

Introduction to Injection Moulding and ProPlasticAdvisor

Injection moulding is a process that has been used in a wide variety of industries for many years. This process uses amorphous and crystalline thermoplastic resins that are heated to a stable temperature and compressed into a mould. The diagram in Figure 1 shows the process of injection moulding.

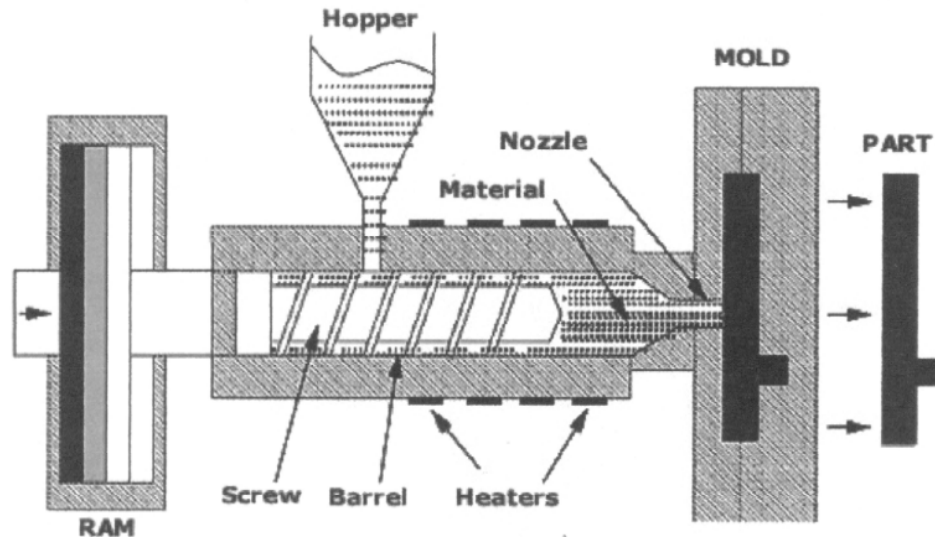


Figure 1 - The Injection Moulding Process

Cold polymer is feed from the hopper into the screw where it is heated; the ram then forces the hot molten polymer into the cold mould. The polymer is then left to cool for a pre-determined time before the mould is opened and the part is ejected. This process can be used to produce both simple and very complex components in a very short time. Examples of familiar products produced by this technique are computer cases, mobile phones casings, garden chairs etc.

One of the main problems with injection moulding is the set up cost; this includes the mould design, tooling required and the polymer used. This makes it suitable for high volume production where the cost of moulds can be amortised over a large number of products. To help to reduce the cost of setting up this process ProEngineer has designed a package called ProPlasticAdvisor. This package can be used to analyse the moulding process and increase confidence in the design before significant costs are incurred in mould manufacture.

Tutorial

In this tutorial you will be introduced to some of the basic functionality of ProPlasticAdvisor. This will include loading the part into the Advisor package, selecting a fill point, selecting a suitable material and producing a report of the product analysed. The first step is to download the part to be analysed from the Internet. Download the part and save it to your working directory. Start ProEngineer and load casing.prt.

ProPlasticAdvisor is a separate package closely linked to ProEngineer. To start an analysis, transfer the part into ProPlasticAdvisor using the following process.

APPLICATIONS ⇒ PLASTIC ADVISOR ⇒ DONE SEL

ProPlasticAdvisor will now open showing the part to be analysed. Holding down the left mouse button and moving the mouse will rotate the view of the part.

The first step in defining and analysis is to position the injection location, where the polymer will enter the mould. Consideration must be taken to find the best place to inject the polymer. In the first exercise we will inject the polymer from the top of the model. Orientate the part as shown in Figure 2.

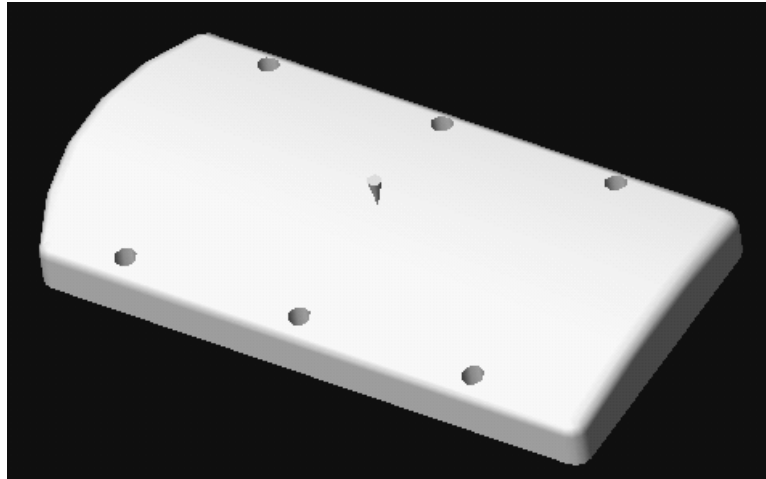



Figure 2 - Initial Fill Location

To select the polymer injection location, click the polymer injection location button  that can be found at the top of the screen then click in the centre of the model. An arrow will appear as shown in Figure 2.

The next step is to define a suitable polymer. ProPlasticAdvisor has a built in polymer materials database with over 4000 polymers to choose from. When selecting a polymer for a product the physical and mechanical properties such as strength, stiffness, hardness, etc. must be considered. The diagram in Figure 3 shows the performance spectrum of plastic materials.

The Performance Spectrum of Plastic Materials

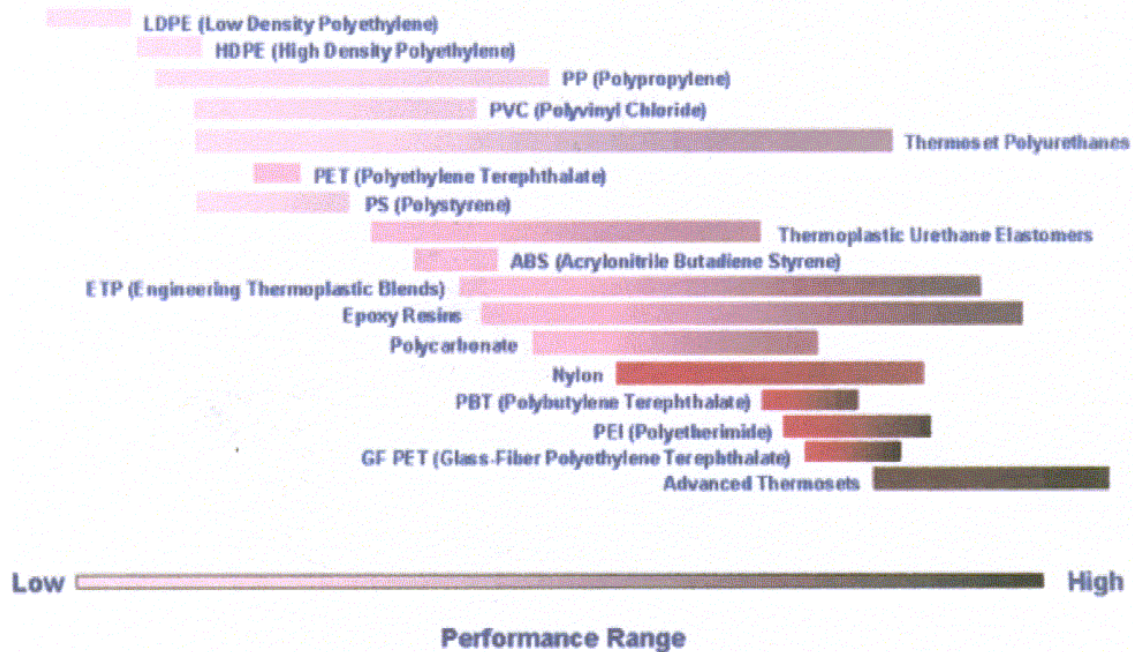






Figure 3 - Polymer Performance Range

The product in this tutorial is the back of a mobile phone. Mobile phones are normally made from polyethylene (PP) so this is the material we will use in this tutorial. To select a material open the materials database by clicking on the mould parameter button . This will display the mould parameters dialogue box where the material needed for this operation can be selected. This is achieved by the following process


Search Material ⇒ polymer family ⇒ PP (polyethylene) ⇒ Supplier ⇒ Amoco Polymers Inc ⇒ 10-1246 [AM800] ⇒ OK

We are now ready to analyse the model. This process is started by clicking on the analyse button . This process will take about 30 seconds to complete, although this depends on the performance of your computer. On completion a results summary box will appear giving information on the process and also stating whether the model is suitable for analysis. The results summary box for this analysis will state that there are weld lines and air traps in this process; we can view the location of these faults by the following method.

Close the results summary box. Click on the Weld line Locations button . This will show red lines on the model, this is where weld lines may form. Weld lines are where two separate flows of material within the mould meet and weld together. They can cause weaknesses in the final product and so should be minimised or avoided.

Click on the Air Traps Location button . This will show blue dots around the bottom edge of the model. This is the location where air may get trapped in the moulding

process. Air pockets can result in incomplete filling of the mould causing weaknesses or poor surface finish in the final product and so should be minimised or avoided.

The software also gives the user access to information on the **RESULTS** pull down menu. This menu will display the confidence of fill, fill time, pressure drop, injection pressure and flow front temp. In this tutorial we are going to pay particular attention to fill time. Click on the **RESULTS** pull down menu and **SHOW > FILL TIME**. When the fill time module appears rotate the mould so the top can be viewed. Then click the play results button . This will play a small animation showing the module filling with polymer and also gives the fill time on the right hand side of the screen.

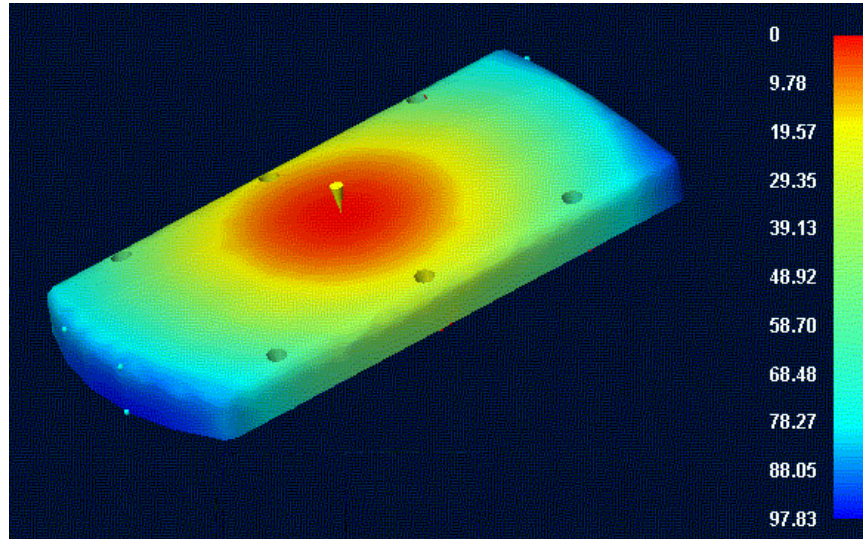


Figure 4 – Fill Time Display

If you are not happy with the set up we can change parameters to try and improve the results. You will now change the polymer injection location to the side of the module as shown in Figure 5 and change the polymer used to HMB 6270 (manufactured by Shell).

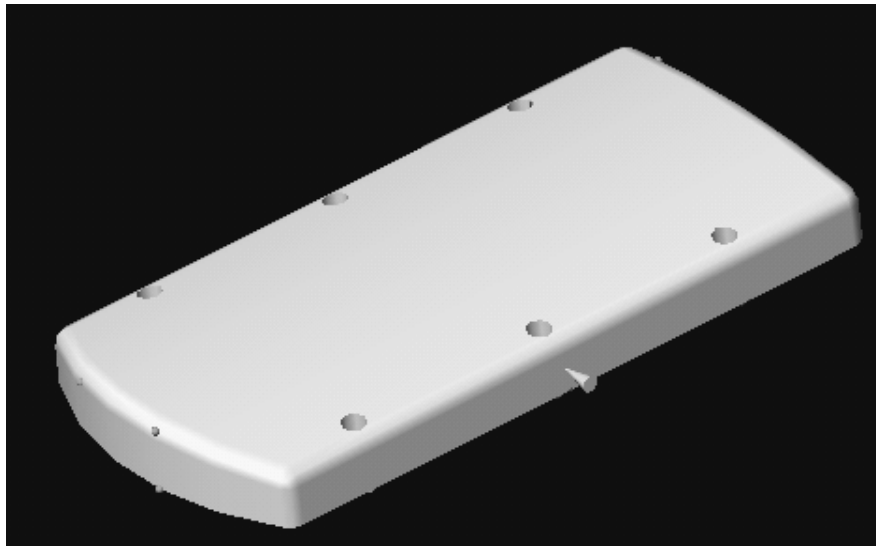



Figure 5 – New fill location

The first step is to click the select button to the left of the screen . Click the polymer injection location arrow and press the delete button on the keyboard. A dialogue box will appear stating if you wish to save the changes click **NO**. Position a new polymer injection location and change the polymer using the method shown earlier and rerun the process. See if you can notice any changes in the results.


ProPlasticAdvisor can produce a web based data sheet showing all the information on the products progress. This is achieved by clicking on the Report button . A dialog box will appear asking for information about the product (

Figure 6). Fill in the required information and tick the required boxes under Contents then **OK**.

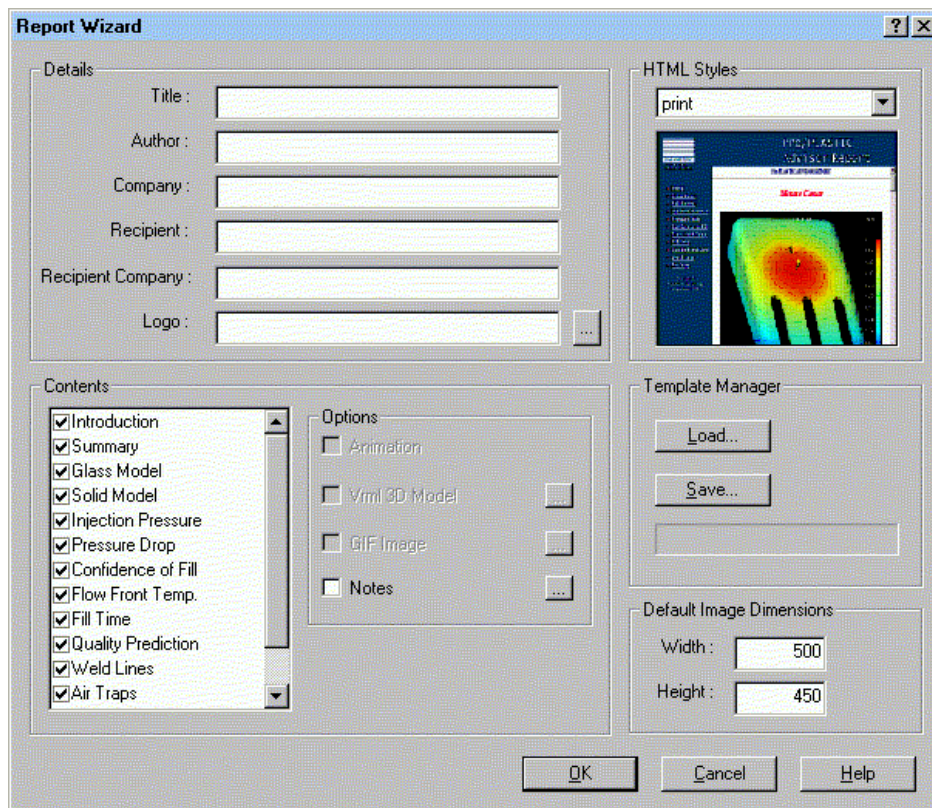


Figure 6 - The Report Generator

You will now need to select a location where you wish to save the report so navigate to your home directory. The program will generate a report in web page format and save all the information in the chosen directory. Your web browser will then start and the web page will load up. If this does not happen or you wish to view the page at a later date then in Windows open the directory and double click on file **start.htm**.