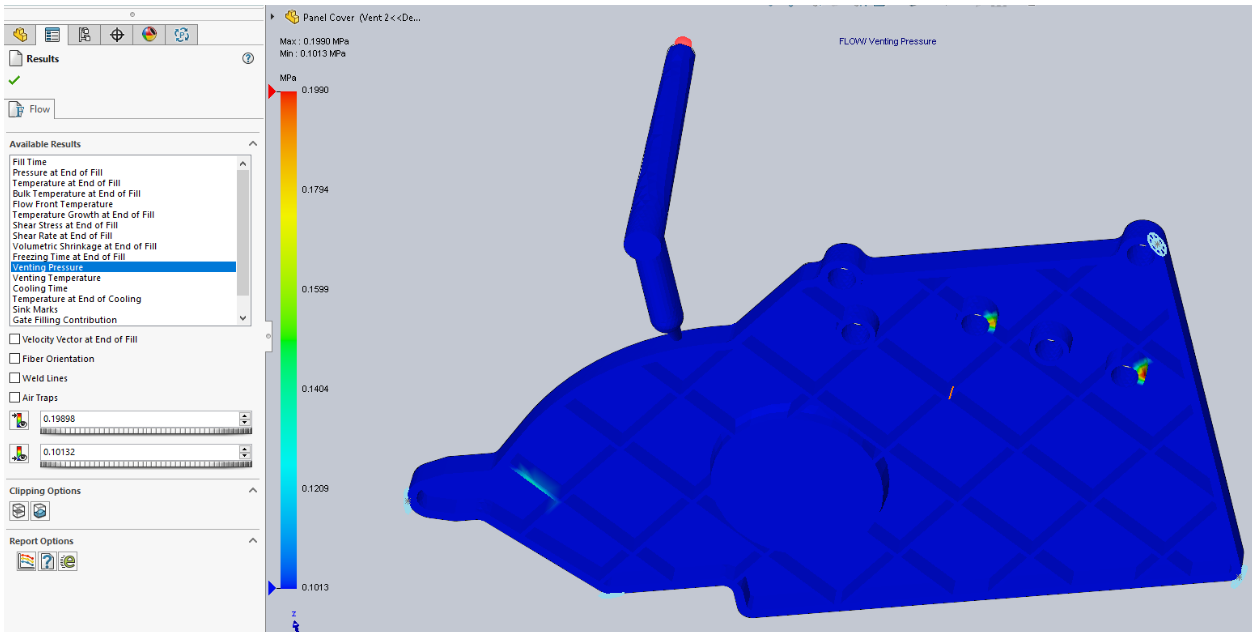
Second, run a fill analysis using the venting analysis option with the vent locations defined.

To specify a vent location, choose existing vertices or sketch points on the model.



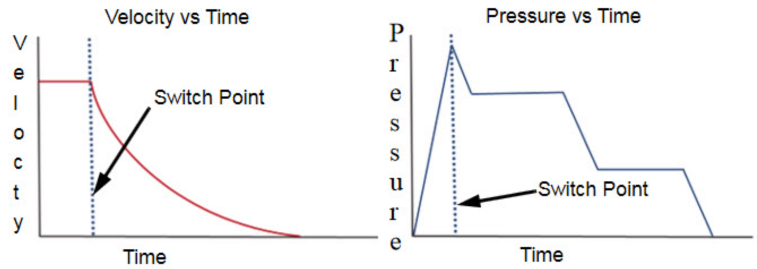
## Vent Analysis Results

The maximum venting pressure is .19898 megapascal well within the acceptable range.



# LESSON 2: PACKING AND COOLING TIMES

In the injection molding process, the transition from the fill stage to the pack stage is characterized by a velocity-controlled fill to a pressure control pack.




Once the part has finished packing, the pressure is reduced, and the part can cool in the mold before ejection.

## Fill to Pack Transition

The fill to pack transition generally occurs when the cavity is filled 95% to 99% before the pack stage begins. The reason why the cavity is not filled to 100% of its full capacity is to prevent flash at the parting lines.

## Pack to Cool Transition

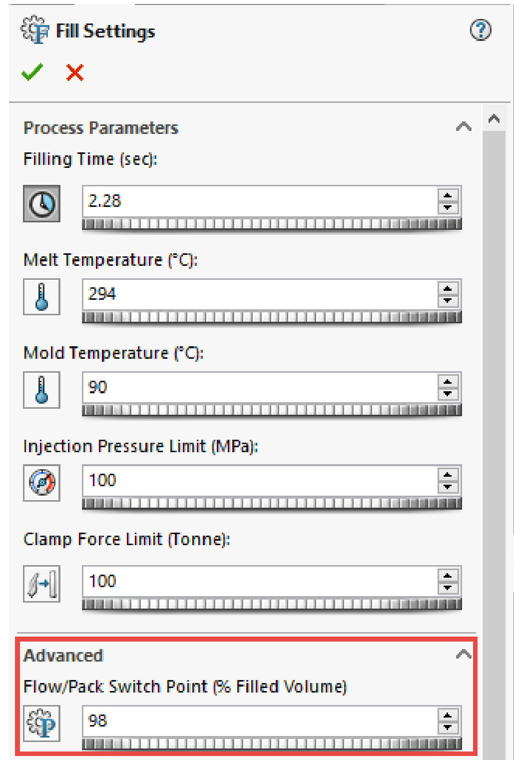
During the pack stage, the plastic in the mold cools and shrinks. To reduce these effects, additional plastic is forced through the mold under a controlled pressure. This causes the cooling, shrinking plastic to be forced against the mold walls. The pack stage occurs as long as additional plastic can flow through the gate. Once additional plastic ceases to flow, the pack stage ends, and the pure cooling stage begins.

## The Cool to Ejection Transition

During the pure cool stage, the pressure is reduced, and the part can cool until it reaches ejection temperature. Ejection temperature should be reached throughout the part not just at the surface. This is because as the part is ejected, pins push the part out of the mold. If the part is still soft in the center during ejection, the pins can perforate or deform the surface of the part.

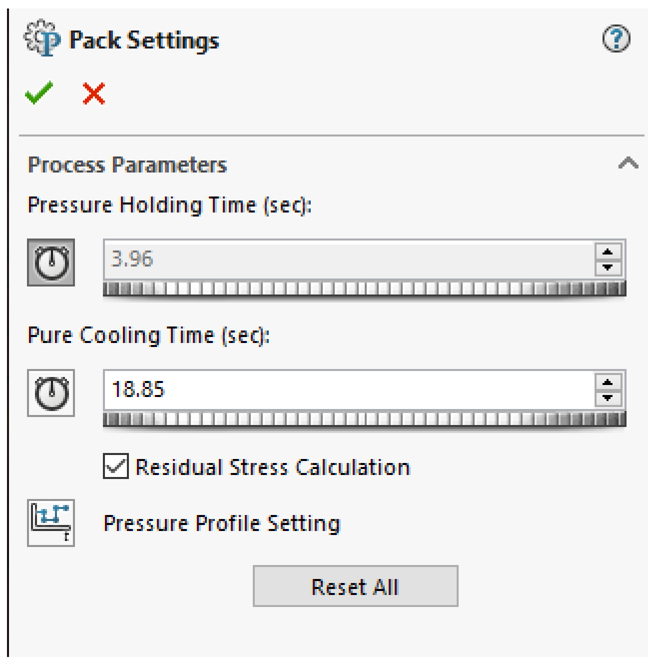
## Adjusting Fill to Pack Transition

The **Flow/Pack Switch Point** setting is found in **Fill Settings** under the **Advanced** section.




## Pack Settings

The molten plastic starts to cool and shrink as soon as it touches the mold walls. To counteract these effects, additional plastic is forced into the mold under a controlled pressure. The amount of pressure applied during the pack stage is often a fraction of the pressure applied by the machine during the fill stage and is staggered down often gradually until the end of the pack stage.

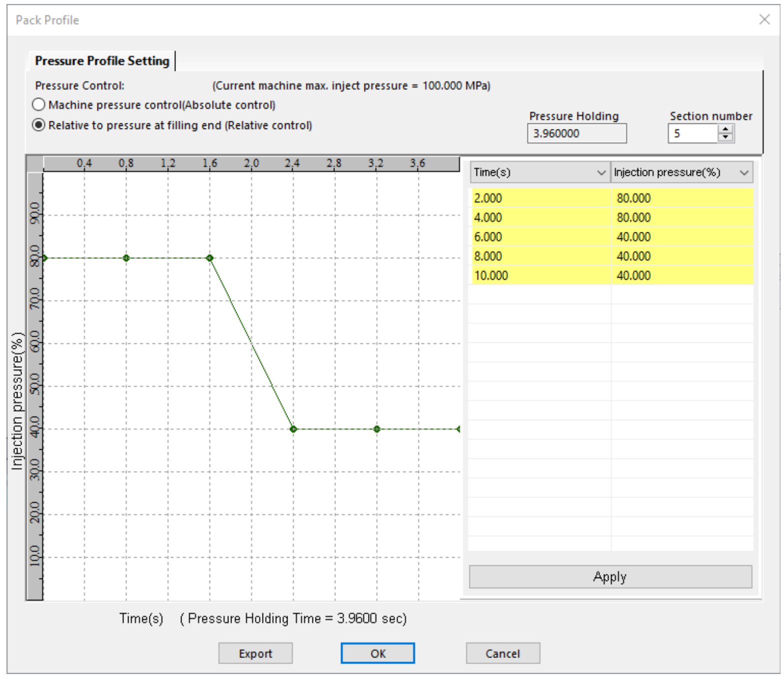



The **Pack Settings** controls the pack time and pure cooling time, as well as the pressure profile settings.

Setting the Pressure Holding Time to a high value such as 40 seconds will allow the user to see when the part will quit gaining mass.

## Pressure Profile Settings

The pressure profile settings determine the packing pressure throughout the pack stage. By default, the pressure profile is set to 80% of the maximum pressure achieved during the fill stage and is reduced to 40% halfway through the pack stage. These values, however, can be changed.

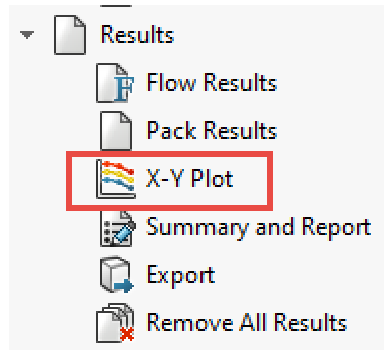



## Pack Results

The **Pack Results** include results for understanding the packing process (**End of Packing**) as well as results for understanding the cooling process (**Post-filling End**):

- Pressure at end of packing
- Temperature at end of packing
- Bulk temperature at end of packing
- Shear stress at end of packing
- Shear rate at end of packing
- Volumetric shrinkage at end of packing
- Temperature at post filling end
- Freezing time at post filling end
- Residual stress at post filling end
- Birefringence at end of packing
- Frozen area at post filling end

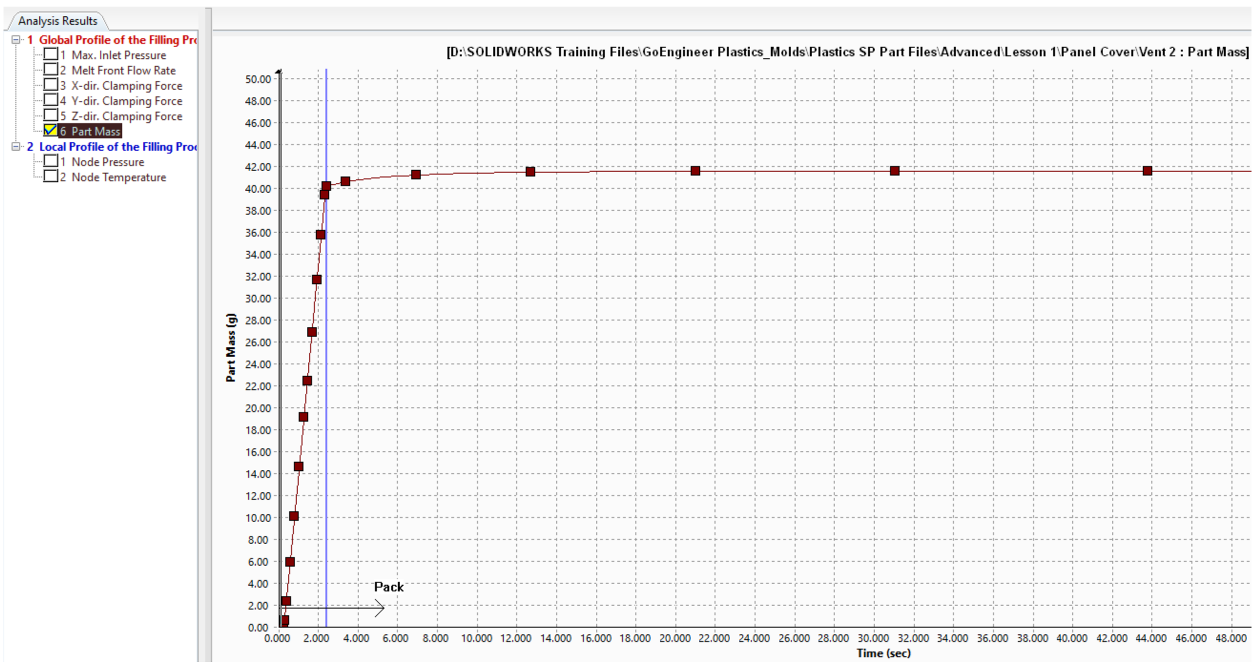
## X-Y Plots




With **X-Y Plots**, parameters can be plotted against time. Viewing the results this way can provide insight into how the system changes throughout the molding process.

The **X-Y Plots** available are:

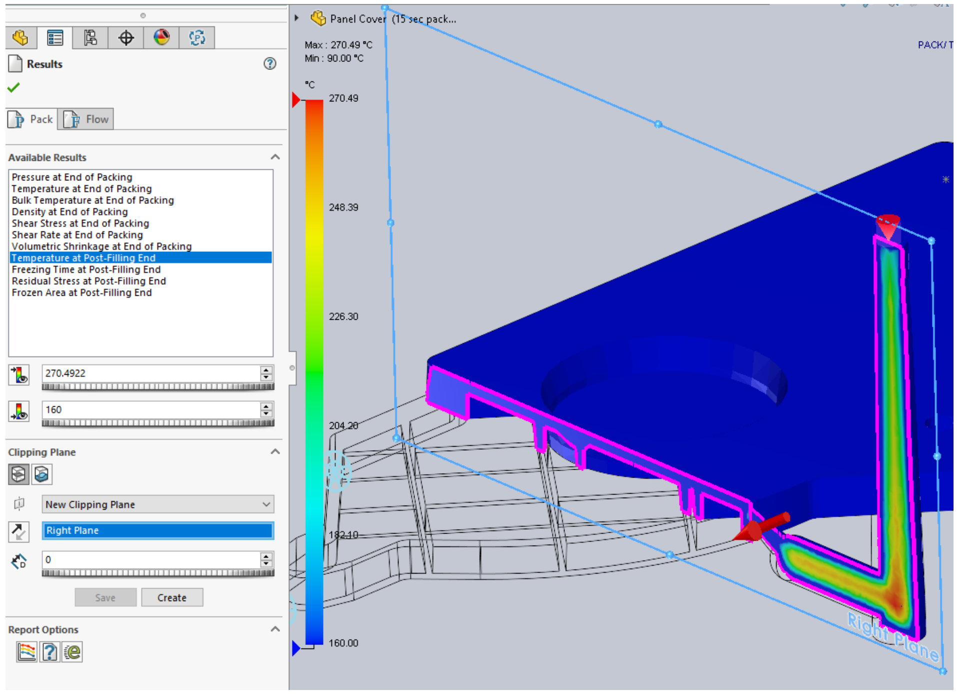
- Max. Inlet Pressure
- Inlet Flow Rate
- Clamp Force
- Part Mass
- Nodal Pressure
- Nodal Temperature



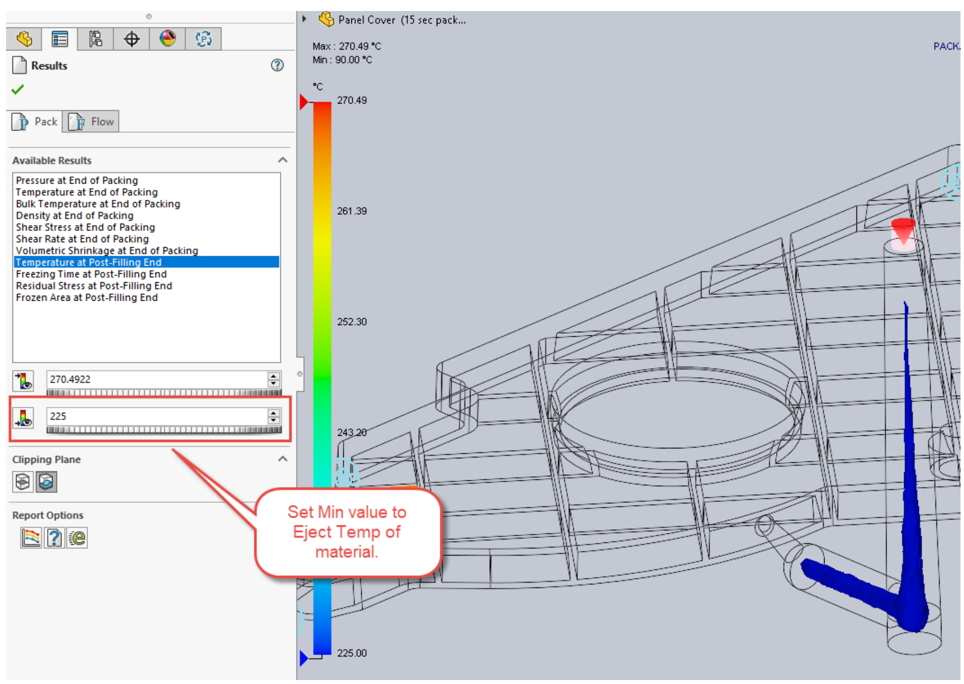

This plot shows that the mass of the part eventually converges to a constant value and appears to be completely leveled off in about 45 seconds. However, from 15 seconds to 45 seconds, the mass of the part only increases by a little more than a gram. Adding 30 seconds to the cycle time may not be worth the increase in part quality.

## Clipping Plane / Isosurface Mode

The **Clip Plane Mode** command is used to view the inside of a result plot across a plane.



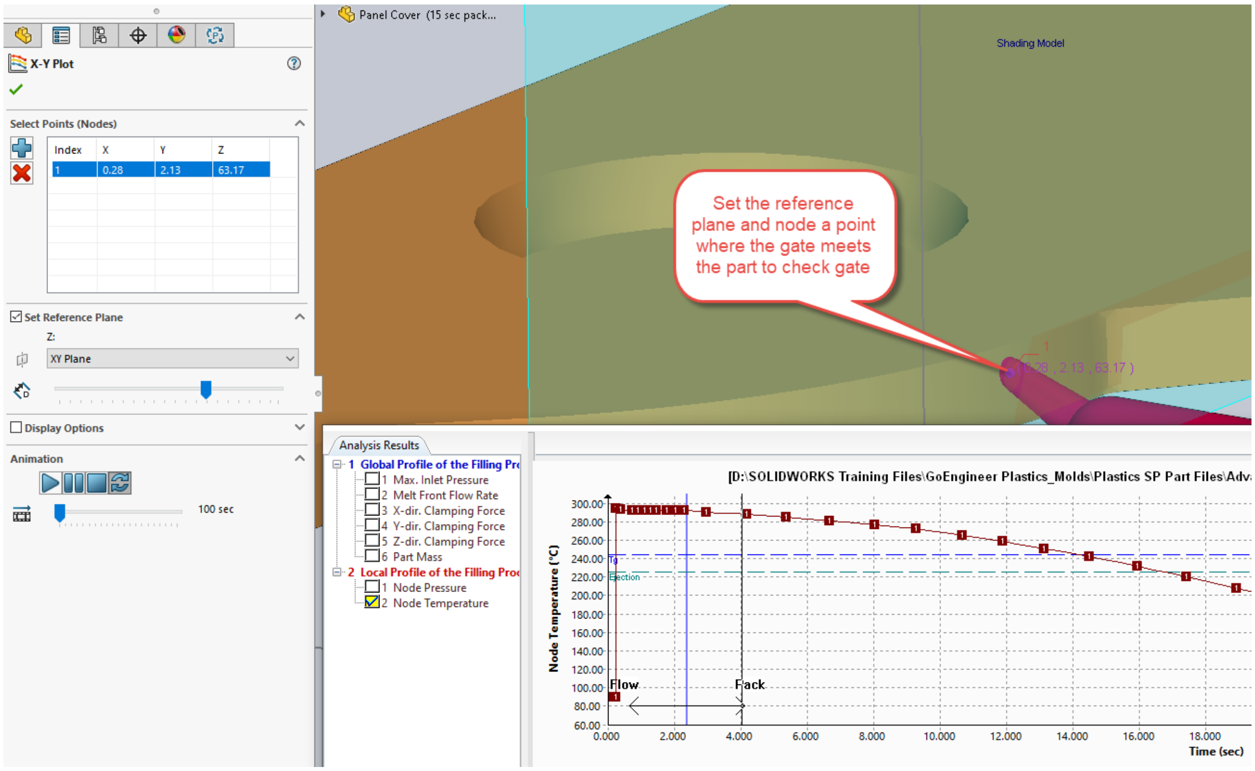

The **Isosurface Mode** command is used to create a surface within a model where a result maintains a constant value.




Selecting Isosurface Mode and setting the minimum value to the eject temperature of the material in the Temperature at Post Filling End Plot will show only the geometry that is still above the eject temperature.

# Nodal Temperature X-Y Plot

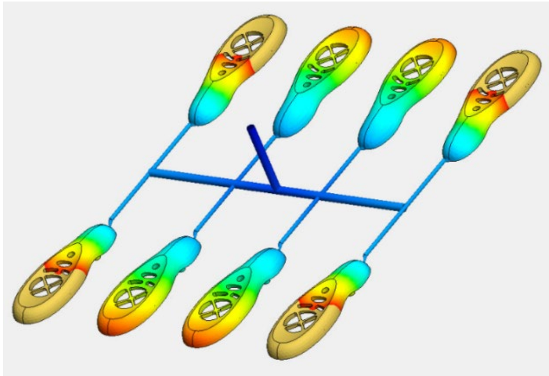
The Nodal Temperature is an X-Y Plot that allows users to plot the temperature of any node against time. It is useful for determining gate freeze time when developing packing and cooling time settings.



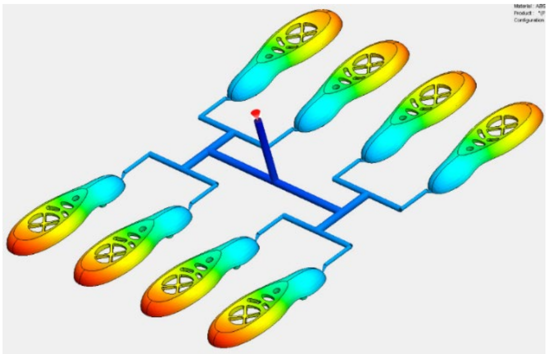

This plot shows the nodal temperature reaches the glass transition temperature between 14 and 16 seconds.

# LESSON 3: MULTIPLE CAVITY MOLDS

Multi-cavity molds can have different types of arrangements. Some mold layouts are better than others with regard to the end product results. A good mold layout achieves equal flow lengths to each part cavity. In this arrangement seen here, the part cavities do not have the same flow length from the sprue.



In this next arrangement, the part cavities have the same flow length from the sprue and theoretically should fill more evenly.

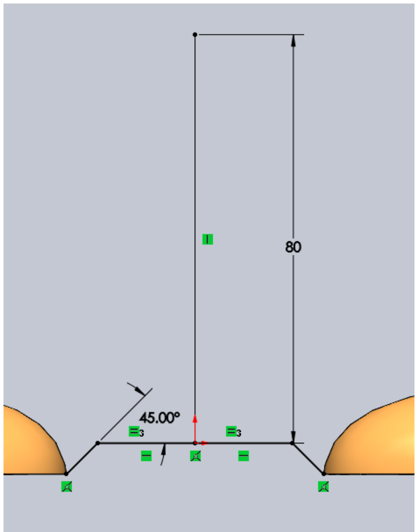


Adding part cavities using powers of two (e.g., 2, 4, 8, and 16) into the mold allows you to maximize the parts per cycle while maintaining equal flow links on each part cavity.

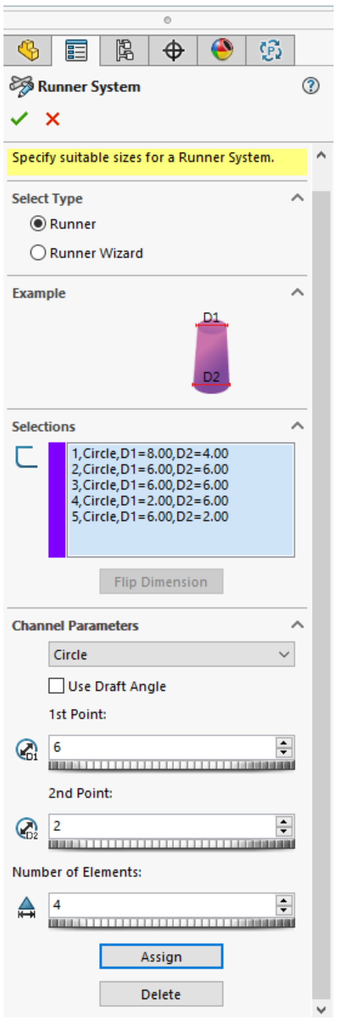
## Runner System

Runner elements are used to distinguish between cavity systems and runner systems. These can be with solid modeled geometry or they can be created based on sketches using the Runner design tools in SOLIDWORKS Plastics.

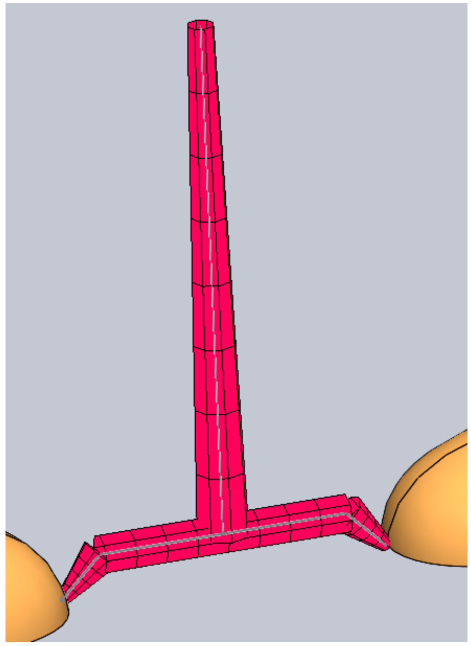
## Runner Channel Design




When using the runner option, runner geometry must be designed using 2D or 3D sketches made from lines and arcs. Sketch entities must be connected to the cavity geometry using relations such as coincident or pierce. These sketch entities are then used to create the components of the runner system: the sprue, the runner, and the gate. Each leg of the runner system needs to be an individual sketch entity.




Each segment of the sketch is selected and assigned a diameter to represent the runner system components.

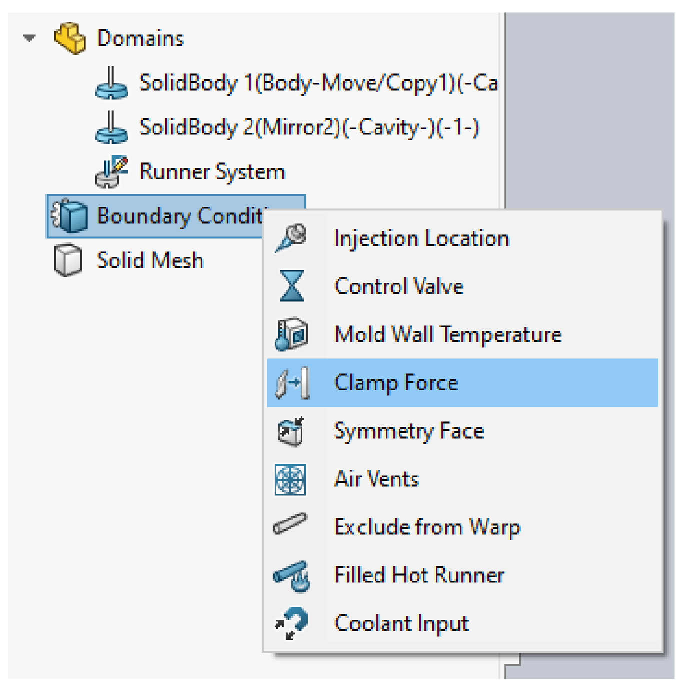


Entering different values in the **1st Point** and **2nd Point** fields will apply a taper to the segment such as with sub-gates or the sprue.

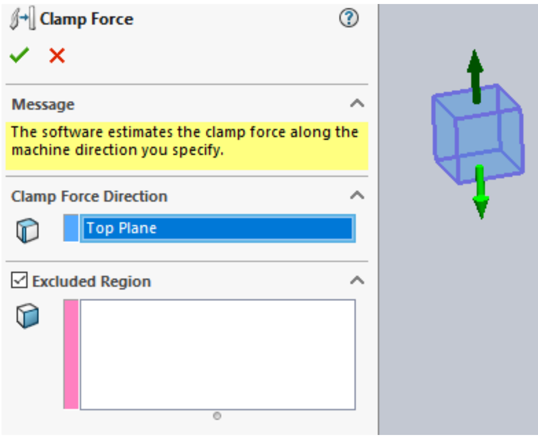
The **Flip Dimension** selection is used to change the direction of the taper if needed.

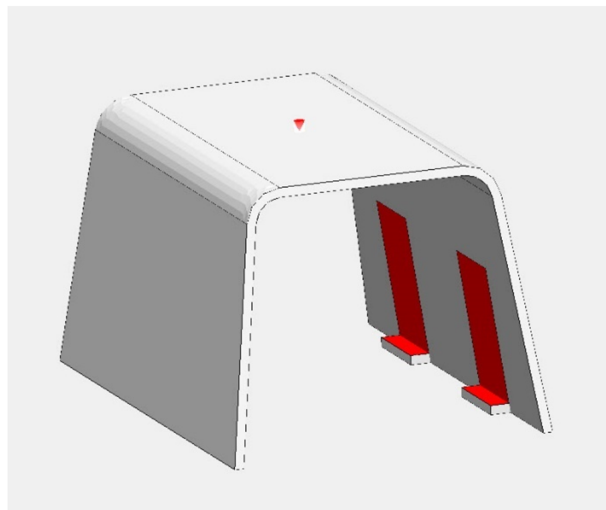
## Clamping Force

Clamping force is the amount of force required to keep the mold closed during the injection molding process. This force doubles with two cavities. Likewise, the force increases by a factor of four with four cavities. Each injection molding machine has a maximum clamping force it can exert. **The clamp force limit** defines the machine's maximum clamping force.



The **Clamp Force** command is used to define two parameters the **Clamp Force Direction** and **Exclude Region.**

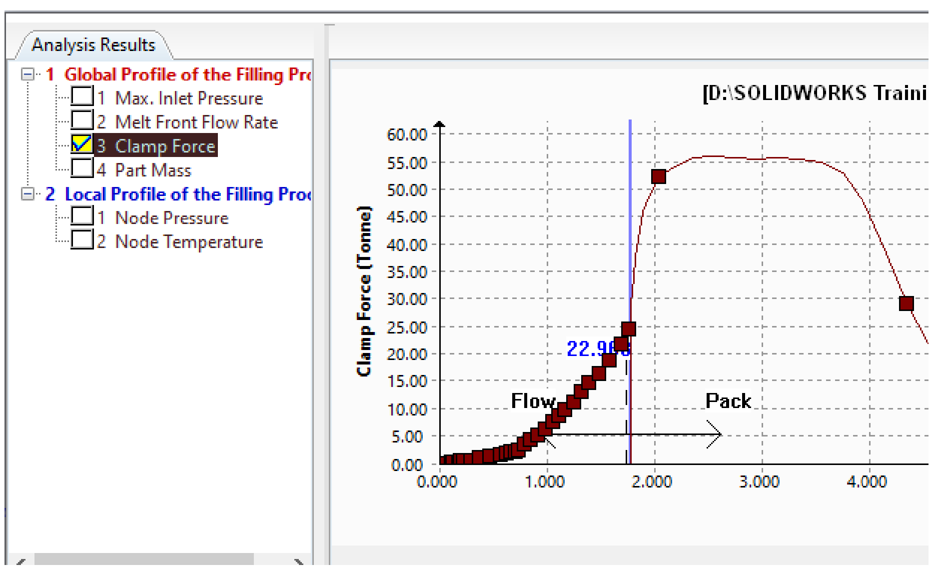


The clamp force **Exclude Region** section is used to define locations of the model that do not contribute to the clamping force such as locations on the model that are molded by slides.

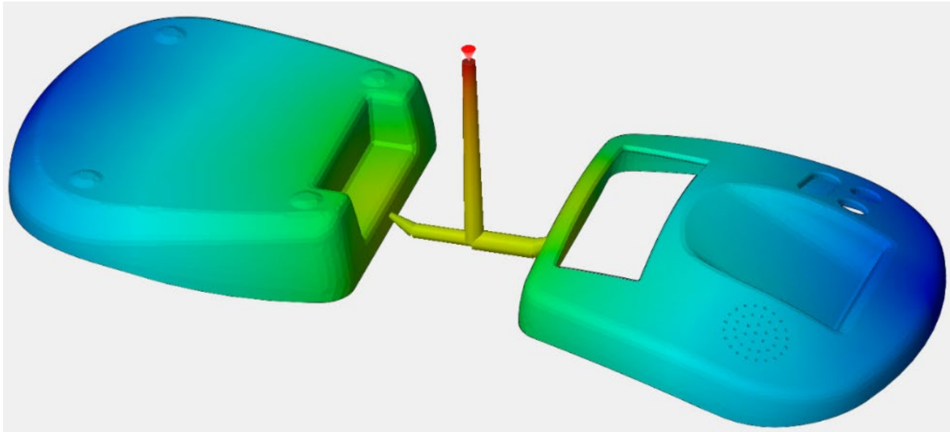
## Clamp Force X-Y Plot

The Clamp Force X-Y Plot tracks the forces required to keep the mold closed over time. The highest clamp force will happen just after the flow to pack transition.




It is a good practice to run Flow + Pack when evaluating clamp force.

## Family Molds

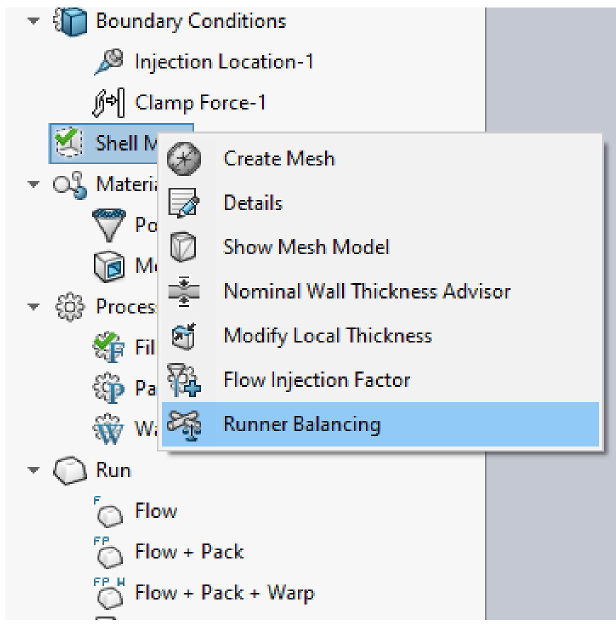


A family mold is a special type of multi-cavity mold containing two or more different parts which may eventually form an assembly. In a family mold, paired parts are created from a single shot so they have the same color and characteristics. Paired parts rarely have the same volume and fill characteristics so the runner system of a family mold must be artificially balanced to ensure uniform pressure distribution in each cavity.

## Runner Balancing

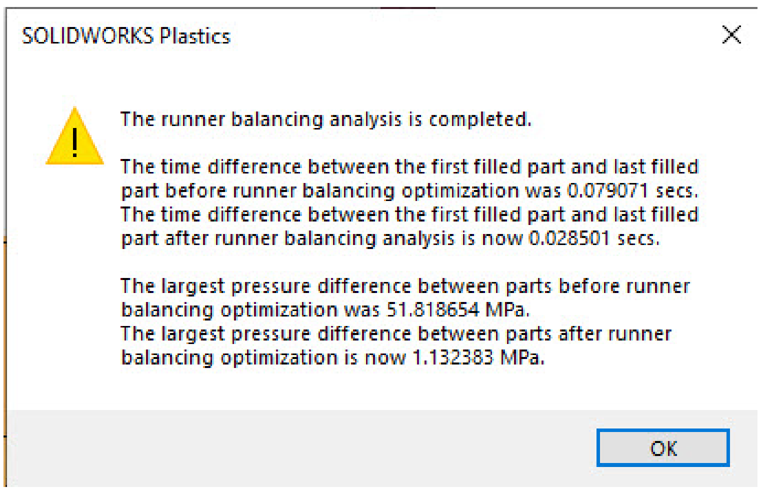
The **Runner Balancing** command is used to automatically adjust the runner diameter to balance the flow between cavities.

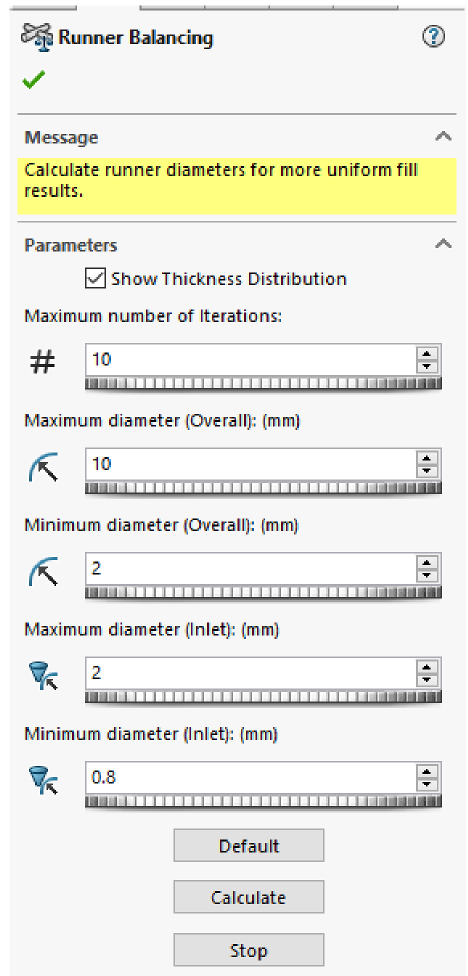
The Runner Balancing command will only work with a shell mesh.



Runner balancing works by setting the Min. and Max. diameters of the runner system and gates. The system will run through the specified number of iterations until the fill time and pressure equalized.

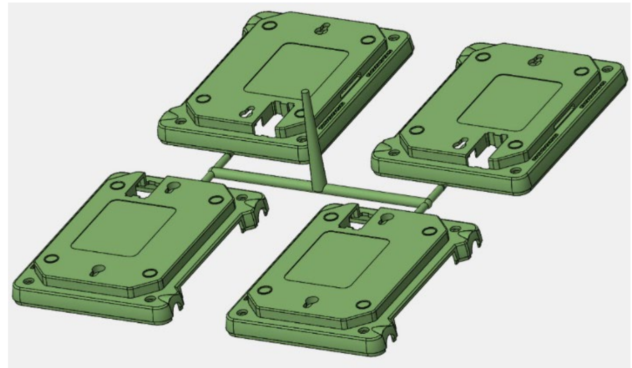
When the analysis is complete, a message appears noting the time difference for fill time and the pressure difference before and after runner balancing.

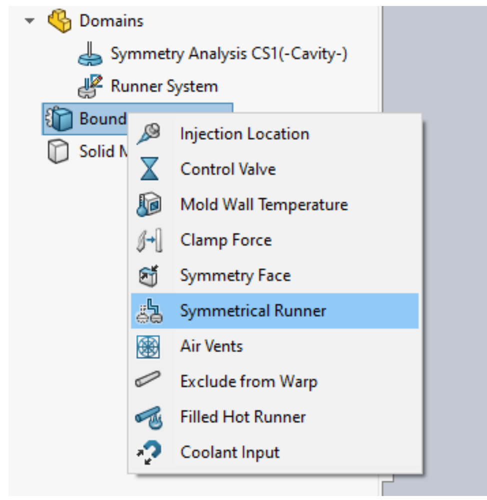


# LESSON 4: SYMMETRY ANALYSIS

Running an analysis on a multi-cavity mold can be computationally intensive. To reduce the processing time, we could take advantage of symmetry which is often seen in multi-cavity molds.





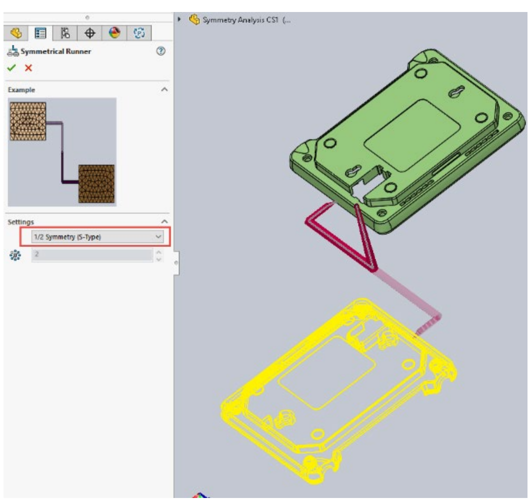

## Symmetrical Runner

The **Symmetrical Runner** command is used with the Runner command.

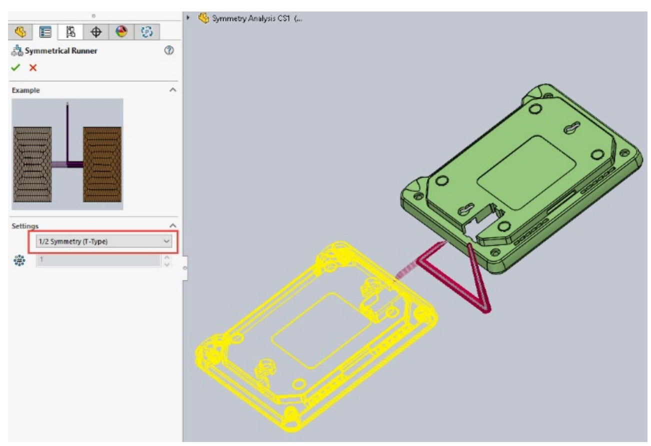
The **Symmetrical Runner** command only works with the solid analysis procedure.

## Symmetrical Runner Layouts

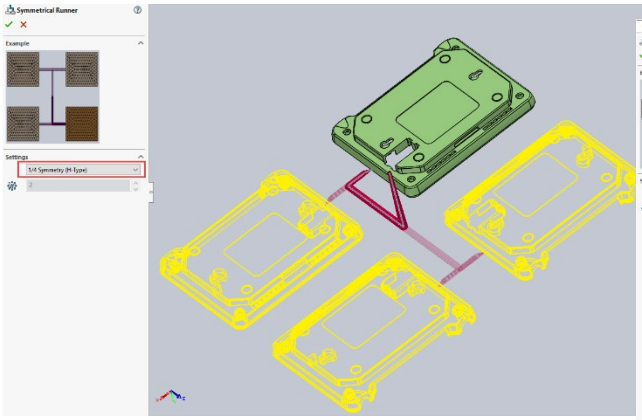
### ½ Symmetry (S-Type)



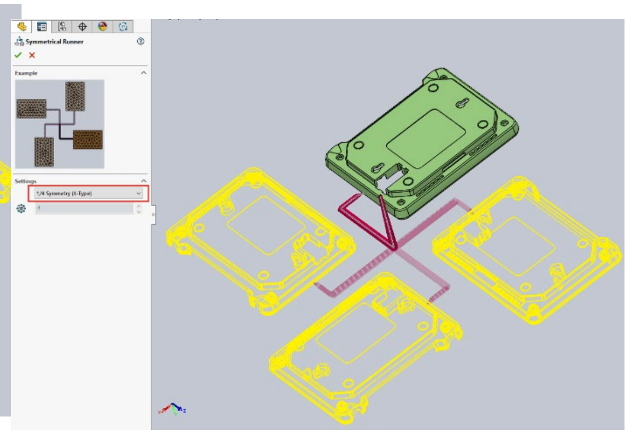

### ½ Symmetry (T-Type)



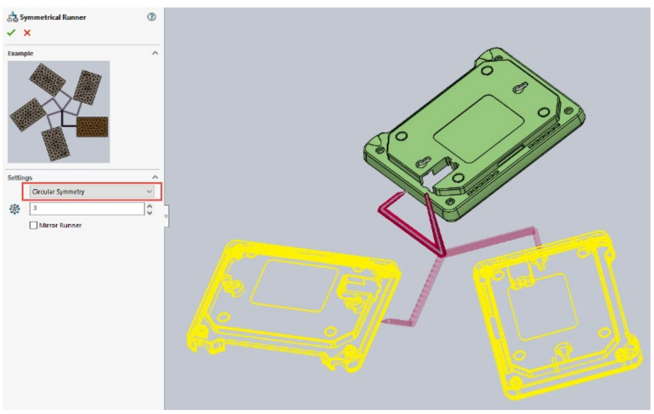
### ¼ Symmetry (H-Type)



### ¼ Symmetry (X-Type)



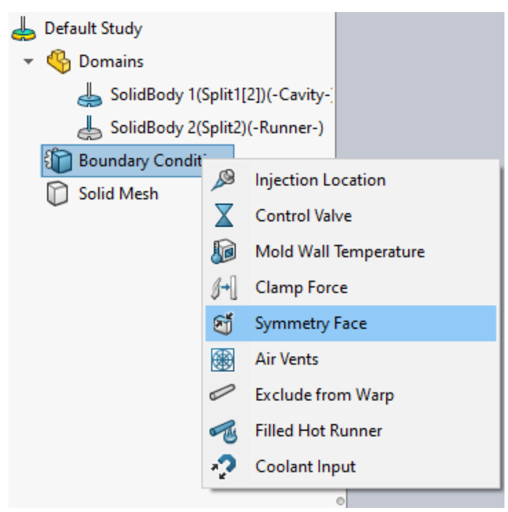
### Circular Symmetry



## Symmetry Face

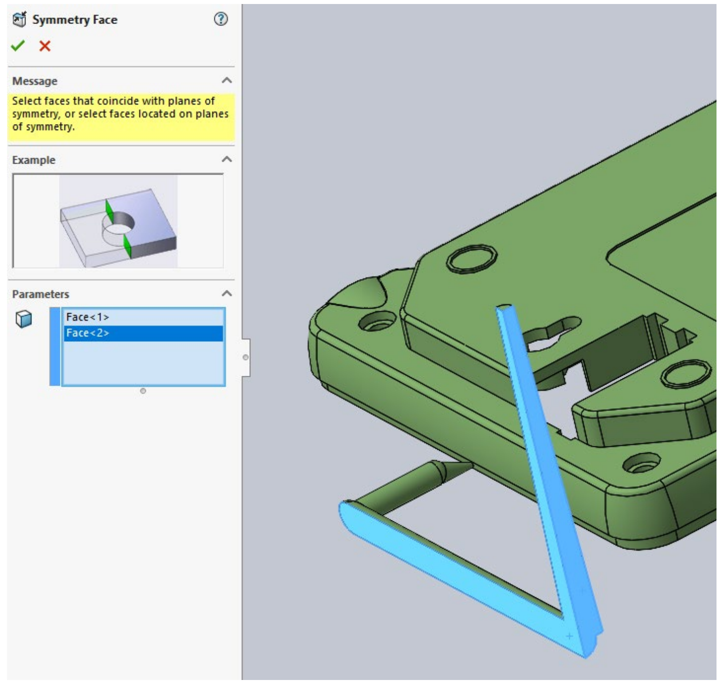
The **Symmetry Face** command is used on faces where flow and heat transfer are both symmetrical. The **Symmetry Face** command is only available for use with solid elements.

The runner system body must be cut to have faces where the symmetry split would be.

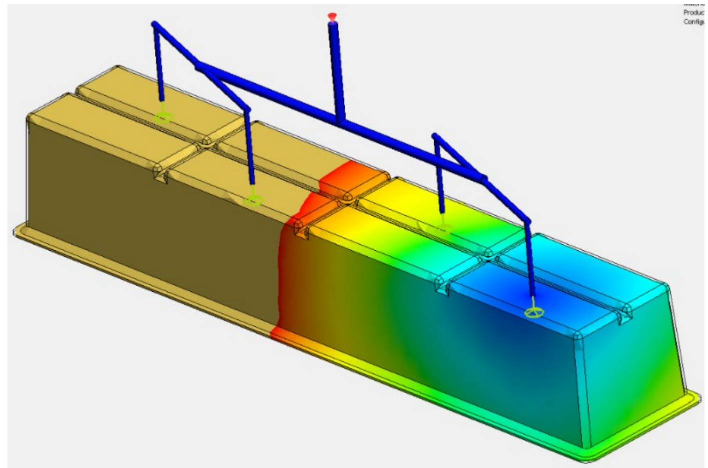


Select the faces of symmetry on the runner system.

The analysis will compute as if there are multiple cavities, but the system will only have to mesh the single part and runner geometry which reduces the mesh elements.

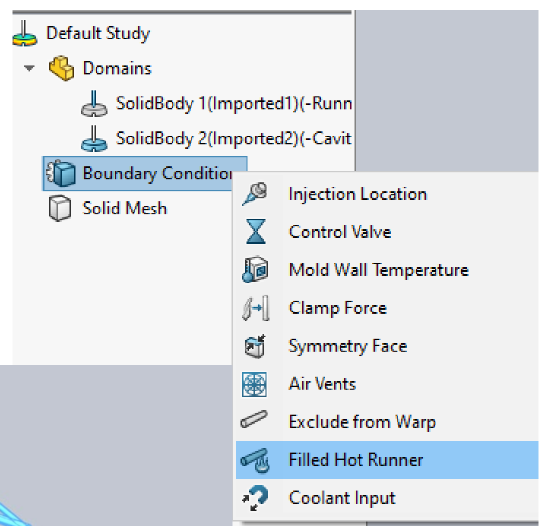


# LESSON 5: VALVE GATES AND HOT RUNNERS



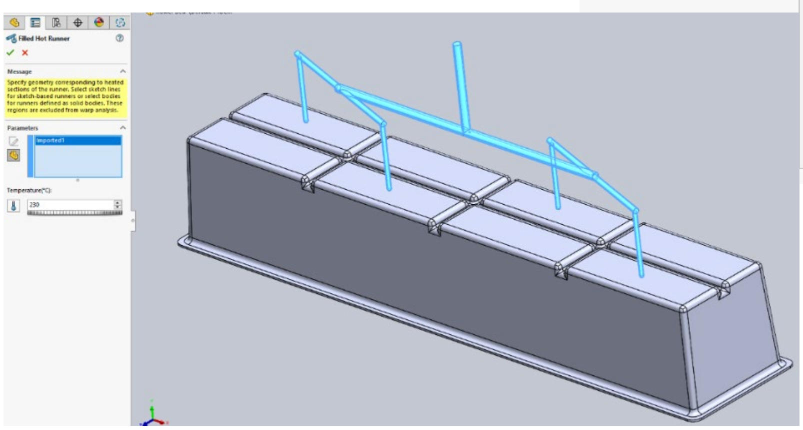
In a typical two or three plate mold, plastic is injected from the machine to the sprue and runner system before it is injected into the cavity. Once cooled, the sprue, runner, and cavity are ejected from the mold. This sprue and runner are often separated, reground, and used again. However, the properties of the resin degrade after it has been reground. To solve this problem, hot runner systems are used. In a typical hot runner system, heating coils keep the resin in the runner system at a controlled temperature. In these systems, the sprue and runner are never ejected, thereby reducing waste. Hot runners have the added benefit of producing parts with exceptional surface finish. Hot runner systems, however, do add considerable upfront and maintenance costs to the mold, so they are not used all the time.

## Filled Hot Runner

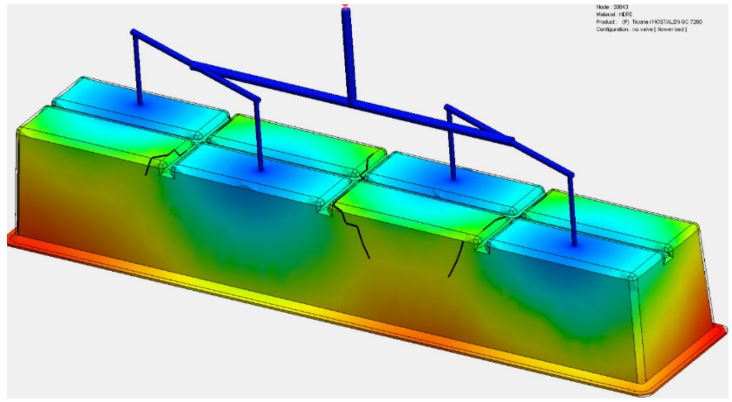



The **Filled Hot Runner** is defined in the **Boundary Conditions** by selecting the solid body geometry.

The hot runner system temperature is set here.

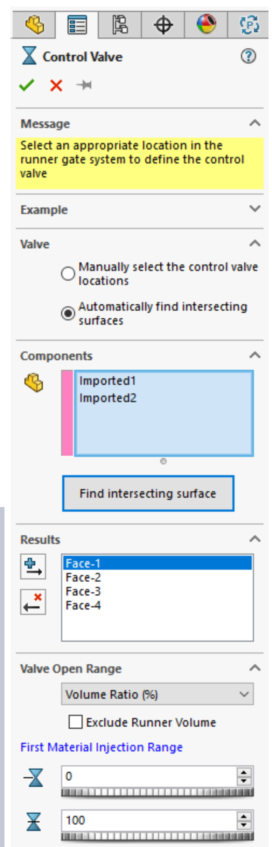


## Valve Gates



In a large part, multiple gates may be required to fill it, however, when multiple gates release plastic at the same time, weld lines are created where the flow fields meet. Weld lines cause surface defects and structural weakness. To fix this problem, valve gates are used. A valve gate is a gate with an on-off switch. Valve gates can produce parts with high levels of surface finish and strength such as car bumpers.

With valve gates, the filling process starts with at least one gate open, and the rest of the valves start closed. Once flow inside the cavity reaches one of the closed gates, that valve opens releasing additional resin in the cavity. This process continues with additional valves if the part is long enough. By using valve gates, the flow field remains continuous thereby reducing weld lines. In SOLIDWORKS Plastics, valve gates are created through the **Control Valve** feature.

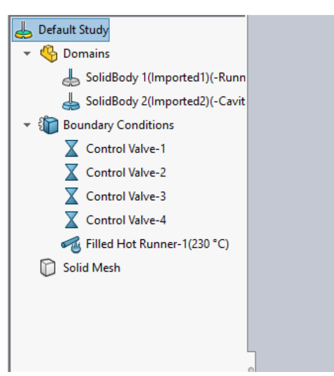



## Control Valve

The **Control Valve** is defined in **Boundary Conditions**.

There are options to manually select the control valve locations or to automatically find intersecting surfaces.

To use automatically find the intersecting surfaces, the body of the cavity and the body of the runner system are selected and then the **Find Intersecting Surface** button is pushed. This will add the control valves automatically on these intersecting surfaces.




Each **Face** is chosen in the **Results** selection pane and the Injection range can

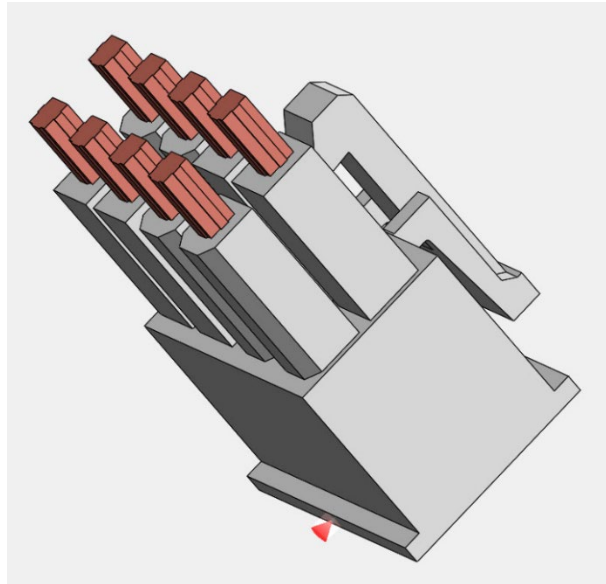
be set. Clicking the **+ sign** will add that control valve.
Once the command is accepted, the **Control Valves** will show in the **Boundary Conditions**.

# LESSON 6: USING INSERTS

An insert is a component that is held inside the cavity while melted resin is allowed to flow around it. This results in a single part made of multiple materials held together by plastic. Cavity domains are always assigned polymer materials while insert domains can be assigned metal or polymer material properties.

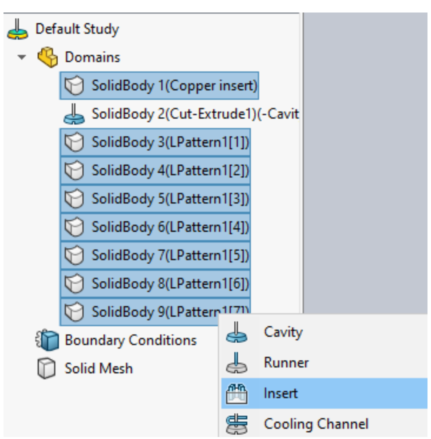
To run an analysis with inserts, you must use a solid element mesh.

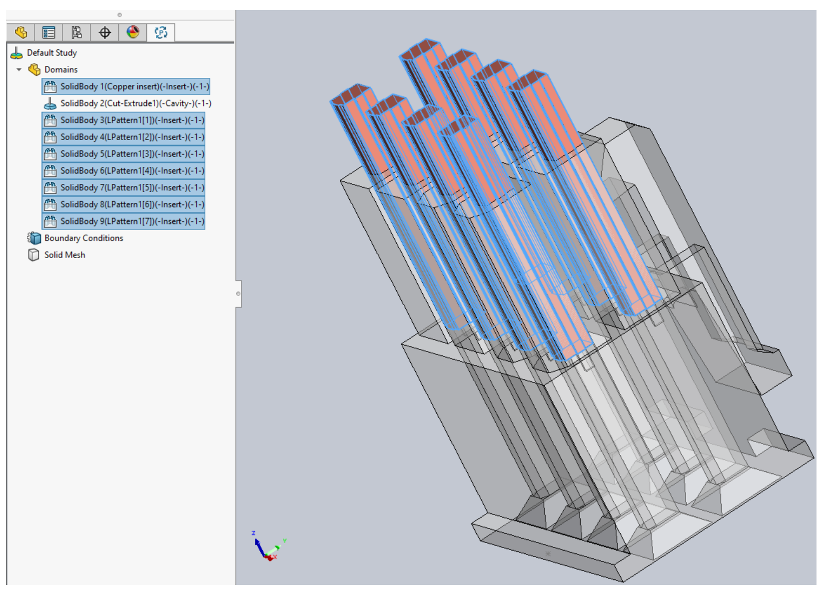
Insert and cavity bodies must fit line to line, meaning no overlap or gaps for the mesh to work properly.



## Insert Domain

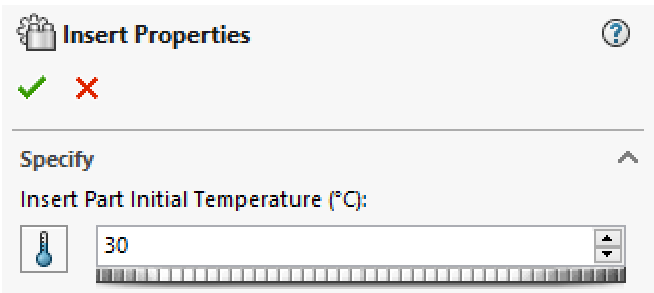
The insert domains are defined by selecting them in the domains tree and assigning them to be an insert.

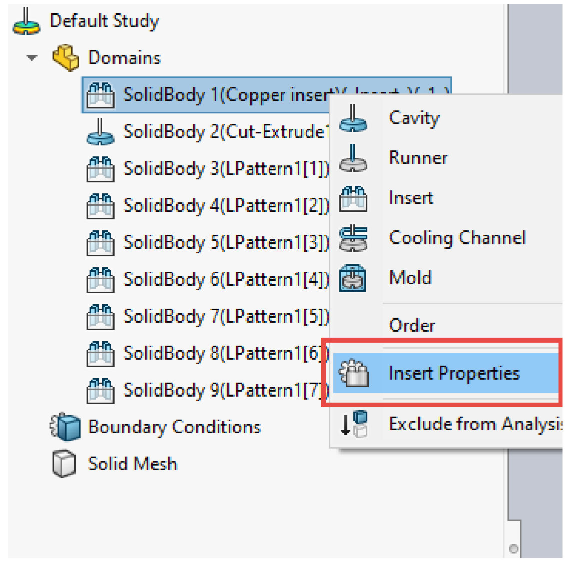


## Insert Properties

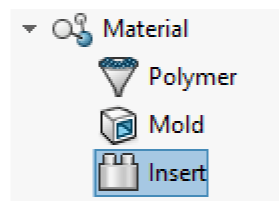
Right-clicking on an Insert domain will allow the user to set the **Insert Properties,** which is the initial temperature of the insert.




## Insert Materials

Materials are applied to inserts through the **Inserts** command. Inserts can be polymers or metals. The polymer library for Inserts uses the same database as the molded resin.

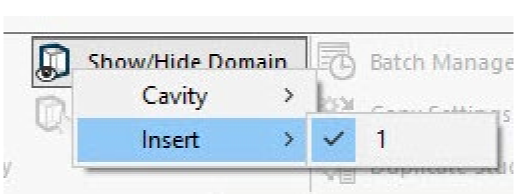



The metal library includes:

- Aluminum alloys
- Copper alloys
- Iron
- Other alloys and metals
- Steel
- Titanium alloys
- Zinc alloys

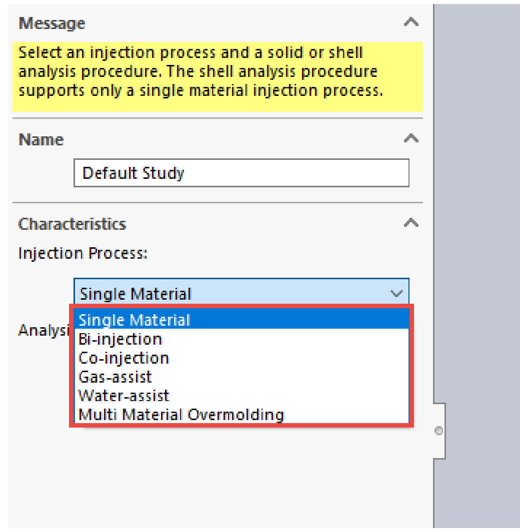
## Hiding Cavities and Inserts

Use **Hide/Show Domain** to hide and show cavities and inserts when viewing the results. All cavities or all inserts can be toggled between hidden and shown.



# LESSON 7: ADVANCED MOLDING PROCESSES

SOLIDWORKS Plastics can simulate advanced molding processes such as bi-injection, co-injection, gas-assisted molding, water-assisted molding, and multi-material overmolding.




## Bi-injection Molding

In the Bi-injection process, two different polymer materials are injected at different locations with each injection process being independently controlled. The materials meet and form a weld line within the cavity.

**Process Set-up:**

- Only the solid analysis procedure supports Bi-injection.
- A single cavity and two materials are allowed.

## Co-injection Molding

For the Co-injection molding process, two different polymer materials are injected sequentially at the same location. Common reasons for co-injection are to use a cheaper fill material for the hidden core of a product or to combine properties of different polymers such as color, feel, or mechanical properties.

**Process Set-up:**

- Supported by the solid analysis procedure only.
- A single cavity and two materials are allowed.

## Gas-assist Molding

In Gas-assist molding, pressurized inert gas, typically nitrogen, is injected into a partially filled or nearly filled cavity. The gas displaces the molten plastic material in thicker regions of the part to complete the filling and to pack the cavity. The result is a hollow part that offers several benefits:

- Less plastic material
- Reduced part weight
- Lower cycle time
- Reduced warpage and distortion
- Improved part appearance

**Process Set-up:**

- The gas assist process is supported by the solid analysis procedure only.
- A single cavity and two materials are allowed.

## Water-assist Molding

In the water-assisted process, water is injected into the melted polymer stream to create a hollow part.

**Process Set-up:**

- Water-assist is supported by the solid analysis procedure only.
- A single cavity and two materials are allowed.

## Multi-Material Overmold

Multi-material overmolding is when two or more different polymer materials are injected within the same cycle. Separate injection nozzles introduce materials sequentially into the cavities. This is achieved by using a special mold that rotates between cycles.

**Process Set-up:**

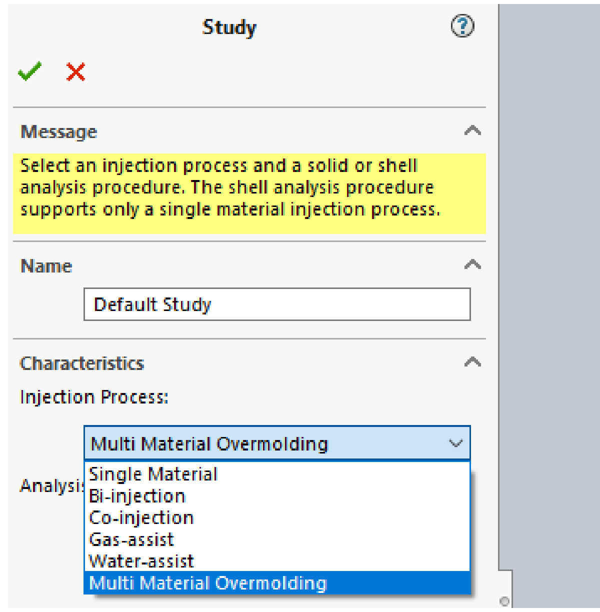
- Only the solid analysis procedure is supported.
- Up to six different materials are allowed.

## Multi-Material Overmold Analysis

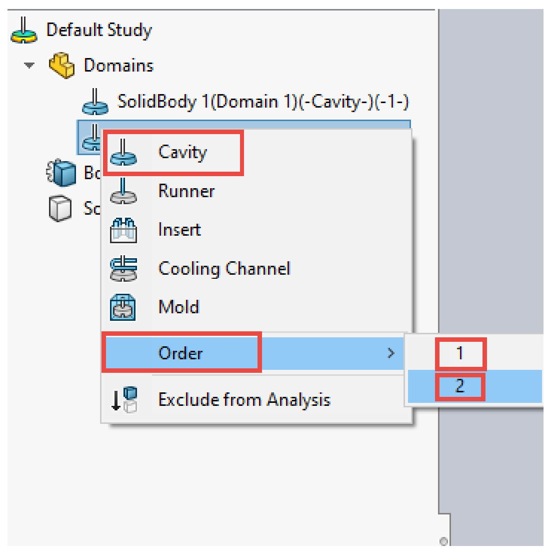
In more complicated molds such as taillights on cars or toothbrush handles, a body will be molded with one type of plastic then molded over with the second type of plastic. There are several ways to do this.

One way is to have the part ejected from the first cavity then rotated into the mold of a second cavity. Multi-shot injection molding is an expensive operation but can be used to create impressive plastic components.

To start a study for multi-shot molding, choose the **Multi-Material Overmolding** option when selecting the **Injection Process.**




## Domains



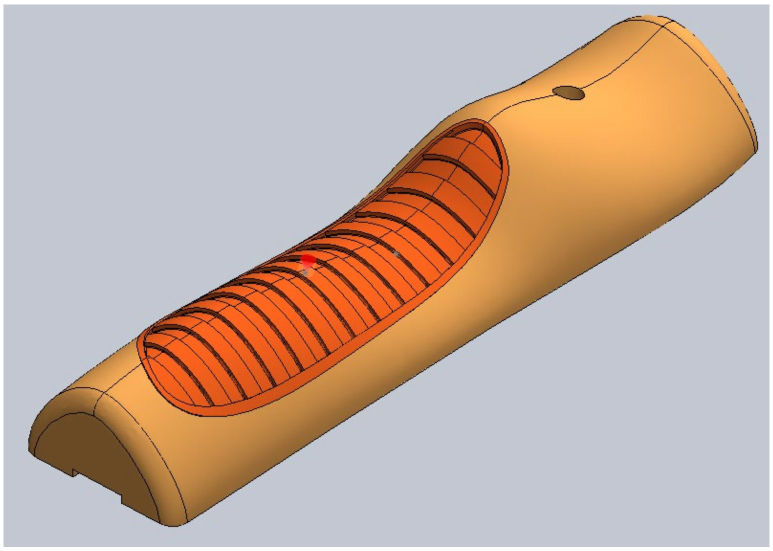

The part file should have at least two bodies for the cavities. Each body will be defined as a **Cavity**.

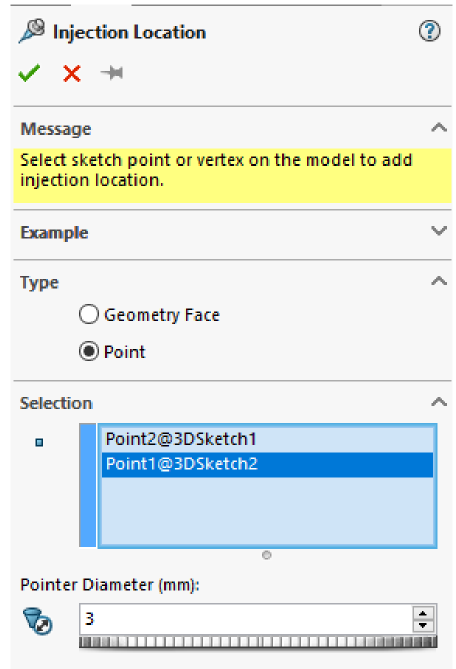
The **Order** must be assigned for each cavity domain. The order is the sequence in which the cavities will be filled. **Order 1** will be injected first; **Order 2** would be 2nd to fill.

Because there two injection molding domains, each will have its own injection location, material, fill, and pack settings.

## Injection Locations

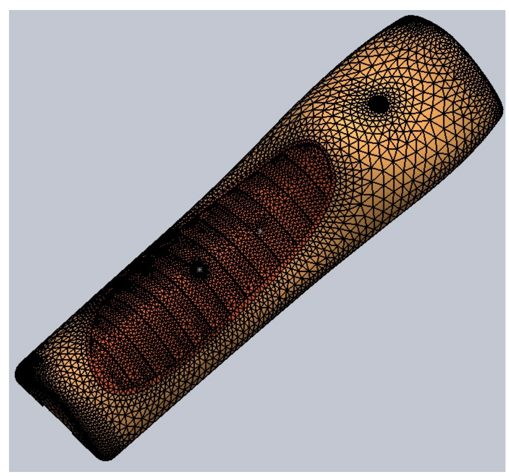
Each cavity will have its injection location defined.






## Mesh

The multi-body parts must fit line-to-line with no overlaps or gaps for the mesh to solve properly. The bodies will mesh together in the same step.

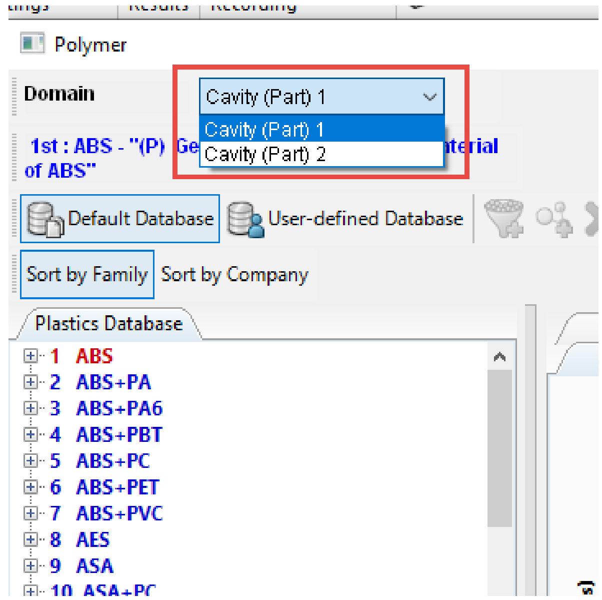


## Materials

A material must be applied to each domain.

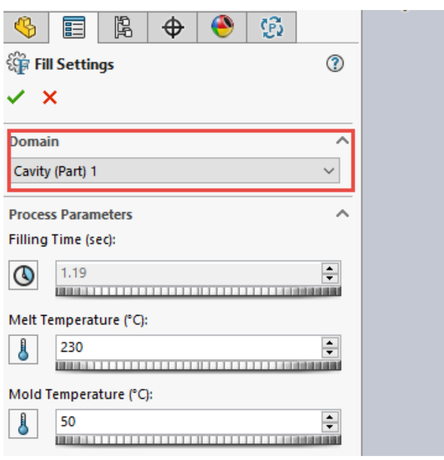
Choose Cavity (Part) 1 and select the material for that cavity.

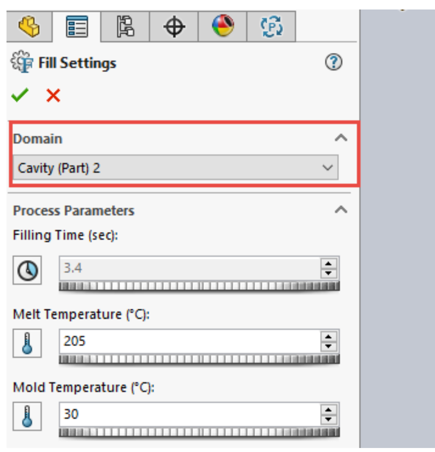
Then choose Cavity (Part) 2 and select the material for it.

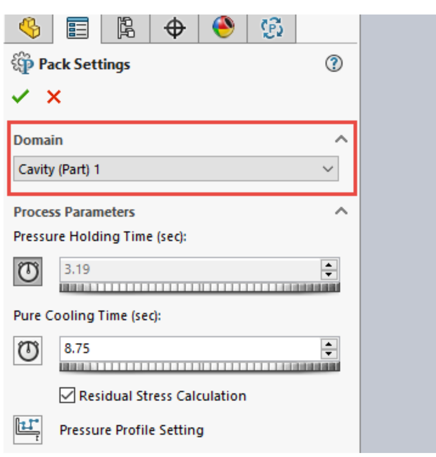


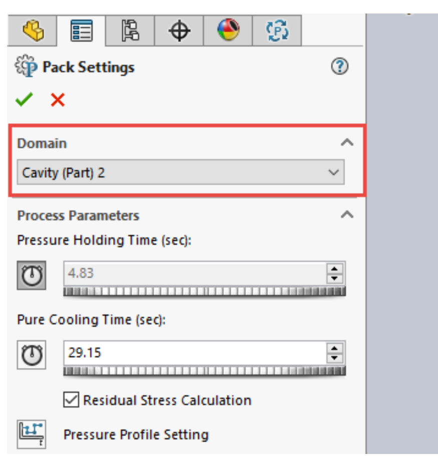

## Fill and Pack Settings

The Fill and Pack Settings change between the two cavities. The default Fill and Pack Settings are derived from the material applied to each cavity.



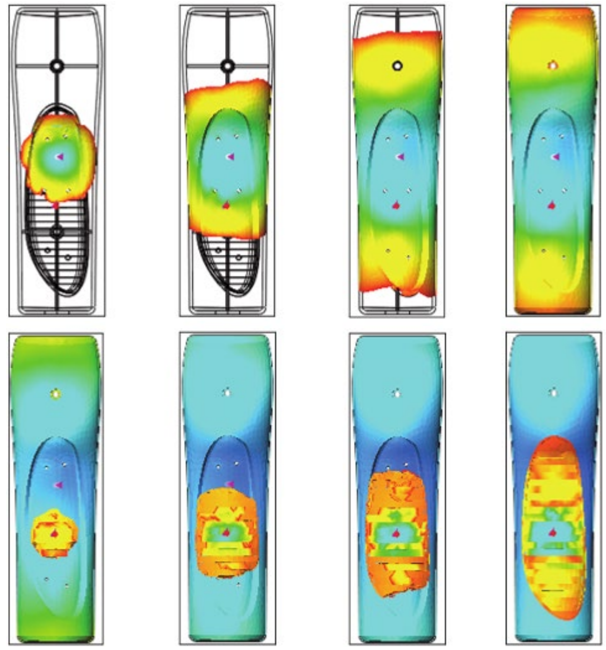



## Fill Time Plot

The Fill Time plot will show the Cavity 1 Domain fill first and then the Cavity 2 Domain will fill after.

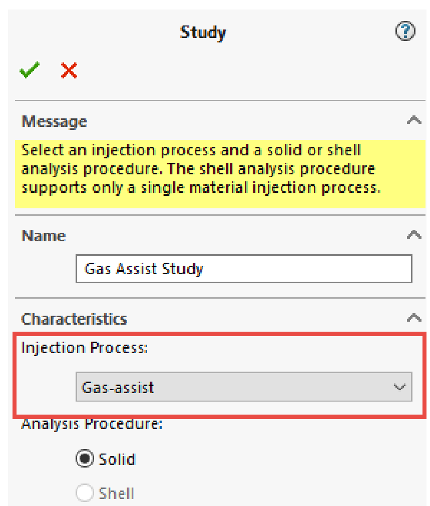


## Gas-Assisted Analysis

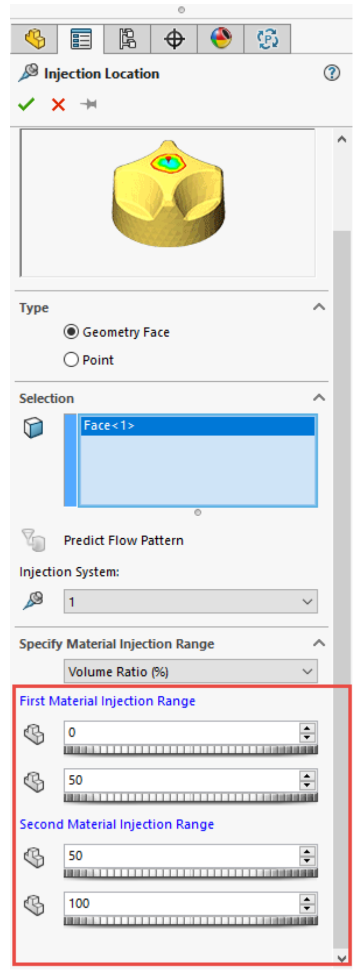
Gas-assisted injection molding is a way to reduce the thickness of parts that would otherwise be thick in nature. In a gas-assisted molding process, a resin is injected into the cavity so that it is about halfway full, a gas is then injected into the mold pushing the resin to the outer walls.

Specifying the **Gas-Assist** injection process will automatically activate some options in the setup.

The Injection Location PropertyManager will have additional settings and the material SelectionManager will have Nitrogen selected as the 2nd material.



# Injection Location

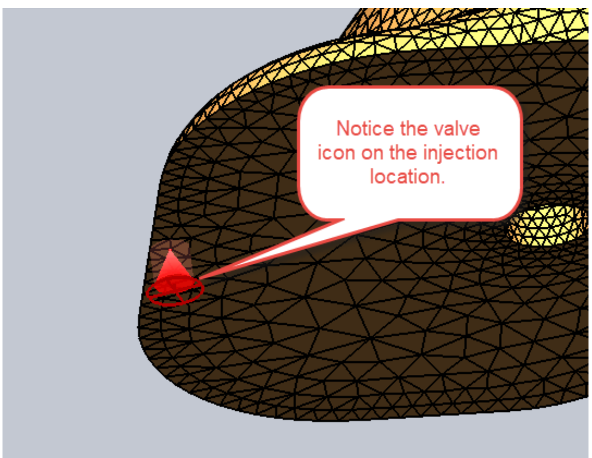



Specify the injection location.

Notice that there are additional settings for the **Material Injection Range**.

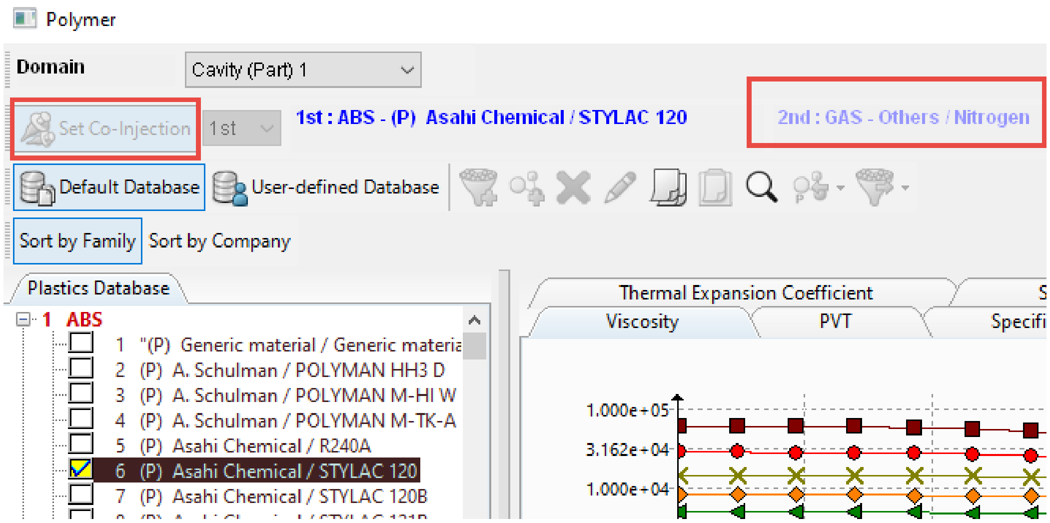
The **First Material Injection Range** is for the plastic resin. This setting is typical from 0 to some percentage of fill around halfway.

The **Second Material Injection Range** is for the Nitrogen gas. This setting will start at the percentage of fill the first material stopped at and continue to 100%.




# Materials

In the case of a gas-assisted injection molding process, two materials must be specified for the same domain: one polymer and the other gas. This is accomplished through the **Set Co-Injection** command which is automatically activated once the study is created with **Gas-assist** specified in the **Injection process**.

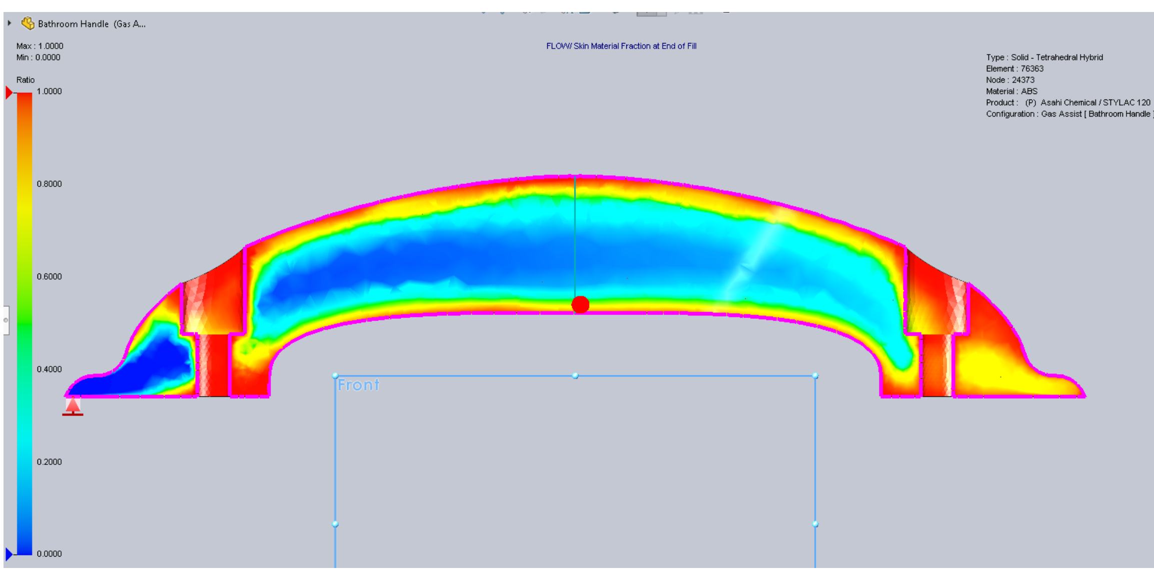



Notice that the 2nd material is already set to Nitrogen. Only the 1st material polymer needs to be set.

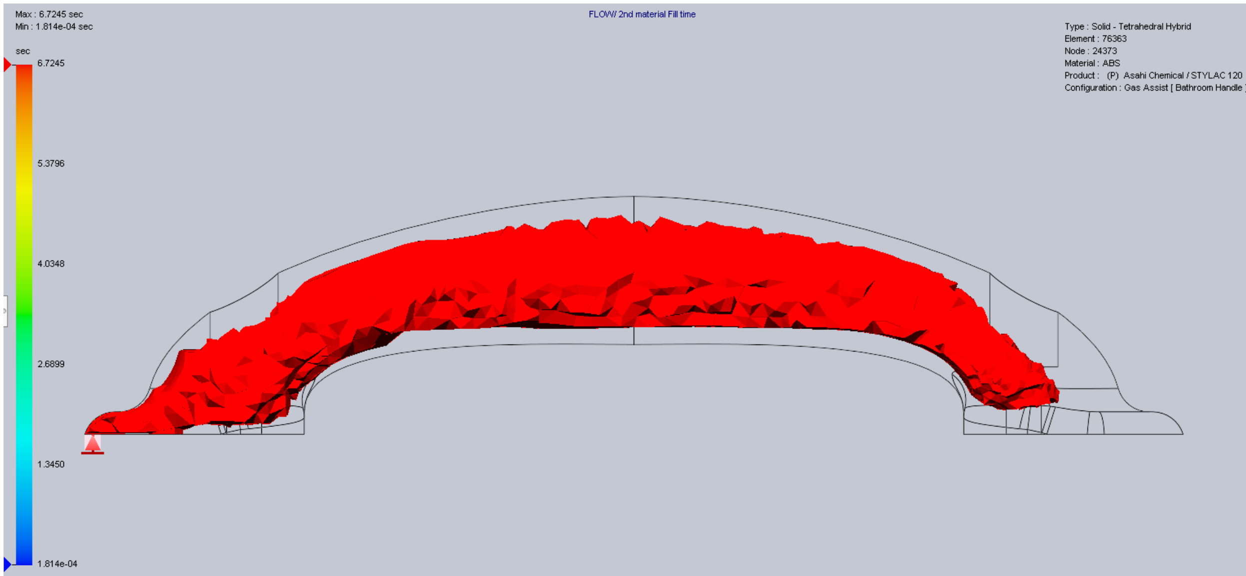
## Results

**Skin Material Fraction at End of Fill s**hows the fraction of the 1st material at the end of fill.

Areas of blue on the outer surface would mean that the Nitrogen broke through the skin layer.



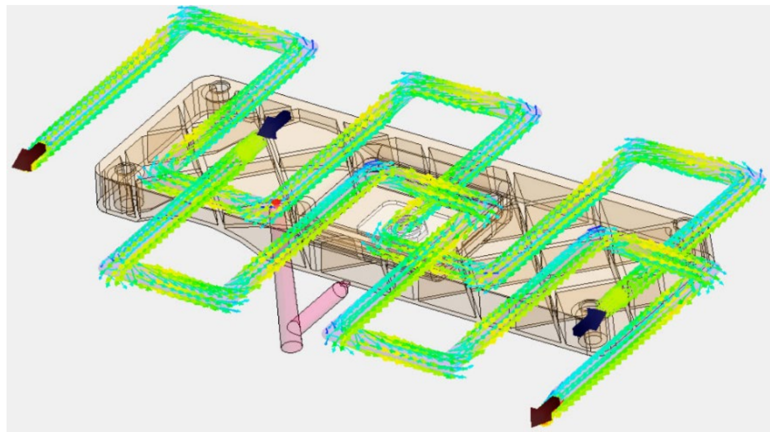
2nd **Material Fill Time** shows the fill time for the Nitrogen gas.



# LESSON 8: COOLING AND WARP ANALYSIS

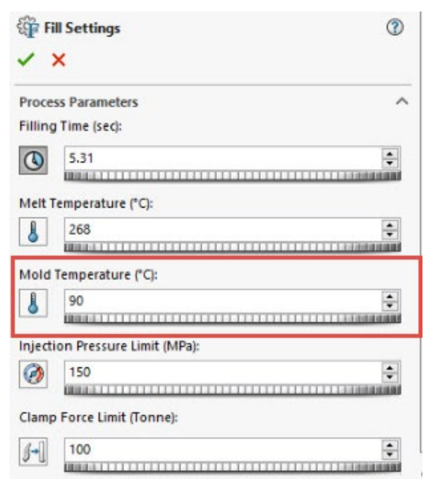
## Cooling

Cooling occurs throughout all stages of the injection molding process from the fill stage through ejection. The rate of cooling has a significant effect on part dimensions and influences part defects. If a part can be cooled uniformly, residual stress within the part can be reduced thereby minimizing the risk of warping and cracking.



The cooling process begins when heat is transferred from the hot liquid resin to the mold walls of the cavity. The heat is then conducted through the metal cavity to the cooling channels. Pumped fluid runs through the cooling channels conducting heat away from the mold. The positioning and properties of the mold material and cooling channels are therefore an important consideration in mold design.

So far, we have assumed that the mold walls stay at a constant temperature as specified by the **Mold Temperature** parameter in **Fill** settings.




The mold temperature, however, is never constant - it changes with time and location. To accurately model the temperature of the walls of the cavity, the cooling channels and the mold bodies must also be modeled.

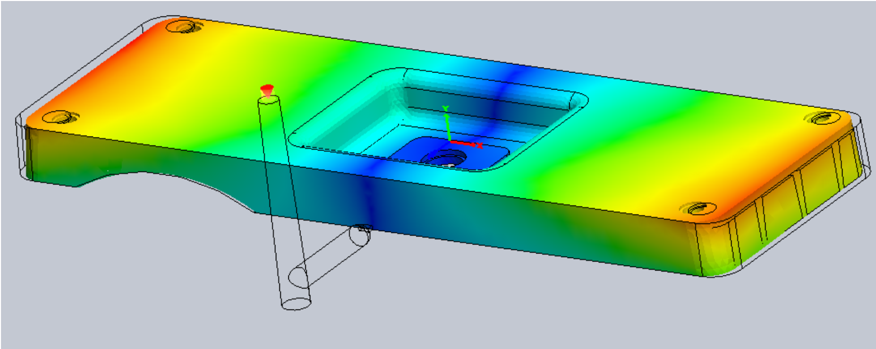
There are two ways to model the mold and the cooling channels.

First, they can be modeled using conventional SOLIDWORKS geometry and specified as cooling channels and molds through domains.

The second way is to start with a sketch that follows the profile of the cooling channels then use the cooling channel and the virtual mode features.

By running a cooling analysis, we will get a more accurate shrinkage prediction and prediction of warpage.

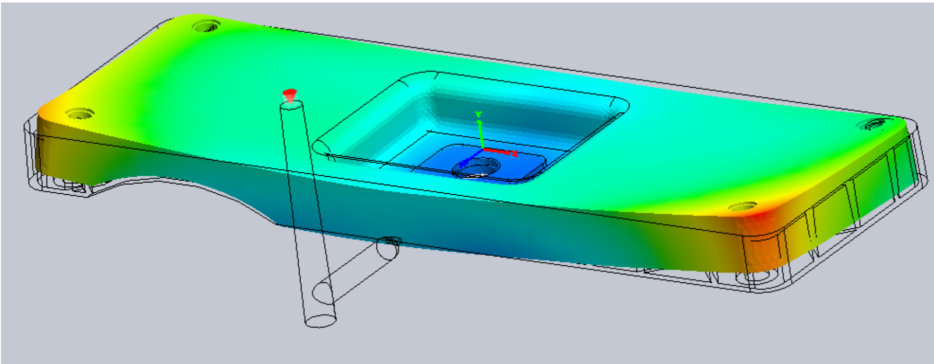
## Shrink and Warp



Shrinkage is inherent in the injection molding process. It occurs because the mold is filled with hot resin under high pressures and then cools to room temperature. This reduction in temperature can cause the molded plastic part to shrink by as much as 20% depending on the material.  Shrinkage is a contributing factor for part warpage; however, warpage is more complicated. Parts warp because of the combination of in-mold residual stress and non-uniform shrinkage.  If a part shrinks uniformly as opposed to non-uniformly, it will not warp but instead becomes smaller.

There are several factors that contribute to warpage:

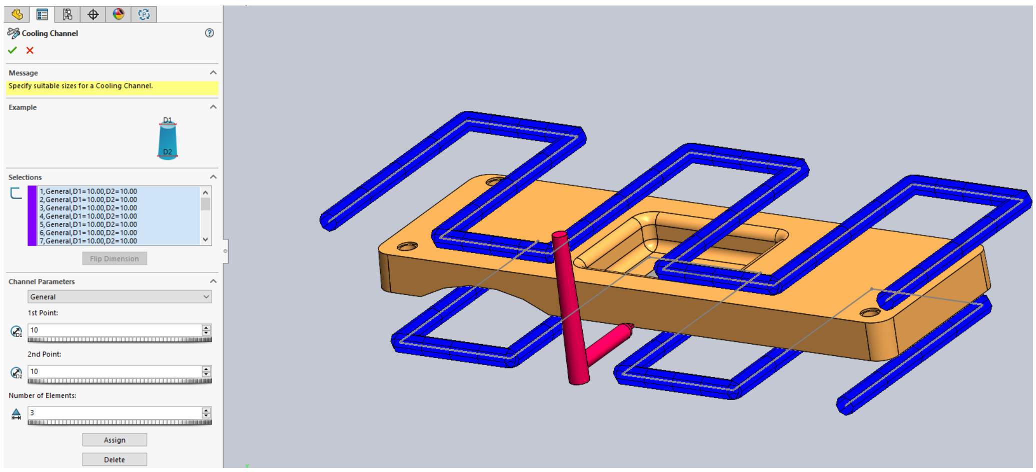
- Variation of temperature as the part cools in the mold.
- Variation of pressure of the melt in the mold.
- Variable rates of shrinkage depending on molecular and fiber orientation.



# Cooling Channel

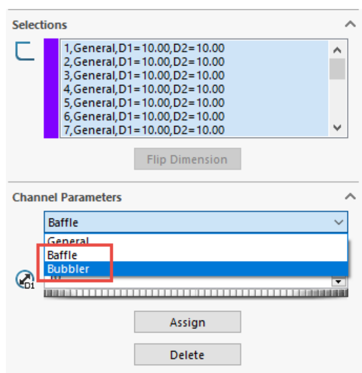
The Cooling Channel domains can be defined with 3D modeled geometry or based on sketch lines.

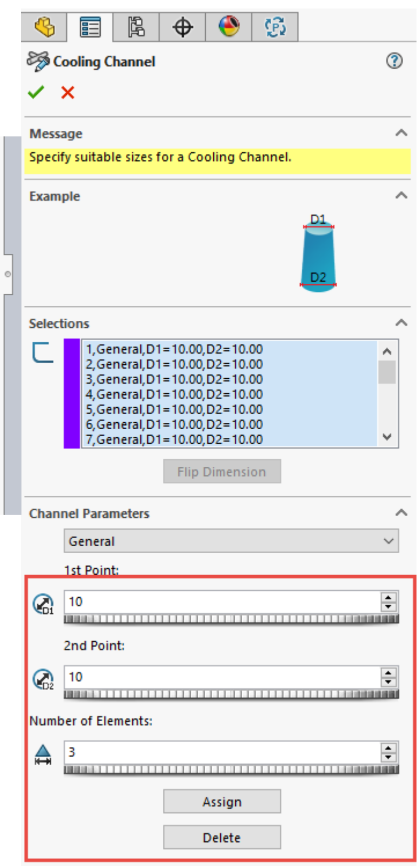
The sketch line method will result in fewer mesh elements which, in turn, will solve the cooling analysis faster.



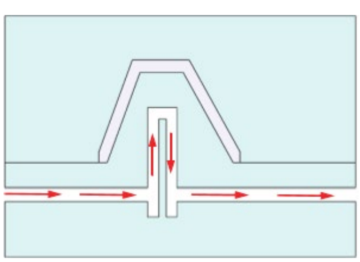
The sketch lines are selected and assigned a diameter. Multiple sketch lines can be selected and assigned at the same time.

There are options to create Baffles and Bubblers during this step as well.

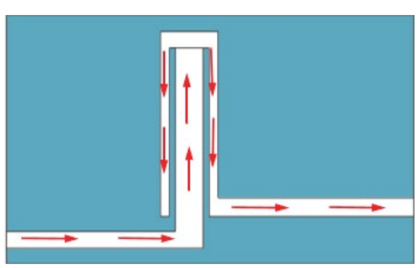


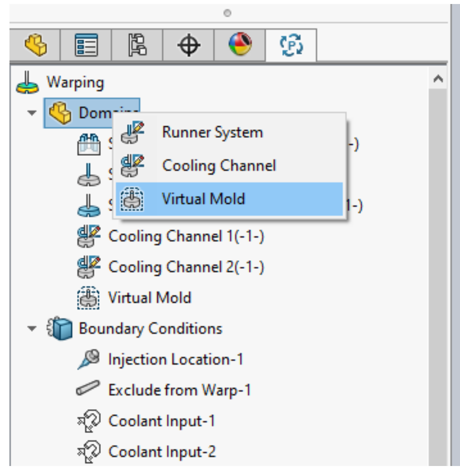

Baffle



Bubbler



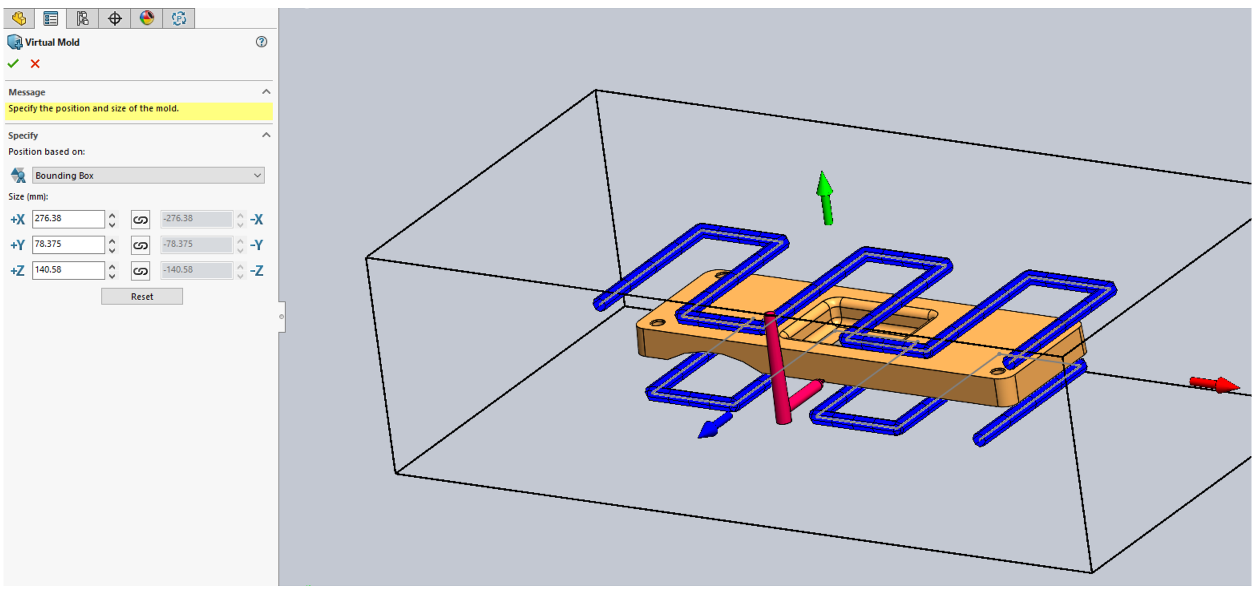
# Virtual Mold



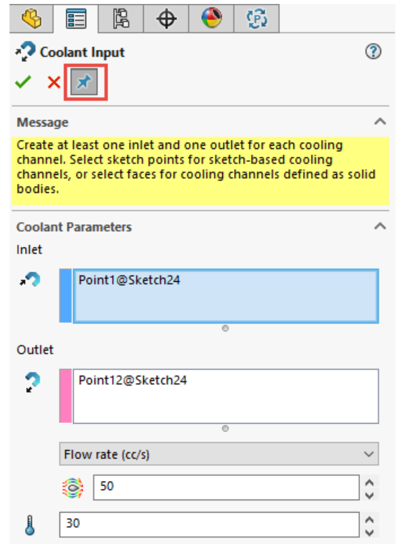

To perform a cooling analysis a **Virtual Mold** will need to be defined. Workflow is easy using a virtual mold because it eliminates the need to have a solid modeled mold core and cavity.

The **Virtual Mold** is defined in the **Domains**.

The mold is created using a bounding box that surrounds the cavity, runner, and cooling channels. This box can be adjusted to a specific size or the red, green, and blue arrows can be dragged to adjust the size.



## Coolant Input

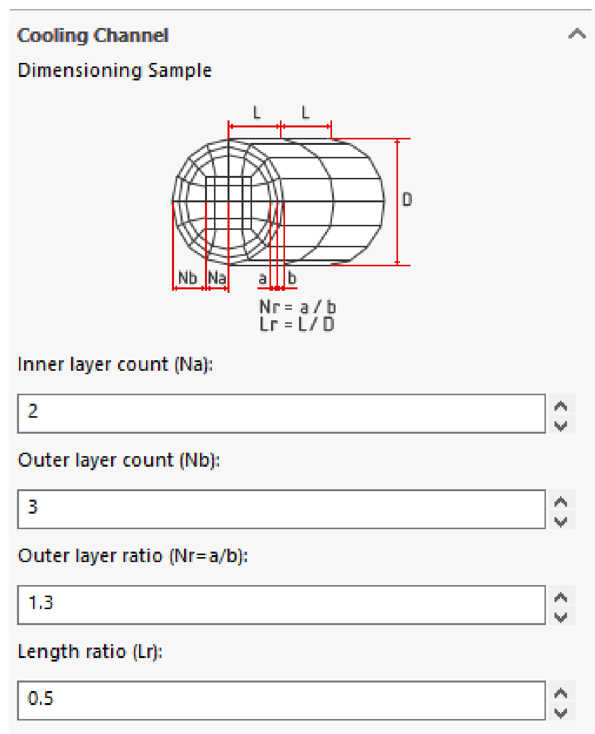



Cooling is simulated within the cooling lines with a fully three-dimensional CFD analysis. Multiple inlets and outlets can exist when simulating the coolant flow. Cooling is simulated by specifying the fluid flow rate and direction. The temperature of the fluid is also specified.

The **Coolant Input** is entered in the **Boundary Conditions**.

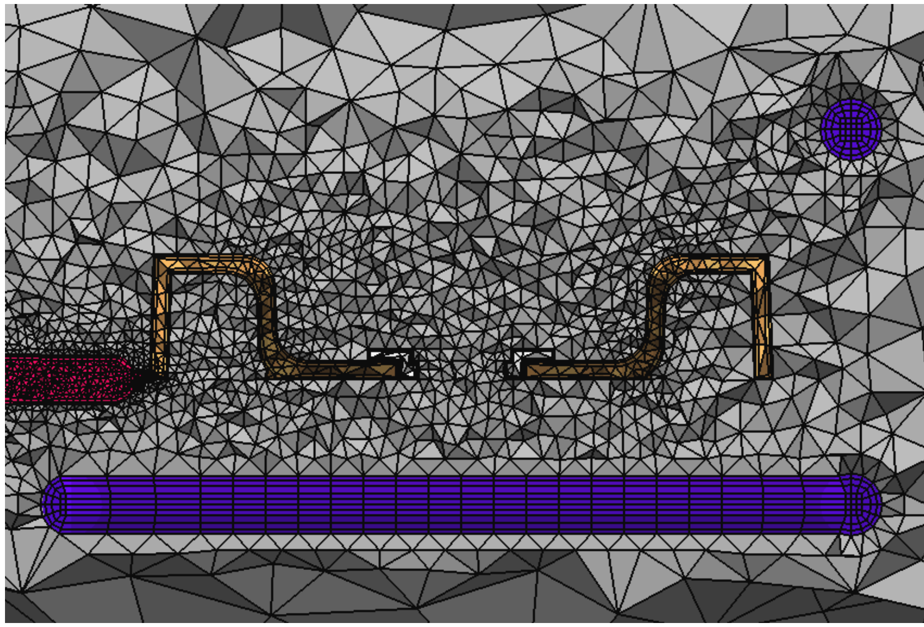
If there are multiple cooling channels, press the **pushpin** at the top of the **Coolant Input** PropertyManager. This will keep the command active until all the channel parameters are defined.

## Mesh



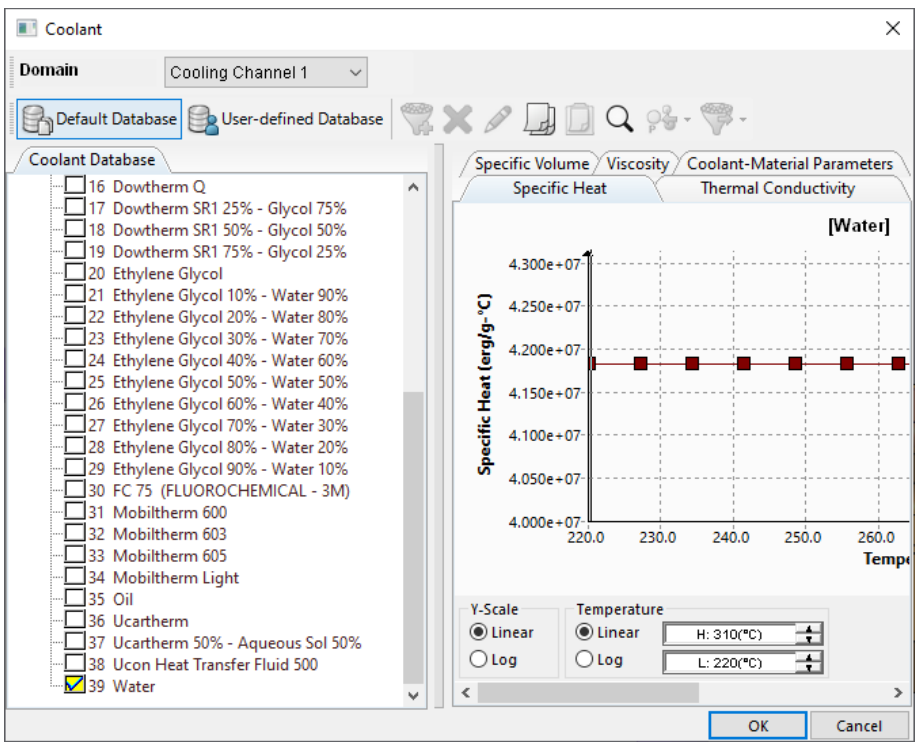

The mesh creation for an analysis with virtual mold and cooling channels still follows the same workflow. There are additional settings for the cooling channel mesh. The default settings will yield a good accurate mesh.

The mesh solver automatically refines the critical areas around the interfaces between the part, runner system, cooling channels, and virtual mold.

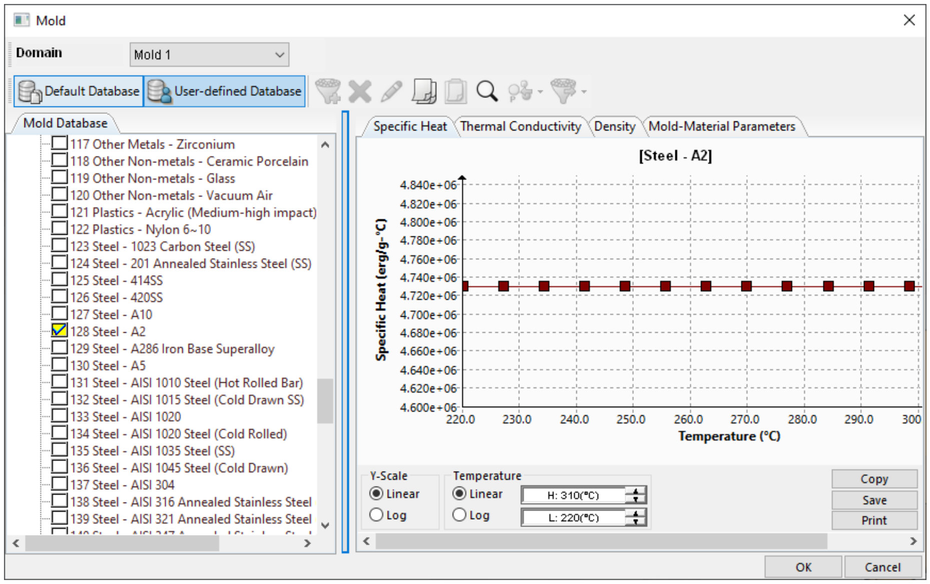


## Coolant Material

Molds are cooled with a variety of fluids including water, water/glycol mixtures, and oil. In many cases, water is the ideal fluid to use because it is inexpensive as well as having a high specific heat, meaning it can absorb a lot of energy without a large change in temperature. Coolant materials are defined through the **Coolant** command.

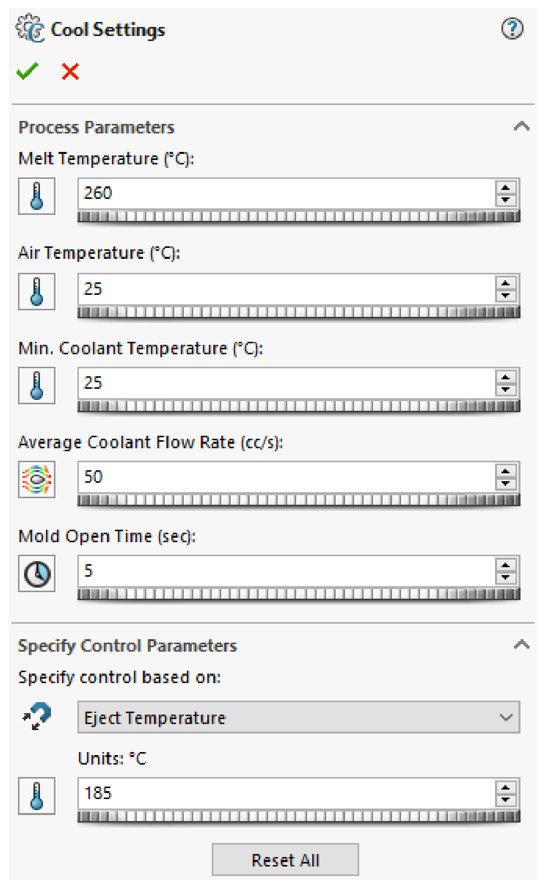


## Mold Material




Heat is conducted from the cavity to the cooling channels through the mold, therefore, the mold material heavily influences the thermal state of the system. Molds can be made of aluminum for low cycle molds or steel for high cycle molds. The mold material is defined through the **Mold** command.

## Cool Settings

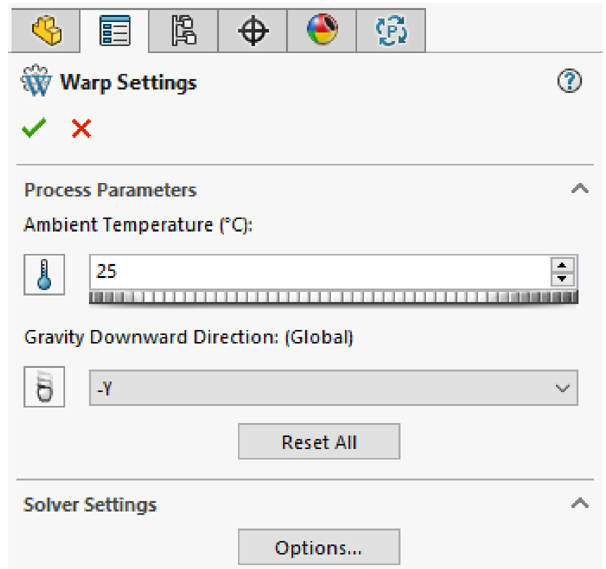



The Cool Settings control the relevant thermal properties of the system. These parameters include:

- **Melt temperature -** The temperature of the resin when the cool simulation starts.
- **Air temperature** – The temperature of the air in the molding shop.
- **Minimum coolant temperature –** The initial temperature of the coolant in each channel.
- **Average coolant flow rate -** The parameter used to specify the flow rate of coolant through the system.
- **Mold open time -** The amount of time that the mold remains open while the part is being ejected.

When the **Eject Temperature** is specified, the cooling time is assumed to be unknown and is solved for during the simulation.

## Warp Settings




The **Warp Settings** specify the conditions of the part at ejection.

**Ambient Temperature** is the environmental temperature the part experiences after it is ejected from the mold.

**Gravity Direction** is the direction in which the part sits as it cools to the ambient temperature.

## Cool Analysis

When a **Cool** analysis is performed first, it is assumed that the cavity is initially filled with resin at **Melt Temperature.** If a **Fill** analysis is then performed, the thermal settings from the **Cool** analysis will be used as input for the **Fill** analysis.

However, if a **Fill** analysis is performed first, the thermal settings from the **Fill** analysis will be used as input for the Cool analysis. Therefore, a final analysis should be run in this order:

**Fill ➡ Cool ➡ Fill ➡ Pack ➡ Warp**

## Cool Results

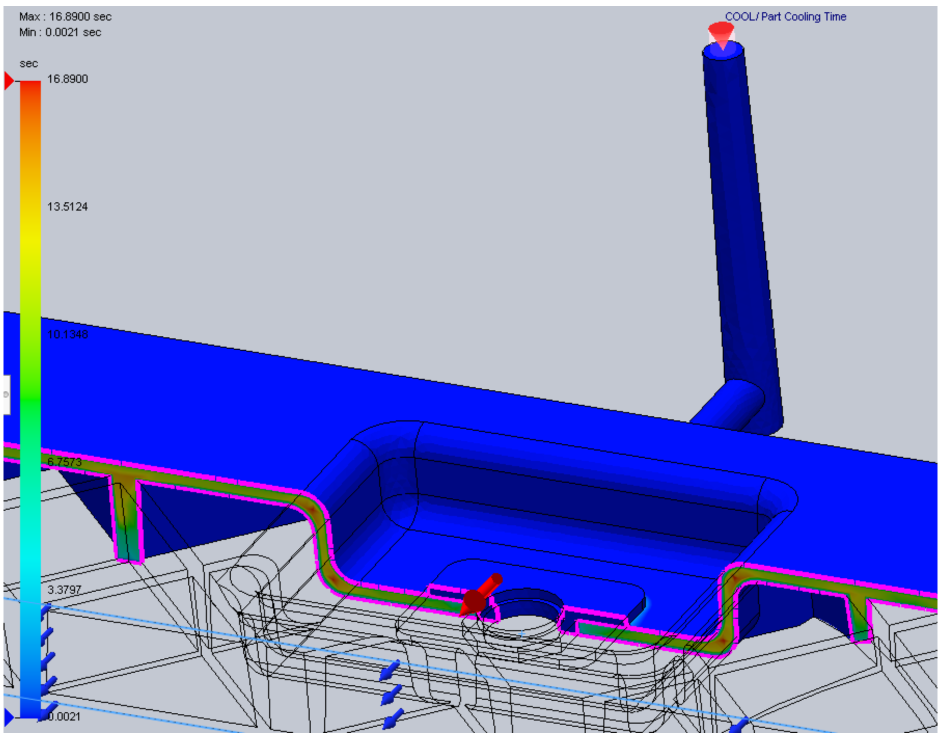
The **Cool Results** include:

- Part cooling time
- Cycle averaged temperature
- Temperature at end of cooling
- Cycle averaged part temperature
- Part temperature at end of cooling
- Cycle averaged mold temperature
- Mold temperature at end of cooling
- Cycle averaged heat flux
- Cycle heat loading

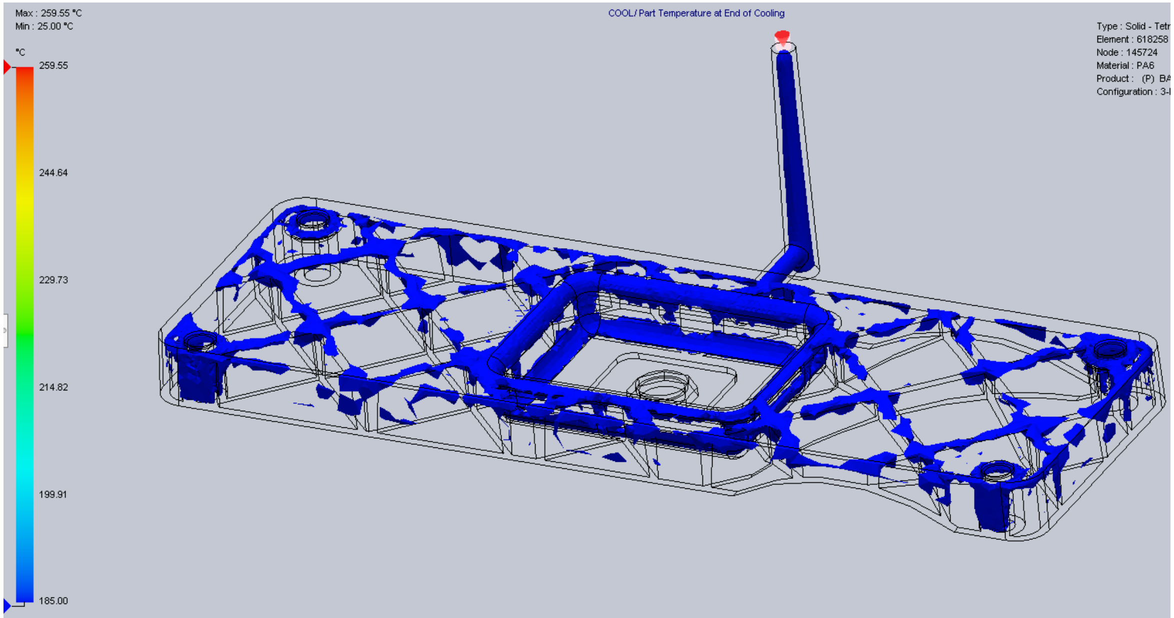
- Pressure of coolant system

The term **end of cooling** refers to a result as measured just before ejection.
The term **cycle averaged** refers to a result that has been averaged over the entire time in the mold.

**Part Cooling Time**
With the Clipping plane option active, the internal part cooling time is displayed.

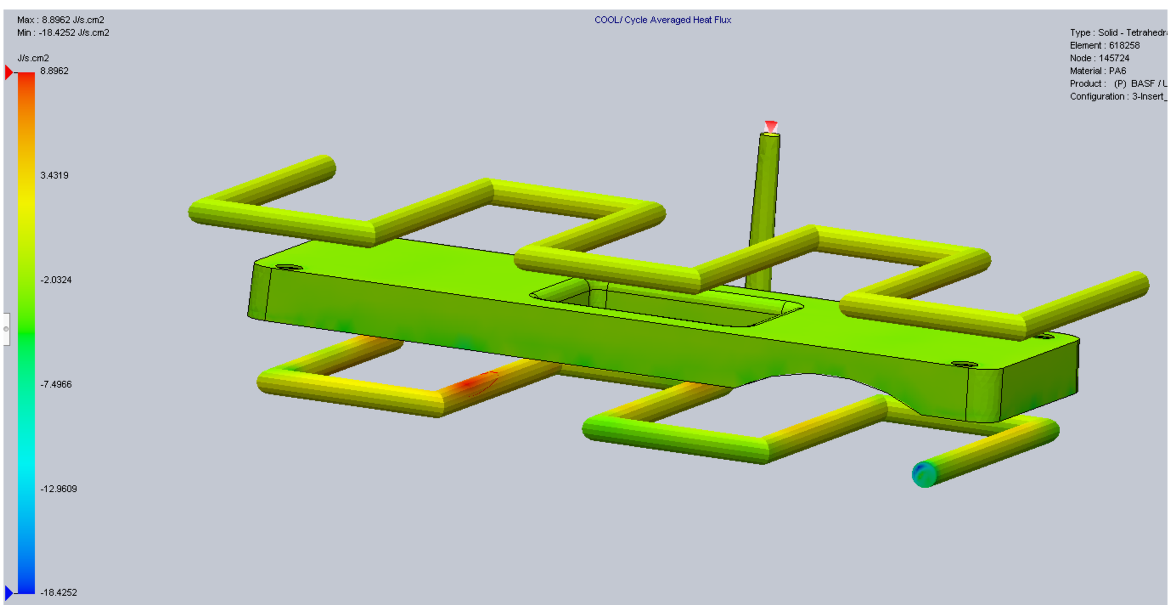



**Temperature at End of Cooling**
With Isosurface active and the minimum set to 185, the areas displayed have a temperature higher than the recommended part ejection temperature. Minimizing the thickness of these areas would reduce overall cooling time and consequently the manufacturing cycle time.




**Cycle Average Heat Flux**

This result shows where the channels are drawing the most thermal energy out of the system which could be useful if the channels need to be redesigned.

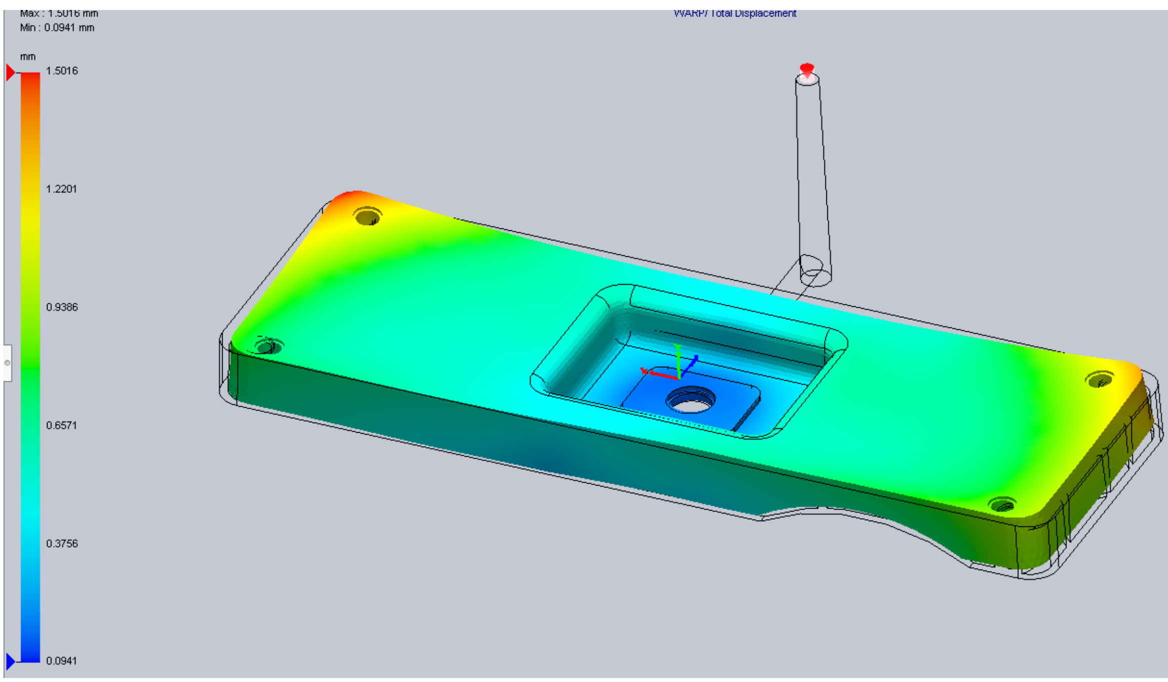


## Warp Results

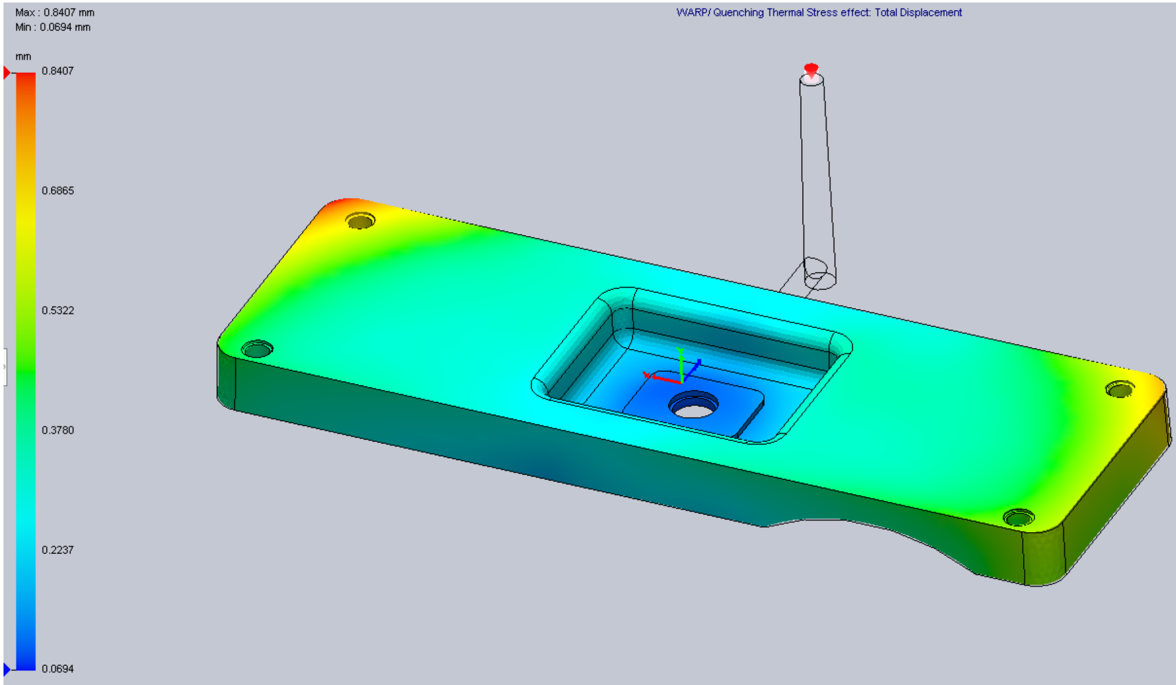
The Warp Results include:

- Total stress displacement
- In-mold residual stress displacement
- Quenching thermal stress displacement
- Total stress displacement without fiber
- Sink mark profile
- In-mold residual Von Mises stress
- Demolding residual Von Mises stress

**Total Stress Displacement**
This plot shows the total amount of the part is predicted to warp after it comes out of the mold and cools to room temperature.

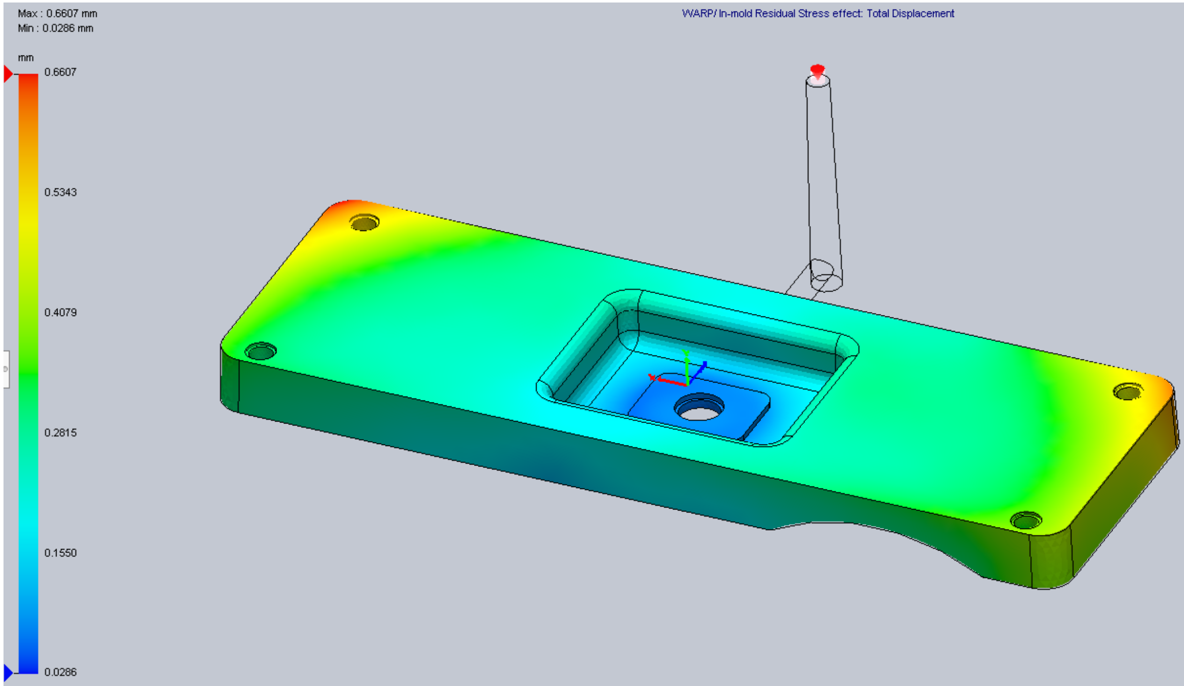



**Quenching Thermal Stress Displacement**
Shows the contribution of warpage due to temperature differences.



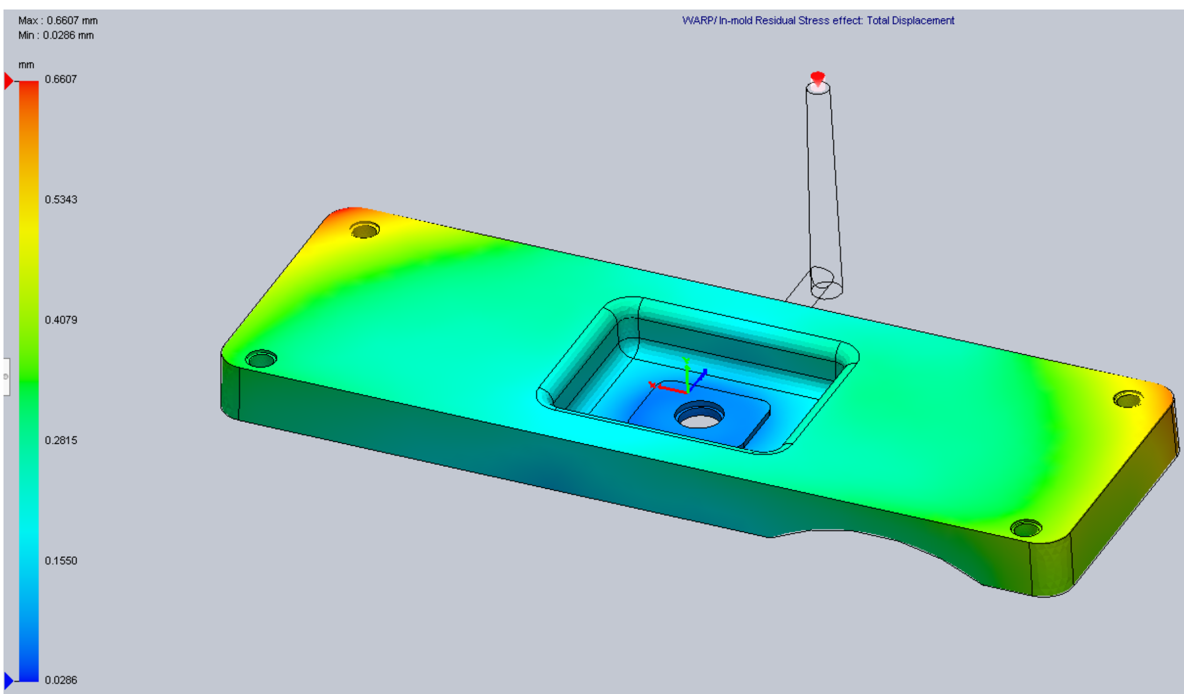

The filling and packing stages of the injection molding process occur under very high pressures. As the part cools it is eventually ejected. These pressures can have a significant effect on part warpage. The process conditions and design factors that reduce stress during cavity filling will help to reduce flow-induced residual stress.

**In-mold Residual Stress Displacement**
This plot shows the amount of displacement which is due to the in-molded stress.




## De-molding Residual Von Mises Stress

Shows the stress which remains in the part after it is ejected from the mold.



**Sink Mark Profile**

The sink mark results from the warp analysis consider the in-mold stress as well as the out-of-mold cooling stress. The sink mark profile obtained for warp analysis is, therefore, more realistic than the prediction from the flow analysis.