G.S.Mandal's MIT Aurangabad

Maharashtra Institute of Technology, Aurangabad

Department of Plastic and Polymer Engineering

LABORATORY MANUAL

List of Experiments

Course Title: CAE for Plastics **Course Code: PPE422** **Class: Final Year BTech PPE**

EXPERIMENT NO.	EXPERIMENT NAME
1	Introduction to CAE for Plastics and software startup with user interface.
2	To practice Fill Analysis for given plastics product for single cavity mould.
3	To practice Fill Analysis for given plastics product for multi cavity mould.
4	To practice Packing Analysis for given plastics product for single cavity mould.
5	To practice Packing Analysis for given plastics product for multi cavity mould.
6	To practice Cooling Analysis for given plastics product for single cavity mould.
7	To practice Cooling Analysis for given plastics product for multi cavity mould.
8	To practice Warping Analysis for given plastics product for single cavity mould.
9	To practice Warping Analysis for given plastics product for multi cavity mould.
10	To create a report based on analysis results.

Mr. M A Ansari
Prepared By

Dr. A Chetterjee
HPPED

Experiment No. 1

Aim: Introduction to CAE for Plastics and software startup with user interface.

Theory:

Refer ebook of **"Mold Flow Design Guide" by Jay Shoemaker published by Hanser**

Visit Preface and relevant chapters.

Procedure:

Creating a project:

All Moldflow analyses are stored in **Projects**. Projects allow you to manage multiple analyses, called **Studies**. A project is created for your filling analysis.
Note: The tutorial assumes that you have opened Autodesk Simulation Moldflow Adviser and you do not yet have a project open.
When you launch Adviser, a Web page and dynamic help are displayed by default. To save space, we will close these items.
Click the **X** on the tab for the Web page to close it.

Click the **X** in the upper right corner of the Dynamic Help pane to close it.

Click Start & Learn > Launch > New Project.

Enter **Quick Start Filling Analysis** in the Project Name field.

Click **Browse** to change the location of the Project (optional).
Click **OK**.

Importing a Model:

In this task, you import an existing CAD model.

Click Home > Import > Import.

The next two steps will only occur when importing for the first time you have imported a model after installation. You are asked to update the location of Autodesk Simulation Moldflow Design Link. After your first time importing, you can start on step four.

Click **OK** to review Autodesk Simulation Moldflow Design Link options.
Click **OK** on the Options dialog box.
Click the Ellipsis button on the Import Wizard dialog box to find the file to import.
Navigate to the location of the Autodesk Inventor file you downloaded.
Select the **Panel.ipt** file and click **Open**.
Click **Next** on the Import Wizard dialog box. The part takes several seconds to import.
Confirm the units are **Millimeters**.

Click **Next** on the Advanced options page.

It takes a few seconds for the cleanup and checks to take place.

Click **Finish** to accept the analysis type as Standard - Dual Domain.

Right-click the imported study in the Project manager and select **Rename**.
Change the name of the study to **Panel End Gate**.
Use the View Cube and Navigation bar on the right to move the model around the screen.

To program mouse buttons, click Applications menu > Options > Mouse. Set the mouse buttons to your liking.

Conclusion:

Studied and gone through the basics of CAE and need of the simulation. Also got a feel of user-interface of simulation software. And how to start a project.

Assignment Questions:

a) What is CAE? Explain the role of simulation in Polymer Engineering.
b) List down the advantages of simulation softwares used in Product and Mold design.

G.S.Mandal's
MIT
A group of Academic & Research Institutions
Aurangabad

Maharashtra Institute of Technology, Aurangabad
Department of Plastic and Polymer Engineering
LABORATORY MANUAL

ACADEMIC YEAR: **PART: I**
COURSE TITLE: CAE for Plastics **COURSE CODE: PPE422**

Experiment No. 2

Aim: To practice fill analysis for given plastics product for single cavity mould.

Theory:

Study of fill analysis with respect to Gate location, Fill Pattern: Weld line position, Air trap locations, Position of vents, Over packing, Shoot Filling.

Refer ebook of **"Mold Flow Design Guide" by Jay Shoemaker published by Hanser**

Visit
Chapter 3, 4 and Chapter 11.

Procedure:

In this tutorial, we simulate the filling phase in a part. The feed system is not included. The filling phase is the most important phase of the injection molding cycle. If there are problems with the filling of a part, the part quality is not optimal. In this tutorial, we will do the following:

Create a project.
Click Start & Learn > Launch > New Project.

Import a model.
Click Home > Import > Import.

Set the injection location.
Rotate the part to see it in 3D.
Click Home > Molding Process Setup > Injection Locations.
Click the edge of the part near the center of the edge.
The exact location of the injection location will also be set by providing the exact co-ordinates.

Select the material and analyze.
Click Home > Molding Process Setup > Select Material.
The analysis wizard opens on the Material page.
After selecting any suitable material, Click *Next* to advance to the Process Settings Tab.

Click *Analyze* to start the Analysis. The Analysis takes a few minutes to analyze.
When the analysis is finished, Review the information in the summary panel.

Review results.
Click *Fill Time* in the Study Tasks list.
Click Home > Results > Results.
Rotate, Pan, and Zoom the model as necessary to inspect the fill time result.
Click Results > Animation > Play to animate the result.
Use the other animation tools to see what they do.

Move the gate.
Click the Down arrow next to the save icon on the Quick Access toolbar and select *Save Study As*.
Enter the new name and click *Save*.
Delete the first injection location and set new location. Then
Click Home > Analysis > Start Analysis.
When the analysis is finished, click *Summary* to close the Summary panel.

Compare results.
Click the *Results* tab to open it.
Double click the study *Panel End Gate* to open the study. Both studies now open.
Click Results > Windows > Tile Vertically to show both studies side by side.
(You could also tile horizontally if you wish.)
Click Results > Locking > Lock All Views , then *Lock All Plots*, and *Lock All Animations*.
By locking, both studies have the same rotation, results and animations.
Click on various parameters for analysis.

Conclusion:
We have performed Fill analysis for the given polymer article.

Assignment Questions:

1) What do you know about fill analysis?
2) Give the significance of Gate location in product design.

Experiment No. 3

Aim: To practice fill analysis for given plastics product for multi cavity mould.

Theory:

Study of fill analysis with respect to Gate location, Fill Pattern: Weld line position, Air trap locations, Position of vents, Over packing, Shoot Filling.

Refer ebook of **"Mold Flow Design Guide" by Jay Shoemaker published by Hanser**

Visit
Chapter 3, 4 and Chapter 11.

Procedure:

In this tutorial, we simulate the filling phase in a part. The feed system is not included. The filling phase is the most important phase of the injection molding cycle. If there are problems with the filling of a part, the part quality is not optimal. In this tutorial, we will do the following:

Create a project.
Click Start & Learn > Launch > New Project.

Import a model.
Click Home > Import > Import.

Duplicate the part

Click Home > Mold type > Multi-Cavity.
Click Home > Geometry > Layout Duplicate.
Enter no.of rows.
Enter row spacing.
Select Edge of selection spacing from.
Enter no.of column.
Click *OK*.
Click Geometry > Layout > Align.

Select X direction as Alignment Direction.
Hold *CTRL* Key and select both injection locations.
Select *Cavity 1/Gate 1* from the Align with part field.
Click *OK*.

Set the parting plane

Click Geometry > Mold > Set Parting Plane.
Rotate the part as necessary to see the injection locations.
Select on one of the injection locations.
Click OK.

Create the mold

In this task, we define the size of the mold. The thickness of the "A" plate determines the length of the sprue.
Click Geometry > Mold > Mold Size.
Enter size … mm for the A Plate thickness.
Enter size …. mm for the B Plate thickness.
Enter *size* .... mm for the X mold dimension.
Enter size ….mm for the Y mold dimension.
Click *OK*.

Create the runners

In this task, you create the runner system using the Runner Wizard.
Click Geometry > Feed System > Runner Wizard.
Click Middle of the mold.

Click Sprue Properties Ellipsis.
Ensure the Sprue type.
Ensure the Sprue shape.
Enter the Start diameter .
Enter the End diameter.
Click *OK*.
Click Runners Properties Ellipsis.
Ensure the Runner type.
Ensure the Runner shape.
Enter Diameter.

Click *OK*.

Click *Gates Properties* Ellipsis.
Ensure the Gate type.
Ensure the Gate shape.
Enter Width.
Enter Thickness.
Ensure the Orientation.
Enter suitable Horizontal length.
Click OK.

Ensure that the layout is *Automatic*.

Click OK.

Set the injection location.
Rotate the part to see it in 3D.
Click Home > Molding Process Setup > Injection Locations.
Click the edge of the part near the center of the edge.
The exact location of the injection location will also be set by providing the exact co-ordinates.

Select the material and analyze.
Click Home > Molding Process Setup > Select Material.
The analysis wizard opens on the Material page.
After selecting any suitable material, Click *Next* to advance to the Process Settings Tab.
Click *Analyze* to start the Analysis. The Analysis takes a few minutes to analyze.
When the analysis is finished, Review the information in the summary panel.

Review results.
Click *Fill Time* in the Study Tasks list.
Click Home > Results > Results.
Rotate, Pan, and Zoom the model as necessary to inspect the fill time result.
Click Results > Animation > Play to animate the result.
Use the other animation tools to see what they do.

Move the gate.
Click the Down arrow next to the save icon on the Quick Access toolbar and select *Save Study As*.
Enter the new name and click *Save*.
Delete the first injection location and set new location. Then
Click Home > Analysis > Start Analysis.
When the analysis is finished, click *Summary* to close the Summary panel.

Compare results.
Click the *Results* tab to open it.
Double click the study *Panel End Gate* to open the study. Both studies now open.
Click Results > Windows > Tile Vertically to show both studies side by side.
(You could also tile horizontally if you wish.)
Click Results > Locking > Lock All Views , then *Lock All Plots*, and *Lock All Animations*.
By locking, both studies have the same rotation, results and animations.
Click on various parameters for analysis.

Conclusion:
We have performed Fill analysis for the given polymer article.

Assignment Questions:

1. Give the challenges while setting up fill analysis for multi cavity mould.

Experiment No. 4

Aim: To practice packing analysis for given plastics product for single cavity mould.

Theory:

In this we set up and run a Filling + Packing analysis on a 2-cavity tool of the panel. We evaluate the fill balance between the parts, the pressure to fill the tool, the shrinkage on the part, and the cooling time.

Refer ebook of **"Mold Flow Design Guide" by Jay Shoemaker published by Hanser**

Procedure:

Create a project.
Click Start & Learn > Launch > New Project.

Import a model.
Click Home > Import > Import.

Set the analysis parameters and run the analysis
Click Home > Molding Process Setup > Analysis Wizard.
Uncheck *Fill* as the analysis sequence.
Check *Fill+Pack* as the analysis sequence, then click *Next*.
Confirm the material.
Set the fill time to *3.5* seconds, then click *Next*.
Review the *Packing profile* and *Automatic cooling time*, then click *Analyze*.
The analysis takes several minutes to analyze.
When the analysis is finished, click Summary to close the Summary panel.

Review results

In this task, you review the Fill time results.

Click Home > Results to display the Results ribbon.
Click Fill Time in the Study Tasks list.

Click Results > Animation > Play to animate the result.
Rotate, pan, and zoom as necessary to see the filling pattern.

Conclusion:

Thus, we have practiced packing analysis for the given polymer article in single cavity mould .

Assignment Questions:

1. What do you know about packing?
2. Give the significance of runner, gate and sprue location in product design.

Experiment No. 5

Aim: To practice packing analysis for given plastics product for multi cavity mould.

Theory:

Study of pack analysis with respect to Gate location, Fill Pattern: Weld line position, Air trap locations, Position of vents, Over packing, Shoot Filling.

Refer ebook of **"Mold Flow Design Guide" by Jay Shoemaker published by Hanser**

Procedure:

In this tutorial, we simulate the filling phase in a part. The feed system is not included. The filling phase is the most important phase of the injection molding cycle. If there are problems with the filling of a part, the part quality is not optimal. In this tutorial, we will do the following:

Create a project.

Click Start & Learn > Launch > New Project.

Import a model.

Click Home > Import > Import.

Duplicate the part

Click Home > Mold type > Multi-Cavity.
Click Home > Geometry > Layout Duplicate.
    Enter no.of rows.
    Enter row spacing.
    Select Edge of selection spacing from.
    Enter no.of column.
    Click *OK*.
Click Geometry > Layout > Align.
    Select X direction as Alignment Direction.
    Hold *CTRL* Key and select both injection locations.

Select *Cavity 1/Gate 1* from the Align with part field.
Click *OK*.

Set the parting plane

Click Geometry > Mold > Set Parting Plane.
Rotate the part as necessary to see the injection locations.
Select on one of the injection locations.
Click OK.

Create the mold

In this task, we define the size of the mold. The thickness of the "A" plate determines the length of the sprue.
Click Geometry > Mold > Mold Size.
Enter size … mm for the A Plate thickness.
Enter size …. mm for the B Plate thickness.
Enter *size* .... mm for the X mold dimension.
Enter size ….mm for the Y mold dimension.
Click *OK*.

Create the runners

In this task, you create the runner system using the Runner Wizard.
Click Geometry > Feed System > Runner Wizard.
Click Middle of the mold.

Click Sprue Properties Ellipsis.
Ensure the Sprue type.
Ensure the Sprue shape.
Enter the Start diameter .
Enter the End diameter.
Click *OK*.
Click Runners Properties Ellipsis.
Ensure the Runner type.
Ensure the Runner shape.
Enter Diameter.
Click *OK*.

Click *Gates Properties* Ellipsis.

Ensure the Gate type.
Ensure the Gate shape.
Enter Width.
Enter Thickness.
Ensure the Orientation.
Enter suitable Horizontal length.
Click OK.

Ensure that the layout is *Automatic*.

Click OK.

Set the injection location.
Rotate the part to see it in 3D.
Click Home > Molding Process Setup > Injection Locations.
Click the edge of the part near the center of the edge.
The exact location of the injection location will also be set by providing the exact co-ordinates.

Select the material and analyze.
Click Home > Molding Process Setup > Select Material.
The analysis wizard opens on the Material page.
After selecting any suitable material, Click *Next* to advance to the Process Settings Tab.
Click *Analyze* to start the Analysis. The Analysis takes a few minutes to analyze.
When the analysis is finished, Review the information in the summary panel.

Review results.
Click *Fill Time* in the Study Tasks list.
Click Home > Results > Results.
Rotate, Pan, and Zoom the model as necessary to inspect the fill time result.
Click Results > Animation > Play to animate the result.
Use the other animation tools to see what they do.

Move the gate.
Click the Down arrow next to the save icon on the Quick Access toolbar and select *Save Study As*.
Enter the new name and click *Save*.
Delete the first injection location and set new location. Then
Click Home > Analysis > Start Analysis.

When the analysis is finished, click *Summary* to close the Summary panel.

Compare results.

Click the *Results* tab to open it.
Double click the study *Panel End Gate* to open the study. Both studies now open.
Click Results > Windows > Tile Vertically to show both studies side by side.
(You could also tile horizontally if you wish.)
Click Results > Locking > Lock All Views , then *Lock All Plots*, and *Lock All Animations*.
By locking, both studies have the same rotation, results and animations.
Click on various parameters for analysis.

Conclusion:

Thus, we have practiced and performed pack analysis for the given polymer article in multi cavity mould.

Assignment Questions:

1. What is runner balancing?
2. Explain Hesitation effect, meld lines, flow leaders and flow deflectors.

Experiment No. 6

Aim: To practice Cooling Analysis for given plastics product for single cavity mould.

Theory:

The cooling analysis calculates a steady state, or average temperatures over the entire mold cycle. At a minimum, all that is modeled is the part, mold boundary, and cooling channels. With this method, it is easy to modify the location of the cooling channels to see the influence of the cooling channel location. This practice uses the model of the parts, feed system, and mold boundary done in the packing analysis tutorial.

Refer ebook of **"Mold Flow Design Guide" by Jay Shoemaker published by Hanser**

Procedure:

Create a project

Import a study

Create the cooling lines

Click Home > Geometry > Cooling System > Cooling Wizard.
Enter the Channel diameter in mm.
Ensure the Layout.
Click Distance between channels and specify a distance in mm.
Enter an Offset from part's boundary in mm.
Enter a Hose length in mm.
Click *OK*.
Rotate the part to 0 0 0 so you can see the FRONT face of the View Cube.
Notice how cooling channels (blue) are inside the mold outline, and hoses are on the outside.
Click View > Visibility > Object Visibility.
Click off the *Mold Outline*.
Click to activate the *Select* tool from the Navigation Toolbar.
Select the two outermost and innermost channels and the hoses that connect to them, as shown. Hold the *Ctrl* key to select multiple items.

Click Geometry > Cooling System > Coolant Inlets.
Click the end of the hoses at the locations to create eight cooling inlets.

Input Process Parameters**:** In this task, we set up and run the cooling analysis.
Click Home > Molding Process Setup > Analysis Wizard.
Uncheck *Fill*.
Check *Cool*.
Click *Next*.
Confirm the material, then click *Next* twice to get to the *Advanced* tab.
Enter *time for* the Injection + Packing + Cooling time.
Click *Analyze* to start the analysis. The analysis takes several minutes to analyze.
When the analysis is finished, click *Summary* to close the Summary panel.

Review Circuit Results: In this task, we interpret different circuit results.
Click Home > Results > Results.
Ensure that the following objects are visible.

a. Cavities
b. Cooling Channels
c. Cooling Hoses
d. Cooling Inlets

If needed click View > Visibility > Object Visibility to turn on the objects.
Rotate, pan, and zoom as necessary to see the circuit results.
Click *Circuit coolant temperature* in the study tasks list.
The increase in temperature is best when less than 2-3° C. Two cavity circuits are upper limit of the guideline indicating changes in the cooling design are desirable.
Click *Circuit flow rate*.
The input per circuit was …. l/min. The pumping equipment needs a capacity of at least … l/min.
Click *Circuit Reynolds Number*.
The results indicate the Reynolds number produced with the input flow rate. Typically a target of 10,000 Reynolds number is desired.

Review Part Temperature Results:
In this task, we interpret the temperature, part results. These results represent the average temperature of the part surface temperature throughout the mold cycle.
Click the *Temperature, part* results.

This result represents the cycle averaged part temperature at the mold surface or plastic/metal interface. The variation in temperature max to min is under 20C which is good. However even the coolest temperature on the part is higher than the target indicating the coolant temperature, is to be studied.

Conclusion:

We have performed cooling analysis for the given polymer article.

Assignment Questions:

a) What is the importance of cooling phase?
b) Why we need control on Mold and Melt temperature?

Experiment No. 7

Aim: To practice Cooling Analysis for given plastics product for multi cavity mould.

Theory:

In this tutorial, we set up and run a warpage analysis. The analysis sequence includes; cooling, filling, and packing. We set up and run the entire sequence.

Refer ebook of **"Mold Flow Design Guide" by Jay Shoemaker published by Hanser**

Procedure:

Create a project

Import a study

Duplicate the part

Click Home > Mold type > Multi-Cavity.
Click Home > Geometry > Layout Duplicate.
Enter no.of rows.
Enter row spacing.
Select Edge of selection spacing from.
Enter no.of column.
Click *OK*.
Click Geometry > Layout > Align.
Select X direction as Alignment Direction.
Hold *CTRL* Key and select both injection locations.
Select *Cavity 1/Gate 1* from the Align with part field.
Click *OK*.

Set the parting plane

Click Geometry > Mold > Set Parting Plane.
Rotate the part as necessary to see the injection locations.
Select on one of the injection locations.
Click OK.

Create the mold

In this task, we define the size of the mold. The thickness of the "A" plate determines the length of the sprue.

Click Geometry > Mold > Mold Size.
Enter size … mm for the A Plate thickness.
Enter size …. mm for the B Plate thickness.
Enter *size* .... mm for the X mold dimension.
Enter size ….mm for the Y mold dimension.
Click *OK*.

Create the runners

In this task, you create the runner system using the Runner Wizard.
Click Geometry > Feed System > Runner Wizard.
Click Middle of the mold.

Click Sprue Properties Ellipsis.
Ensure the Sprue type.
Ensure the Sprue shape.
Enter the Start diameter .
Enter the End diameter.
Click *OK*.
Click Runners Properties Ellipsis.
Ensure the Runner type.
Ensure the Runner shape.
Enter Diameter.
Click *OK*.

Click *Gates Properties* Ellipsis.
Ensure the Gate type.
Ensure the Gate shape.
Enter Width.
Enter Thickness.
Ensure the Orientation.
Enter suitable Horizontal length.
Click OK.

Ensure that the layout is *Automatic*.

Click OK.

Set the injection location.
Rotate the part to see it in 3D.
Click Home > Molding Process Setup > Injection Locations.
Click the edge of the part near the center of the edge.
The exact location of the injection location will also be set by providing the exact co-ordinates.

Input Process Parameters

Click Home > Molding Process Setup > Analysis Wizard.
Uncheck *Fill*.
Check *Cool*.
Check *Fill+Pack*.
Check *Warp*.
Click *Next*.
Select the material.
Ensure the mold temperature.
Ensure the Melt temperature.
Ensure the injection time.
Ensure the Mold-open time, then click *Next*.
Enter *time in* seconds as the Injection + Packing + Cooling time.
Click *Analyze*.
When the analysis is finished, click *Summary* to close the Summary panel.

Create the cooling lines

Click Home > Geometry > Cooling System > Cooling Wizard.
Enter the Channel diameter in mm.
Ensure the Layout.
Click Distance between channels and specify a distance in mm.
Enter an Offset from part's boundary in mm.
Enter a Hose length in mm.
Click *OK*.
Rotate the part to 0 0 0 so you can see the FRONT face of the View Cube.
Notice how cooling channels (blue) are inside the mold outline, and hoses are on the outside.

Click View > Visibility > Object Visibility.
Click off the *Mold Outline*.
Click to activate the *Select* tool from the Navigation Toolbar.
Select the two outermost and innermost channels and the hoses that connect to them, as shown. Hold the *Ctrl* key to select multiple items.
Click Geometry > Cooling System > Coolant Inlets.
Click the end of the hoses at the locations to create eight cooling inlets.

Review Circuit Results: In this task, we interpret different circuit results.
Click Home > Results > Results.
Ensure that the following objects are visible.

e. Cavities
f. Cooling Channels
g. Cooling Hoses
h. Cooling Inlets

If needed click View > Visibility > Object Visibility to turn on the objects.
Rotate, pan, and zoom as necessary to see the circuit results.
Click *Circuit coolant temperature* in the study tasks list.
The increase in temperature is best when less than 2-3° C. Two cavity circuits are upper limit of the guideline indicating changes in the cooling design are desirable.
Click *Circuit flow rate*.
The input per circuit was …. l/min. The pumping equipment needs a capacity of at least … l/min.
Click *Circuit Reynolds Number*.
The results indicate the Reynolds number produced with the input flow rate. Typically a target of 10,000 Reynolds number is desired.

Review Part Temperature Results:
In this task, we interpret the temperature, part results. These results represent the average temperature of the part surface temperature throughout the mold cycle.
Click the *Temperature, part* results.
This result represents the cycle averaged part temperature at the mold surface or plastic/metal interface. The variation in temperature max to min is under 20C which is good. However even the coolest temperature on the part is higher than the target indicating the coolant temperature, is to be studied.

Conclusion:

Thus, we have practicd cooling analysis for the given polymer article in multi cavity mould

Assignment Questions:

a. What do you understand by balanced cooling?

Experiment No. 8

Aim: To practice Warping Analysis for given plastics product for single cavity mould.

Theory:

A multi-cavity mold has two or more cavities. The cavities may be identical, to produce multiple copies of a part in a single injection cycle, or they may be different (a family mold), to produce related parts in a single injection cycle.
A multi-cavity mold has the capacity to produce several finished parts per injection cycle. It is suitable for producing parts in high volume, or for producing parts that are small.
In this tutorial we will:

Procedure:

Create a project

Import a study

Set the injection location.
Rotate the part to see it in 3D.
Click Home > Molding Process Setup > Injection Locations.
Click the edge of the part near the center of the edge.
The exact location of the injection location will also be set by providing the exact co-ordinates.

Input Process Parameters

Click Home > Molding Process Setup > Analysis Wizard.
Uncheck *Fill*.
Check *Cool*.
Check *Fill+Pack.*
Check *Warp*.
Click *Next*.
Select the material.
Ensure the mold temperature.

Ensure the Melt temperature.
Ensure the injection time.
Ensure the Mold-open time, then click *Next*.
Enter *time in* seconds as the Injection + Packing + Cooling time.
Click *Analyze*.
When the analysis is finished, click *Summary* to close the Summary panel.

Review Warpage Results:

In this task, you interpret warpage indicator results to determine the relative magnitude of warpage.
Click View > Visibility > Object Visibility.
Click off all objects except *Cavities*.
Click Home > Results > Results.
Click *Warpage indicator, all effects* in the study tasks list.
Click Results > Results > Plot Properties
Click the *Deflection* tab, set the Scale factor to *5* and Click *OK*.
This result shows the relative magnitude of the warpage for the parts. Green areas and red areas are are to be checked for Nominal max. deflection. The deflection is exaggerated five times so it can be seen better.
Rotate Pan and zoom to inspect the warpage.
Display the other warpage indicators.
Compare them to the *all effects* plot. The plot that looks most like the all effects plot has the greatest influence on the warpage.

Conclusion:

Thus, we have studied and practiced warpage analysis for the given polymer article in single cavity mould.

Assignment Questions:

a) What is warpage? Troubleshoot this defect.
b) Explain the role of processing parameters in warpage occurrence.

Experiment No. 9

Aim: To practice Warping Analysis for given plastics product for multi cavity mould.

Theory:

A multi-cavity mold has two or more cavities. The cavities may be identical, to produce multiple copies of a part in a single injection cycle, or they may be different (a family mold), to produce related parts in a single injection cycle.
A multi-cavity mold has the capacity to produce several finished parts per injection cycle. It is suitable for producing parts in high volume, or for producing parts that are small.
In this tutorial we will:

Procedure:

Create a project

Import a study

Duplicate the part

Click Home > Mold type > Multi-Cavity.
Click Home > Geometry > Layout Duplicate.
Enter no.of rows.
Enter row spacing.
Select Edge of selection spacing from.
Enter no.of column.
Click *OK*.
Click Geometry > Layout > Align.
Select X direction as Alignment Direction.
Hold *CTRL* Key and select both injection locations.
Select *Cavity 1/Gate 1* from the Align with part field.
Click *OK*.

Set the parting plane

Click Geometry > Mold > Set Parting Plane.
Rotate the part as necessary to see the injection locations.
Select on one of the injection locations.
Click OK.

Create the mold

In this task, we define the size of the mold. The thickness of the "A" plate determines the length of the sprue.
Click Geometry > Mold > Mold Size.
Enter size … mm for the A Plate thickness.
Enter size …. mm for the B Plate thickness.
Enter *size* …. mm for the X mold dimension.
Enter size ….mm for the Y mold dimension.
Click *OK*.

Create the runners

In this task, you create the runner system using the Runner Wizard.
Click Geometry > Feed System > Runner Wizard.
Click Middle of the mold.

Click Sprue Properties Ellipsis.
Ensure the Sprue type.
Ensure the Sprue shape.
Enter the Start diameter .
Enter the End diameter.
Click *OK*.
Click Runners Properties Ellipsis.
Ensure the Runner type.
Ensure the Runner shape.
Enter Diameter.
Click *OK*.

Click *Gates Properties* Ellipsis.
Ensure the Gate type.
Ensure the Gate shape.

Enter Width.
Enter Thickness.
Ensure the Orientation.
Enter suitable Horizontal length.
Click OK.

Ensure that the layout is *Automatic*.

Click OK.

Set the injection location.
Rotate the part to see it in 3D.
Click Home > Molding Process Setup > Injection Locations.
Click the edge of the part near the center of the edge.
The exact location of the injection location will also be set by providing the exact co-ordinates.

Input Process Parameters

Click Home > Molding Process Setup > Analysis Wizard.
Uncheck *Fill*.
Check *Cool*.
Check *Fill+Pack.*
Check *Warp*.
Click *Next*.
Select the material.
Ensure the mold temperature.
Ensure the Melt temperature.
Ensure the injection time.
Ensure the Mold-open time, then click *Next*.
Enter *time in* seconds as the Injection + Packing + Cooling time.
Click *Analyze*.
When the analysis is finished, click *Summary* to close the Summary panel.

Review Warpage Results:

In this task, you interpret warpage indicator results to determine the relative magnitude of warpage.
Click View > Visibility > Object Visibility.

Click off all objects except *Cavities*.
Click Home > Results > Results.
Click *Warpage indicator, all effects* in the study tasks list.
Click Results > Results > Plot Properties
Click the *Deflection* tab, set the Scale factor to *5* and Click *OK*.
This result shows the relative magnitude of the warpage for the parts. Green areas and red areas are are to be checked for Nominal max. deflection. The deflection is exaggerated five times so it can be seen better.
Rotate Pan and zoom to inspect the warpage.
Display the other warpage indicators.
Compare them to the *all effects* plot. The plot that looks most like the all effects plot has the greatest influence on the warpage.

Conclusion:

Thus, we have studied and practiced warpage analysis for the given polymer article in multi cavity mould.

Assignment Questions:

a) How will you optimize product design so as to minimize warpage?

Experiment No. 10

Aim: To create a report based on analysis results.

Theory: Create a new report containing several results.

Procedure:

Click > (Home tab > Reporting panel > Reports) to open the Reports tab.

Click > (Reports tab > Reports panel > Report Wizard).

Note The Report Generation Wizard allows you to include results from different studies within your project in the report.

For example, you may want to compare the warpage results for mating parts. If both of these studies are saved in the same project, you can add these studies to the Selected studies to include them in the report.

If there are multiple studies in your project, click Next.

The Data Selection page appears.

Select the Fill time result and click Add.

The study name with the selected result has been added to the Selected data box.

Add the Pressure at end of fill and Weld lines results.

Click Next.

The Report Layout page appears.

The HTML document format is already selected in the Report format drop-down box because this is the default format.

The Report items pane shows the results and associated options that have been selected for inclusion in the final report.

Select the Fill time result from the Report items pane.

The Item details pane becomes active.

Select the Descriptive text checkbox.

Click Edit.

The Plot Note is displayed in the Report Item Description dialog. You can edit the Plot Note before including it in the report.

Enter "This results in an acceptable fill time" at the end of the Report Item Description text box.

Click OK to accept this alteration.

Note: Changes made to Plot Notes at this stage will only appear in the report and not affect the Plot Note in the study.
Ensure the Descriptive text box is selected for the remaining results.
Click Generate.

Conclusion: Thus, we have practiced to create a report based on analysis results.