Introduction to Injection Moulding

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.

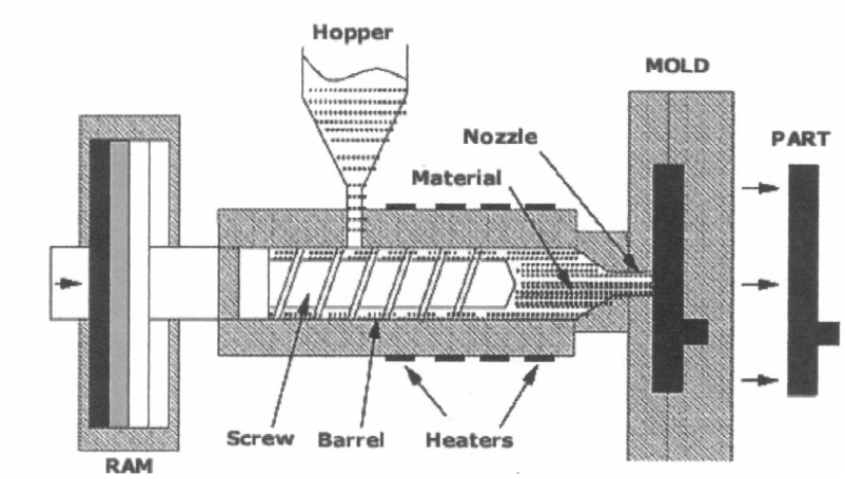


Figure 1 : The Injection Moulding Process

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 and the tooling required. 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 Pro Plastic Advisor. This is an optional package – there is no guarantee any system (outside of Staffordshire University) will include the software. This package can be used to analyse the moulding process and increase confidence in the design before significant costs are incurred in mould manufacture.

Pro Plastic Advisor Analysis

In this tutorial you will be introduced to some of the basic functionality of Pro Plastic Advisor. 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. The part name is casing.prt and can be found at http://www.staffs.ac.uk/~entdgc/WildfireDocs. Download the part and save it to your working directory. Start ProEngineer and load this part.

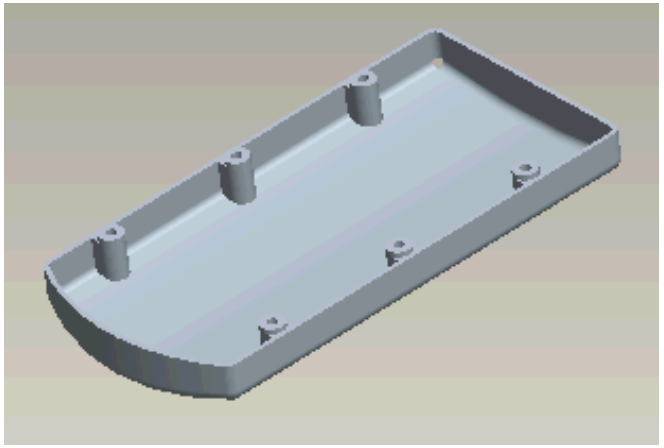


Figure 2 : Casing Part

Pro Plastic Advisor is a separate package closely linked to Pro Engineer. To start an analysis, transfer the part into Pro Plastic Advisor using the following process APPLICATIONS > PLASTIC ADVISOR then press the middle mouse to miss out the injection point definition (for now).

Pro Plastic Advisor will now open. To show the part to be analysed choose WINDWO > CASING. Holding down the left mouse button and moving the mouse will rotate the view of the part. The middle mouse button zooms in/out and the right button pans. You can also use the arrow keys to rotate the view.

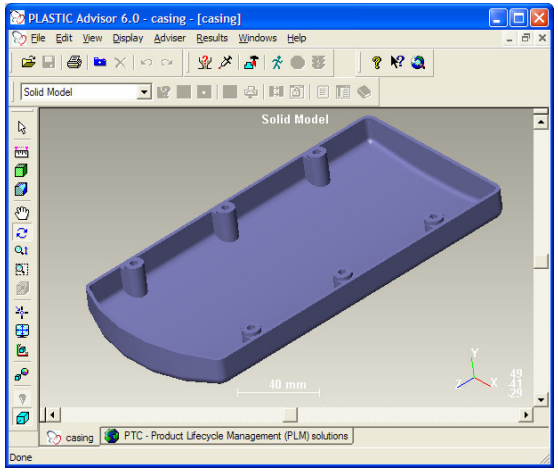


Figure 3 : Pro Plastic Advisor Window

Injection Location

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 4.

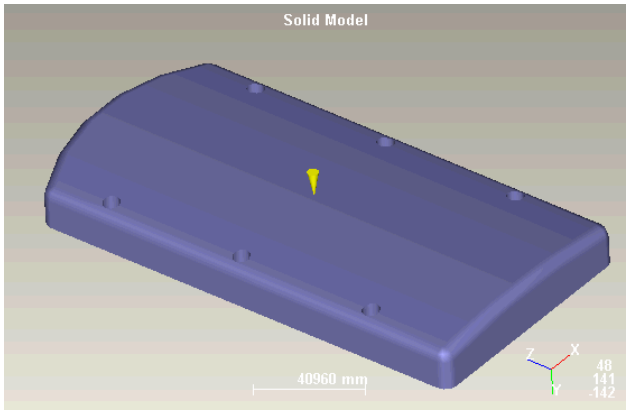


Figure 4 - Initial Fill Location

To select the polymer injection location, click the polymer injection location icon  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 4. Is this a good location from a design perspective? Injection locations will leave marks on the product so you would normally choose to inject on the inside of a product.

Material Definition

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 5 shows a generalised performance spectrum of plastic materials.

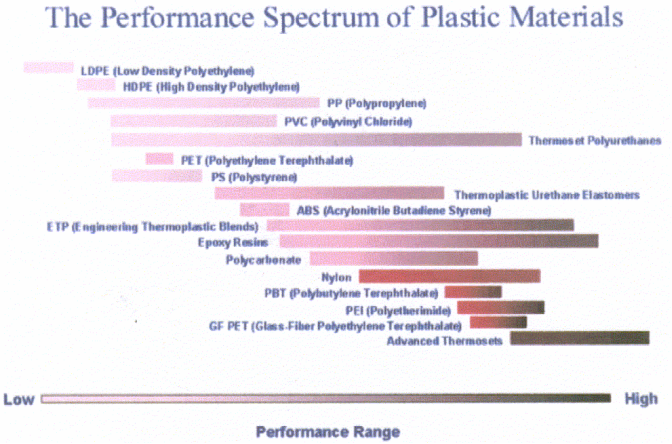


Figure 5 - Polymer Performance Range

The product in this tutorial is the back of a mobile phone. Mobile phones are often made from ABS so this is the material we will used 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. Click on specific material and the choose the material shown in Figure 6. Click on the DETAILS button to see some of the properties of this material.

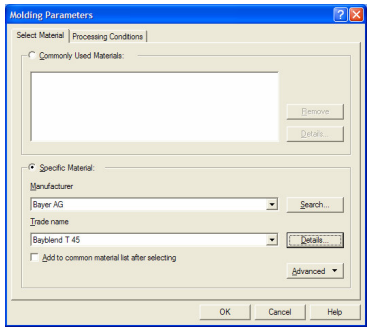


Figure 6 : Moulding Parameters Dialog

Analysing the Model

We are now ready to analyse the model. This process is started by ADVISER > ANALYSIS SELECTION or by clicking on  icon. The dialog box displayed allows you to choose the type of analysis required. Choose plastic flow analysis and START.

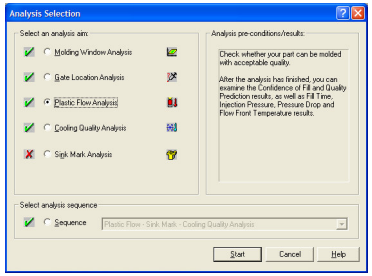


Figure 7 : Analysis Selection Dialog

This process will take about 30 seconds to complete, although this depends on the performance of your computer. During the analysis the part will be displayed transparent and will gradually fill with material starting from the injection point. 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 show amber traffic lights indicating there are possible problems - in particular weld lines and air traps may be created. Press the More… button to show a help page explaining these results in more detail.

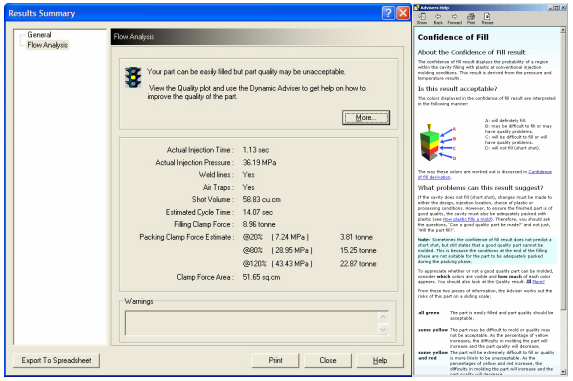


Figure 8 : Results Summary

Close the results summary box. The main graphics display will have changed to show confidence of fill which is entirely green indicating the part should fill correctly. Click on the Weld line Locations button  . This will show red lines on the model 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.

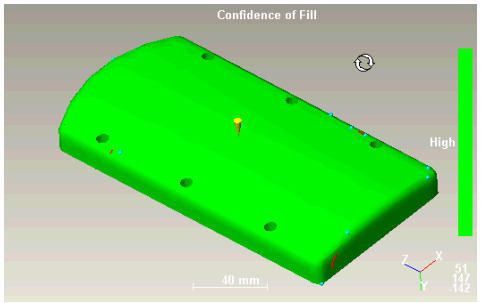


Figure 9 : Weld Lines and Air Traps

Remember it is rare to achieve a perfect mould analysis and good mould can be achieved even when nominal problems are reported. This analysis for example may well achieve a good mould even with these reported problems.

To improve your understanding of the analysis you can choose ADVISER > ADVISER or the icon. This displays a dialog with two tabs – choose the Quality Prediction tab. Areas of the model are shown in yellow showing problems exist.

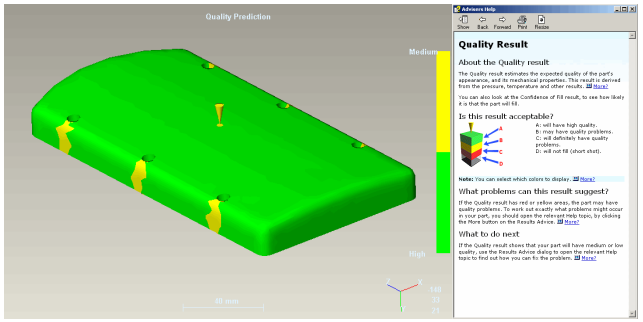


Figure 10 : Quality Prediction

Click with the RIGHT mouse button on a yellow area to get info on the problem. The dialog warns that the cooling time is too high and clicking on More… will give advice on the problem. It suggests you could…

- Decrease the melt temperature.
- Decrease the mold temperature.
- Make the problem area thinner.

You may have noticed that both the weld lines and the low quality area occur around the screw holes. One of the design goals for plastic injected parts is to maintain constant wall thickness. The material is much thicker in this area – could this be a source of the problem? Could a design changes in the part help?

The software also gives the user access to additional information on the RESULTS > SHOW menu. This menu will display the confidence of fill, fill time, pressure drop, injection pressure and flow front temp. Let's look at fill time. Choose RESULTS > SHOW > FILL TIME. When any time based result is displayed the 'video' control buttons are active at the top of the window. These let you view the results over time. 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.

Generating Reports

Pro Plastic Advisor 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 wizard will appear asking for information about the product and allowing you to choose what analysis elements you want to include. You will now need to select a location where you wish to save the report. Since the report will contain many files it is a good idea to keep them all together by creating a new directory. The program will generate a report in web page format and save all the information in the chosen directory. The web page will load up in the analysis software. 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 **index.htm**.

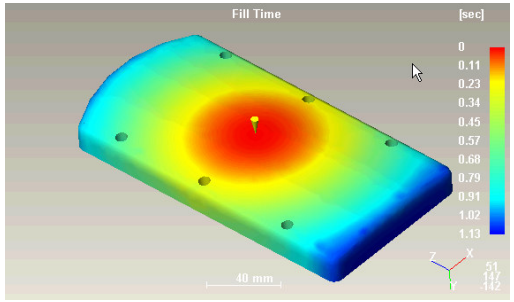


Figure 11 – Fill Time Display

Review

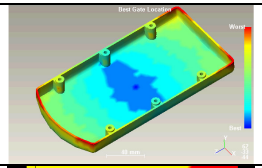
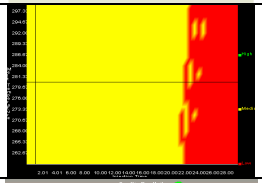
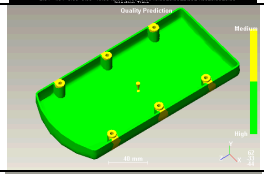
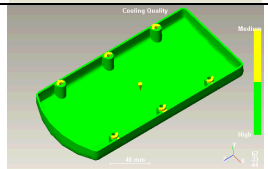
So what should you have learnt?

- How to start a mould flow analysis.
- How to position injection points and select materials.
- How to perform analyses.
- How to understand results.
- How to generate reports.

Any problems with these? Then you should go back through the tutorial – perhaps several times – until you can complete it without any help.

What Next?

Now you have a general idea of how to operate the software you may want to try a correctly structured analysis either on this or another part of your own. The table below shows the sequence of steps you should undertake to correctly perform a full analysis and generate all possible feedback.

Sequence Description	Analysis Result
Select material based on experience or databases such as CES.	Bayer AG Bayblend 45
Perform Gate Analysis to find best position for injection point.	
Position injection point and perform a moulding window analysis to determine best material conditions.	
Use the conditions specified and performs a plastic flow analysis to determine mould ability.	
Perform a Cooling Quality analysis.	
Perform a Sink Mark analysis.