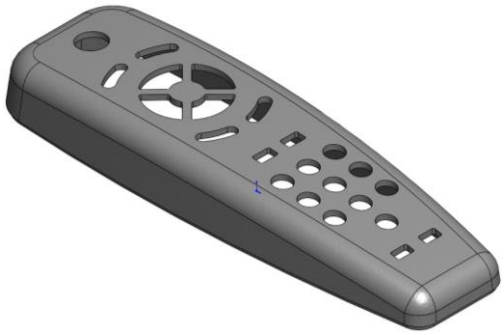
CHAPTER 12

SOLIDWORKS Plastics Flow Analysis

When a plastic part is made by injection molding, the plastic pellets are loaded into a hopper. They get heated and melted into liquid resin and then forced into a cavity to fill it (Fill Stage). This is when the shrinkage starts to take place. To minimize the shrinking, additional liquid resin is forced into the cavity under constant pressure (Pack Stage). The plastic starts to solidify in the mold, and within a few seconds when it reaches the ejection temperature (Cool Stage), it is then ejected from the mold.

This lesson will walk us through the 5 basic steps of performing the flow analysis: Injection Location, Mesh, Material, Run and View the analysis results.

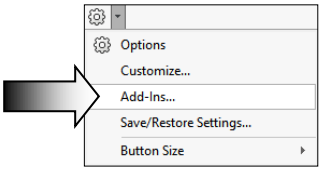
1. Opening a part document:



Click **File, Open**.

Open the part document named:
Plastics_Flow Analysis.sldprt

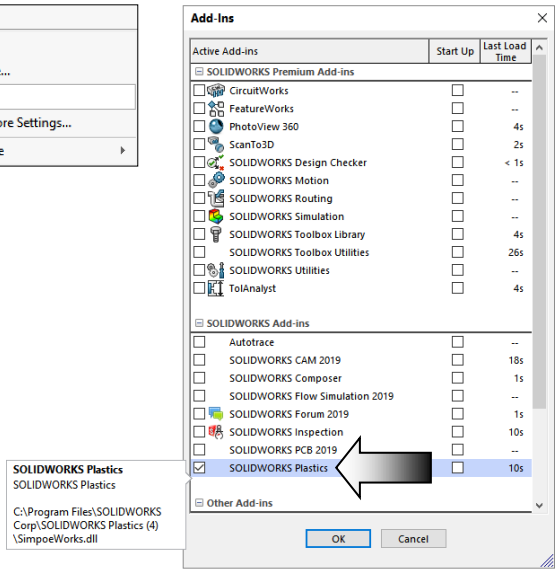
(This model has 2 configurations: With Runner & Gate and Without Runner & Gate.
The Without Runner & Gate is the active configuration).



2. Enabling Plastics:

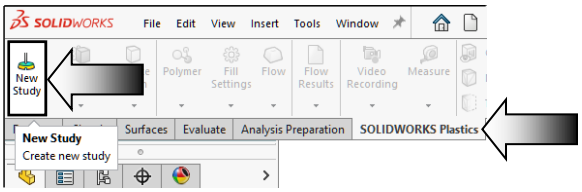
Click the drop down arrow next to the gear symbol (Options) and select **Add-Ins** (arrow).

Under SOLDWORKS Add-Ins, enable the checkbox for **SOLIDWORKS-Plastics** (arrow).



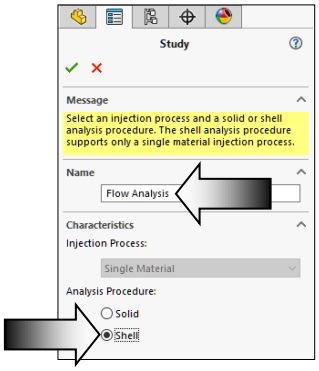
Two new tabs: **SOLIDWORKS Plastics** and **Analysis Preparation** are added to the CommandManager.

3. Setting up the Mesh:



Switch to the **SOLIDWORKS Plastics** tab and select: **New Study** (arrow).

For Study Name, enter: **Flow Analysis**.

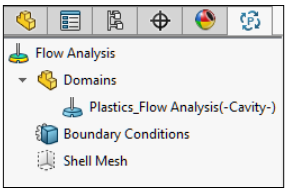


For Analysis Procedure, select: **Shell** (arrow).

Click **OK**.

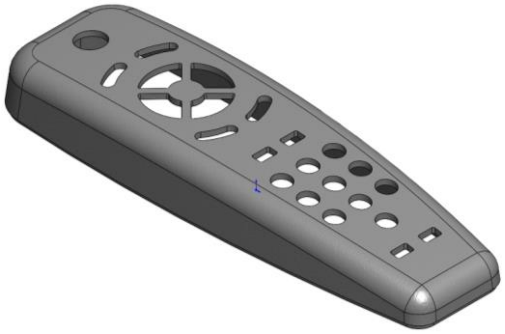
A PlasticManager tree appears on the left side of the screen.

4. Viewing the PlasticsManager tree:



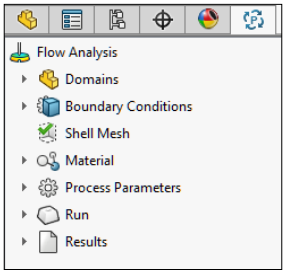
The PlasticsManager tree contains the information regarding the simulation study such as:

*** Domains**
Represents a volume in space through which you simulate a heat or fluid flow.



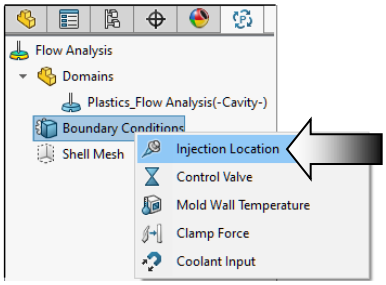
*** Boundary Conditions**
Settings for specifying the injection location.

*** Shell Mesh**
The shell analysis procedure requires a shell mesh, which is appropriate for thin-walled parts with uniform thickness.



Several other tools such as Material, Process Parameters, Run, and Results will be added to the PlasticsManager tree after the Shell Mesh is created.

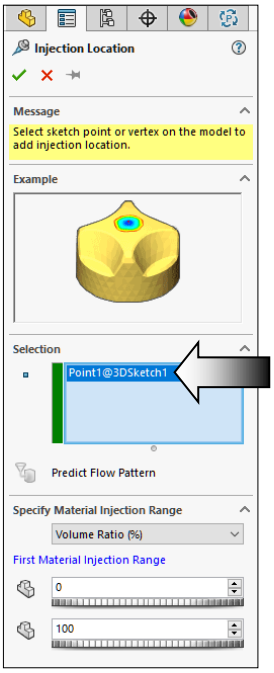
5. Adding an Injection Location:

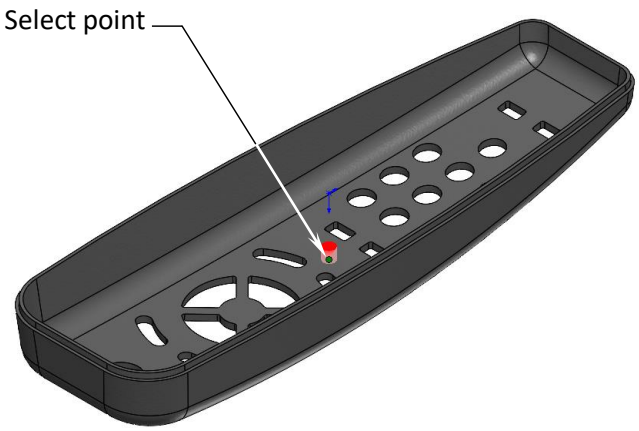


Injection locations introduce polymer material at the specified melt temperature into the cavity.

Expand the **Boundary Condition** feature and double click on **Injection Location** (arrow).

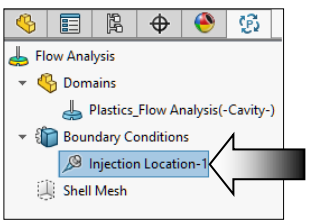
For injection location, select the <u>sketch point</u> as noted (3D Sketch1).





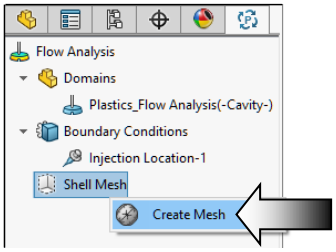
(A red conical pointer appears at the injection location. Its diameter can only be changed using the solid analysis procedure).

Click **OK**.



The Injection Location is captured and saved under the Boundary Conditions section.

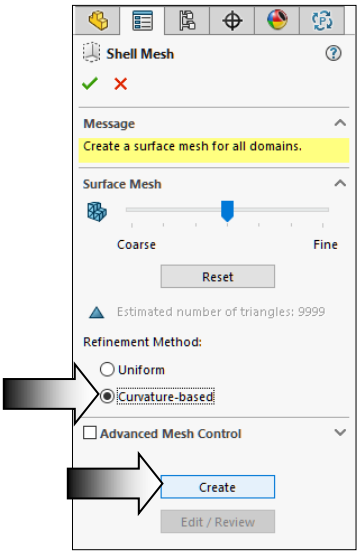
6. Creating a mesh:

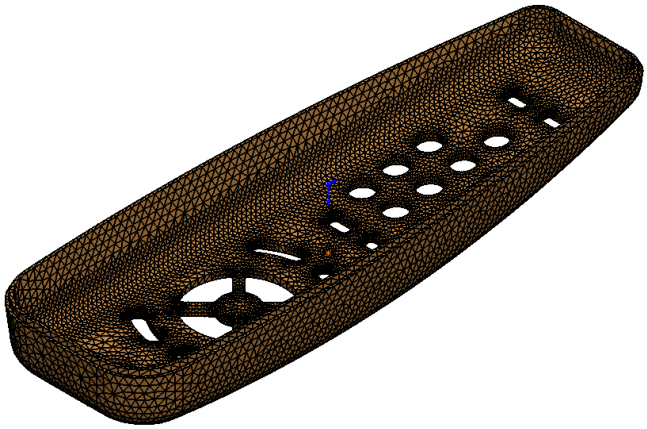


A mesh subdivides each domain of the simulation model, the cavity, runner system, cooling channel, inserts, and mold, into discrete cells.

Within each cell, SOLIDWORKS applies the appropriate conservation equations. The conservation equations compute the flow of melted polymer and heat, simulate phase change as the melt cools, and predict residual stresses and their effect on the unconstrained part shape.

For Surface Mesh, use the **default** mesh density.



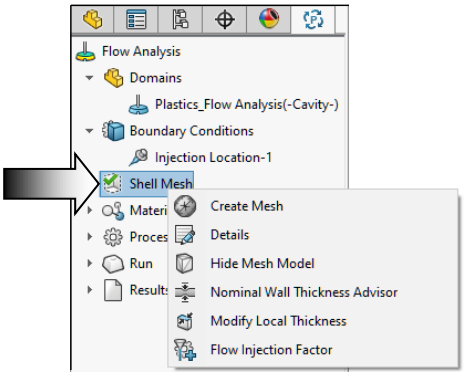


For Refinement Method, select **Curvature-Based**.

Click **Create**.

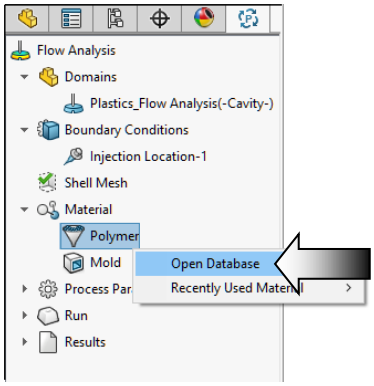
Click **OK**.

A green check mark on the Shell Mesh node indicates that a shell mesh is created.



After a shell mesh is created, several commands appear under the Shell Mesh node. Right-click on the Shell Mesh icon to access these commands.

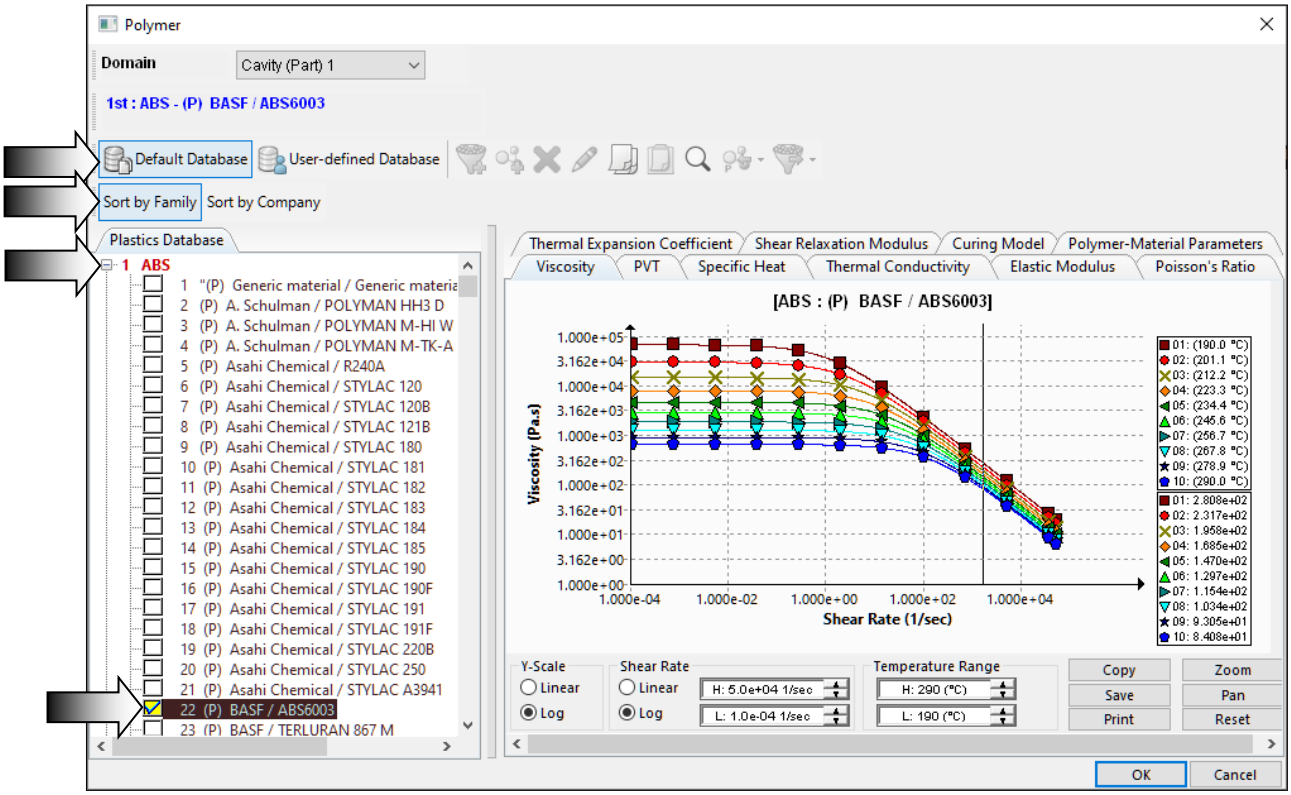
7. Selecting a Polymer:



The default material data base offers thousands of materials and the polymers are organized by the family and company. They can be applied to the mold cavities and mold inserts.

From the **PlasticsManager** tree, expand the **Material** option and double click on **Polymer** to Open Database.

Select the following:
Default Database, Sort by Family, ABS, 22 (P) BASF / ABS6003 (arrows).

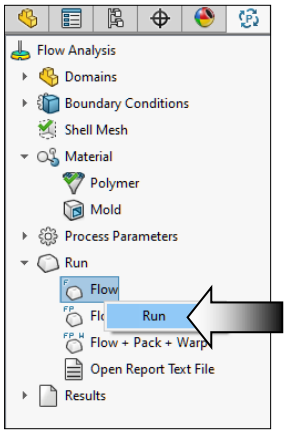


Click **OK**.

Depending on the material selected for the cavity, insert, mold, or cooling channel domain, the complete list of material properties appears in the selected material's dialog box.

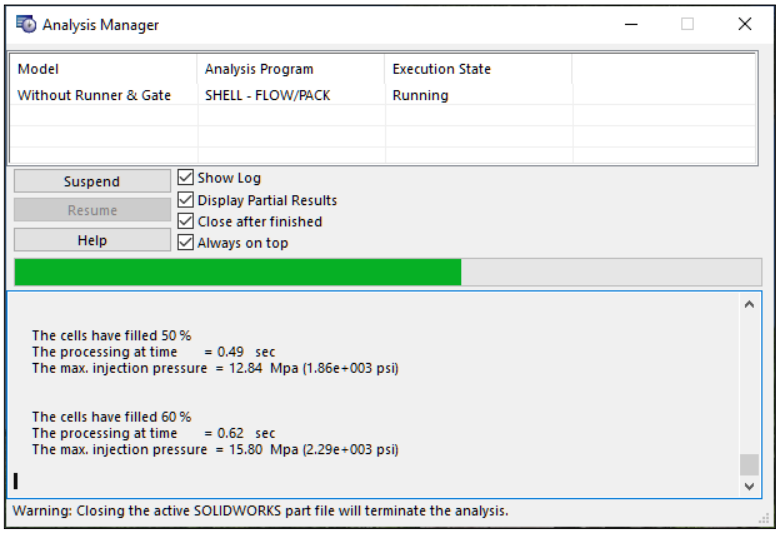
At this point, the information needed to run an analysis is completed. The next step is to run the flow analysis and interpret the results.

8. Running the Flow Analysis:

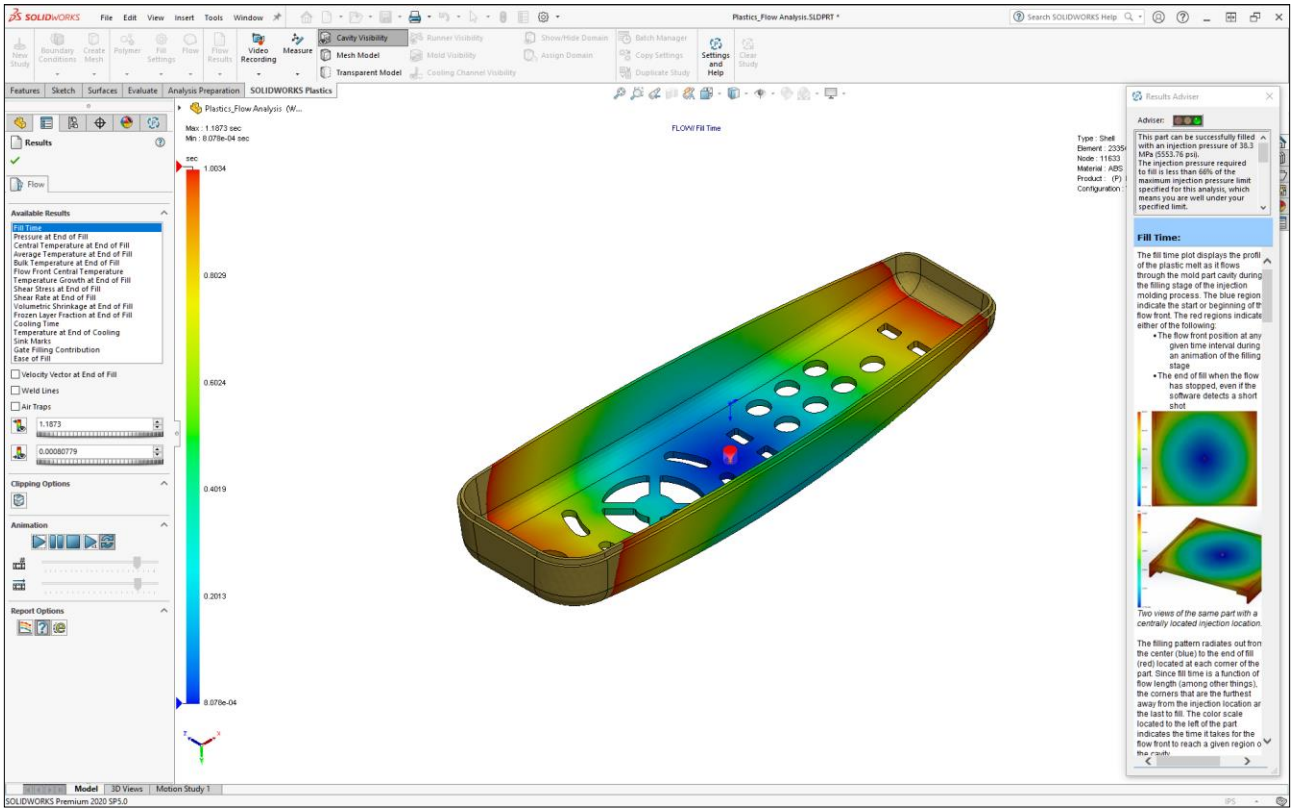


From the **PlasticsManager** tree, expand the **Run** feature. Right-click **Flow** and select: **Run** (arrow).

The Analysis Manager dialog-box appears and the analysis is started. This analysis may take several minutes.

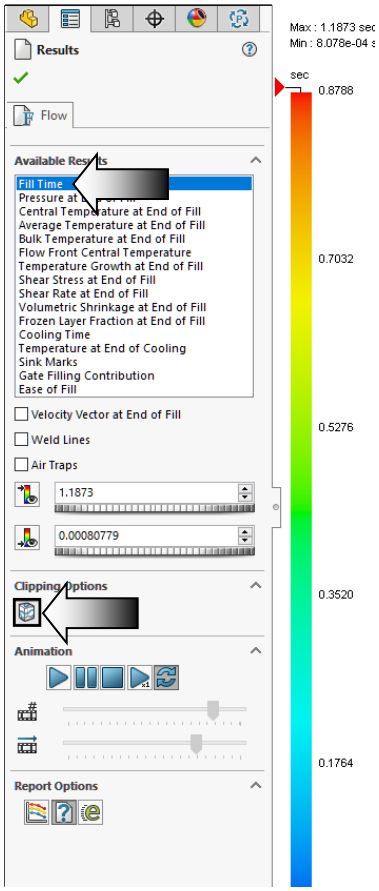


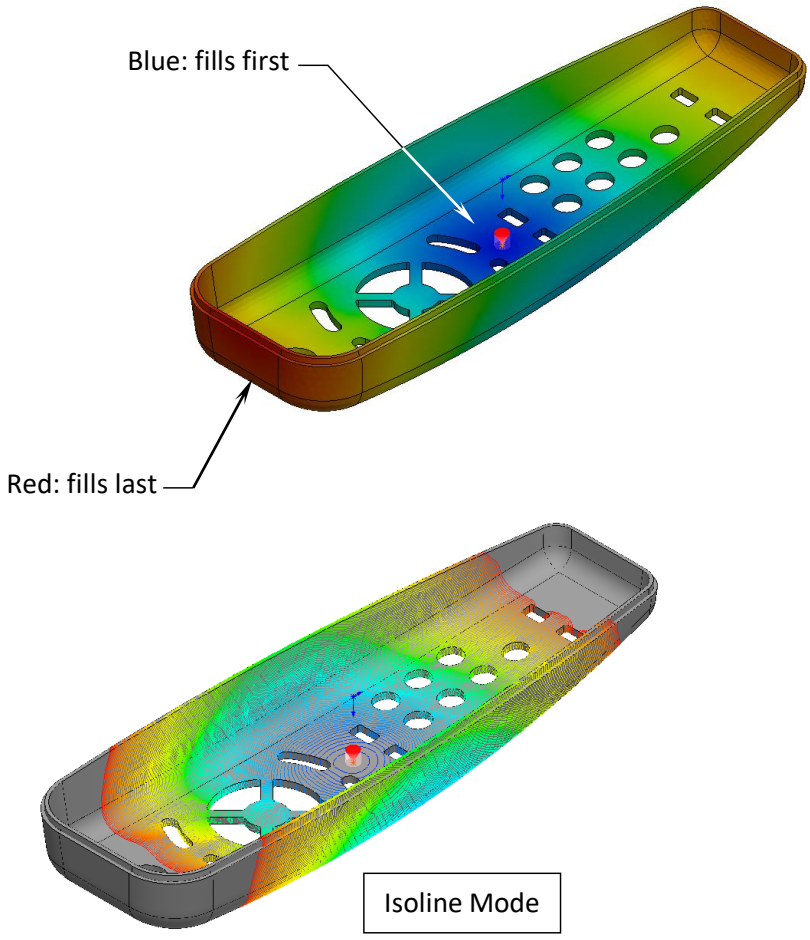
When the analysis is completed, the Flow Results are displayed on the left side of the Plastics-Manager tree. We will take a look at some of them in the next steps.



9. Viewing the Fill Time plot:

Use the **Fill Time** plot to view the profile of the liquid plastic as it flows through the cavity of the mold. The **blue** color regions are the first areas to fill and the **red** are the last one to fill.

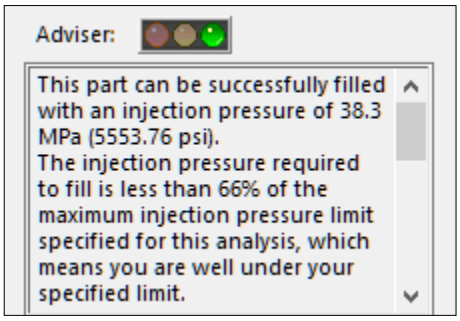




The Isoline Mode available for Shell Mesh, indicates where particular value is constant.

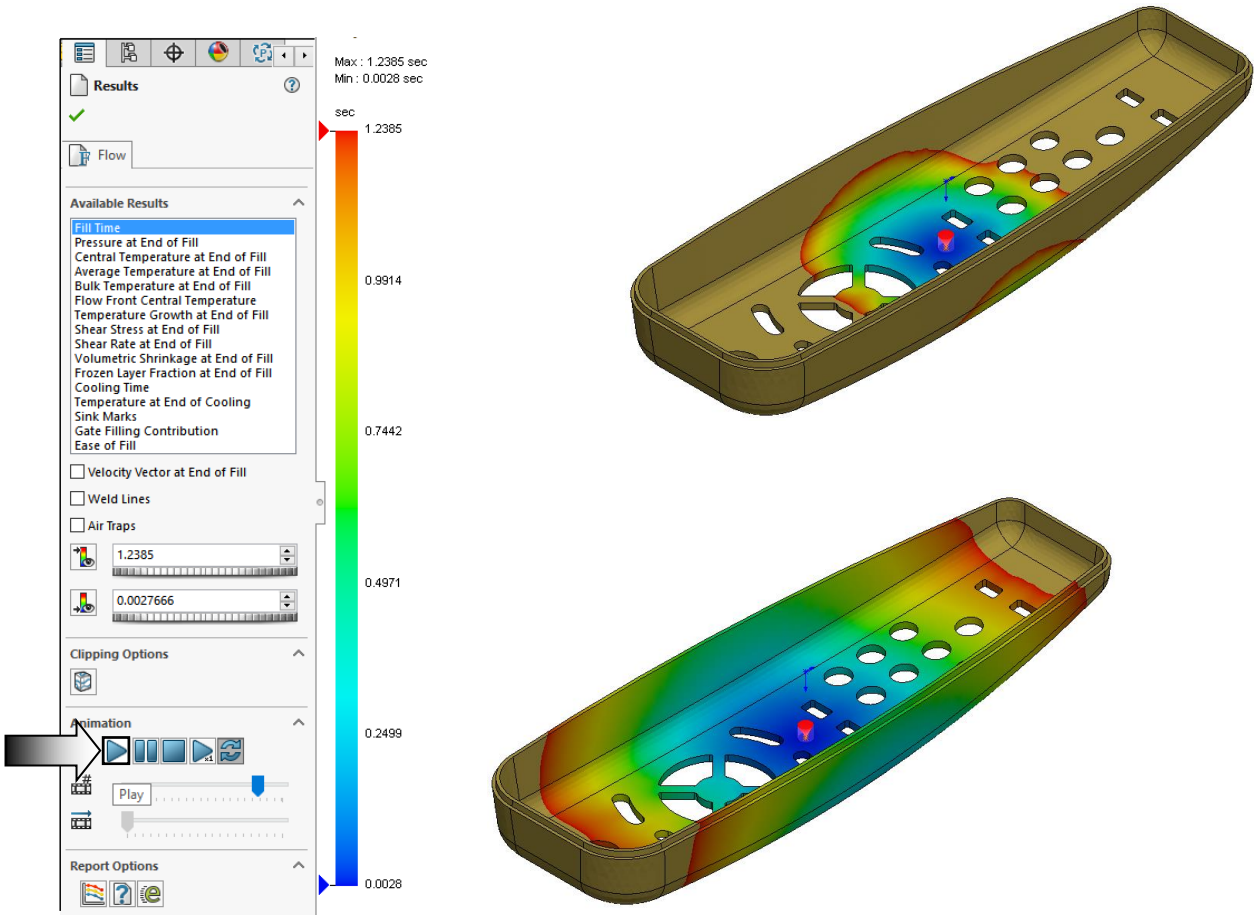
Under **Clipping Options**, click the **Isoline** button (arrow).
The Isoline Mode only available for Shell Mesh.
Isoline plots the regions of plastic material where results fall within the range of min and max values.

The maximum and minimum values of the results shown on the active plot and the **Results Adviser** appears on the right side indicating the part can be successfully filled with an injection pressure of 38.3 MPa (5553.76 psi).



10. Animating the results:

In the same **Results** window, under **Animation**, click **Play** (arrow).



The melt flow front position is animated showing the filling stage. Use the tools in the Animation section to control the animation speed, to pause, stop, or loop the animation.

Starting from the Injection Location, the **blue** color areas get filled first and the **red** color areas are filled last.

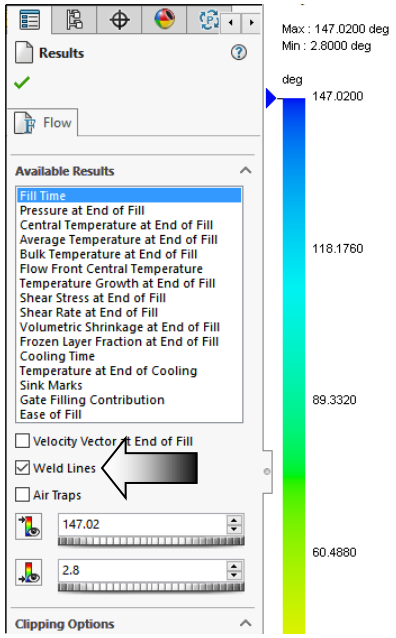
Click the **Stop** (square) button to stop the animation but keep the Results dialog box open for the next step.

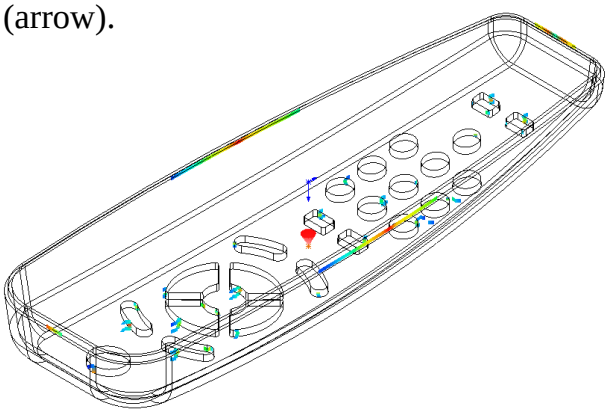
11. Displaying the Weld Lines:

Weld Lines are formed when two or more flow fronts come together. They appear when there are multiple injection locations or multiple wall thicknesses in the part. To avoid the weld lines, either move the injection locations or make changes in the plastic part, but they cannot be eliminated if there are through holes in the part.

Click **Control + 6** to change to the bottom orientation.

Click the **Weld Lines** checkbox (arrow).

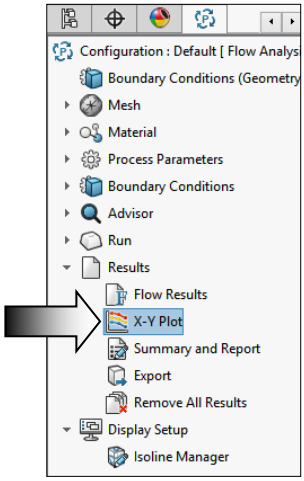


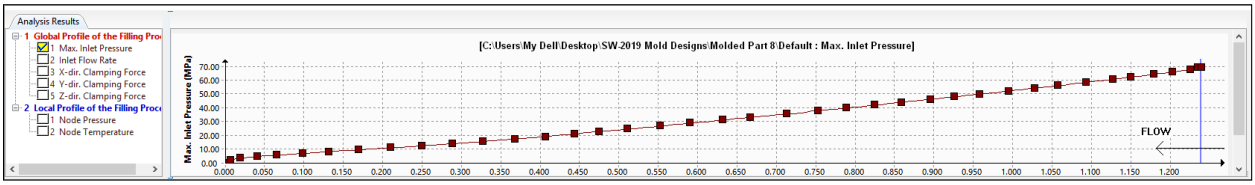


The plot displays the weld line locations and the angle of the flow field as the weld line form.

Click **OK** to close the Flow Results.

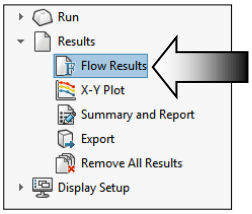
Under Results, double-click the **X-Y Plot** (arrow).
The X-Y plots are used to visualize key results such as the evolution of the Clamp force as a function of the cycle time, or to view pressure buildup at the injection point.





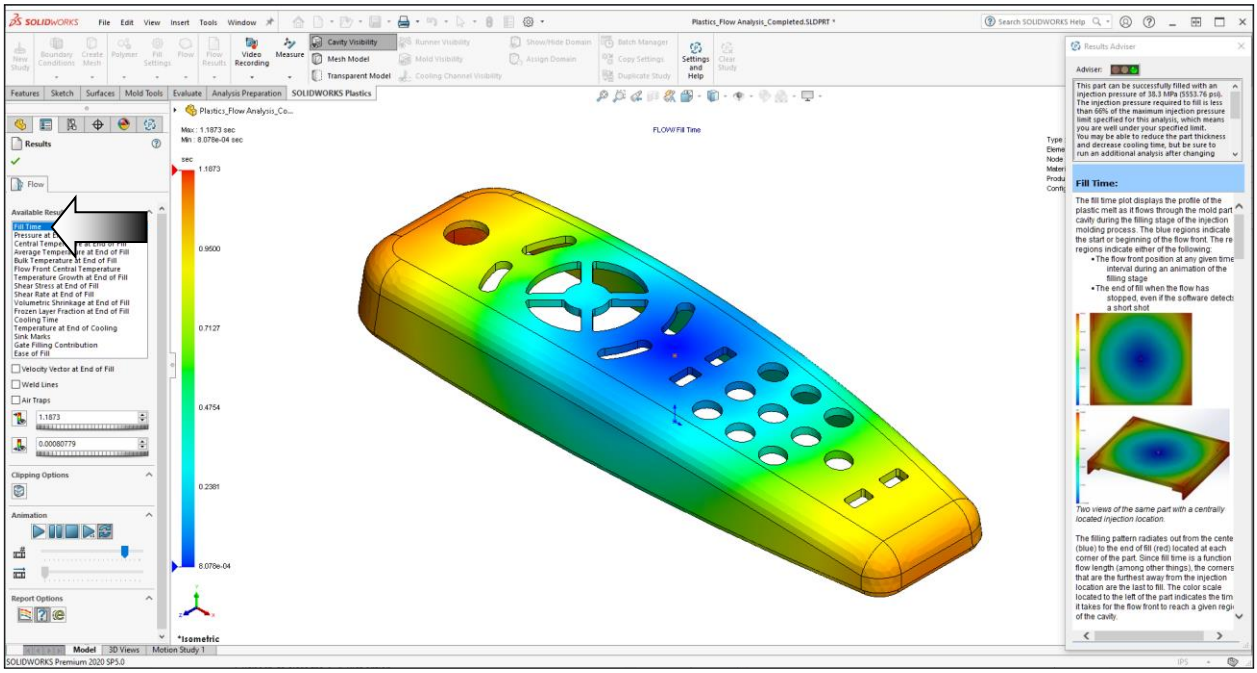
Click **OK** to close the X-Y Plot option.

12. Viewing the results:

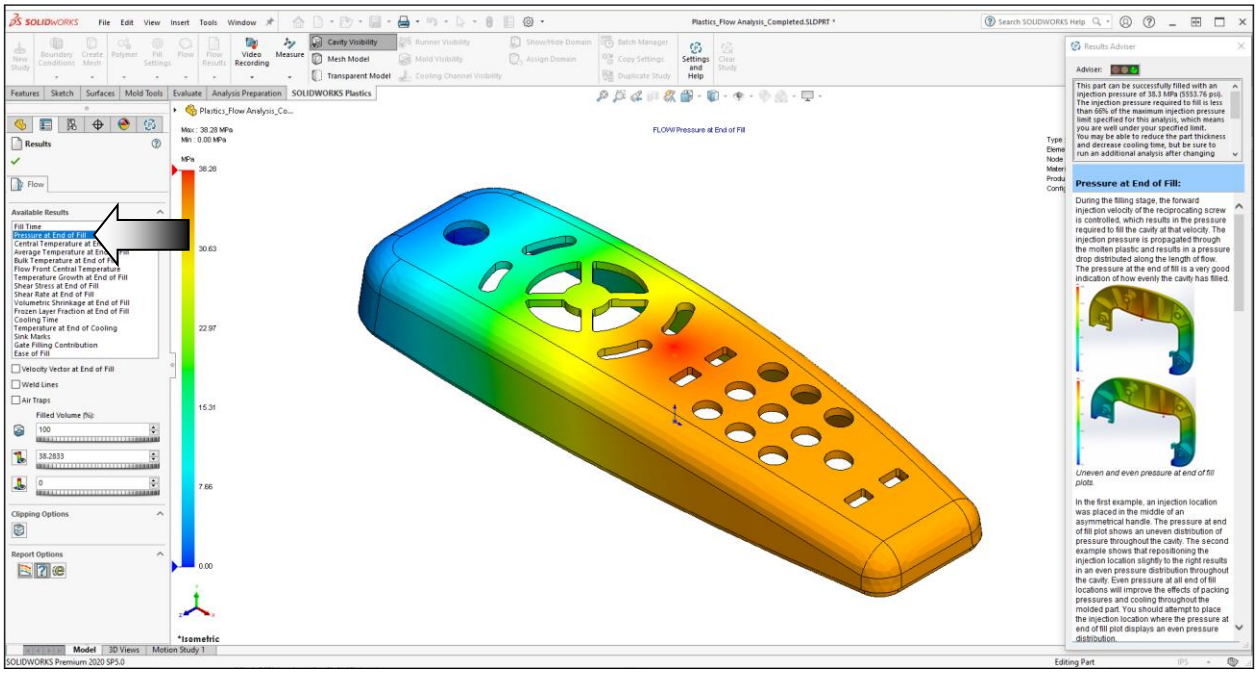


Double-click on **Flow Results** to see all other result plots.

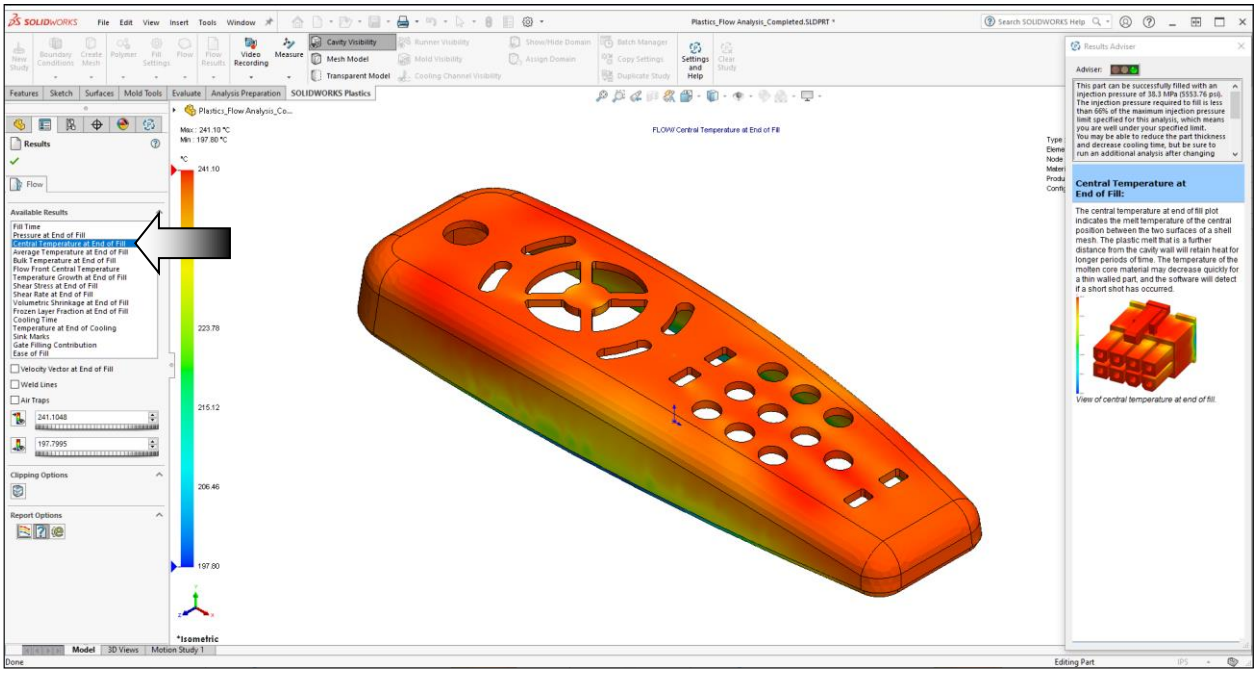
The Fill Time results



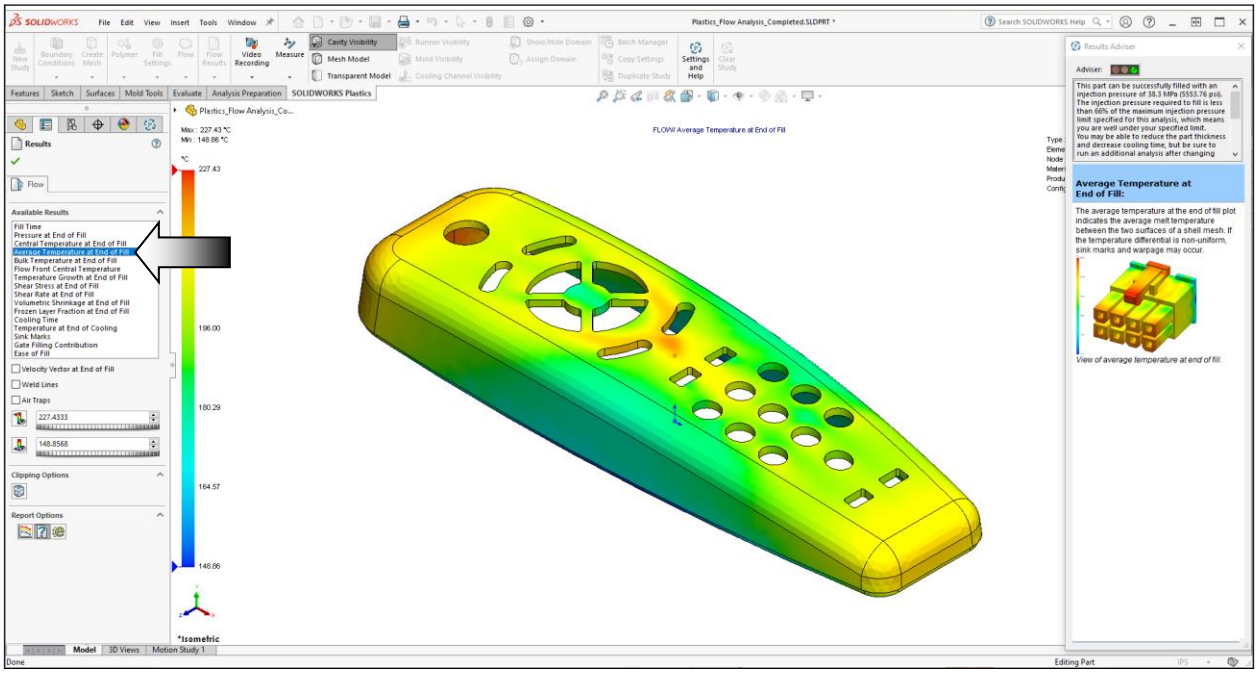
The Pressure at End of Fill results



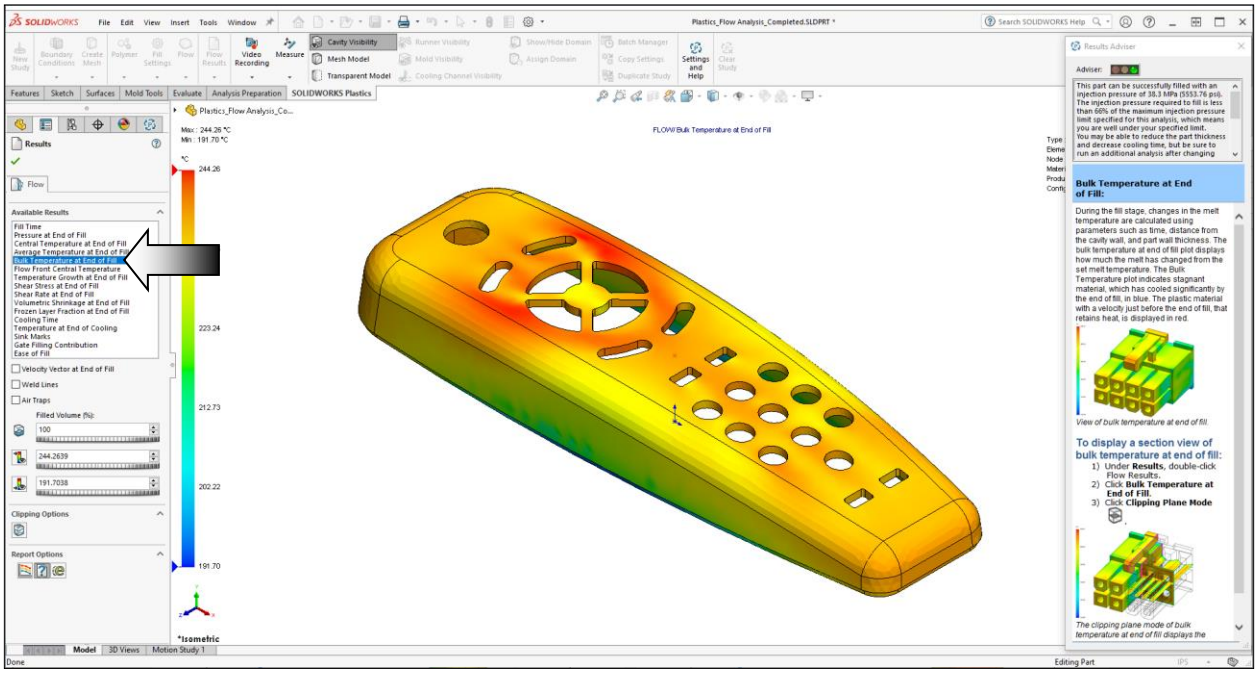
The Central Temperature at End of Fill results



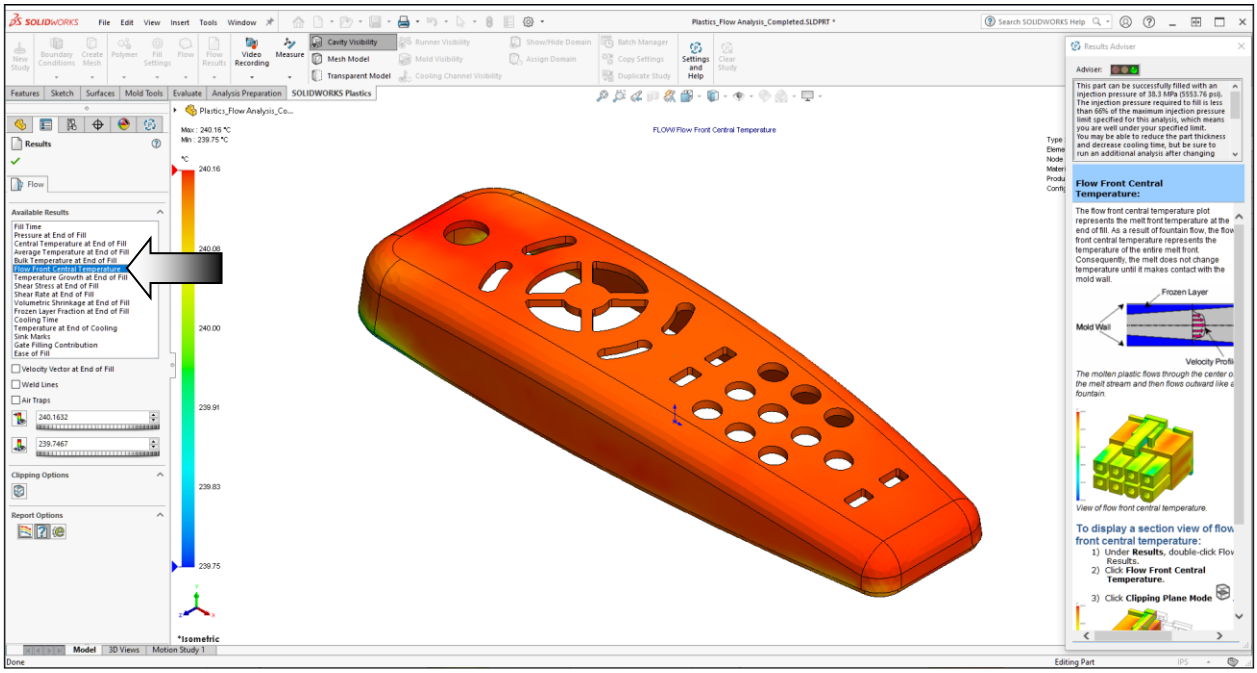
The Average Temperature at End of Fill results



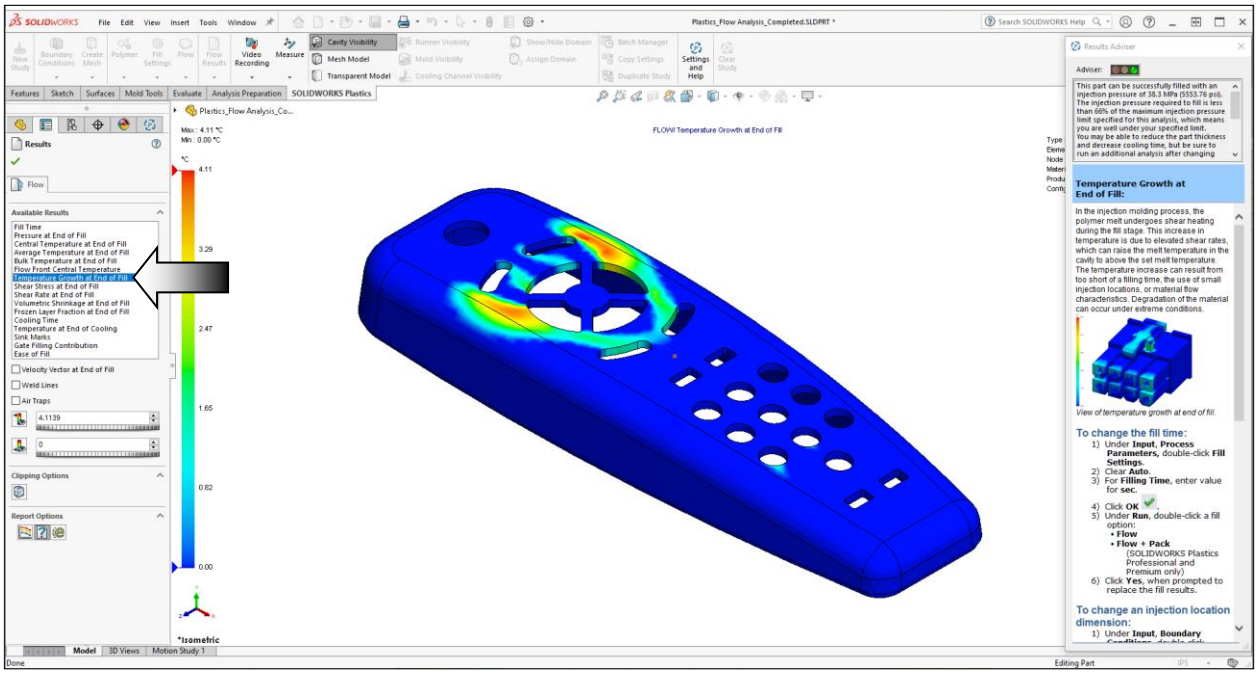
The Bulk Temperature at End of Fill results



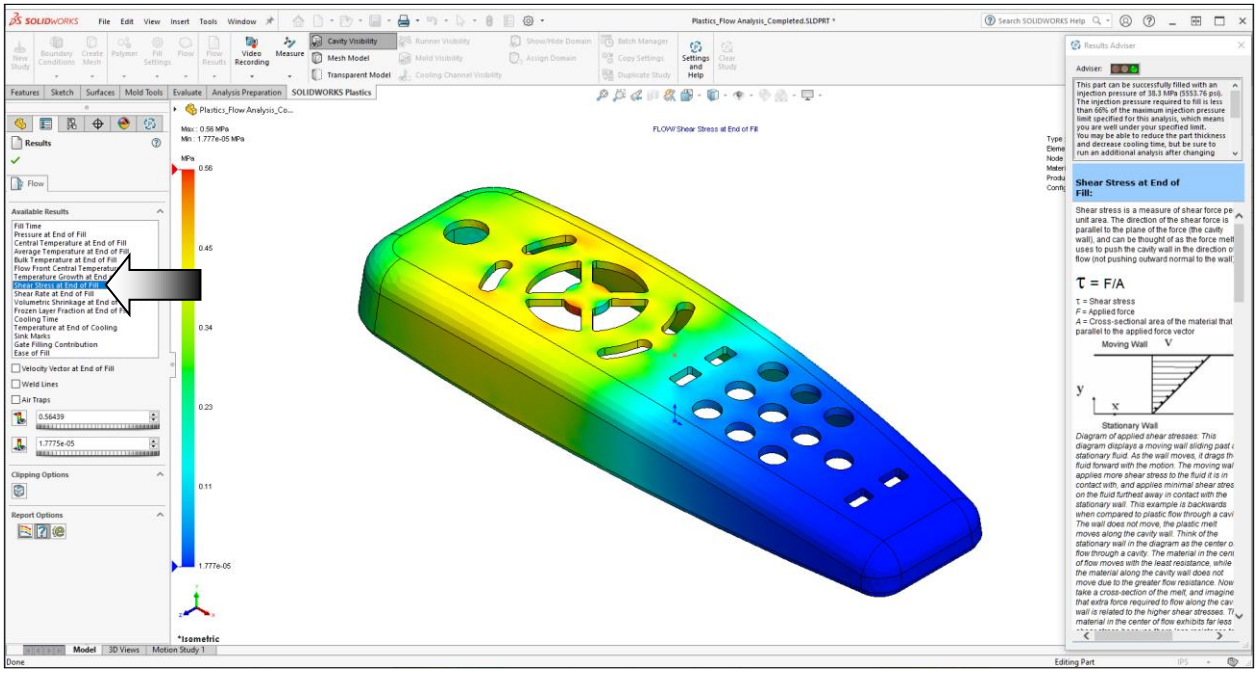
The Flow Front Central Temperature results



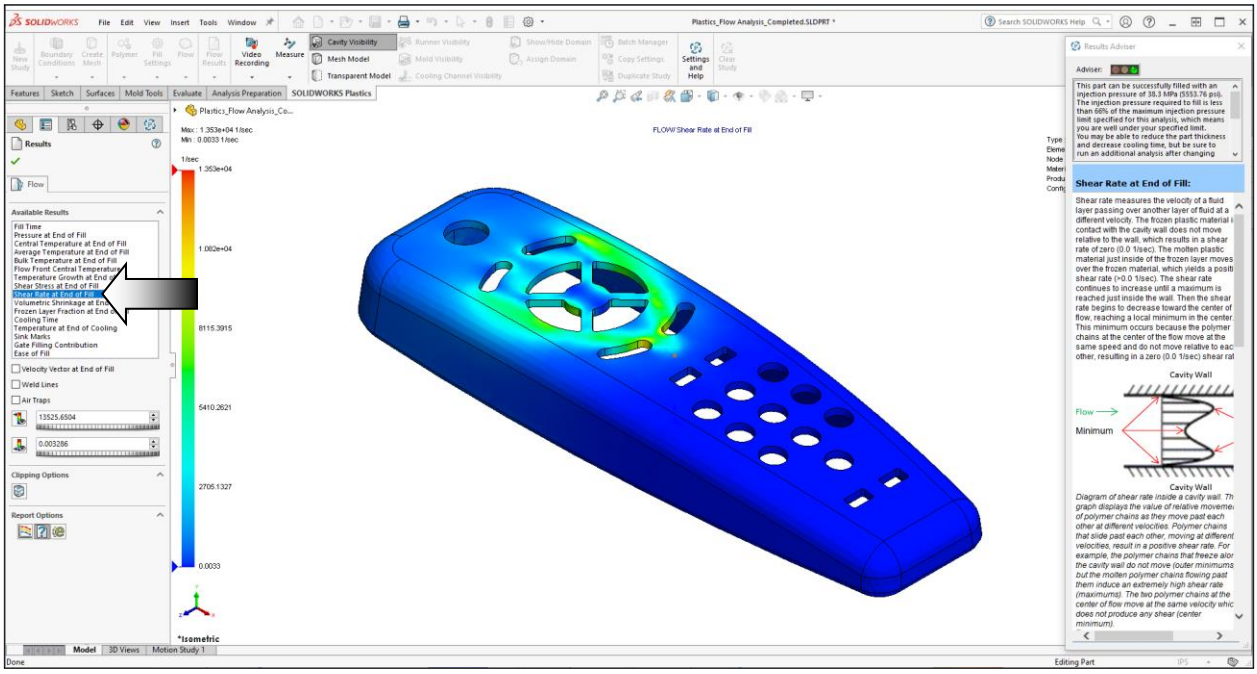
The Temperature Growth at End of Fill results



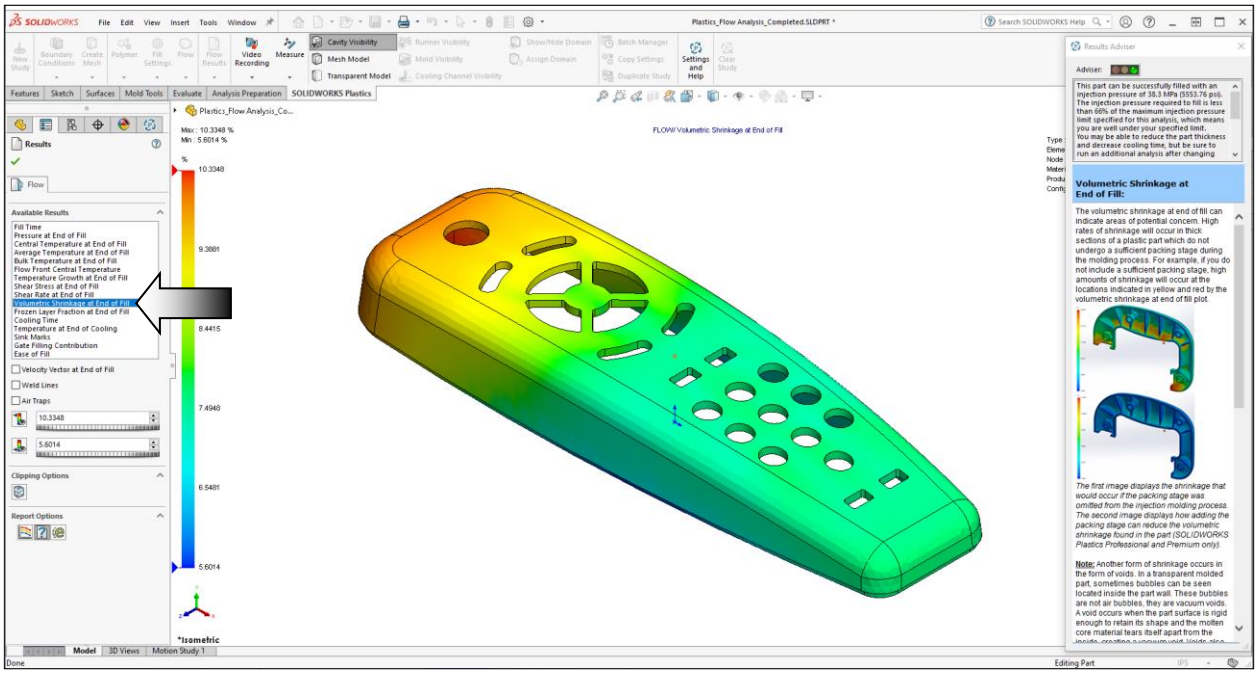
The Shear Stress at End of Fill results



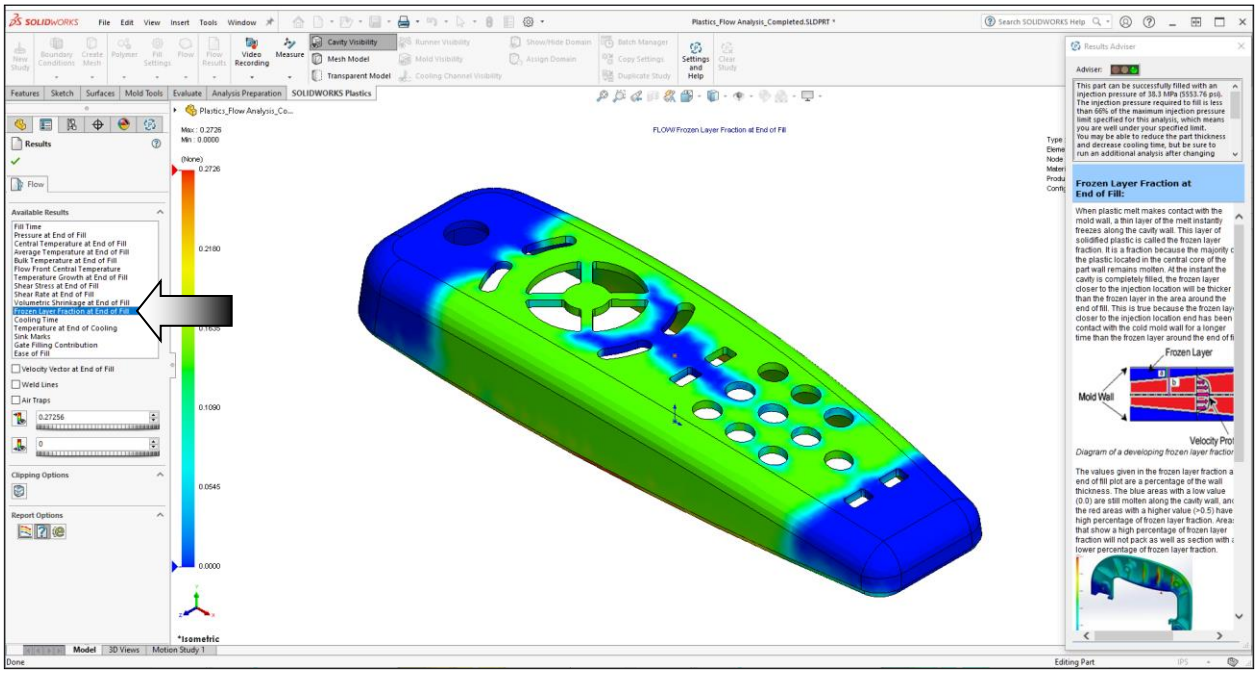
The Shear Rate at End of Fill results



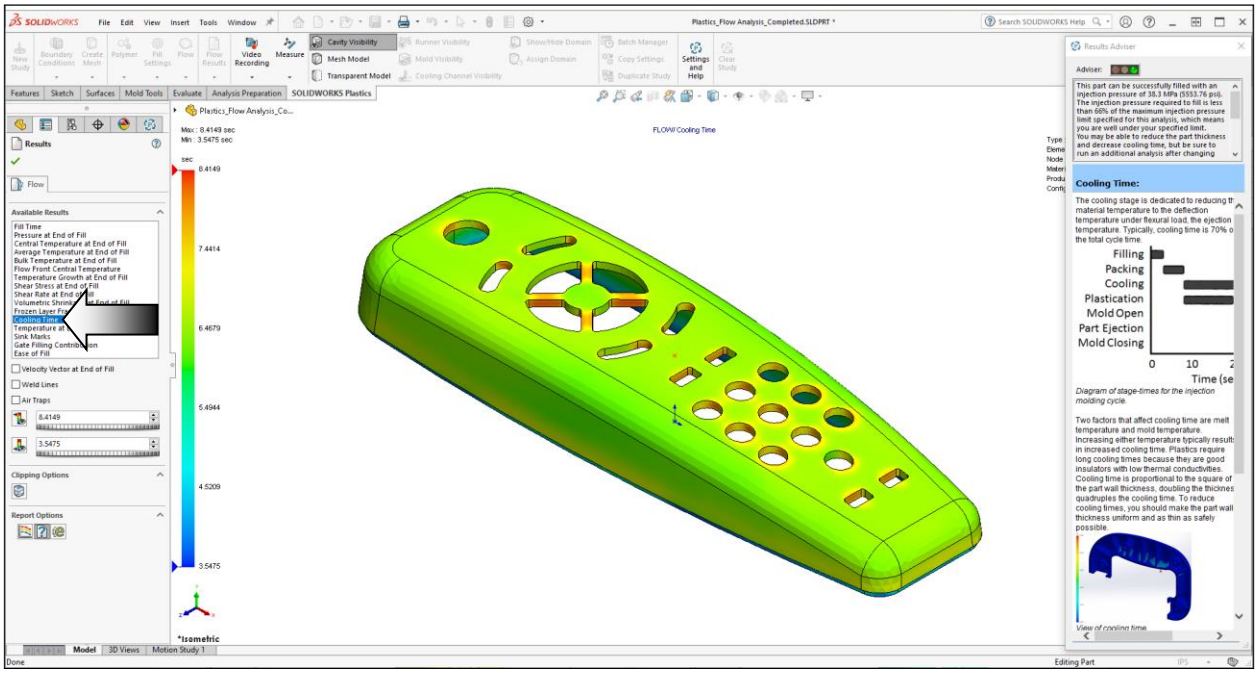
The Volumetric Shrinkage at End of Fill results



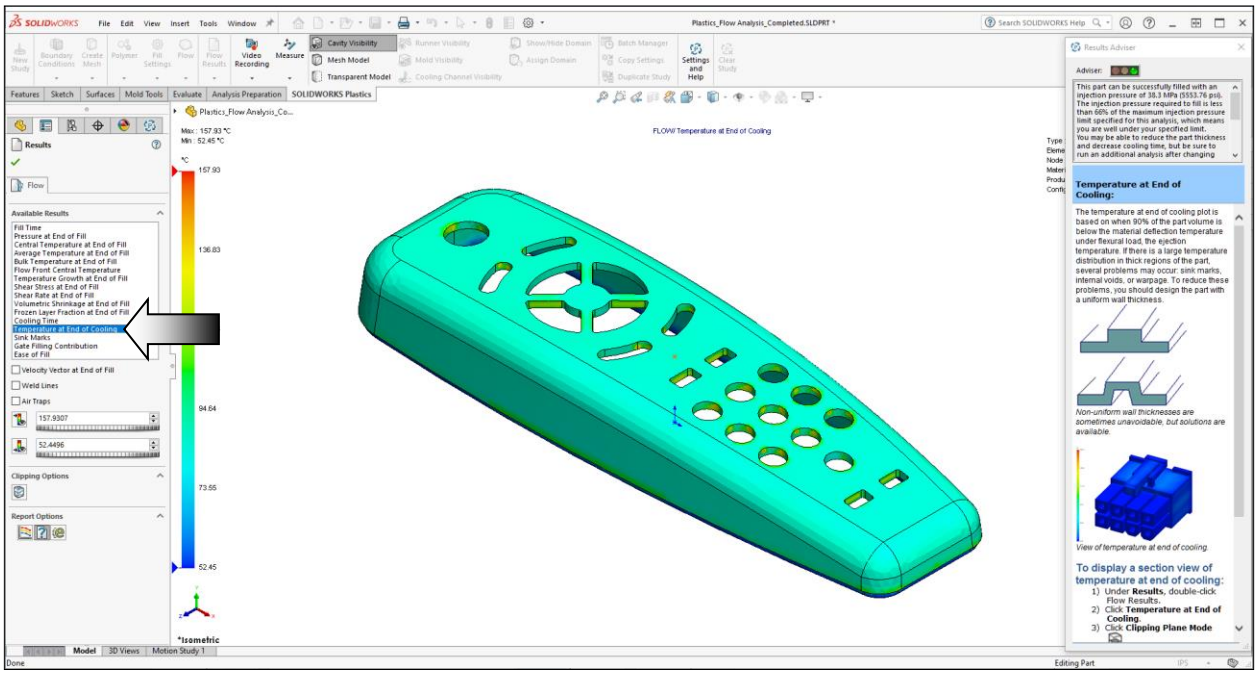
The Frozen Layer Fraction at End of Fill results



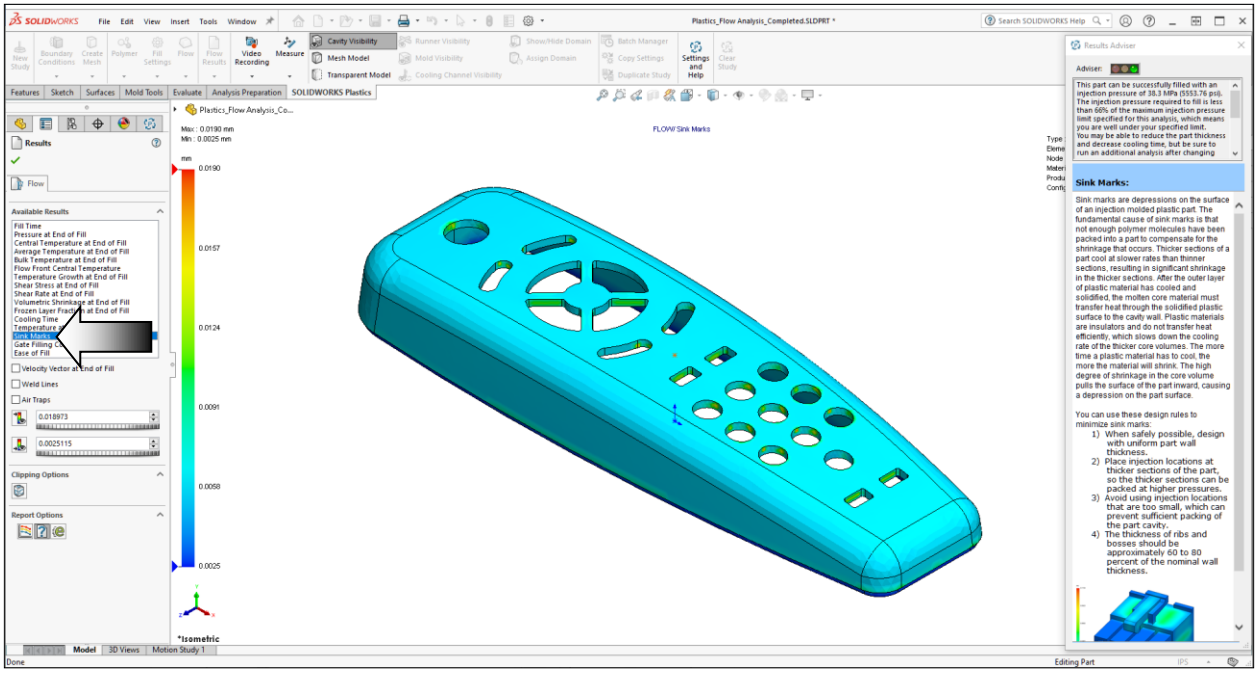
The Cooling Time results



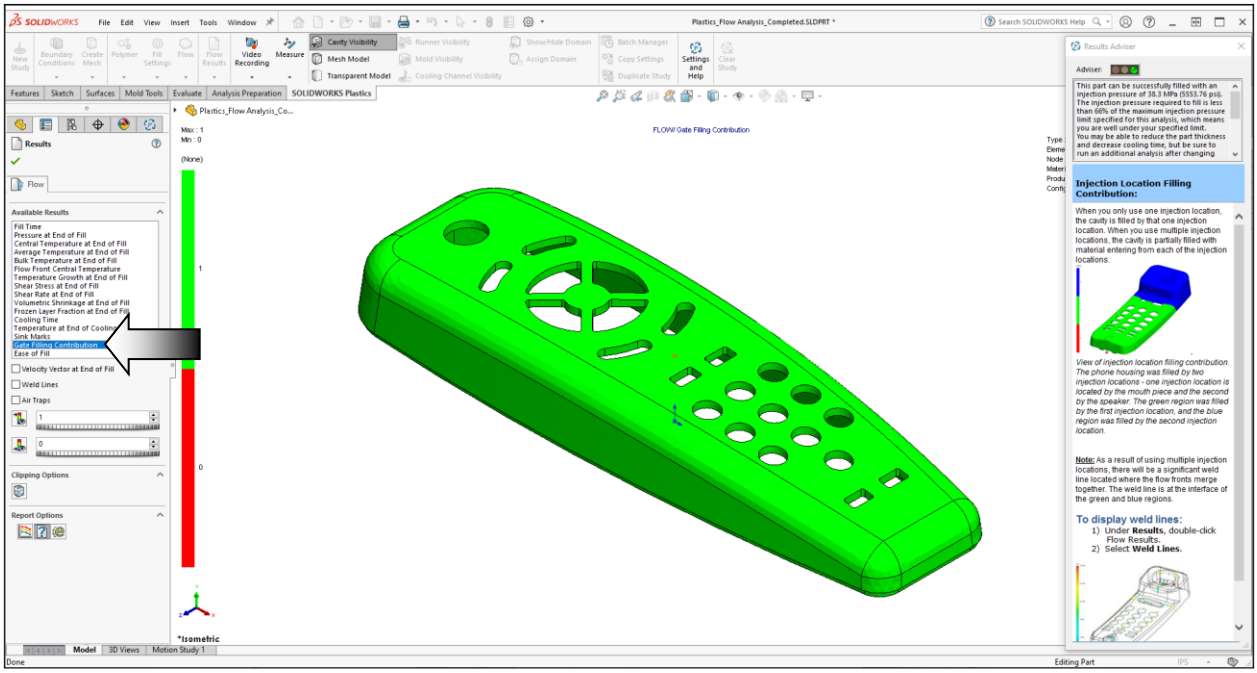
The Temperature at End of Cooling results



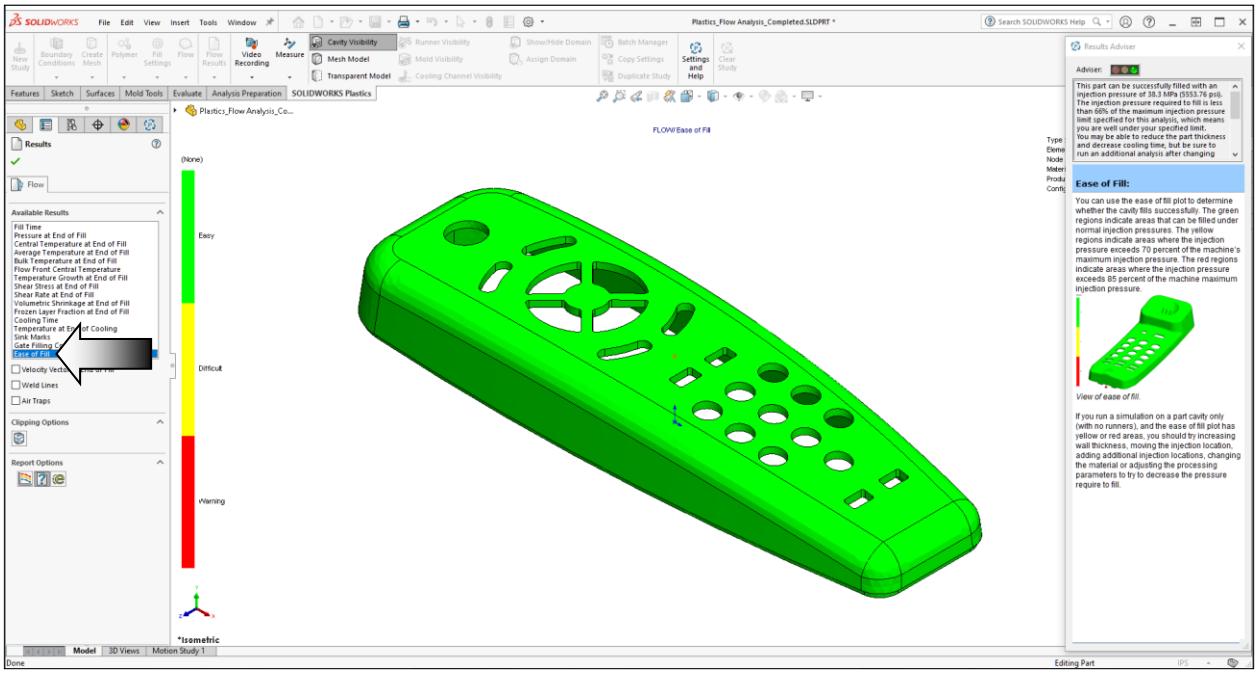
The Sink Marks results



The Gate Filling Distribution results



The End of Fill results



At this point, the Green color in the part indicates that it should be filled easily based on the references such as: Material, Injection Location and other parameters provided.

13. Saving your work:

Select **Files, Save As**.

Enter: **Plastics_Flow Analysis_Completed.sldprt** for the file name.

Click **Save**.