# CHAPTER 11

# Using SOLIDWORKS Plastics

SOLIDWORKS Plastics Standard,
SOLIDWORKS Plastics Professional, and
SOLIDWORKS Plastics Premium are available as separately purchased products that can
be used with SOLIDWORKS Standard, SOLIDWORKS Professional, and SOLIDWORKS
Premium.

SOLIDWORKS Plastics simulates how melted plastic flows during the injection
molding process to predict manufacturing related defects on parts and
molds. A new study is created to define the key characteristics of
your simulation, specifically whether you are simulating single
or multiple materials, and whether you are choosing the solid
or shell analysis procedure.
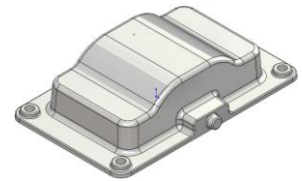
Configuration
Single Cavity

## 1. Opening a part document:

Select **File, Open**.

Open the part document named:
**Basin.sldprt**
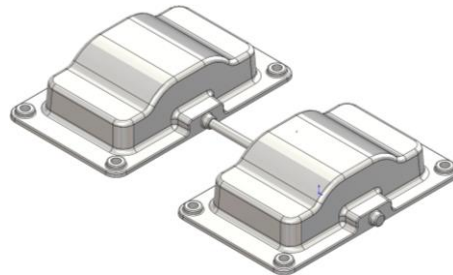
Configuration
Double Cavity

## 2. Enabling SOLIDWORKS Plastics:

This part has 2 configurations, Single Cavity
and Double Cavity.

The Double Cavity will take a lot more time
to analyze. The Single Cavity will be used
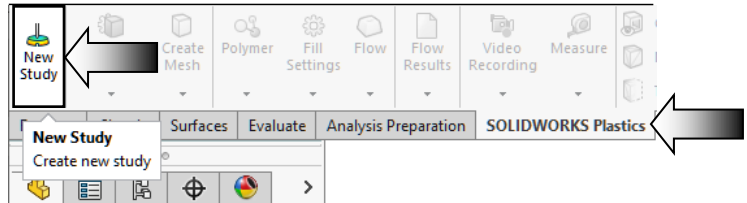and it is the active configuration.

To enable SOLIDWORKS Plastics, select:
**Tools, Customize** and click the
**SOLIDWORKS Plastics** checkbox.

Click **OK**.

**SOLIDWORKS Plastics**
SOLIDWORKS Plastics

C:\Program Files\SOLIDWORKS
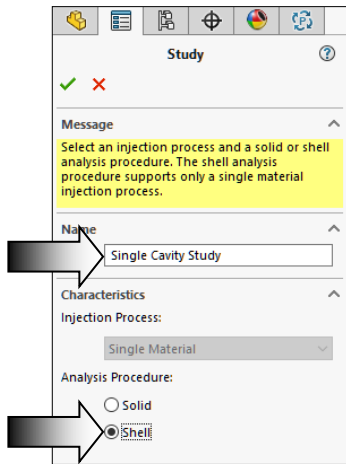Corp\SOLIDWORKS Plastics (4)
\SimpoeWorks.dll
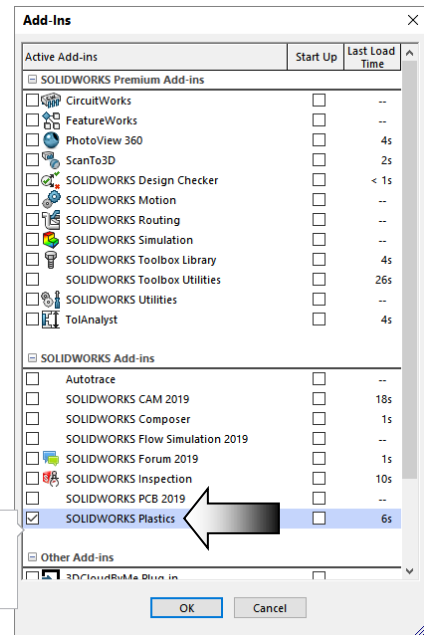
## 3. Creating a new study:

Switch to the **SOLIDWORKS Plastics** tool tab (arrow).

Click **New Study** (arrow).

For Study Name, use the default **Single Cavity Study**.

For Analysis Procedure, select **Shell**.

The shell analysis procedure can only support a single domain of the Cavity type. The shell procedure cannot accurately model grid-like or strip-like configurations, or parts with step-changes in thickness.

Click **OK**.

The **PlasticsManager** becomes available on the tree.

The three nodes: **Domains**, **Boundary Conditions** and **Shell Mesh** are displayed on this tree.
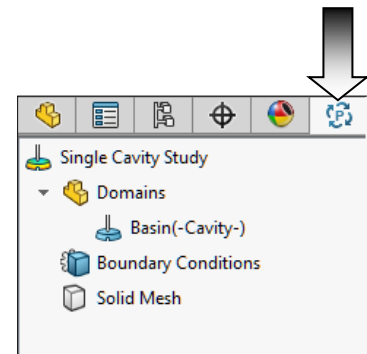
A Domain represents a volume in space through which you simulate a heat or fluid flow. For the analysis to proceed, you must assign a Domain type to each body or exclude from the analysis the bodies without a Domain type. With the shell analysis procedure, the only supported Domain type for a solid body is cavity.

## 4. Specifying the injection location:

Melted plastic material flows into the part cavity through injection locations.

The Injection Location introduces polymer material at the specified melt temperature into the cavity.

Right click **Boundary Conditions** and select: **Injection Location**.
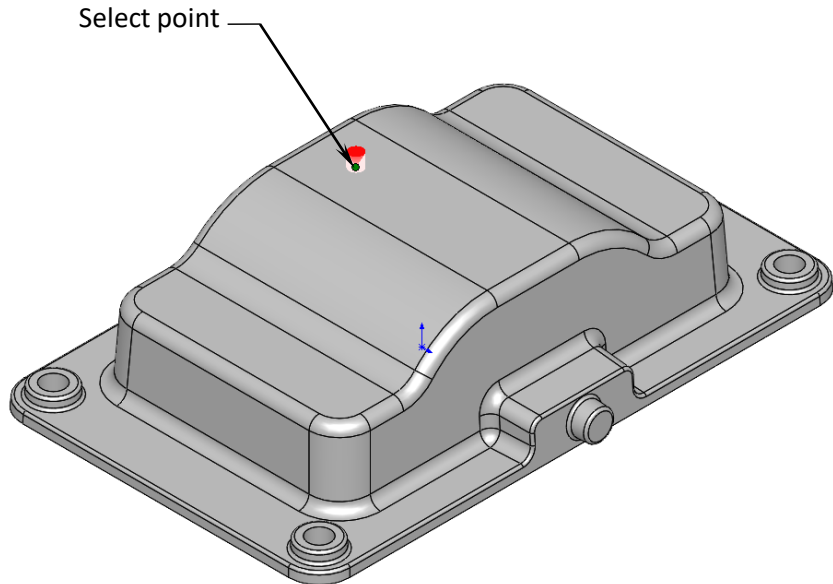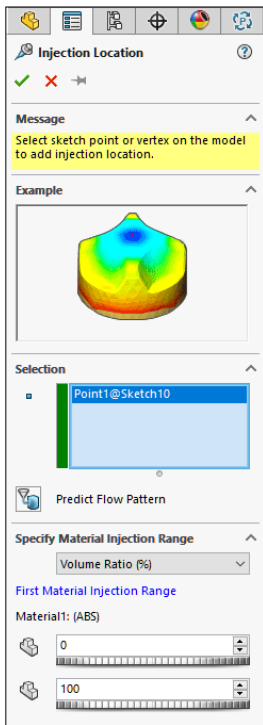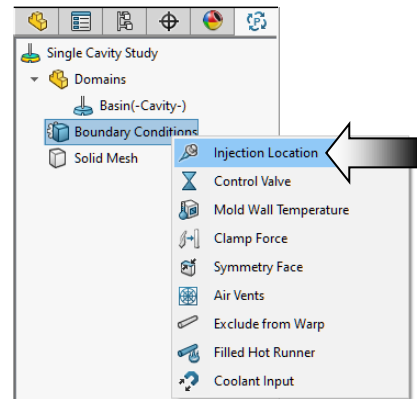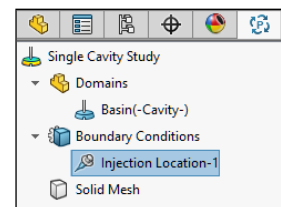
Change to the Isometric view (Control+7)

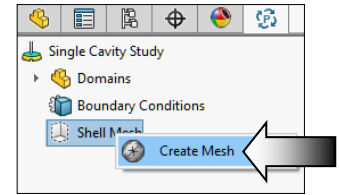For Face Selection, select the **Sketch Point** as indicated.

Click **OK**.

The Injection Location is created and displayed on the tree.

## 5. Creating a Shell Mesh:

Shell analysis is the most computationally efficient for thin walled parts with moderate thickness variations and no narrow flow channels.

Right-click **Shell Mesh** and select **Create Mesh**.

For Surface Mesh, use the <u>default</u> Mesh Density.

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

Select **Curvature-based** local refinement to refine the mesh near the holes, the injection location, and the corners. Refining the mesh in critical regions improves the accuracy of results, but increases the computational time.
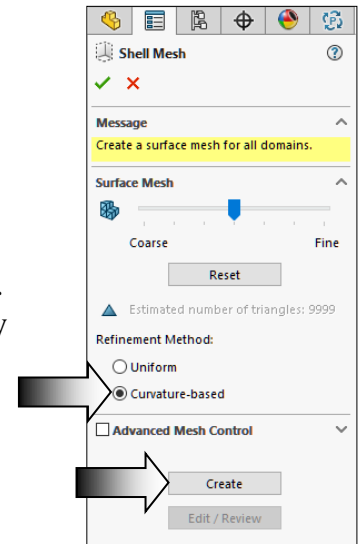
Click **Create**.
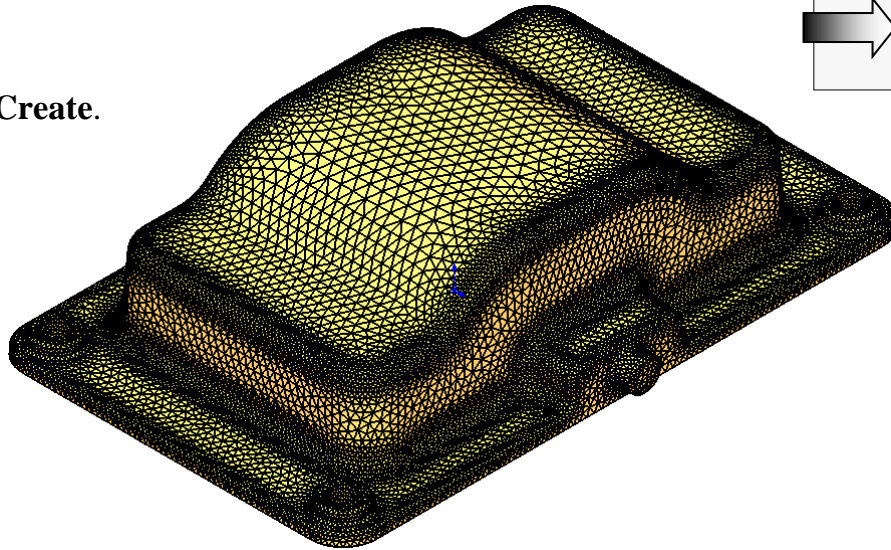
A surface mesh with the current settings is created and displayed in the graphics area. The Create button is activated again when you modify the current mesh settings.

Click **OK**.

## 6. Assigning material:

Expand the **Material** option on the PlasticsManager tree.

Right-click **Polymer** and select **Open Database**.

Select the **Default Database** option (arrow).



Under Plastics Database, expand **ABS** and click the fisrt checkbox for: **Generic material of ABS**.

Click **OK**.

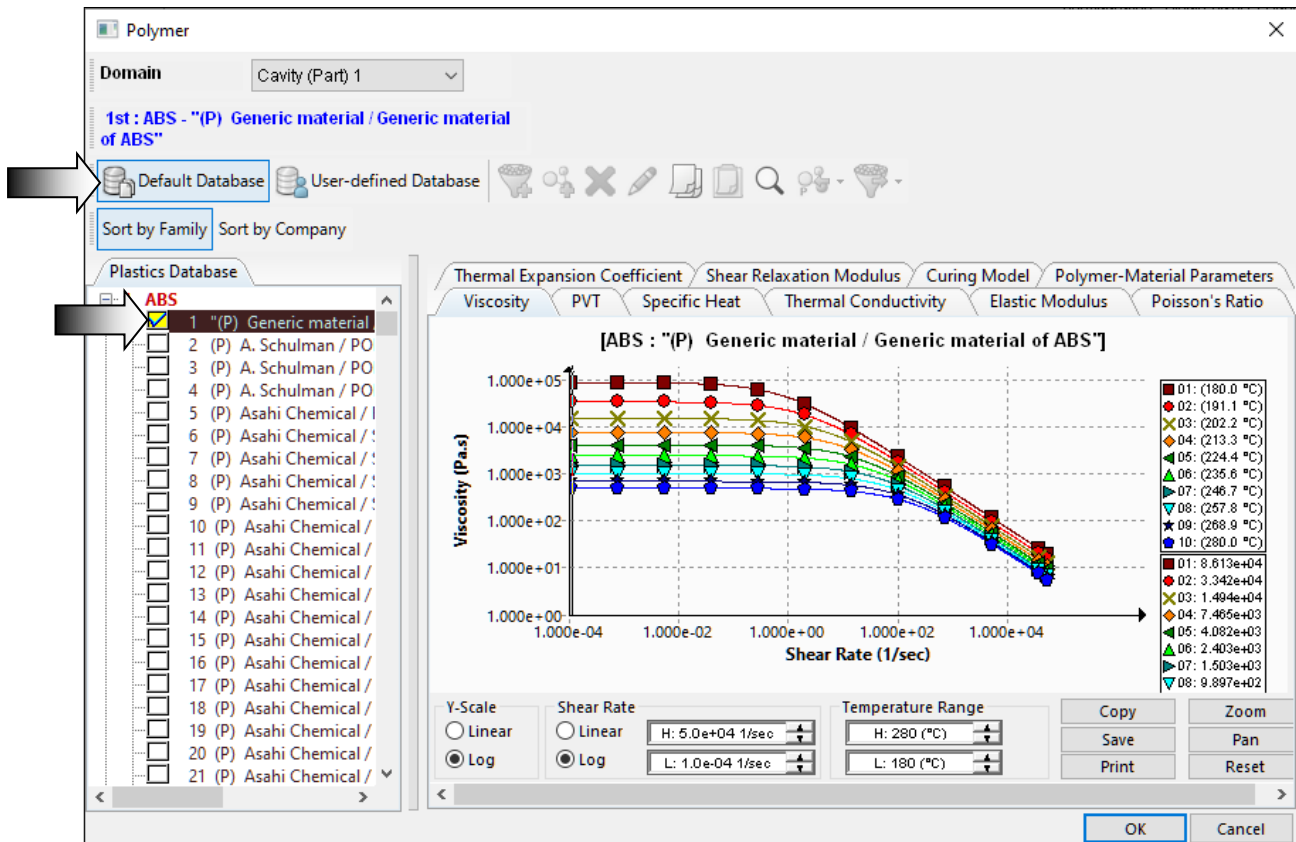The Polymer icon now has a green check mark, indicating a material has been assigned to the part.

## 7. Running the flow analysis:

This step predicts the filling pattern of the material melt flow, as it fills the cavity.

In the PlasticsManager tree, expand **Run**.

Right-click Flow and select **Run** to start the flow analysis.

A progress bar appears showing the status of the Flow analysis. It may take a few minutes to complete the analysis.
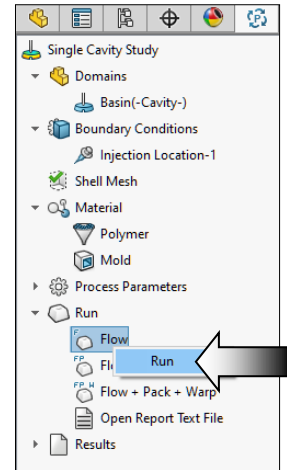
After the Flow analysis is completed, the **Results** Property-Manager lists the plots under the **Available Results** section.
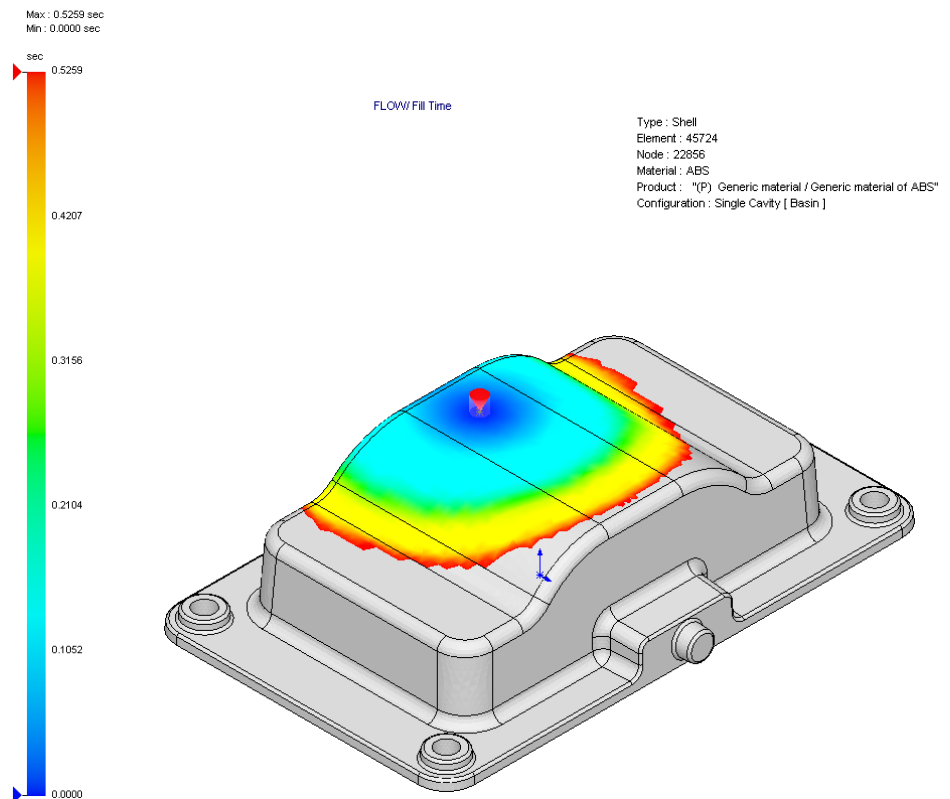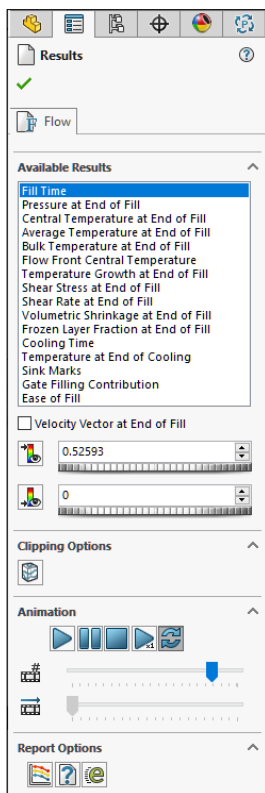
## 8. Viewing the results:

The first available result is the **Fill Time** plot.

The results shows the time when each runner and cavity domain is filled.
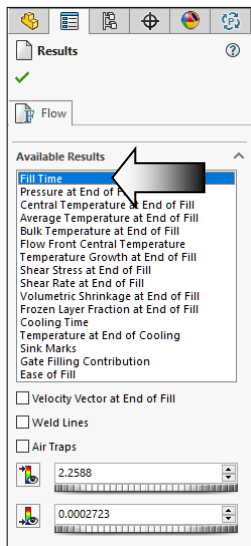
Click **Play Animation** to view the cavity is being filled with the material.
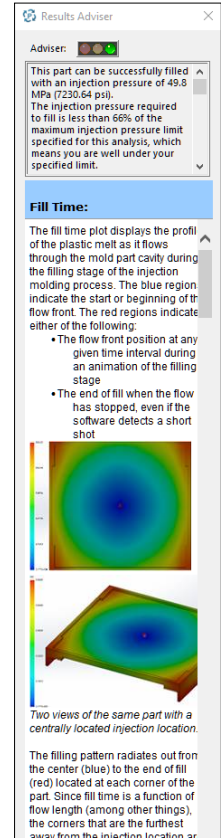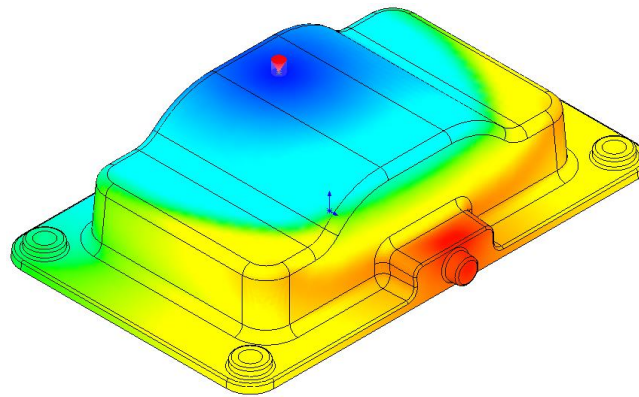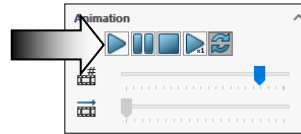


The **Advisor** button is automatically enabled to display the **Results Adviser** on the right-hand side of the screen.
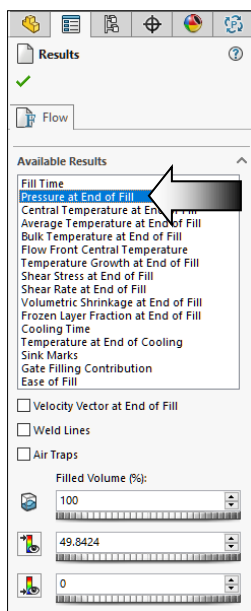
<u>**The Pressure at End of Fill Plot**</u>
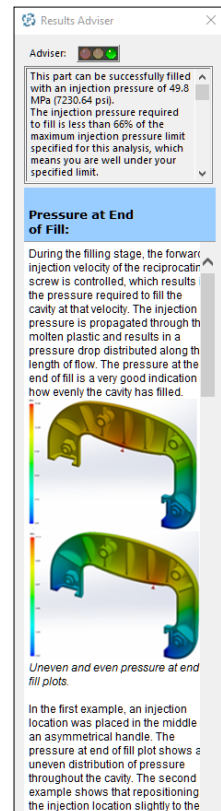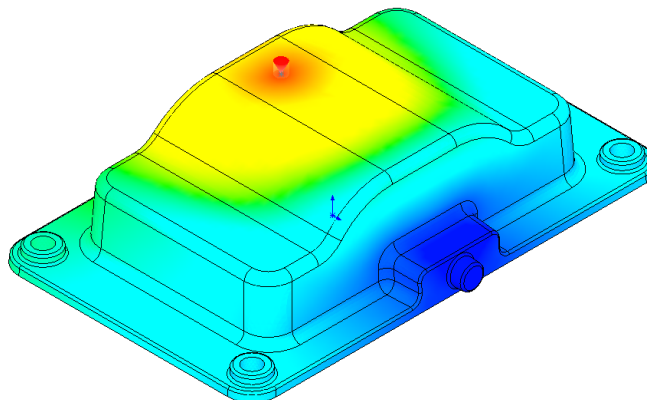
Select other Flow Results such as **Temperature at End of Cooling** or **Ease of Fill**.

Review the associated descriptions in the **Results Adviser** window.

## The Temperature at End of Cooling Result Plot



The Advisor indicates this part can be successfully filled with an injection pressure of 49.8 MPa (7230.64 psi). The injection pressure required to fill is less than 66% of the maximum injection pressure limit specified for this analysis, which means you are well under your specified limit.

## The Ease of Fill Result Plot



The result has a uniform green color indicating that the cavity fills successfully based on thegeometry, the injection location, and the default process parameters.

Click **OK** to exit the Results.

# 9. Viewing the report text file:

You can view a summary of the analysis and generate a project report to communicate key findings and results of the simulation to the team members.

To open the Summary and Report PropertyManager, do one of the following:

\* In the PlasticsManager tree, expand Results, and double-click **Summary and Report**.
or
\* In the Plastics CommandManager, click **Generate Report**.
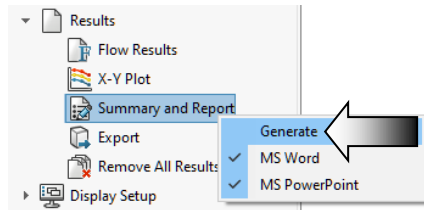
A text file of the report is also generated and stored under **Run / Open Report Text File**.

Expand **Open Report Text File**.

Right-click **Flow Text** and select **Open**.

Internet Explorer is used to display the report.

The report includes the information regarding the analysis such as: Solver Parameter Information, Process Information, Material Data Information, Geometric Data Information, and Filling Stage Results Summary.

# FLOW ANALYSIS #
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
Dassault Systemes Injection Molding Simulation

Copyright (C) 1995 ~ 2019
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
\* System machine information : Number of Processor= 16
# Maximum number of treads of the run-time system= 16
# Number of processor available of running process= 16
\* FLOW-PACK Analysis ...
>>>>> Engineer Version - Shell Style Analysis <<<<<
@@@ Shell FLOW-PACK Analysis V2019 @@@
>>>>> Xplorer Version - Shell Style Analysis <<<<<

# Solver parameter information ......

(\*)Quick-Fill Calculation = 1.00e+000
Solver Type(1: AMG, 2: PCG-AMG) = 1.00e+000
(\*)Pressure/flow-rate relative error tolerance(1.E-4 ~ 1.E-2) = 5.00e-003
(\*)Melt-temp. error tolerance(0.01 ~ 1.) = 1.00e-001 (3.22e+001 oF)
(\*)Pressure/flow-rate iteration relaxation factor(0.1 ~ 0.5) = 4.00e-001
(\*)Melt-temp. iteration relaxation factor(0.5 ~ 1.) = 1.00e+000
(\*)Max. No. of non-linear iterations(40 ~ 100) = 4.00e+001
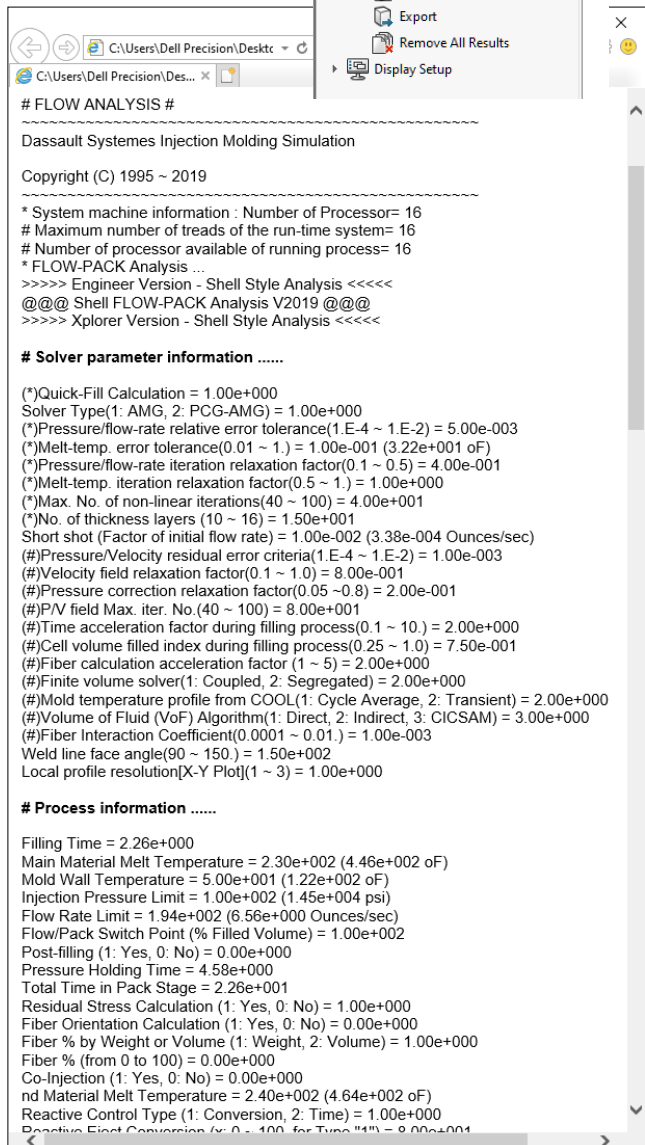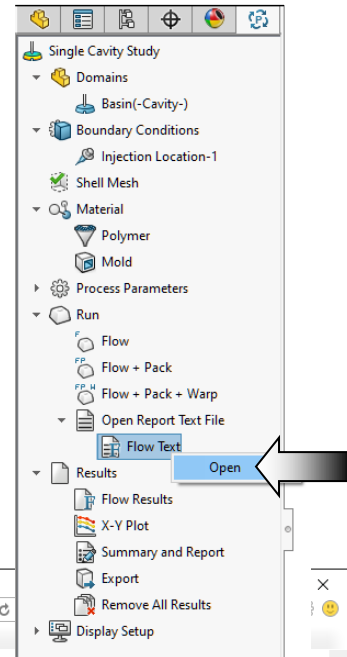(\*)No. of thickness layers (10 ~ 16) = 1.50e+001
Short shot (Factor of initial flow rate) = 1.00e-002 (3.38e-004 Ounces/sec)
(#)Pressure/Velocity residual error criteria(1.E-4 ~ 1.E-2) = 1.00e-003
(#)Velocity field relaxation factor(0.1 ~ 1.0) = 8.00e-001
(#)Pressure correction relaxation factor(0.05 ~0.8) = 2.00e-001
(#)P/V field Max. iter. No.(40 ~ 100) = 8.00e+001
(#)Time acceleration factor during filling process(0.1 ~ 10.) = 2.00e+000
(#)Cell volume filled index during filling process(0.25 ~ 1.0) = 7.50e-001
(#)Fiber calculation acceleration factor (1 ~ 5) = 2.00e+000
(#)Finite volume solver(1: Coupled, 2: Segregated) = 2.00e+000
(#)Mold temperature profile from COOL(1: Cycle Average, 2: Transient) = 2.00e+000
(#)Volume of Fluid (VoF) Algorithm(1: Direct, 2: Indirect, 3: CICSAM) = 3.00e+000
(#)Fiber Interaction Coefficient(0.0001 ~ 0.01.) = 1.00e-003
Weld line face angle(90 ~ 150.) = 1.50e+002
Local profile resolution[X-Y Plot](1 ~ 3) = 1.00e+000

# Process information ......

Filling Time = 2.26e+000
Main Material Melt Temperature = 2.30e+002 (4.46e+002 oF)
Mold Wall Temperature = 5.00e+001 (1.22e+002 oF)
Injection Pressure Limit = 1.00e+002 (1.45e+004 psi)
Flow Rate Limit = 1.94e+002 (6.56e+000 Ounces/sec)
Flow/Pack Switch Point (% Filled Volume) = 1.00e+002
Post-filling (1: Yes, 0: No) = 0.00e+000
Pressure Holding Time = 4.58e+000
Total Time in Pack Stage = 2.26e+001
Residual Stress Calculation (1: Yes, 0: No) = 1.00e+000
Fiber Orientation Calculation (1: Yes, 0: No) = 0.00e+000
Fiber % by Weight or Volume (1: Weight, 2: Volume) = 1.00e+000
Fiber % (from 0 to 100) = 0.00e+000
Co-Injection (1: Yes, 0: No) = 0.00e+000
nd Material Melt Temperature = 2.40e+002 (4.64e+002 oF)
Reactive Control Type (1: Conversion, 2: Time) = 1.00e+000

## 10. Saving your work:

Select **File, Save As**.

Enter: **Basin_Completed.sldprt** for the file name.

Click **Save**.

SOLIDWORKS saves information such as the details of the study, the domain assignment, boundary conditions such as injection location and mesh parameters, with the part document.

The Plastics analysis settings and results are saved in the same location as the part file.

Close all documents.