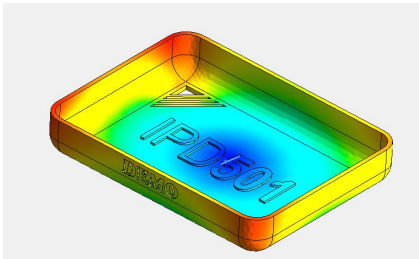# SOLIDWORKS PLASTICS
### Rev 2.2 18.02.06

Introduction

SolidWorks Plastics is an add-on feature to the standard SolidWorks package. It is a computer-aided engineering (CAE) program used to calculate and run injection molding simulations, allowing for examination of flow patterns, weld lines, and cooling rates. This guide aims to introduce the basic functionality of the program and to allow you to explore the ideas behind design for injection molding.

Assignment

Following the walkthrough below, create an injection molding simulation using SolidWorks Plastics. You are encouraged to use the part you will be injection molding. Please submit a PDF with pictures of some of your plots and describe any interesting things you try or find.

Walkthrough

1) Design a part as you would normally model in **SolidWorks** with features like extrude, revolve, loft, etc (get as fancy as you would like). For this demo, we will download the sample part from MEAM.Design.
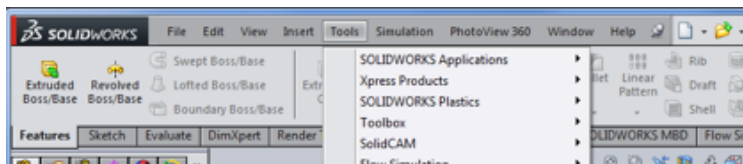
2) Check to make sure **SolidWorks Plastics** is turned on.

   a. Select **Tools** from the action bar (pictured above)

   b. Select **Add-Ins**... in the dropdown

   c. Check the boxes for **SolidWorks Plastics** in the window ➔

   d. Press **OK**
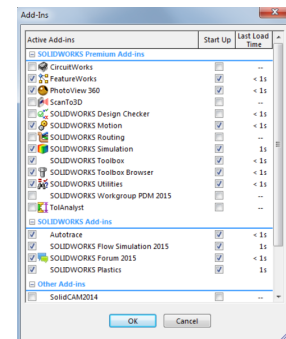
3) Select the **SolidWorks Plastics** tab on the ribbon.

4) Select **Getting Started Wizard** and a side panel will appear to the right of the screen.
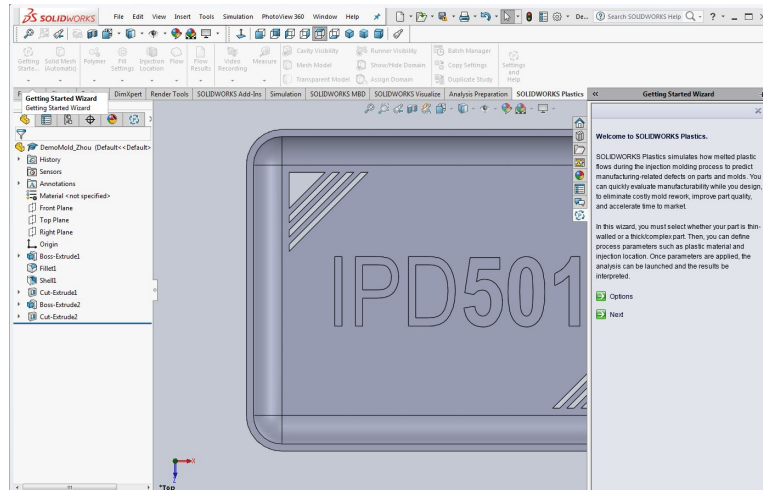
5) Select **Next.**

6) You will be prompted with the question of which type of mesh you would like to make. In our case, we will select **Shell Mesh** because of the thin walls in the part.

7) The manager panel (left panel on SW) will now prompt us in making the mesh.

   a. The **Domains** tab will appear.

   b. The part should be listed in the window labeled as a cavity.

   c. You can show and hide the cavities to ensure they are connected

      i. If the part is broken up into multiple parts, go back to your model and check that each feature of the part is merged.

   d. Click the **blue arrow**.

   e. The **Surface Mesh** tab will appear.

   f. Adjust the **Triangle size** and select **Mesh**. You will want to iterate through several different triangle sizes to find one the covers the part in a tight mesh similar to the middle example below.
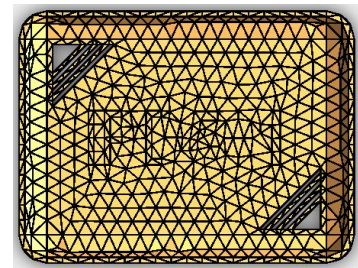
i. Click the **blue arrow**.

j. **Summary** window will appear providing feedback on the mesh. For a good part the aim would be to keep intersections and non-manifolds at zero.

k. Click the **green check**.

l. Click the **green check** again.
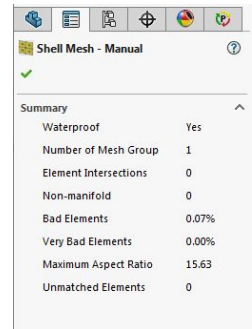
8) Move back to the **Wizard** panel (right panel in SW) and select **Next**.

9) The **Material** tab will open.

   a. Select **Change Material.**

   b. The **Polymer** window will appear (below). You will be able to select from a large range of plastics for your simulation. In this demo, we will use **HDPE** (high-density polyethylene).

   c. Click **OK** in the **Polymer** window.

   d. Select **Next** on the Wizard panel.
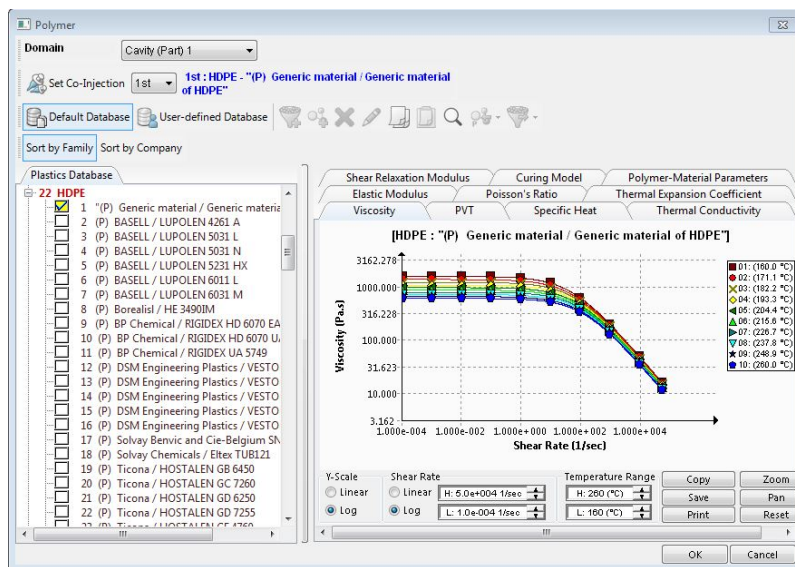
10) The **Process Parameters** tab will open.

   a. Select **Change Fill Settings.**

   b. The **Fill Settings** Panel (left panel in SW) will appear. You have the control to change the temperature and pressures of the injection.

   c. Set **Injection Pressure Limit** to 80 MPa to match the Morgan Press G-100T.

   d. Click the **green check** in the **Fill Settings** panel.

   e. Select **Next** on the wizard panel.
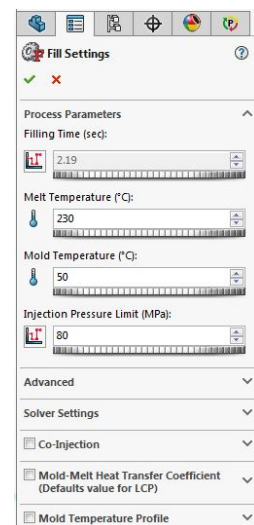
11) The **Injection Location** tab will open.

   a. Select **Add Injection Location.**

b. The **Injection Location** panel (below) will appear. This is where we set the injection spot/s for the part.

c. Click the area on the part where you would like the injection point.

d. Select the **Add Location** button (needle icon) on the Injection Location panel.

    i. It is possible to add multiple injection points. We will only be using one in this exercise, however.

e. Click the **green check** in the **Injection Location** panel.

f. Select **Next** on the wizard panel.
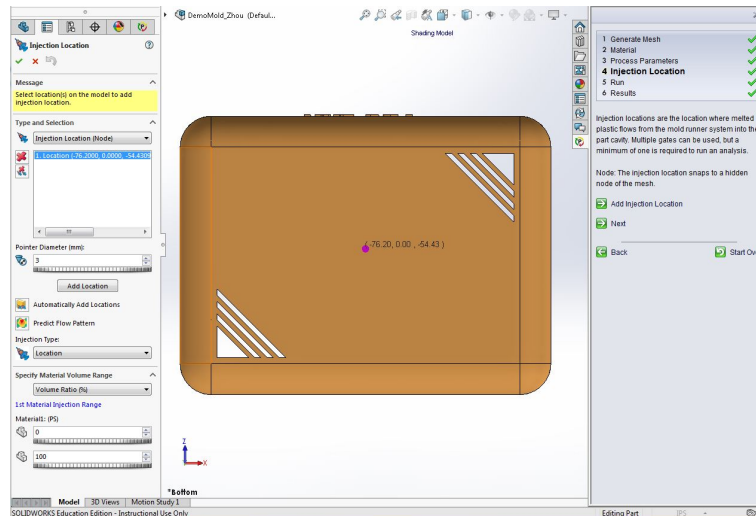
12) The **Run** tab will open.

a. Select **Run Simulation.**

b. Watch as the injection calculates and the part fills.

    i. Note: this may be SLOW…
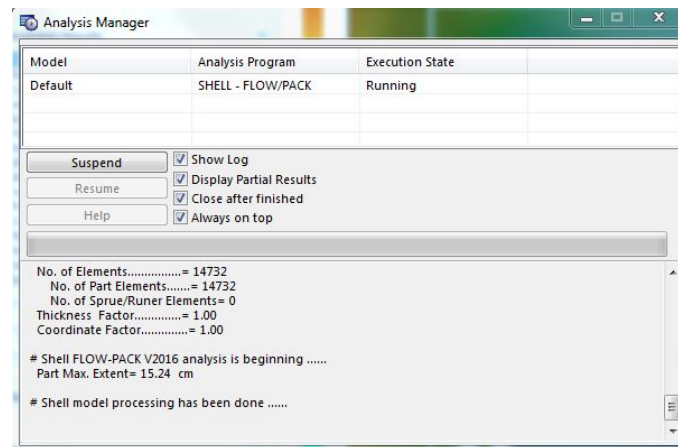


13) Once the analysis is complete..

a. Click the **play** button on the **Results** panel (Left Panel in SW) and watch the fill simulation from multiple angles and look at the different feature.

b. The **Results Advisor Window** (left) also provide feedback on the part and some explanation to help read your results.
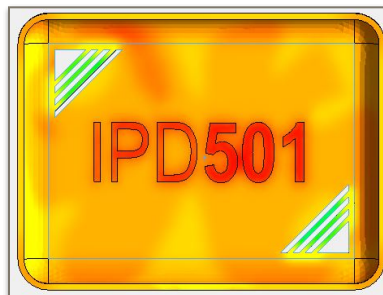
c. Select some of the different plots from the list.
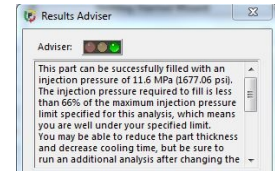
　　i. The **Fill Time** plot maps the time as the plastics flows through the part.

　　ii. The **Cooling Time** plot shows time it take sections of the part to cool. In our case we may be concern with **warping** in the lettering.

　　iii. The **Sink Marks** plot shows the measurable effects of the warping that could occur.

　　iv. The **Gate Filling Contribution** plot maps the location of material for multiple injection points.

　　v. The **Ease of Fill** plot is the check off if the part is good to go.

d. Checking the **Weld Lines** box shows where two flows of material met. These are key weak spots in the part that may require redesign.
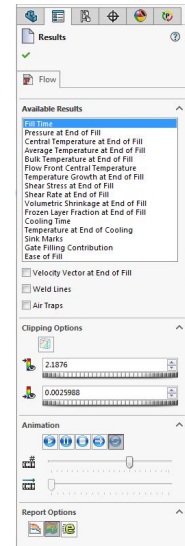
14) Now play around and try different things like…

a. Move or add injection sites.

b. Change the materials.

c. Add some features on the part.

d. Make an assembly with a flow channel and fill multiple parts at the same time.

Good Luck!