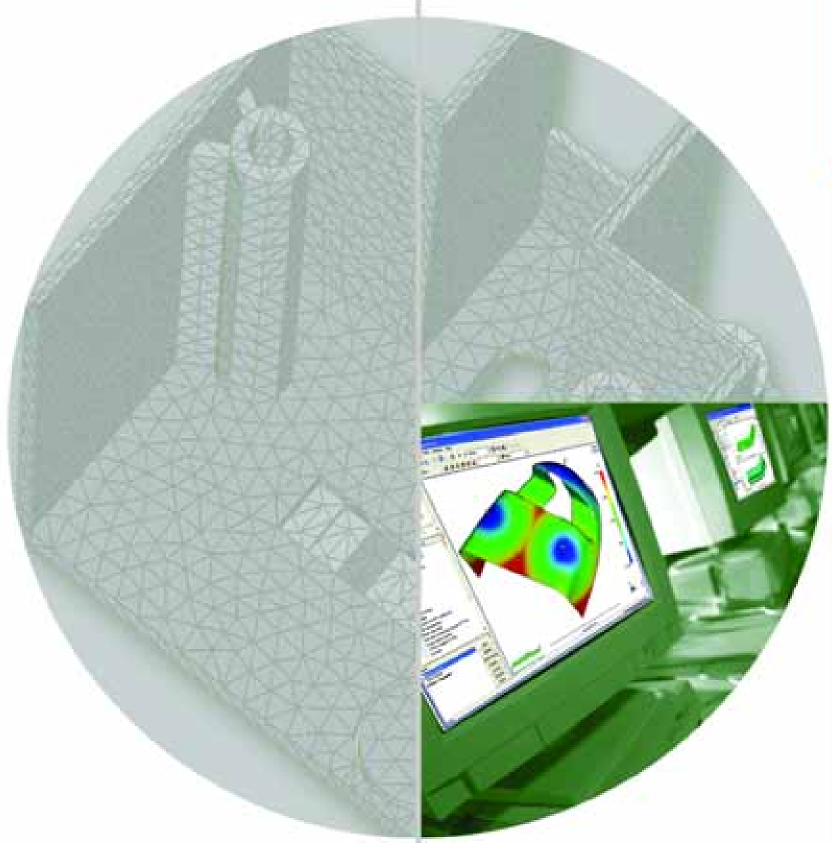
Moldflow Plastics Insight®

Release 6.0



Simulation Fundamentals Training Practice

Simulation Fundamentals

PRACTICE FOR MPI 6.0

March 2006

Published by Moldflow Corporation,

Contents

About this manual

The Simulations Fundamentals, Practice manual is designed with the new Moldflow user in mind. In creating this manual, our goal was to introduce you to some basic plastic flow and design principles in addition to skills needed to translate, analyze and interpret models.

There is a significant amount of information in this manual, more information than can be absorbed during the class. This manual should be useful as a handy desk reference when back in the office.

Using this manual

This manual is separated into several chapters and appendices. Each of the chapters covers a specific topic and includes the following sections:

Aim

Describes the learning objectives of the chapter.

Why Do It

Outlines the reasons for following the prescribed guidance, suggestions, and methodology within the chapter.

Overview

A complete outline of what will be covered within the chapter.

Practice

This section contains hands-on exercises used to reinforce what was learned. The practice section guides the user through the steps necessary to complete a project.

Formatting used in this manual

Tasks

To perform a step on the computer

1. When the Task icon is shown, below it is a list of numbered steps to complete the task.

   1.1. Tasks can have a sub-step,

   - A bulleted list provides information on a step, or a non-sequential actions to be done,

     - A second level bulleted list to provide information on a sub-step.

2. A task is used in the practice section of a chapter to indicate steps to be done on the computer.

Bulleted lists

- A bulleted list contains a number of items that have no particular order.
- It does not represent a list of steps that have to be followed in sequence.

Ruled paragraph

```
Text from a computer screen is shown between ruled lines.
```

Tip

A tip is a useful piece of information that is normally associated with a task or procedure. Something that can be done to make a task easier or more efficient.

Note

A note is generally used to highlight some background or theoretical information.

Training files setup

The files required for the Simulation Fundamental class are organized into several folders. Each folder has the files necessary for one chapter. The table below shows the required folders, translation and study files, and results necessary for the class. In each folder, there will be a *.mpi file with the same name as the folder. The mpi file is the database of the Project pane in Synergy. All the results that need to be run will be provided in class. However if for some reason the results are not available, they can be obtained by analyzing the necessary studies.

Table 1: Files Required for the Simulation Fundamentals Class

Folder name	Translation and Study Files Needed	Results needed
Basic_Packing	3d_3snap_cover_packing.sdy snap_cover_packing.sdy	
Create_Reports	grabit_center_gate.sdy grabit_end_gate.sdy	Flow analysis on both parts.
Extra	cover.sdy	
Fiber_flow	cover_fiber.sdy manifold_fiber	Flow analysis on both parts.
Flow_Leaders	window_cover.sdy	
Gate_placement	cover.sdy door_panel.sdy paper_holder.sdy phone.sdy	
Gate_Runner_Design	box_lid.sdy snap_cover_runner_modeling.sdy	
Guided_Project	base_fixed.sdy base_mesh.sdy	
Magics_STLs	00 front.stl 01 FixWizard.stl 02 Normals.stl 03 Stitch.stl 04 Stitch + normals.stl 05 holes.stl 06 shells.stl 07 overlaps.stl 08 Fixing_test.stl 09 ChildCarSeat.stl 10 ChildCarSeat_result.stl Q-base.igs	
Material_Searching	cover.sdy	

Table 1: Files Required for the Simulation Fundamentals Class

Folder name	Translation and Study Files Needed	Results needed
Modeling_Tools	speedo_fusion.sdy speedo_md.sdy	
Molding_Window	cover.sdy door_panel_mw.sdy phone_mw_cent.sdy phone_mw_end.sdy	
Projects	Boot.igs Cap.igs Change_tray.igs Cover.igs Drawer.igs Dustpan.igs Grabit.igs Light_holder.ige Paper_Holder.igs phone.igs reel.igs Snap_Cover.igs	
QuickFCW	snap3_cover.igs snap_cover.igs	
Results_Interpretation	door_panel.sdy manifold.sdy	Flow analysis on both parts.
Translation_Cleanup	cover.igs dustpan.stl housing.step housing_cleanup.sdy Manifold.igs snap_cover.igs snap_cover.prt snap_cover.step snap_cover.stl snap_cover.x_t snap_cover_rad.igs	
Valve_Gates	tub.sdy	

CHAPTER 1

Injection Molding Overview

There is no practice for this subject.

CHAPTER 2

Finite Element Overview

There is no practice for this subject.

CHAPTER 3

Moldflow Design Principles

There is no practice for this subject.

CHAPTER 4

Introduction to Synergy

Aim

To learn the many features of the MPI/Synergy user interface.

Why do it

The MPI/Synergy graphical user interface provides you with a quick, easy-to-use method of preparing, running and post-processing an analysis of a particular part design. This chapter will introduce you to the various aspects of the user interface and show you how they can be used, as well as getting you up and running with a project and interacting with a model.

Overview

In this exercise, you will review and use the many interface features provided in MPI/Synergy, including:

- Menus
- Panel, Project pane
- Panel, Study tasks
- Panel, Tools
- Layers
- Toolbars
- Context menu
- Display window
- Preferences
- Toolbars
- Working with projects
- Entity selection
- Properties
- Model manipulation
- Wizards

Practice - Introduction to Synergy

Follow the steps below to practice using some of the features of Synergy.

To create a new project

1. Select **File ➡ New Project…**.
2. Click **Browse…** and navigate to the **My MPI 6.0 Projects\MPI_Fundamentals** folder.
3. Enter **Synergy_practice** in the **Project name** text box, and click **OK**.

> The files may not be located in the folder mentioned above. If you are unable to locate the files, consult with your instructor.

To import a model stored in a study file

1. Select **File ➡ Import…**
   - Or, right-click the mouse in the **Project View** and select **Import…** from the top of the context menu.
2. Navigate to the **My MPI 6.0 Projects\MPI_Fundamentals\extra** folder.
3. Click on the file **cover.sdy** to select it, and click **Open**.
   - The cover model will be imported into MPI.
4. Rotate the model and investigate its features.

:

To define the mouse modes

The first thing that you will do in Synergy is to define the mouse modes, and how the mouse works in conjunction with additional keystrokes, as follows:

1. Select **File ➡ Preferences…**
2. Click the **Mouse** tab to display the options for configuring the mouse modes.
3. Set the following options:
   - Middle: To **Rotate.**
   - Middle+Shift: To **Center.**
   - Middle+Ctrl: To **Dynamic Zoom.**
   - Right: To **Pan.**
   - Right + Ctrl: To **Mouse Apply.**
   - Wheel **- Dynamic Zoom**.
   - Wheel + Shift **- Pan X**.
   - Wheel + Ctrl **- Pan Y**.
   - Initial mode for new windows: To **Select**.
4. Leave the remaining options at their default values and click **OK**.

In addition to configuring how the mouse works when using MPI, to save time, you can also define a toolbar. This allows you to have quick access to commonly used functions.

To define a toolbar

1. Select **View ➡ Toolbars ➡ Customize…**
2. Click **New** and enter **Mytoolbar** in the Toolbar name text box.
3. Click **OK**.
    - A blank toolbar will appear at the top left of the screen.
4. Click the **Commands** tab in the Customize dialog.
5. Now you will drag and drop the following commands from the **Customize** dialog to the newly created toolbar:

> Place the cursor over each icon to see the tool tip, which will display the name of the command. You can also click on the icon to see a description at the bottom of the dialog

5.1. From the **File** category:

- ➤ Save As.
- ➤ Export.

5.2. From the **Edit** category:

- ➤ Assign Property.
- ➤ Save Image.

5.3. From the **View** category:

- ➤ Lock All Views.
- ➤ Lock All Animations.
- ➤ Lock All Plots.
- ➤ Unlock Views.
- ➤ Unlock All Animations.
- ➤ Unlock All Plots.
- ➤ Units.

The toolbar should look similar to the image in Figure 1 below.

Figure 1: My Toolbar

6. Click **OK** to close the menu.

> If the layer pane is not display you can open it by clicking the Layers icon (**View ➡ Layer...**).

To create a new layer

1. Click the icon in the Layers pane, to add a new layer.
2. Enter **Rim** as the layer name.
   - When created, the layer name should be highlighted and ready to edit. If not, click on the layer to rename it.

In this step, you will assign the rim section of the cover model to the Rim layer that you just created.

To assign the rim to the new layer

1. Enter –90 –90 0 (-90 **space** -90 **space** 0) into the **Enter Rotation Angles** text box.
   - The rotation angles text box is on the **Viewpoint** toolbar. The toolbar may not be turned on.

Figure 2: Viewpoint toolbar

2. De-select the **Perspective** tool in the **Viewer** toolbar, only if it is currently depressed.
3. Select the rim section of the model by creating a banded selection (click, hold and drag the left mouse button with the select icon , as indicated in Figure 3 below. Start from the lower left corner and drag to the upper right).

> If the banded selection is not high enough on the rim, not all of the rim elements will be selected. If the banding goes higher than the rim, part of the top of the part will also be selected and should not be.
>
> If you have selected too much, click in the display area off the model and reselect.
>
> When properly selected, most of the interior elements shown in Figure 4 will be selected and moved to the new **Rim** layer. These extra elements in the center of the part will be moved back to the **New Triangles** layer.

4. Ensure that the Rim layer is highlighted in the Layers pane, and then click **(Assign Layer**).
   - This assigns all selected elements to the Rim layer.
5. Enter 0 0 0 into the **Enter Rotation Angles** text box.
6. De-select the **New Triangles** layer in the Layers pane.
   - Notice that there are more elements on the Rim layer than required.

7. Click the **Select** tool  and create a bounding box to select all elements inside the rim, as indicated in Figure 4 below.
8. Highlight the **New Triangles** layer, and then click **Assign** to assign the selected elements back to the New Triangles layer.

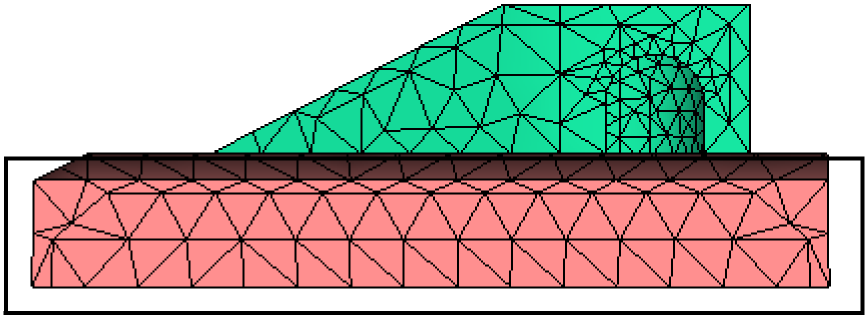


Figure 3: Rim section selected

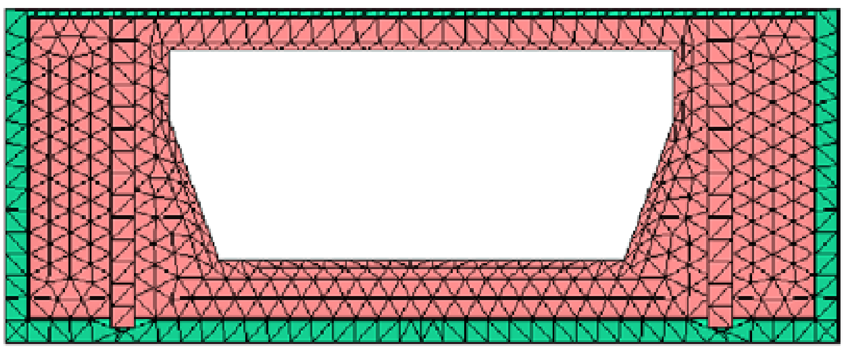
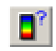


Figure 4: Elements inside rim band selected

To set the thickness of the rim layer

1. Ensure that only the **Rim** layer is visible.
2. Click **Ctrl + A** to select the entire rim.
3. Select **Edit ➧ Assign Property…**
4. Click **New ➧ Part surface (Fusion).**
5. Select **Specified** in the Thickness drop-down list, and enter **2** in the text box that appears.
6. Enter **2.0 mm Rim,** in the **Name** text box.
   - This allows you to uniquely identify the part property at a later time.
7. Click **OK** twice.
8. Click **Edit ➧ Remove unused properties**.
   - This will remove any properties that have no entities assigned to them. Depending on the editing being done, this could be a significant number.

To check the model thickness

1. Select the **New Triangles** and **Rim** layers so they are both visible.
2. Click the **Tools** tab.
3. Click the Mesh diagnostic icon .
4. Select **Thickness Diagnostic** and click **Show**.
5. Use the **Query Result** tool  and click on the rim.
   - Thickness values at the selected location will be displayed.

   5.1. Hold down the control key and click on the model to display multiple thickness values at one time.

To save an image of the model

1. Click the **Save Image** tool in Mytoolbar, or alternatively, you can **select Edit ➡ Save Image To File**.
2. Select where you want to save the image in the **Save in** drop-down list.
3. Enter a name for the image in the **File name** text box.
4. Select the image format from the **Save as type** drop-down list, and click **Save**.

To save the model

1. Click the **Save As** tool in Mytoolbar.
   - This will allow you to save the study under a new name.
2. Enter **Cover 2mm rim**.
3. Click **Save**.
4. Click the **Tasks** tab.
   - The new study will be updated in the Project View.

Competency check - Introduction to Synergy

Find the menu in which the following commands are located.

1. Copying entities ______________________

2. Save Image to File ______________________

3. Lock all Views ______________________

4. Split a display window ______________________

5. Save XY Plot curve Data ______________________

6. Generate a Mesh ______________________

7. Set the Molding Process ______________________

8. Create Curves ______________________

Evaluation Sheet - Introduction to Synergy

Find the menu in which the following commands are located.

1. Copying entities	Modeling
2. Save Image to File	Edit
3. Lock all Views	View
4. Split a display window	Window
5. Save XY Plot curve Data	Results
6. Generate a Mesh	Mesh
7. Set the Molding Process	Analysis
8. Create Curves	Modeling

CHAPTER 5

How to Use Help

Aim

The aim of this chapter is to learn the many features of the MPI online help system.

Why do it

The MPI online help provides a wealth of information on using MPI, from solver theory to how to interpret analysis results. This chapter will introduce you to the various ways the help can be used.

Overview

In this chapter, you will be introduced and practice with accessing the many help features provided in MPI / Synergy, including:

- Help homepage.
- Help contents.
- Help index.
- Help full-text search.
- Panel / Dialog help.
- Context-sensitive (What's this?) help.
- Favorites.
- Help commands.

Practice - How to use Help

Preparing to access help

If you did the Practice for the Introduction to Synergy chapter, do the task below. If not, skip to the next Task titled **To create a new Project.**

To open a sample project and study

1. Open the project called **synergy_practice.mpi** located in the folder **My MPI 6.0 Projects\MPI_Fundamentals** on your PC in the folder called **synergy_practice**
2. Open the **cover.sdy** study by double-clicking on it in the Project pane.
3. Skip the next 2 tasks. The next task you need to do is reviewing the tutorials.

To create a new project

1. Select **File ➡ New Project…**.
2. Click **Browse**… and navigate to the **My MPI 6.0 Projects\MPI_Fundamentals** folder.
3. Enter **synergy_practice** into the Project name field.
4. Click **OK**.

The files may not be located in the folder mentioned above. If necessary, consult with your instructor.

To import a study

1. Select **File ➡ Import…**, or, right-click the mouse in the **Project View** and select **Import…** from the top of the context menu.
2. Navigate to the **My MPI 6.0 Projects\MPI_Fundamentals\projects\extra** folder.
3. Click on the file **cover.sdy** to select it, and click **Open**.
   - The cover model will be imported into MPI.
4. Rotate the model and investigate its features.

Using the help features provided in MPI/Synergy

To review the online tutorials

1. Select **Help ➡ Tutorials**.
   - A help window opens listing the available MPI Tutorials. The Synergy window is automatically resized so that the tutorial and the Synergy window appear side by side.

2. Click on a tutorial to open it.
3. Use the navigation tools at the bottom of the tutorial window to browse the individual pages of the tutorial.
   - You can return to the Course Map by clicking the Map button in the top-left corner of the window.
4. When you have finished reviewing the tutorial contents, click the close button at the top right corner of the window.

: **To review the help homepage and help system functional tabs**

1. Select **Help ➨ Search Help…**
   - A help window opens containing the help system homepage topic.

   1.1. Explore the featured topics by clicking on the buttons on the homepage.

   1.2. To return to the homepage, click the Home button on the help toolbar.
2. Select the **Contents tab** at the top left of the help window.

   2.1. Explore the books and topics in the help contents by double-clicking on various entries.
   - ➤ The **Glossary**, and **Troubleshooting and design advice** books, for example, provide valuable general reference material about terminology, part design, and flow analysis in general.
3. Select the **Index tab** at the top left of the help window.
   - The help index provides a large selection of search terms that can assist you with finding information in the help system. **For example**, use the **Index tab** to find information about:

   3.1. Switch-over pressure, pressure control points.

   3.2. Valve gate, editing polymer timings.

   3.3. Troubleshooting, high volumetric shrinkage.

   3.4. Solving jetting.
4. Select the **Search tab** at the top left of the help window.
   - If you have difficulty finding the information you are looking for using the Contents and Index tabs, you can also use the **Search tab** to perform a full-text search of the entire online help system.
   - To refine your search, you can use multiple words or phrases in quotation marks “”. For a full list of tips about searching, use the link, **Contents tab ➨ Plastics Insight Home ➨ Using Help ➨ +Search tab ➨ Tips for searching the Help.**
5. Close the help window by clicking the close button at the top right corner of the window.

To access dialog specific help

1. Double-click the words **Process Settings** in the Study Tasks pane on the left of the application window.
   - The Process Settings Wizard dialog opens.
2. Click the **Help** button at the bottom right of the dialog.
   - A help window opens containing the help topic for the current Wizard page.
3. Click the **What's This** button  at the top right corner of the Process Settings Wizard dialog, and then click on one of the dialog controls. You can see that this provides a quick method of accessing help about a particular dialog control.
4. Close the Process Settings Wizard dialog by clicking the **Cancel** button.

To access help about toolbar items

1. Click the **What's This** tool 
   - Or select **Help ➡ What's This**.
2. Click on a particular toolbar item that you would like to be explained.

Competency check - How to use Help

Use the online help to find the Keyboard shortcuts for:	
1. Save Image to File	**1**
2. Move/Copy	**2**
3. Query Entities	**3**
4. To close a dialog without saving	**4**
5. Show Diagnostics	**5**
6. Job Manager	**6**

Name 4 of the tutorials provided with Synergy

1. ____

2. ____

3. ____

4. ____

Evaluation Sheet - How to use Help

Use the online help to find the Keyboard shortcuts for:	
1. Save Image to File	**1 Ctrl + F**
2. Move/Copy	**2 Alt +O, M**
3. Query Entities	**3 Ctrl + Q**
4. To close a dialog without saving	**4 Esc**
5. Show Diagnostics	**5 Ctrl + D**
6. Job Manager	**6 Ctrl + J**

Name 4 of the tutorials provided with Synergy

1. **Getting started**
2. **Mesh Editing**
3. **Post Processing**
4. **Automating MPI**

CHAPTER 6

Quick Cool-Flow-Warp Analysis

Aim

The aim of this exercise is to complete a Cool, Flow, and Warp analysis on an imported IGES file, and to create a report.

Why do it

Performing this quick analysis will show the overall procedure for running a cool + flow +warp analysis in MPI. Every project is different, but the basic steps in this chapter are typical for any analysis project. The different stages of the analysis process will not be described in detail; however, other units in this training course will show the analysis process in more detail.

Overview

There are two different models to choose from, a model to be used with a Fusion mesh and one using a 3D mesh. In the process of completing this project, you will do the following:

1. Open a project.
2. Import an IGES file.
3. Mesh the IGES file.
4. Diagnose the mesh.
5. Fix the mesh.
6. Verify mesh quality.
7. Set the injection location.
8. Model a runner and cooling system.
9. Prepare the Cool, Flow, and Warp analysis sequence.
10. Select a material.
11. Run the analysis sequence.
12. Review results.
13. Create a report.

Practice - Quick Cool-Flow-Warp Analysis

This chapter has two models to choose from and are described below. Pick one to work on.

Table 2: Models used for the quick Cool-Flow-Warp analysis

Description	Model
Snap Cover: starts on page 33	
The **Snap cover** model uses a Fusion mesh. Use this model if you will primarily be using a mesh type of Fusion or midplane in your analysis work.	
3 Snap Cover: starts on page 53	
The **3 Snap cover** model uses a 3D mesh. Use this model if you will primarily be using a mesh type of 3D.	

Snap Cover

Open project

A project is the highest level of organization within Synergy’s project management system. All information contained within a project is stored in a single directory. You can import and analyze as many models as you wish within a single project.

In this exercise, you will open a project that has already been created for you.

To open a project

1. Click the file open icon or **File ➨ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\QuickCFW** and double click the project file **quickCFW.mpi**.

Preparing the Mesh

Import IGES

The IGES file to be imported should just contain surface entities of the plastic part itself. It must NOT be of the entire mold design. It should also be a good, fully-trimmed model of the part.

: To Import the IGES file

1. Right-click in the **Project** pane and select **Import**.
   - The Import command can also be accessed by the icon or **File ➨ Import.**
2. Click on the file **Snap Cover.IGS**, and click **Open**.
3. Select **Fusion** as the mesh type and click **OK**.
   - The imported model should look like Figure 5.

> The IGES model shown in Figure 5 has been displayed using the Net setting. IGES surfaces can also be displayed as:
>
> - Transparent.
> - Solid.
> - Transparent + Net.
> - Solid + Net.
> - Net.
>
> To try these different display options, click **File ➨ Preferences**, select the **Default Display** tab and then experiment with the **Surface** setting.

4. Right-click on the imported Study in the Project pane, and rename it **Snap Cover**.

5. Use the model manipulation tools to investigate the IGES model, such as.
   - Rotate.
   - Pan.
   - Center.
   - Zoom.

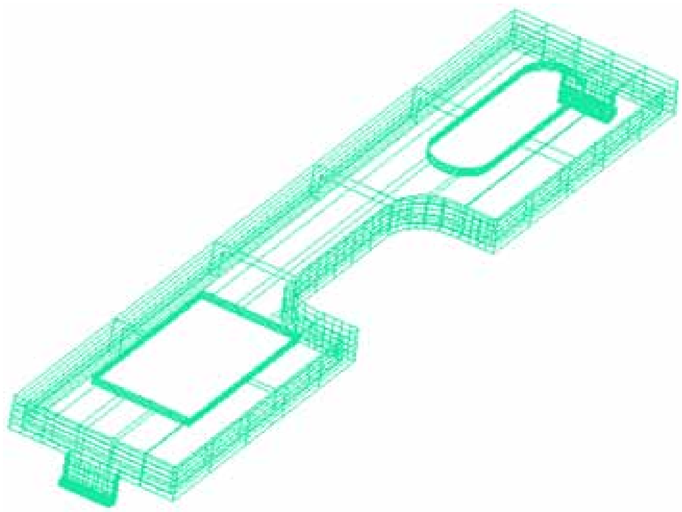


Figure 5: IGES Representation of the Snap Cover Model

Mesh IGES file

If your IGES model, when meshed, has very high aspect ratio triangles, the mesh will be distorted, and the analysis results will be less accurate. For a triangular mesh, the aspect ratio is the ratio of the length of the longest side, to the height perpendicular to that side. As a general rule, this ratio should be less than 6:1.

On the **Generate Mesh** dialog, the following **Advanced options** categories can be set:

- Edge Length.
- Mesh Control.

Edge length is the main option you can use to control and adjust the mesh density. The various mesh options are discussed further in the model translation chapter.

To mesh the IGES file

1. Click **Mesh** ➡ **Generate Mesh**, or double-click the **Mesh** icon in the Study Tasks pane.
2. Enter **3.8** (mm) in the Global edge length text box.
3. Click **Mesh Now** to begin the meshing procedure.
   - It will only take a few moments to create the meshed model. After the mesh is created, notice that there are two new layers in the **Layers** pane.

4. Check the **Logs** box.

- One is located in the study tasks list, the other is at the bottom right corner of the document window. Both do the same thing.

5. Right click on the **New Triangles** layer. Select **Hide all other layers**.

- Your model should now appear as shown in Figure 6.

> The mesh on this part is course. Normally it would have more elements. The element count has been kept down primarily to keep the analysis time very low.

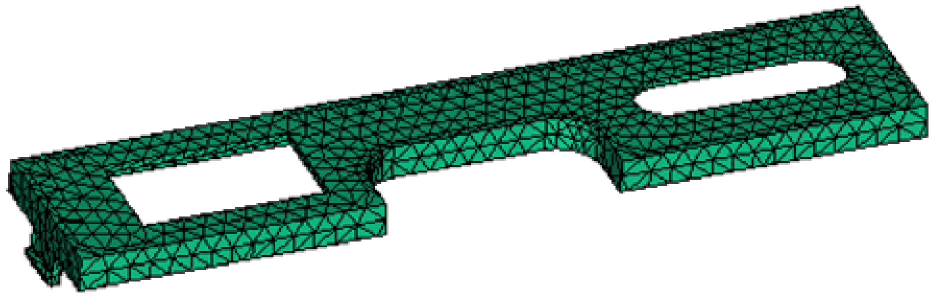


Figure 6: Meshed snap cover model

Check mesh

Once the IGES model file has been meshed, you can quickly check the quality of the mesh using a mesh statistics report. The Mesh Statistics report is divided into six main sections:

- Entity counts.
- Edge details.
- Orientation details.
- Intersection details.
- Surface triangle aspect ratio.
- Match ratio.

Generally, the Surface triangle aspect ratio and Match ratio sections are the two most critical aspects of the mesh, and the most likely to cause problems. The match ratio relates to whether the elements on the top and bottom surfaces of a Fusion mesh, match to one another spatially. A match ratio of 85% or better is critical for good flow analysis results. A match ratio of 90% or better is needed for accurate warpage results.

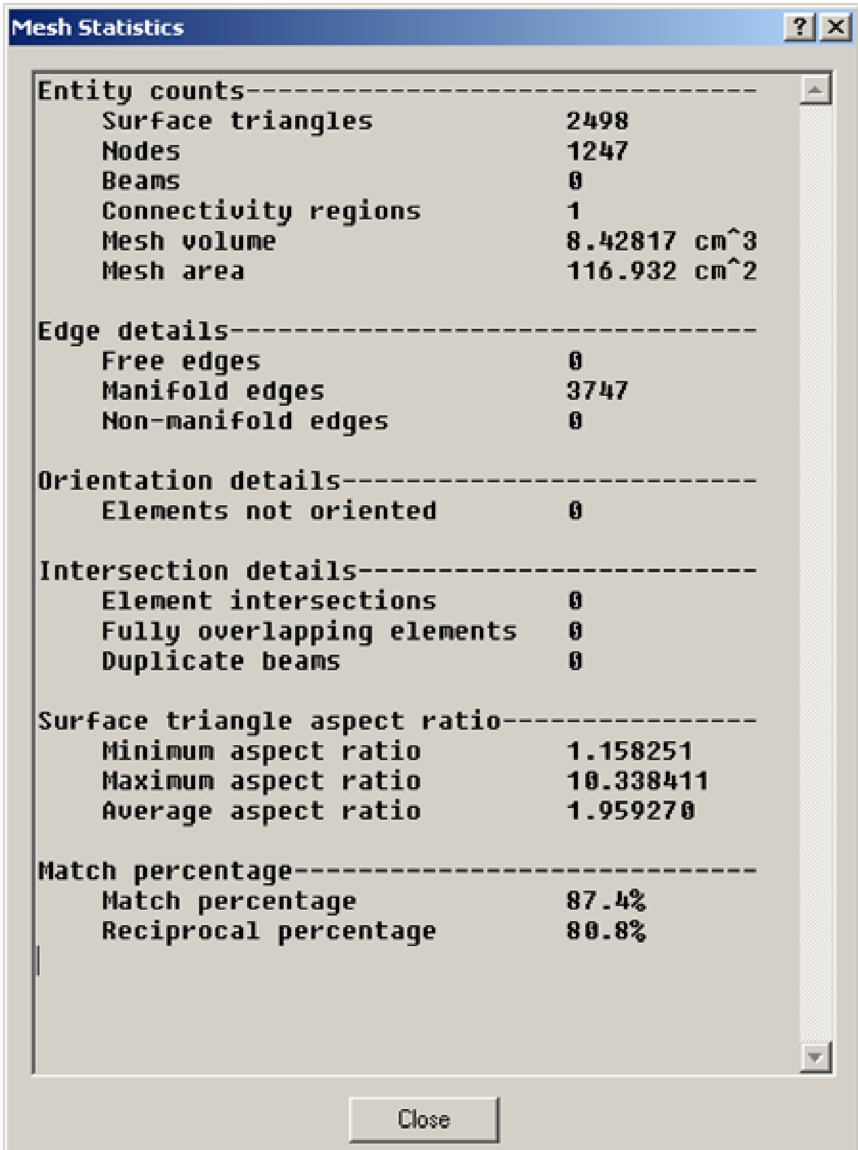


Figure 7: Mesh Statistics Report

To check the mesh for errors

1. Click **Mesh ➡ Mesh Statistics**.
2. Scan the Mesh Statistics report for any quality issues, as shown in **Figure 7** above.
   - The report indicates that the mesh is reasonably clean and problem free, although it does show that the model contains a maximum aspect ratio of above 6:1, which may affect the accuracy of the results. The mesh match ratio is also important. In this case, it is OK as the ratio is above 85%.
3. **Close** the Mesh Statistics report.

Fix the Mesh

Mesh Repair Wizard

The mesh repair wizard is a tool that can find and fix most of the problems on your model. Each page of the wizard detects a different type of problem and lets you decide if you want to fix it or not. The pages of the wizard include:

- Stitch Free Edges.
- Fill Holes.
- Overhang.
- Degenerate Elements.
- Flip Normals.
- Fix Overlap.
- Collapsed faces.
- Aspect Ratio.
- Summary.

Most models do not have serious problems that the wizard can't address at least partly. Many pages of the wizard have a tolerance the user can set in order to adjust how the part will work. This may often need be done.

To use the mesh wizard

1. Click **Mesh ➨ Mesh Repair Wizard…**.
2. Click **Next** on the **Stitch Free Edges** page as there are no problems.
3. Click **Next** on the **Fill Hole** page as there are no problems.
4. Click **Next** on the **Overhang** page as there are no problems.
5. Select **Specific** on the **Degenerate Elements** page.
6. Enter **0.7** mm as the tolerance.
7. Select the **Show Diagnostics** checkbox.
   - Elements that are less than the tolerance are displayed in blue.
   - A Diagnostic navigator toolbar is displayed.
8. Click the first diagnostic icon  on the Diagnostic navigator toolbar.
   - The model will zoom and center on the first element in the list of problems.
   - The model may not be rotated to see the element.

   8.1. Rotate the model around to see the problem.
   - ➤ Zoom in or out as necessary to see the problem.

   8.2. Click the next diagnostic icon  to view the next problem area.

   8.3. Continue to look at the problem areas.

9. Click the **Fix** button.
   - Several elements are fixed, and the diagnostic plot disappears because there are no more degenerate element problems for the specified tolerance value.
10. Click **Next.**
   - This will fix the problems (if they exist) and go to the next page.
   - The **Skip** button goes to the next page without fixing.
11. Click **Next** on the **Flip Normal** page as there are no problems.
12. Click **Next** on the **Fix Overlap** page as there are no problems.
13. Click **Next** on the **Collapsed faces** page as there are no problems.
14. Click **Next** on the **Aspect Ratio** page as there are no elements that are above 10:1.
   - The elements that still have an aspect ratio above 6:1 will be fixed manually.
15. Click **Close** on the **Summary** page after reviewing the changes made.
16. Zoom in on the problem areas of the model that the degenerate elements fixed.
   - Make sure that the geometry was not corrupted by the Mesh Repair Wizard. Any time an “Automatic” tool is used, there is a chance the model can be corrupted.

Manual Mesh Repair

To display high aspect ratio elements

1. Click the tools tab 

on the panel on the left of the screen.
2. Click the mesh diagnostic icon  in the toolbox and select **Aspect ratio diagnostic**.
   - This diagnostic can also be opened by the command **Mesh ➨ Mesh Diagnostics ➨ Aspect Ratio Diagnostic**.
3. Ensure **6** is in the **Minimum** text box.
4. Ensure that **Display** is selected in the drop-down list
5. Click **Show**.
6. Click the first diagnostic icon  on the Diagnostic navigator toolbar.

   6.1. Rotate the model around to see the problem.
   - ➤ Zoom in or out as necessary to see the problem.

   6.2. Click the next diagnostic icon  to view the next problem area.

   6.3. Continue to look at the problem areas.

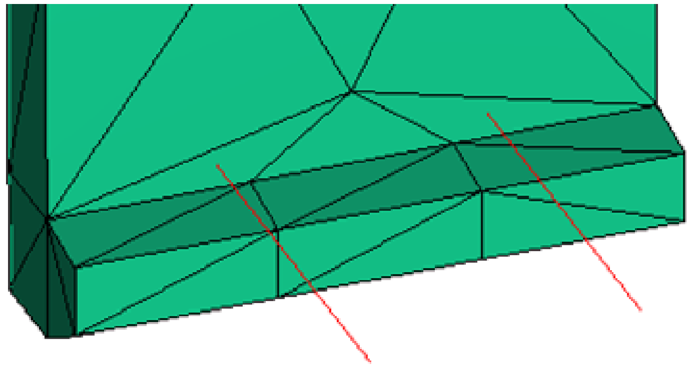


Figure 8: High aspect ratio elements

In Figure 8 above, there is a colored line pointing normal to the element. The color of the lines indicates the severity of the problem. Red lines highlight the highest aspect ratio elements, and blue lines the lowest aspect ratios above the specified threshold value of 6:1.

Fix Mesh Aspect Ratio

As a general rule, the ratio of the longest edge of an element to its height (aspect ratio) should be less than 6:1. Aspect ratios greater than 6:1 should not be used because high aspect ratios can cause solution convergence problems and affect the accuracy of the results. Long thin elements should be avoided when the pressure, temperature and velocity of the flow might vary rapidly.

To fix the elements with an aspect ratio greater than 6:1, use the diagnostic navigator to go to the first problem. In the toolbox on the Tools tab, there are four groups of tools commonly used to fix meshes. These include:

-  Create/Beam/Tri/Tetra.
-  Nodal Mesh Tools.
-  Edge Mesh Tools.
-  Global Mesh Tools.

If there is a significant number of elements with problems to be fixed, first move the problem elements to a **Diagnostic results** layer by checking **Place results in diagnostics layer**. Next use the **Expand** command in the **Layers** pane with the default level of 1 to show the elements directly surrounding the problem elements. Once the problem elements are identified, you can use the mesh tools to fix the problem and repair the mesh.

For more information on checking and correcting mesh problems, enter **Mesh** on the Index tab of the online help.

To fix the high aspect ratio elements

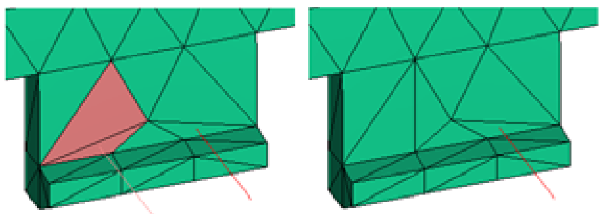
1. Click the first diagnostic icon on the Diagnostic navigator toolbar.
2. Rotate and zoom the part as necessary to see the problem.
3. Click the edge mesh tools icon in the toolbox, then select **Swap edge**.
   - Swap edge fixes problems by taking two elements that share an edge and re-meshing the elements.

   3.1. Click on the high aspect ratio element.

   3.2. Click on the element above the selected element, sharing the edge.

   3.3. Click **Apply** on the Tools pane.

   - Figure 9 shows elements before and after swapping.



Selected elements before swapping **Elements After swapping**

Figure 9: Swapping edges

4. Fix all other high aspect ratio elements.
   - Use the diagnostic navigator to move to the other elements.
   - When all element aspect ratios are below 6:1, the legend on the right side of the screen disappears.
5. Click the save icon or click **File ➡ Save Study.**

.

Depending on how you are zooming in and out on your part, you may loose the centering on an area of interest. The dynamic navigator helps prevent this situation.

In all cases, an easy way to zoom in an area to make it easier to work on is to use the **Center** command followed by the **Dynamic zoom** command .This will quickly magnify the required area. The **Center** command ensures that when an area is magnified, or the part is rotated, that area will not rotate off the screen.

To make it easy to use the **Center** and **Dynamic zoom** commands, program the mouse to use those commands.

The **Apply** action on the Tools pane can also be performed using the mouse by assigning this action on the **Mouse** tab of the **Preferences** dialog. A useful mapping for this “**Mouse Apply**” action is **Right + CTRL**.

The **Apply** action can also be performed via the context menu that appears when you right-click in the model display window.

On tools panel, a yellow highlight is used to indicate which text box is currently active. Many dialogs have automatic text box switching. For instance, when using the Swap Edge command, the focus (highlight) will automatically move from the 1st to the 2nd text box after you select the first triangle.

To recheck the model

1. Click **Mesh ➡ Mesh Statistics**.

2. Verify that there are no problems.

2.1. Be sure to check orientation details.

➤ Often in the process of fixing a mesh the orientation becomes inconsistent.

2.2. If necessary, fix the element orientations using the menu command **Mesh ➡ Orient All**.

Verify the mesh quality

Even though the mesh statistics indicated the mesh was good, there are two diagnostics that you should review: the **Thickness** diagnostic, and the **Fusion Mesh Match** diagnostic. The statistics indicate the match ratio is acceptable because it is above 90%, but this plot will show where the elements are not matched.

To plot mesh thickness

1. Click the Mesh diagnostic icon  in the toolbox, then select **Thickness Diagnostic**.
2. Click Show.
3. The thickness range on the part should be 1.1 mm to 1.7 mm.
4. Click query result icon  on the Results or Viewer toolbars, or use the command **Results ➨ Query Result**.

   4.1. Hold down the CTRL key and click on several locations on the model.

   - A label will appear indicating the thickness at the location picked. Holding the CTRL key will allow more than one label at a time to be displayed.

To plot mesh match ratio

1. Click the Mesh diagnostic icon  in the toolbox, then select **Fusion Mesh Match Diagnostic**.
2. Click Show.
   - The blue areas are matched and must be the vast majority of the part.
   - The Red areas are not matched. Elements that are on a face that are opposite an intersecting wall should not be matched.
   - Green areas are elements that represent the "thickness" or edge of the part.
   - Areas of non-uniform matching could possibly be improved by meshing finer.

Create the runner and cooling systems

Create runners

The next task requires you to create a runner system for a single-cavity tool. You will do this by setting an injection location on the model, and then using the Runner System Wizard to create the runner with a diameter of 4 mm and a gate orifice of 1.5 mm. The injection location should be set in the position indicated by the yellow cone in Figure 10.

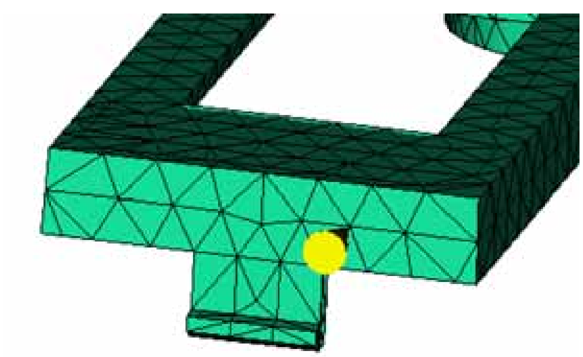


Figure 10: Snap cover injection location

To create the runner

1. Enter a rotation of **-30 70 50** in the **Enter rotation angles** field on the viewpoint toolbar.
2. Zoom in as necessary to see the injection location as shown in Figure 10.
3. Double-click the injection location icon on the Study Tasks list.
4. Set the injection location as shown in Figure 10.
5. Click the save icon or click **File ➨ Save Study.**

To create the runner

1. Click **Modeling ➨ Runner System Wizard…**
2. Define the **sprue position**.

   2.1. Enter **0** in the **X**: text box.

   2.2. Enter –**50** in the **Y**: text box.
3. Enter **-6.35** in the **Parting Plane Z[1]**: text box.
   - This is the Z value for the bottom edge of the part.
   - This is the Z location the runners will be created in.
4. Click **Next**.
5. Define the **sprue size**.

   5.1. Enter **3.97** in the **Orifice diameter** text box.

   ➤ This is a standard sprue orifice.

   5.2. Enter **2.38** in the **Included Angle** text box.

   ➤ This is a taper for a standard sprue.

   5.3. Enter **60** in the **Length** text box.
6. Enter **4.0** in the **Runner Diameter** text box.
7. Click **Next**.
8. Define the **Gate**.

   8.1. Enter **1.5** in the Side gates **Orifice diameter** text box.

   8.2. Enter **15** in the **Included Angle** text box.

   8.3. Click the **Angle** option under the Included angle text box, and enter **45**.
9. Click **Finish** to create the runner.
10. Click the save icon or click **File ➨ Save Study.**

Create Cooling System

After creating the runner system, you should create a cooling system to effectively remove heat from the mold-cavity during the molding cycle. In this task, you will create 2 cooling lines with a 10 mm channel diameter, sitting 25 mm away from the part.

Refer to the **Preparing your model ➡ Modeling in MPI**, and **Troubleshooting and design advice ➡ Cooling system related design advice** books in the online Help **Contents** tab for more information on modeling in MPI.

To create the cooling system

1. Click **Modeling ➡ Cooling Circuit Wizard…**
2. Ensure that **10** mm is selected in the **Channel diameter** to use drop-down list.
3. Enter **25** mm in the **How far above and below part** text box.
4. Select **X-axis** aligned and click **Next**.
5. Enter **2** in the Number of channels text box.
6. Enter **30** mm in the **Distance between** channel centers text box.
7. Enter **50** mm in the **Distance to extend beyond** part text box.
8. Check the **Connect channels with hoses** box.
9. Click **Finish** to create the cooling circuit.
10. Click the save icon  or click **File ➡ Save Study**.

The runner and cooling circuit should be modeled as indicated in Figure 11.

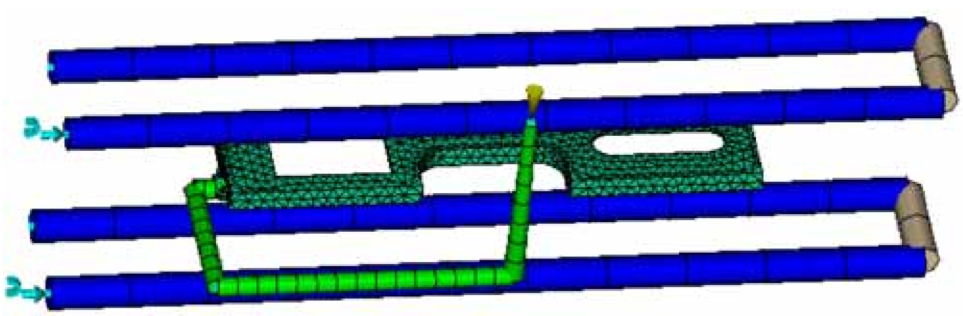


Figure 11: Snap cover modeled runner and cooling lines

Mold Boundaries

A mold boundary defines the size of the mold. From an analysis perspective, it defines the boundary of the cooling analysis. The physical size of the mold boundary is not critical. It is best and easiest to just surround the entire part and water lines with a cube boundary.

To create a mold boundary

1. Click **Modeling ➡ Mold Surface Wizard**.
2. Specify **300** mm for all dimensions.

3. Click **Finish**.

The mold boundary is created using two new layers. The first contains curves and regions, the second one contains the nodes and elements of the automatically generated mesh. It is convenient to re-organize the components of the mold boundary layers. The **Elements** layer should only contain elements and made transparent. The nodes need to be moved to the **Regions** layer.

When you create a mold boundary in the shape of a cube, the number of elements in the resultant mesh is minimized. The mold boundary is meshed by placing 5 rows of elements along the narrowest side, and using that edge length for the other sides. Since the sides are of equal length, the number of elements produced is the minimum possible.

To clean up the layers

1. Highlight the **Mold block surface (default) (Elements)** layer.
2. Right-click and select **Hide all other layers**.
3. Type **Ctrl + B**, or click **Edit ➡ Select by ➡ Properties**.
4. Click **OK**.
   - Node is the first entity type in the list so it is automatically selected.
5. Highlight the **Mold block surface (default) (Regions)** layer and click the assign icon .
6. Highlight the **Mold block surface (default) (Elements)** layer.
7. Click the **Layer display** icon .
8. Select **Triangle element** from the Entity type pull down menu.
9. Select **Transparent** in the **Show as** field and click **Close**.
10. Turn on and off layers.

    10.1. Turn off the Mold block layers.

    10.2. Turn on the layers:

    - New triangles.
    - Runner System.
    - Channel (default) #1.
    - Channel (default) #2.
11. Click the save icon or click **File ➡ Save Study**.

Running the analysis

Analysis sequence

In this task, you will set the analysis sequence to be performed on the model. In this exercise, you will run a **Cool ➡ Flow ➡ Warp** analysis sequence. The analysis sequence can be set either by selecting a command in the Analysis menu, or by double-clicking on the Analysis Sequence icon in the Study Tasks pane.

To set the analysis sequence

1. Double-click the **Analysis Sequence** icon in the Study Tasks pane.
2. Select the **Cool + Flow + Warp** analysis sequence.
3. Click **OK**.

The newly selected sequence is updated in the Study Tasks pane, and the corresponding icons are updated in the Project View pane.

The Select Analysis Sequence dialog may not display all the available analysis sequences by default. Click **More…** to view the full list of available analysis sequences and, if desired, add additional sequences to the default list.

Select Material

The next pre-processing task is to select a material for analysis. For this design, you will select the polymer named Cycolac GPT5500: GE Plastics (USA). Similar to the analysis sequence selection task in the previous step, you can do this either by selecting a command in the Analysis menu, or by double-clicking on the Select Material icon in the Study Tasks pane. Refer to the on-line Help for more information on material selection and properties.

To select a material

1. Double-click the **Select Material** icon in the Study Tasks pane.
2. Click the **Manufacturer** drop-down list and select **GE Plastics (USA)**.
3. Click the **Trade name** drop-down list and select **Cycolac GPT5500**.
4. Click **OK**.

Process Settings

In this task, you will specify the process settings for analysis. If you look at the Process Settings icon in the Study Tasks pane, you will see that this task already has a green check mark. This is because every analysis has a set of default inputs based on the material that you selected. This is also indicated by the word “(Default)” in the name of this Study Task item. For this quick introductory analysis, you will accept the default mold surface temperature, melt temperatures and mold-open time, and specify an injection time and injection + packing + cooling time.

To specify the process settings

1. Double-click the **Process Settings** icon in the Study Tasks pane.
   - The Process Settings Wizard opens at the **Cool Settings** page, which shows the analysis settings for the first analysis in the currently selected analysis sequence.
2. Ensure **Specified** is selected in the **Injection + packing + cooling** time drop-down list, and enter **15** in the associated text box to the right.
3. Click **Next**.
4. Select **Injection time** in the **Filling Control** drop-down list.

   4.1. Enter **1** in the time text box.
5. Click **Next**.
6. Check:

   6.1. The **Isolate cause of warpage** box.

   6.2. Click **Finish**.
   - Notice that the word **(User)** appears in the name of the process settings task in the Study Tasks pane. This indicates that you have changed one or more of the default settings.

Analyze

You have now performed all of the pre-processing tasks for this model. In this task, you will perform the analysis, and view the screen output file as the analysis proceeds. This file allows you to check the inputs that were specified, the current progress of the analysis, the status of the analysis as indicated by any warning or error messages, and also some text-based results.

To run the analysis

1. Double-click the **Analyze Now** icon in the Study Tasks pane.
2. Click **OK** in the **Select Analysis Type** prompt.
   - This box may not appear. It depends on your preferences.
3. Look at the screen output file to track the analysis progress.
   - The analysis should only take a couple of minutes to complete.
4. Turn off the log files when the analysis is complete.

Review the Results

In this section, you will use the post-processing features of MPI to view the following results:

- Temperature (top), part result.
- Fill time result.
- Bulk temperature result.
- Pressure result.

- Volumetric shrinkage (at ejection) result.
- Deflection, all effects and variants results.

The **online Help** provides information on interpreting each of the results created by MPI. To access this information, first display the result that you are interested in reviewing, click on the results display window to select it, and then press **F1** to open the help for that specific result.

To display cooling results

1. Deselect the Channel and Runner layers in the Layers pane so that you can see the part clearly.
2. Select the **Temperature (top), part** cooling analysis result in the Study Tasks pane.
   - This plot represents the cycle-averaged mold surface temperature on the part.
3. Click the **Vertical Split** icon to split the window into two.

Make sure that the logs are unchecked so the Split windows icons are available.

4. Display the **Temperature (top), part** result in the second window and view both sides of the part.
   - Notice how the core side runs hotter than the cavity side.
5. Click the **Vertical Split** icon again to remove the split window display.
6. Save an image for the report.
   - **6.1.** Rotate and scale the part to clearly see important aspects of the part.
   - **6.2.** Right-click over the displayed results' name in the study tasks list and select **Add Report image**.
   - **6.3.** Click **OK** to accept the default name.
   - **6.4.** Click **OK** to accept the screen shot properties.
   - An image will be created and placed in the study tasks list under the result's name.

To display Fill time results

1. Click on the **Runner system** layer.
2. Select the **Fill time** result, click the animate icon it and check the filling pattern.
   - With this plot you can check for balanced flow within the part, what areas fill early or late, where weld lines and air traps will form, etc. This is one of the most widely used plots.

3. Save an image for the report.

    3.1. Rotate and scale the part to clearly see important aspects of the part.

    3.2. Right-click over the displayed results' name in the study tasks list and select **Add Report image**.

    3.3. Click **OK** to accept the default name.

    3.4. Click **OK** to accept the screen shot properties.

    - An image will be created and placed in the study tasks list under the result's name.

To display temperature results

1. Select the **Bulk temperature (end of filling)** result and investigate the temperatures.

    - This represents the velocity-weighted temperature average through the thickness of the polymer at the end of fill. Because of the velocity weighting, areas of low temperature have a very low velocity, and areas of high temperature have a high velocity.

2. Save an image for the report.

    2.1. Rotate and scale the part to clearly see important aspects of the part.

    2.2. Right-click over the displayed results' name in the study tasks list and select **Add Report image**.

    2.3. Click **OK** to accept the default name.

    2.4. Click **OK** to accept the screen shot properties.

    - An image will be created and placed in the study tasks list under the result's name.

To display pressure results

1. Select the **Pressure** result and animate it to view the pressure history.

    - This result shows how the pressure in the part changes over time, from the beginning of fill to ejection.

2. Save an image for the report.

    2.1. Rotate and scale the part to clearly see important aspects of the part.

    2.2. Right-click over the displayed results' name in the study tasks list and select **Add Report image**.

    2.3. Click **OK** to accept the default name.

    2.4. Click **OK** to accept the screen shot properties.

    - An image will be created and placed in the study tasks list under the result's name.

To display volumetric shrinkage results

1. Select the **Volumetric shrinkage (at ejection)** result to view the part shrinkage.
   - This result shows the volume change of each element of the part. The trend is high shrinkage at the end of fill, and low shrinkage at the gate.
2. Save an image for the report.

   2.1. Rotate and scale the part to clearly see important aspects of the part.

   2.2. Right-click over the displayed results' name in the study tasks list and select **Add Report image**.

   2.3. Click **OK** to accept the default name.

   2.4. Click **OK** to accept the screen shot properties.

   - An image will be created and placed in the study tasks list under the result's name.

To display warpage results

1. Click **Vertical Split** and **Horizontal Split** icons to create a four-window display.
2. Use the tool **View ➡ Lock ➡ All Views**.
   - This will synchronize the rotation, panning and zooming of all windows.
3. Display the following results, beginning in the top-left window and working in a clockwise direction, as indicated in Figure 12:

   3.1. Deflection, all effects: Z Component,

   3.2. Deflection, differential cooling: Z component,

   3.3. Deflection, orientation effects: Z component,

   3.4. Deflection, differential shrinkage: Z component.

4. Click the Warpage visualization tools icon on the Results toolbar.

   4.1. Click the **scale** tool.

   4.2. Set the scale to **100**.

   4.3. Uncheck **X** and **Y**.

   4.4. Set **Apply on all deflection plots in this study**.

   4.5. Click **Apply**.

   - The plot should look similar to Figure 12.

The deflection plots show how much the part is going to warp. The "all effects" plot shows the total warpage, and the others indicate the relative contribution of various causes to the total warpage.

5. Save an image for the report.

   5.1. Unsplit the window.

   5.2. Rotate and scale the part to clearly see important aspects of the part.

   5.3. Right-click over the displayed results' name in the study tasks list and select **Add Report image**.

   5.4. Click **OK** to accept the default name.

   5.5. Click **OK** to accept the screen shot properties.

   - An image will be created and placed in the study tasks list under the result's name.

   5.6. Repeat this for all 4 warp plots.

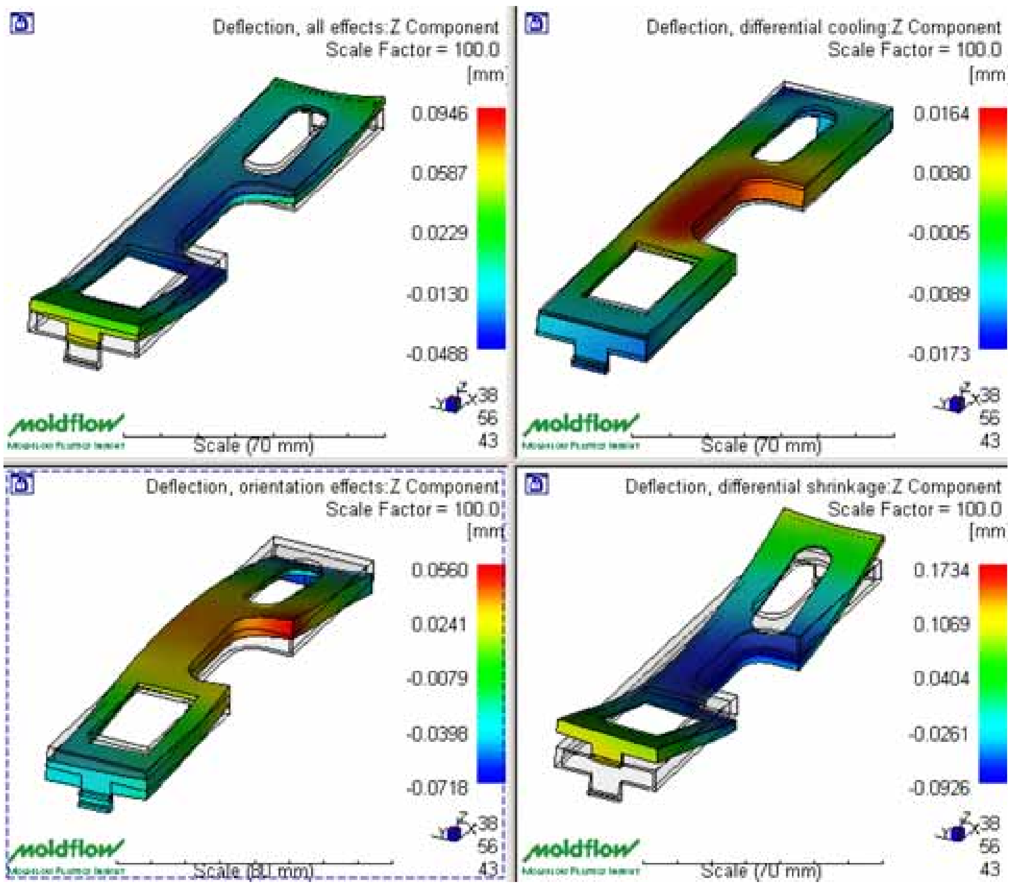


Figure 12: Warp deflection results in a split window display

Writing the report

Once you are satisfied with the analysis results, you may need to let other people know about your findings. In this task, you will finish a report based on some of the results that you have just displayed. When the images were created above, a report was automatically created. Just some details need to be added. The report will be created in HTML format, which allows for easy dissemination and viewing in an internet browser.

To view the existing report

1. Click **Report** ➨ **View**.
   - This will open the existing report in Synergy's HTML browser. The links on the left side of the form correspond to the file names of the images as they were created.
2. Scroll through the report to see the plots that were added.
3. Click the Exit icon ☒ in the upper right corner of the window when done.

To Edit the report

1. Click **Report** ➨ **Edit**.
2. Click **Next** twice.
   - The third page should be displayed.
3. Set the standard template to Contemporary.
4. Check **Cover page**.

   4.1. Click the **Properties** button next to the **Cover page** check box.

   4.2. Enter in the information.

   4.3. Click **OK**.
5. Click on each plot in the list.

   5.1. Add descriptive text for the plot.
   - When the Descriptive text box is displayed, you can't go back to Synergy to check out any information. If you need to enter detailed information, write it up in a text editor and paste it into this field.
6. Change the order of the plots by highlighting a plot and clicking the Move Down or Move Up buttons.
7. Click **Generate** to re-build the report.

3 Snap Cover

Open project

A project is the highest level of organization within Synergy's project management system. All information contained within a project is stored in a single directory. You can import and analyze as many models as you wish within a single project.

In this exercise, you will open a project that has already been created for you.

To open a project

1. Click the file open icon or **File** ➡ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Quickcfw** and double click the project file **quickcfw.mpi**.

Preparing the Mesh

Import IGES

The IGES file to be imported should just contain surface entities of the plastic part itself. It must NOT be of the entire mold design. It should also be a good, fully-trimmed model of the part.

: **To import the IGES file**

1. Right-click in the **Project** pane and select **Import**.
    - The Import command can also be accessed by the icon or **File** ➡ **Import.**
2. Click on the file **Snap3 Cover.IGS**, and click **Open**.
3. Select **Fusion** as the mesh type and click **OK**.
    - Initially, the part will be meshed like a Fusion model, then the mesh will be converted.
    - The imported model should look like Figure 13.

The IGES model shown in Figure 13 has been displayed using the Net setting. IGES surfaces can also be displayed as:

- ➤ Transparent.
- ➤ Solid.
- ➤ Transparent + Net.
- ➤ Solid + Net.
- ➤ Net.

To try these different display options, click File ➡ Preferences, select the **Default Display** tab and then experiment with the **Surface** setting.

4. Right-click on the imported Study in the Project pane, and rename it **Snap3 Cover**.
5. Use the model manipulation tools to investigate the IGES model, such as.
   - Rotate.
   - Pan.
   - Center.
   - Zoom.

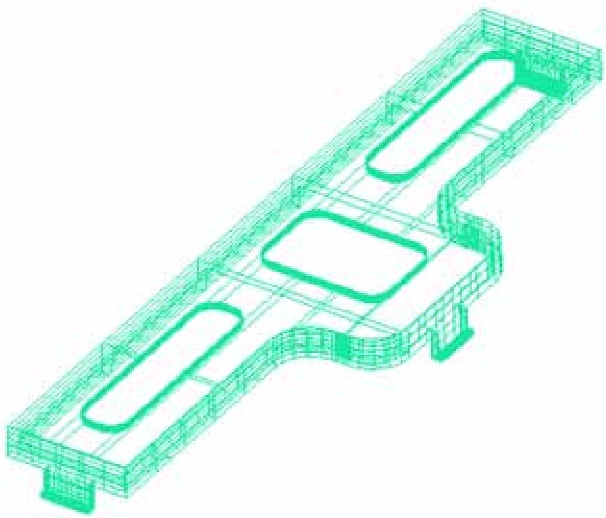


Figure 13: IGES Representation of the Snap Cover Model

Mesh IGES file

If your IGES model, when meshed, has very high aspect ratio triangles, the mesh will be distorted, and the analysis results will be less accurate. For a triangular mesh, the aspect ratio is the ratio of the length of the longest side, to the height perpendicular to that side. As a general rule, this ratio should be less than 6:1.

On the **Generate Mesh** dialog, the following **Advanced options** categories can be set:

- Edge Length.
- Mesh Control.

Edge length is the main option you can use to control and adjust the mesh density. The various mesh options are discussed further in the model translation chapter.

To mesh the IGES file

1. Click **Mesh** ➡ **Generate Mesh**, or double-click the **Mesh** icon in the Study Tasks pane.
2. Enter **3.8** (mm) in the Global edge length text box.
3. Click **Mesh Now** to begin the meshing procedure.
   - It will only take a few moments to create the meshed model. After the mesh is created, notice that there are two new layers in the **Layers** pane.

4. Ucheck the **Logs** box.
   - One is located in the study tasks list, the other is at the bottom right corner of the document window. Both do the same thing.
5. Right click on the **New Triangles** layer. Select **Hide all other layers**.
   - Your model should now appear as shown in Figure 14.

> The mesh on this part is course. Normally it would have more elements. The element count has been kept down primarily to keep the analysis time very low.

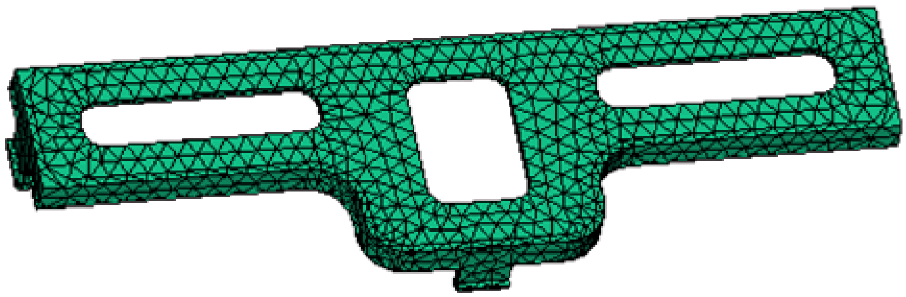


Figure 14: Meshed snap3 cover model

Check mesh

Once the IGES model file has been meshed, you can quickly check the quality of the mesh using a mesh statistics report. The Mesh Statistics report is divided into six main sections:

- Entity counts.
- Edge details.
- Orientation details.
- Intersection details.
- Surface triangle aspect ratio.
- Match ratio.

Generally, the Surface triangle aspect ratio is the most critical aspect of the mesh, and the most likely to cause problems. The aspect ratio for a Fusion mesh being converted to 3D should be well below 30:1 as a maximum. For this part, the aspect ratio will be taken below 6:1, in part to highlight some mesh cleanup tools in Synergy.

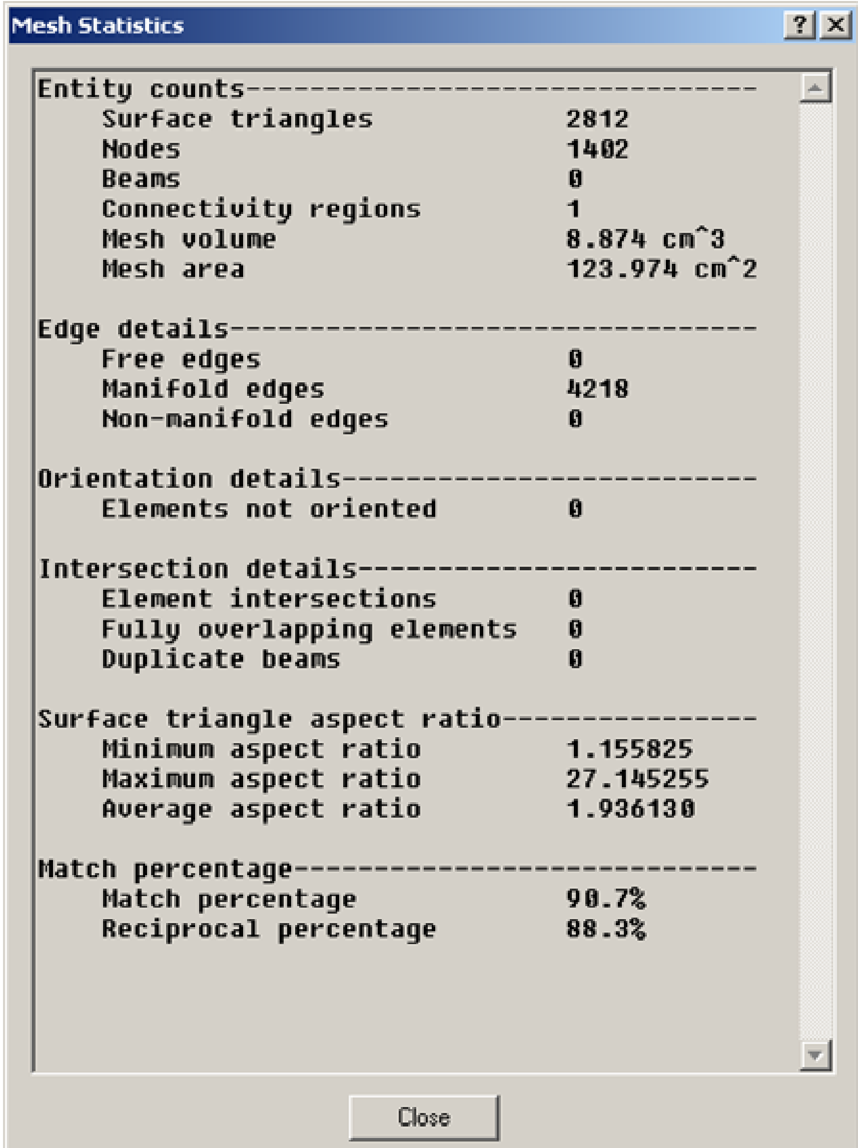


Figure 15: Mesh Statistics Report

To check the mesh for errors

1. Click **Mesh** ➡ **Mesh Statistics**.
2. Scan the Mesh Statistics report for any quality issues, as shown in **Figure 15** above.
    - The report indicates that the mesh is reasonably clean and problem free, although it does show that the model contains a maximum aspect ratio of above 6:1.
3. **Close** the Mesh Statistics report.

Fix the Mesh

Mesh Repair Wizard

The mesh repair wizard is a tool that can find and fix most of the problems on your model. Each page of the wizard detects a different type of problem and lets you decide if you want to fix it or not. The pages of the wizard include:

- Stitch Free Edges.
- Fill Holes.
- Overhang.
- Degenerate Elements.
- Flip Normals.
- Fix Overlap.
- Collapsed faces.
- Aspect Ratio.
- Summary.

Most models do not have serious problems that the wizard can't address at least partly. Many pages of the wizard have a tolerance the user can set in order to adjust how the part will work. This may often need be done.

To use the mesh wizard

1. Click **Mesh ➡ Mesh Repair Wizard…**.
2. Click **Next** on the **Stitch Free Edges** page as there are no problems.
3. Click **Next** on the **Fill Hole** page as there are no problems.
4. Click **Next** on the **Overhang** page as there are no problems.
5. Select **Specific** on the **Degenerate Elements** page.
6. Enter **0.7** mm as the tolerance.
7. Select the **Show Diagnostics** checkbox.
   - Elements that are less than the tolerance are displayed in blue.
   - A Diagnostic navigator toolbar is displayed.
8. Click the first diagnostic icon  on the Diagnostic navigator toolbar.
   - The model will zoom and center on the first element in the list of problems.
   - The model may not be rotated to see the element.

   8.1. Rotate the model around to see the problem.
   - ➤ Zoom in or out as necessary to see the problem.

   8.2. Click the next diagnostic icon  to view the next problem area.

   8.3. Continue to look at the problem areas.

9. Click the **Fix** button.
   - Several elements are fixed, and the diagnostic plot disappears because there are no more degenerate element problems for the specified tolerance value.
10. Click **Next**.
    - This will fix the problems (if they exist) and go to the next page.
    - The **Skip** button goes to the next page without fixing.
11. Click **Next** on the **Flip Normal** page as there are no problems.
12. Click **Next** on the **Fix Overlap** page as there are no problems.
13. Click **Next** on the **Collapsed faces** page as there are no problems.
14. Click **Next** on the **Aspect Ratio** page as there are no elements that are above 10:1.
    - The elements that still have an aspect ratio above 6:1 will be fixed manually.
15. Click **Close** on the **Summary** page after reviewing the changes made.
16. Zoom in on the problem areas of the model that the degenerate elements fixed.
    - Make sure that the geometry was not corrupted by the Mesh Repair Wizard. Any time an “Automatic” tool is used, there is a chance the model can be corrupted.

Manual Mesh Repair

To display high aspect ratio elements

1. Click the tools tab 

on the panel on the left of the screen.
2. Click the mesh diagnostic icon  in the toolbox and select **Aspect ratio diagnostic**.
   - This diagnostic can also be opened by the command **Mesh ➨ Mesh Diagnostics ➨ Aspect Ratio Diagnostic**.
3. Ensure **6** is in the **Minimum** text box.
4. Ensure that **Display** is selected in the drop-down list
5. Click **Show**.
6. Click the first diagnostic icon  on the Diagnostic navigator toolbar.

   6.1. Rotate the model around to see the problem.
   - ➤ Zoom in or out as necessary to see the problem.

   6.2. Click the next diagnostic icon  to view the next problem area.

   6.3. Continue to look at the problem areas.

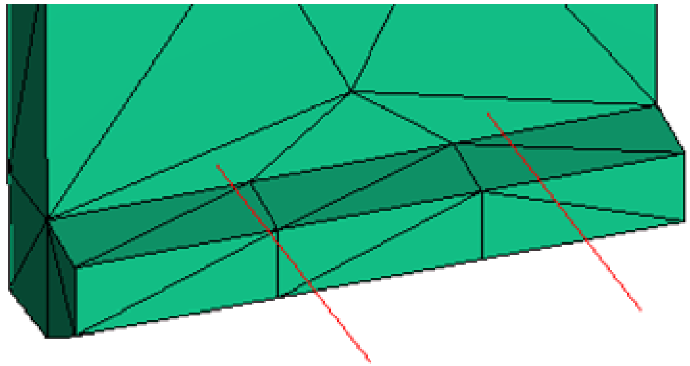


Figure 16: High aspect ratio elements

In Figure 16 above, there is a colored line pointing normal to the element. The color of the lines indicates the severity of the problem. Red lines highlight the highest aspect ratio elements, and blue lines the lowest aspect ratios above the specified threshold value of 6:1.

Fix Mesh Aspect Ratio

As a general rule, the ratio of the longest edge of an element to its height (aspect ratio) should be less than 6:1 for a Fusion mesh. When converting to 3D the aspect ratio can be as high as 30:1 but for this example it is being taken down to 6:1. Long thin elements should be avoided when the pressure, temperature and velocity of the flow might vary rapidly.

To fix the elements with an aspect ratio greater than 6:1, use the diagnostic navigator to go to the first problem. In the toolbox on the Tools tab, there are four groups of tools commonly used to fix meshes. These include:

-  Create/Beam/Tri/Tetra.
-  Nodal Mesh Tools.
-  Edge Mesh Tools.
-  Global Mesh Tools.

If there is a significant number of elements with problems to be fixed, first move the problem elements to a **Diagnostic results** layer by checking **Place results in diagnostics layer**. Next use the **Expand** command in the **Layers** pane with the default level of 1 to show the elements directly surrounding the problem elements. Once the problem elements are identified, you can use the mesh tools to fix the problem and repair the mesh.

For more information on checking and correcting mesh problems, enter **Mesh** on the Index tab of the online help.

 **To fix the high aspect ratio elements**

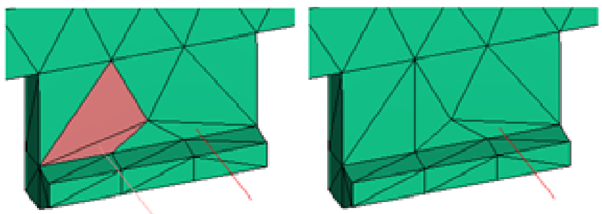
1. Click the first diagnostic icon  on the Diagnostic navigator toolbar.
2. Rotate and zoom the part as necessary to see the problem.
3. Click the edge mesh tools icon  in the toolbox, then select **Swap edge**.
   - Swap edge fixes problems by taking two elements that share an edge and re-meshing the elements.

   3.1. Click on the high aspect ratio element.

   3.2. Click on the element above the selected element, sharing the edge.

   3.3. Click **Apply** on the Tools pane.

   - Figure 17 shows elements before and after swapping.



Selected elements before swapping | **Elements After swapping**

Figure 17: Swapping edges

4. Fix all other high aspect ratio elements.
   - Use the diagnostic navigator to move to the other elements.
   - When all element aspect ratios are below 6:1, the legend on the right side of the screen disappears.
5. Click the save icon  or click **File ➡ Save Study.**

.

Depending on how you are zooming in and out on your part, you may loose the centering on an area of interest. The dynamic navigator helps prevent this situation.

In all cases, an easy way to zoom in an area to make it easier to work on is to use the **Center** command followed by the **Dynamic zoom** command .This will quickly magnify the required area. The **Center** command ensures that when an area is magnified, or the part is rotated, that area will not rotate off the screen.

To make it easy to use the **Center** and **Dynamic zoom** commands, program the mouse to use those commands.

The **Apply** action on the Tools pane can also be performed using the mouse by assigning this action on the **Mouse** tab of the **Preferences** dialog. A useful mapping for this "**Mouse Apply**" action is **Right + CTRL**.

The **Apply** action can also be performed via the context menu that appears when you right-click in the model display window.

On tools panel, a yellow highlight is used to indicate which text box is currently active. Many dialogs have automatic text box switching. For instance, when using the Swap Edge command, the focus (highlight) will automatically move from the 1st to the 2nd text box after you select the first triangle.

To recheck the model

1. Click **Mesh ➡ Mesh Statistics**.

2. Verify that there are no problems.

2.1. Be sure to check orientation details.

- Often in the process of fixing a mesh the orientation becomes inconsistent.

2.2. If necessary, fix the element orientations using the menu command **Mesh ➡ Orient All**.

Create a 3D mesh

Now that the surface mesh has been created, it must be converted into a 3D mesh.

To create the 3D mesh

1. Click the **Tasks** tab on the panel.
2. Right-click on the **Fusion Mesh** icon in the Study tasks.

   2.1. Highlight **Set Mesh Type**.

   2.2. Select **3D**.
3. Double-click the 3D Mesh icon .
   - Notice the Edge length parameters are unchanged.
4. Click the **Tetra Refinement** button.
   - This page controls the creation of the 3D tetrahedral mesh.
   - No adjustments are to be made.
5. Click **Mesh Now**.
   - The conversion process will take a few seconds to complete.

Mesh Repair Wizard

The mesh repair wizard is a tool that can find and fix most of the problems on your 3D model. Each page of the wizard detects a different type of problem and lets you decide if you want to fix it or not. This will find issues that are related strictly to a 3D mesh. The first page of the wizard lists all the items to be checked. Unchecking an item in the list will stop the wizard for looking at that item. By default all the items are checked. The wizard checks and repairs:

- Inverted tetras.
- Collapsed faces.
- Insufficient refinement through the thickness.
- Internal long edges.
- Tetras with extremely large volumes.
- Tetras with high aspect ratios.
- Tetras with a small angle between faces.

Most models do not have serious problems that the wizard can’t address at least partly. Many pages of the wizard have a tolerance the user can set in order to adjust how the part will work. This may often need to be done.

To use the mesh wizard

1. Click **Mesh** ➡ **Mesh Repair Wizard…**.
2. Click **Next** on the **Options** page.

3. Examine the **Overview** page, then click **Next** when done.

4. Click **Next** on the **Inverted tetras** page.

5. Click **Next** on the **Collapsed faces** page.

6. Click **Next** on the **Insufficient layers through thickness** page.

7. Click **Next** on the **Internal long edges** page.

8. Click **Next** on the **Tetras with Large Volume** page.

9. On the **Aspect ratio (tetras)** page:

   9.1. Set the aspect ratio tolerance to **40**.

   9.2. Click **Update**.

   9.3. Click **Fix**.

   9.4. Repeat clicking **Fix** until there are **0** elements with an aspect ratio higher than 40 or no more elements are being fixed with a click of the Fix button.

   - An aspect ratio of 50 is good. It was changed primarily to show the process of fixing a problem.

10. On the **Included angles of tetras** page:

    10.1. Click **Fix**.

    10.2. Repeat clicking **Fix** until there are **0** elements with an included angle less than 2 degrees or no more elements are being fixed with a click of the Fix button.

11. Click **Next** then **Close** on the **Summary** page.

12. Click the save icon  or click **File ➡ Save Study.**

Create the runner and cooling systems

Create runners

The next task requires you to create a runner system for a single-cavity tool. You will do this by setting an injection location on the model, and then using the Runner System Wizard to create the runner with a diameter of 4 mm and a gate orifice of 1.5 mm. The injection location should be set in the position indicated by the yellow cone in Figure 18.

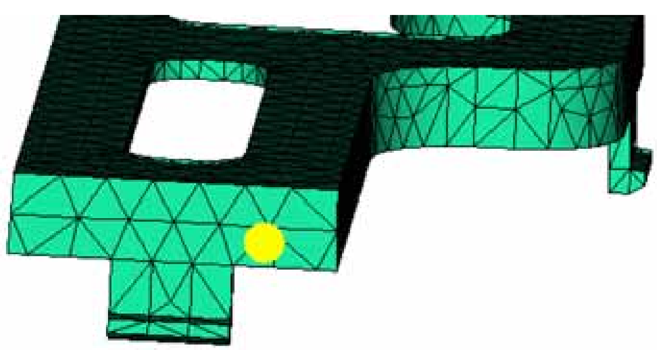


Figure 18: Snap cover injection location

To create the runner

1. Enter a rotation of **-30 70 50** in the **Enter rotation angles** field on the viewpoint toolbar.
2. Zoom in as necessary to see the injection location as shown in Figure 18.
3. Double-click the injection location icon on the Study Tasks list.
4. Set the injection location as shown in Figure 18.
5. Click the save icon or click **File ➡ Save Study.**

To create the runner

1. Click **Modeling ➡ Runner System Wizard…**
2. Define the **sprue position**.

   2.1. Enter **0** in the **X**: text box.

   2.2. Enter –**50** in the **Y**: text box.
3. Enter **-6.35** in the **Parting Plane Z[1]**: text box.
   - This is the Z value for the bottom edge of the part.
   - This is the Z location the runners will be created on.
4. Click **Next**.
5. Define the **sprue size**.

   5.1. Enter **3.97** in the **Orifice diameter** text box.

   ➤ This is a standard sprue orifice.

   5.2. Enter **2.38** in the **Included Angle** text box.

   ➤ This is a taper for a standard sprue.

   5.3. Enter **60** in the **Length** text box.
6. Enter **4.0** in the **Runner Diameter** text box.
7. Click **Next**.
8. Define the **Gate**.

   8.1. Enter **1.5** in the Side gates **Orifice diameter** text box.

   8.2. Enter **15** in the **Included Angle** text box.

   8.3. Click the **Angle** option under the Included angle text box, and enter **45**.
9. Click **Finish** to create the runner.
10. Click the save icon or click **File ➡ Save Study.**

Create Cooling System

After creating the runner system, you should create a cooling system to effectively remove heat from the mold-cavity during the molding cycle. In this task, you will create 2 cooling lines with a 10 mm channel diameter, sitting 25 mm away from the part.

> Refer to the **Preparing your model ➨ Modeling in MPI**, and **Troubleshooting and design advice ➨ Cooling system related design advice** books in the online Help **Contents** tab for more information on modeling in MPI.

To create the cooling system

1. Click **Modeling ➨ Cooling Circuit Wizard…**
2. Ensure that **10** mm is selected in the **Channel diameter** to use drop-down list.
3. Enter **25** mm in the **How far above and below part** text box.
4. Select **X-axis** aligned and click **Next**.
5. Enter **2** in the Number of channels text box.
6. Enter **30** mm in the **Distance between** channel centers text box.
7. Enter **50** mm in the **Distance to extend beyond** part text box.
8. Check the **Connect channels with hoses** box.
9. Click **Finish** to create the cooling circuit.
10. Click the save icon or click **File ➨ Save Study.**

The runner and cooling circuit should be modeled as indicated in Figure 19.

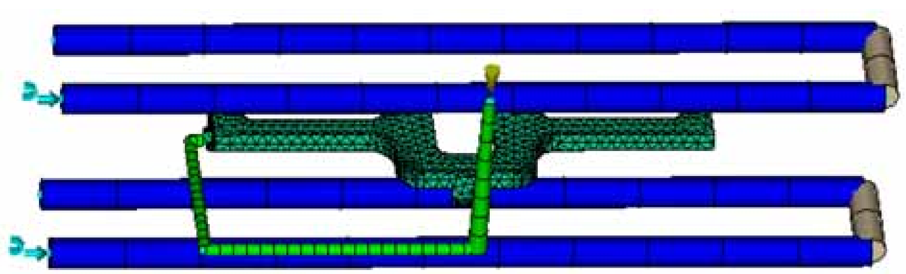


Figure 19: Snap cover modeled runner and cooling lines

Mold Boundaries

A mold boundary defines the size of the mold. From an analysis perspective, it defines the boundary of the cooling analysis. The physical size of the mold boundary is not critical. It is best and easiest to just surround the entire part and water lines with a cube boundary.

To create a mold boundary

1. Click **Modeling ➨ Mold Surface Wizard**.
2. Specify **300** mm for all dimensions.
3. Click **Finish**.

The mold boundary is created using two new layers. The first contains curves and regions, the second one contains the nodes and elements of the automatically generated mesh. It is convenient to re-organize the components of the mold boundary layers. The **Elements** layer should only contain elements and made transparent. The nodes need to be moved to the **Regions** layer.

When you create a mold boundary in the shape of a cube, the number of elements in the resultant mesh is minimized. The mold boundary is meshed by placing 5 rows of elements along the narrowest side, and using that edge length for the other sides. Since the sides are of equal length, the number of elements produced is the minimum possible.

To clean up the layers

1. Highlight the **Mold block surface (default) (Elements)** layer.
2. Right-click and select **Hide all other layers**.
3. Type **Ctrl + B**, or click **Edit ➡ Select by ➡ Properties**.
4. Click **OK**.
   - Node is the first entity type in the list so it is automatically selected.
5. Highlight the **Mold block surface (default) (Regions)** layer and click the assign icon .
6. Highlight the **Mold block surface (default) (Elements)** layer.
7. Click the **Layer display** icon .
8. Select **Triangle element** from the Entity type pull down menu.
9. Select **Transparent** in the **Show as** field and click **Close**.
10. Turn on and off layers.

    10.1. Turn on the layers:

    - New Tetras.
    - Runner System.
    - Channel (default) #1.
    - Channel (default) #2.

    10.2. Turn off the Mold block layers.
11. Click the save icon or click **File ➡ Save Study.**

Running the analysis

Analysis sequence

In this task, you will set the analysis sequence to be performed on the model. In this exercise, you will run a **Cool ➡ Flow ➡ Warp** analysis sequence. The analysis sequence can be set either by selecting a command in the Analysis menu, or by double-clicking on the Analysis Sequence icon in the Study Tasks pane.

To set the analysis sequence

1. Double-click the **Analysis Sequence** icon in the Study Tasks pane.
2. Select the **Cool + Flow + Warp** analysis sequence.
3. Click **OK**.

The newly selected sequence is updated in the Study Tasks pane, and the corresponding icons are updated in the Project pane.

The Select Analysis Sequence dialog may not display all the available analysis sequences by default. Click **More…** to view the full list of available analysis sequences and, if desired, add additional sequences to the default list.

Select Material

The next pre-processing task is to select a material for the analysis. For this design, you will select the polymer named 5824S: Huntsman Chemical Company. Similar to the analysis sequence selection task in the previous step, you can do this either by selecting a command in the Analysis menu, or by double-clicking on the Select Material icon in the Study Tasks pane. Refer to the on-line Help for more information on material selection and properties.

To select a material

1. Double-click the **Select Material** icon in the Study Tasks pane.
2. Click the **Manufacturer** drop-down list and select **Huntsman Chemical Company**.
3. Click the **Trade name** drop-down list and select **5824S**.
4. Click **OK**.

Process Settings

In this task, you will specify the process settings for the analysis. If you look at the Process Settings icon in the Study Tasks pane, you will see that this task already has a green check mark. This is because every analysis has a set of default inputs based on the material that you selected. This is also indicated by the word “(Default)” in the name of this Study Task item. For this quick introductory analysis, you will accept the default mold surface temperature, melt temperatures and mold-open time, and specify an injection time and injection + packing + cooling time.

To specify the process settings

1. Double-click the **Process Settings** icon in the Study Tasks pane.
   - The Process Settings Wizard opens at the **Cool Settings** page, which shows the analysis settings for the first analysis in the currently selected analysis sequence.
2. Ensure **Specified** is selected in the **Injection + packing + cooling** time drop-down list, and enter **15** in the associated text box to the right.
3. Click **Next**.

4. Select **Injection time** in the **Filling Control** drop-down list.

   4.1. Enter **1** in the time text box.

5. Click **Next**.
6. Uncheck **Use mesh aggregation and 2nd-order tetrahedral elements**.
7. Click **Advanced options**.

   7.1. Check **Isolate cause of warpage** box.

   7.2. Click **OK**.

8. Click **Finish**.

Analyze

You have now performed all of the pre-processing tasks for this model. In this task, you will perform the analysis, and view the screen output file as the analysis proceeds. This file allows you to check the inputs that were specified, the current progress of the analysis, the status of the analysis as indicated by any warning or error messages, and also some text-based results.

To run the analysis

1. Double-click the **Analyze Now** icon in the Study Tasks pane.
2. Click **OK** in the **Select Analysis Type** prompt.
   - This box may not appear. It depends on your preferences.
3. Look at the screen output file to track the analysis progress.
   - The analysis takes about 20-30 minutes to complete.
4. Turn off the log files when the analysis is complete.

Review the results

In this section, you will use the post-processing features of MPI to view the following results:

- Mold internal temperature result.
- Fill time result.
- Temperature (3D) result.
- Pressure result.
- Volumetric shrinkage (3D) result.
- Deflection, all effects and variants results.

The **online Help** provides information on interpreting each of the results created by MPI. To access this information, first display the result that you are interested in reviewing, click on the results display window to select it, and then press **F1** to open the help for that specific result.

To display cooling results

1. Deselect the **Channel** and **Runner layers** in the Layers pane so that you can see the part clearly.

2. Click the **Edit Cutting Plane** icon on the viewer toolbar.

   2.1. Check the **Plane YZ**.

   2.2. Uncheck the **Plane XY**

   2.3. Click **Close**.

3. Select the **Mold internal temperature** cooling analysis result in the Study Tasks pane.

   - This plot represents the cycle-averaged mold temperature across the mold.

4. Click the **Move cutting plane** icon on the viewer toolbar.

   4.1. Uncheck Show active plane.

   4.2. Click-hold and drag the left mouse button up and down to move the cutting plane.

   4.3. Rotate the part as necessary to see both the cavity and core sides of the part.

5. Save an image for the report.

   5.1. Rotate, scale the part and move the cutting plane to clearly see important aspects of the part.

   5.2. Right-click over the displayed results' name in the study tasks list and select **Add Report image**.

   5.3. Click **OK** to accept the default name.

   5.4. Click **OK** to accept the screen shot properties.

   - An image will be created and placed in the study tasks list under the result's name.

6. Click the **Edit Cutting Plane** icon on the viewer toolbar.

   6.1. Uncheck the **Plane YZ**.

   6.2. Click **Close**.

To display fill time results

1. Click on the **Runner system** layer.

2. Select the **Fill time** result, click the animate icon and check the filling pattern.

   - With this plot you can check for balanced flow within the part, what areas fill early or late, where weld lines and air traps will form, etc. This is one of the most widely used plots.

3. Save an image for the report.

   3.1. Rotate and scale the part to clearly see important aspects of the part.

   3.2. Right-click over the displayed results' name in the study tasks list and select **Add Report image**.

   3.3. Click **OK** to accept the default name.

   3.4. Click **OK** to accept the screen shot properties.

   - An image will be created and placed in the study tasks list under the result's name.

To display temperature results

1. Select the **Temperature (3D)** result and investigate the temperatures.
   - This represents the temperature across the thickness of the part at several times during the cycle (animated through time).
2. Turn on the cutting plane in the **ZX** direction.
3. Move the cutting plane across the part.
4. Animate the results with the cutting plane set at a location of your choosing.
5. Save an image for the report.

   5.1. Rotate and scale the part to clearly see important aspects of the part.

   5.2. Right-click over the displayed results' name in the study tasks list and select **Add Report image**.

   5.3. Click **OK** to accept the default name.

   5.4. Click **OK** to accept the screen shot properties.

   - An image will be created and placed in the study tasks list under the result's name.

6. Click the **Edit Cutting Plane** icon  on the viewer toolbar.

   6.1. Uncheck the **Plane ZX**.

   6.2. Click **Close**.

To display pressure results

1. Select the **Pressure** result and animate it to view the pressure history.
   - This result shows how the pressure in the part changes over time, from the beginning of fill to ejection.

2. Save an image for the report.

   2.1. Rotate and scale the part to clearly see important aspects of the part.

   2.2. Right-click over the displayed results' name in the study tasks list and select **Add Report image**.

   2.3. Click **OK** to accept the default name.

   2.4. Click **OK** to accept the screen shot properties.

   - An image will be created and placed in the study tasks list under the result's name.

To display volumetric shrinkage results

1. Select the **Volumetric shrinkage (3D)** result to view the part shrinkage.

   - This result shows the volume change of each element of the part. The trend is high shrinkage at the end of fill, and low shrinkage at the gate.

2. Turn on the cutting plane in the ZX direction.

3. Move the cutting plane across the part.

4. Save an image for the report.

   4.1. Rotate and scale the part to clearly see important aspects of the part.

   4.2. Right-click over the displayed results' name in the study tasks list and select **Add Report image**.

   4.3. Click **OK** to accept the default name.

   4.4. Click **OK** to accept the screen shot properties.

   - An image will be created and placed in the study tasks list under the result's name.

5. Click the **Edit Cutting Plane** icon on the viewer toolbar.

   5.1. Uncheck the Plane ZX.

   5.2. Click **Close**.

To display warpage results

1. Click **Vertical Split** and **Horizontal Split** icons to create a four-window display.

2. Use the tool **View ➨ Lock ➨ All Views**.

   - This will synchronize the rotation, panning and zooming of all windows.

3. Display the following results, beginning in the top-left window and working in a clockwise direction, as indicated in Figure 20:

   3.1. Deflection, all effects: Z Component.

   3.2. Deflection, differential cooling: Z component.

   3.3. Deflection, differential shrinkage: Z component.

   3.4. Deflection, orientation effects: Z component.

4. Click the Warpage visualization tools icon  on the Results toolbar.

4.1. Click the scale tool.

4.2. Set the **scale** to **20**.

4.3. Uncheck **X** and **Y**.

4.4. Set **Apply on all deflection plots in this study**.

4.5. Click **Apply**.

- The plot should look similar to Figure 20.

The deflection plots show how much the part is going to warp. The “all effects” plot shows the total warpage, and the others indicate the relative contribution of various causes to the total warpage. Orientation is zero because the material is not fiber filled. If the material were fiber filled and the fiber flow analysis was run, then this plot would show the influence of the fiber filler.

5. Save an image for the report.

5.1. Unsplit the window.

5.2. Rotate and scale the part to clearly see important aspects of the part.

5.3. Right-click over the displayed results’ name in the study tasks list and select **Add Report image**.

5.4. Click **OK** to accept the default name.

5.5. Click **OK** to accept the screen shot properties.

- An image will be created and placed in the study tasks list under the result’s name.

5.6. Repeat this for all 4 warp plots.

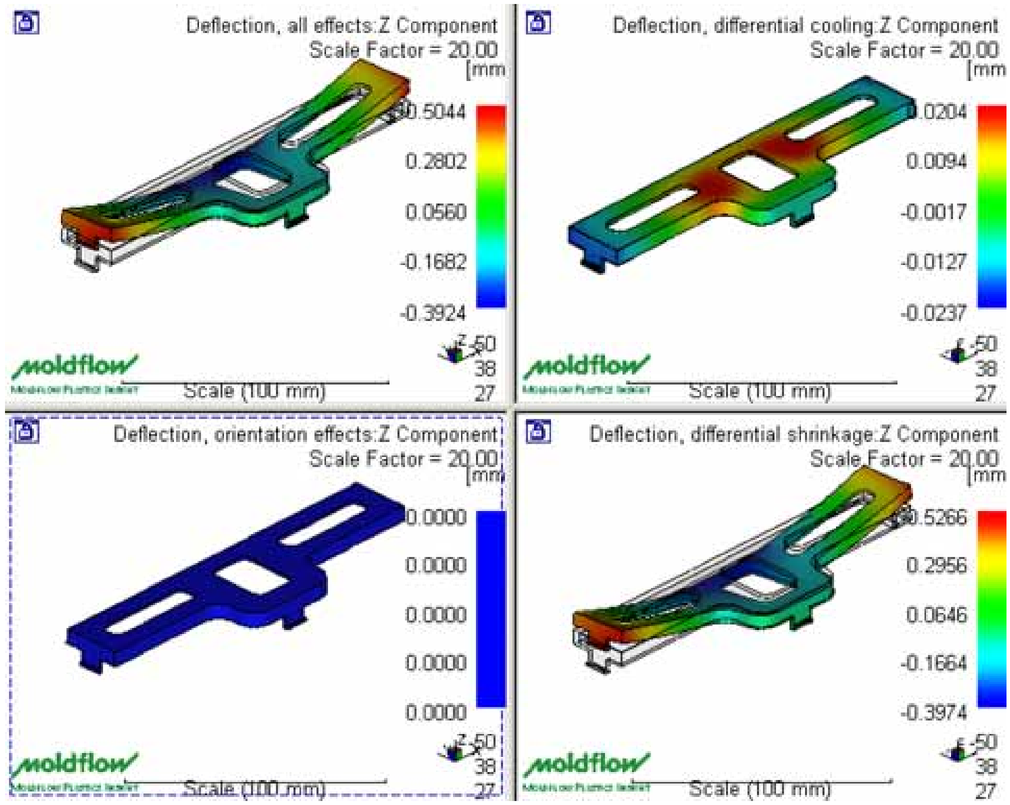


Figure 20: Warp deflection results in a split window display

Writing the report

Once you are satisfied with the analysis results, you may need to let other people know about your findings. In this task, you will finish a report based on some of the results that you have just displayed. When the images were created above, a report was automatically created. Just some details need to be added. The report will be created in HTML format, which allows for easy dissemination and viewing in an internet browser.

To view the existing report

1. Click **Report** ➡ **View**.
   - This will open the existing report in Synergy's HTML browser. The links on the left side of the form correspond to the file names of the images as they were created.
2. Scroll through the report to see the plots that were added.
3. Click the Exit icon in the upper right corner of the window when done.

To Edit the report

1. Click **Report** ➡ **Edit**.
2. Click **Next** twice.
   - The third page should be displayed.

3. Set the standard template to Contemporary.
4. Check **Cover page**.

   4.1. Click the **Properties** button next to the **Cover page** check box.

   4.2. Enter in the information.

   4.3. Click **OK**.
5. Click on each plot in the list.

   5.1. Add descriptive text for the plot.

   - When the Descriptive text box is displayed, you can't go back to Synergy to check out any information. If you need to enter detailed information, write it up in a text editor and paste it into this field.
6. Change the order of the plots by highlighting a plot and clicking the Move Down or Move Up buttons.
7. Click **Generate** to re-build the report.

Competency check - Quick Cool-Flow-Warp Analysis

Question:
When should the mesh be checked for problems and why?
Answer:

For Fusion, What analysis type (Cool, Flow or Warp, create the following results?	
• Frozen layer fraction	
• Average temperature, part	
• Bulk temperature	
• Deflection, all effects: Deflection	
• Volumetric shrinkage	
• Temperature, (top), part	
• Maximum temperature, part	
• Time to freeze	
• Temperature profile, part	

For 3D, What analysis type (Cool, Flow or Warp, create the following results?	
• Deflection, all effects:Z Component	
• Time to freeze, part (3D)	
• Freeze time	
• Temperature, part	
• Velocity (3D)	
• Pressure	
• Temperature, part	

Evaluation Sheet - Quick Cool-Flow-Warp Analysis

Question:
When should the mesh be checked for problems and why?
Answer:
It should always be checked after meshing the part. It could also be checked after the mesh has been fixed to make sure all errors have been addressed and no new errors are present. The mesh needs to be checked for problems because a bad mesh can affect analysis results.

For Fusion, What analysis type (Cool, Flow or Warp, create the following results?	
• Frozen layer fraction	Flow
• Average temperature, part	Cool
• Bulk temperature	Flow
• Deflection, all effects: Deflection	Warp
• Volumetric shrinkage	Flow
• Temperature, (top), part	Cool
• Maximum temperature, part	Cool
• Time to freeze	Flow
• Temperature profile, part	Cool

For 3D, What analysis type (Cool, Flow or Warp, create the following results?	
• Deflection, all effects:Z Component	Warp
• Time to freeze, part (3D)	Cool
• Freeze time	Flow
• Temperature, part	Cool
• Velocity (3D)	Flow
• Pressure	Flow
• Temperature, part	Cool

CHAPTER 7

Flow Analysis Steps

There is no practice for this subject.

CHAPTER 8

Model Requirements

There is no practice for this subject.

CHAPTER 9

Model Translation and Cleanup

Aim

The aim of this chapter is to learn how to import a model from a CAD system, check the model for errors, and then clean up any mesh problems that may be present.

Why do it

There are many ways to get geometry into MPI for **running** an analysis. Most of them involve taking a model from a CAD system, reading it into MPI, meshing it if necessary, checking the mesh quality, and finally fixing any mesh problems. The majority of the models analyzed in MPI are imported using this method.

Overview

Translation and cleanup of a model involves these basic steps:

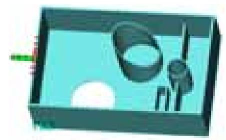
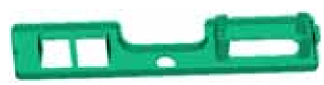
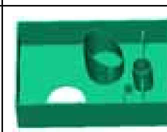
1. Import a CAD model in one of many usable formats.
2. Mesh the imported geometry.
3. Check the mesh for errors.
4. Use the Mesh Repair Wizard and mesh cleanup tools to fix the model.

These steps will be discussed in detail in this chapter. This chapter focuses on Fusion primarily but also discusses how the translation process is different between Fusion and midplane or 3D. The extra steps involved for preparing 3D or midplane models is also discussed.

Practice - Model Translation and Cleanup

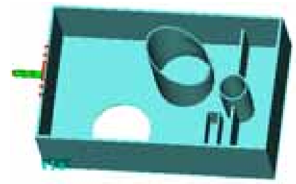
This chapter has several models that are used. The first is for mesh cleanup practice, the second section contains models to look at different mesh densities before cleaning and the last section contains optional models.

Table 3: Models used for Translation and cleanup

Description	Model
Mesh tools practice, must complete	
Housing: starts on **page 87** This part is already meshed. There are several areas on the part where the mesh needs to be fixed. This part was constructed to give you practice using many of the mesh cleanup tools discussed above.	
Primary Models, Pick one to complete	
Cover: starts on **page 103** The cover is a **Fusion** model. It will be meshed with several mesh densities then one will be used and cleaned up.	
Manifold: starts on **page 113** The manifold is a **3D** part. It is initially meshed as a Fusion model at several different densities, then will be converted to 3D and the 3D mesh will be checked and cleaned.	
Optional Models, translate for extra practice	
Snap Cover: starts on **page 123** This part has small features in it that need to be meshed with chord height. There are several versions of the part including: • Iges, with and without small unwanted radii. • STL. • STEP. • Parasolid. • Pro/ENGINEER.	
Dustpan: starts on **page 127** This part is an IGES file that will be meshed, cleaned then converted to a midplane model.	
Housing: starts on **page 129** This version of the housing is a STEP file. It will be meshed with different chord heights to see the effect on the mesh.	

Housing

This version of the housing model has been meshed for you. The part has been separated into a number of layers. Each layer has a problem that needs to be fixed using mesh cleanup tools.



Open Study

To open a project

1. Click the file open icon or **File** ➡ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Translation_cleanup**.
2. Double click the project file **Translation_cleanup.mpi**.

To open the study

1. Double click on the **Housing Cleanup** study in the Project pane.
2. Rotate, pan and zoom in and out on the model to view the geometry.
3. Turn on and off the layers to see what entities are on the various layers.
4. Click **File** ➡ **Save Study as**.

   4.1. Enter the name **Housing Cleanup Practice**.

   4.2. Click **Save**.

   - You will be practicing with a copy so the original is not touched so you can go back and finish it later.

Cleanup layer Fix 1

In the center of the edge, the mesh is not very uniform. Nodes are going to be added then aligned and swapped to fix the corner.

To prepare the model

1. Right click on the **Fix 1** layer in the layer pane.
2. Select **Hide all other layers** from the context menu.
3. Enter the rotation **46, -5, -10** into the **Enter rotation angles** field of the viewpoint toolbar.
   - Commas or spaces can be used as a delimiter when entering the rotation.
4. Click the **Fit to Window** icon to scale the visible entities.

To Insert nodes near the edge

1. Click on the **Tools** tab on the panel.
2. Click the **Nodal Mesh Tools** icon in the toolbox.

3. Select **Insert node**.
4. Refer to Figure 21A.
   4.1. Click on the node labeled 1.
   4.2. Click on the node labeled 2.
   4.3. Click **Apply**.
   4.4. Repeat for nodes:
   - 2 and 3.
   - 2 and 4
   - Nodes should have been added labeled 5 to 7 in Figure 21B.

Remember, **Apply** can be done 3 ways:

- In the tool that is in the pane.
- Right click in the display area and select Apply from the context menu.
- If your mouse was programmed with the command Mouse Apply. A handy mouse combination for Mouse Apply is the Ctrl + Right mouse click.

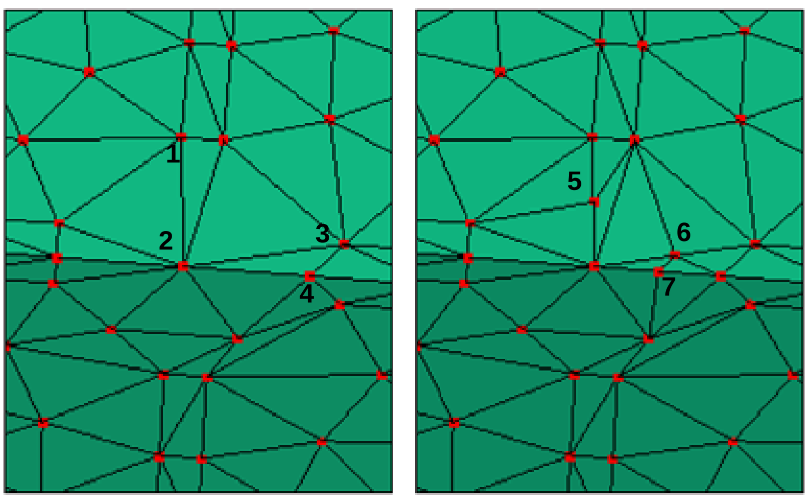


Figure 21: Insert nodes near the corner

To swap elements

1. Press the F6 key, or click the **Edge Mesh Tools** icon in the toolbox.
   1.1. Select **Swap edge**.

2. Refer to Figure 22A.

   2.1. Click on the element labeled 1.

   2.2. Click on the element labeled 2.

   2.3. Click **Apply**.

   - The edges are swapped so they look like elements 3 and 4 in Figure 22B.

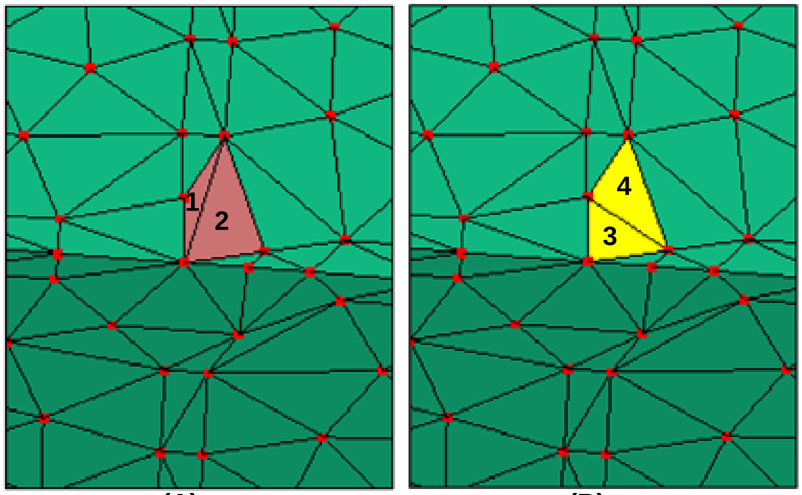


Figure 22: Swapping element edges

To Align the nodes near the corner

1. Press the F11 key, or click the **Nodal Mesh Tools** icon  in the toolbox.

   1.1. Select **Align nodes**.

2. Refer to Figure 23A.

   2.1. Click on the node labeled 1.

   2.2. Click on the node labeled 2.

      ➤ These nodes define the line definition.

   2.3. Hold the Ctrl key and Click on nodes 3 and 4.

   2.4. Click **Apply**.

   - Nodes 3 and 4 get moved to the position of nodes 6 and 5 in Figure 23B.

   2.5. Continue to align nodes until your model looks like Figure 23B.

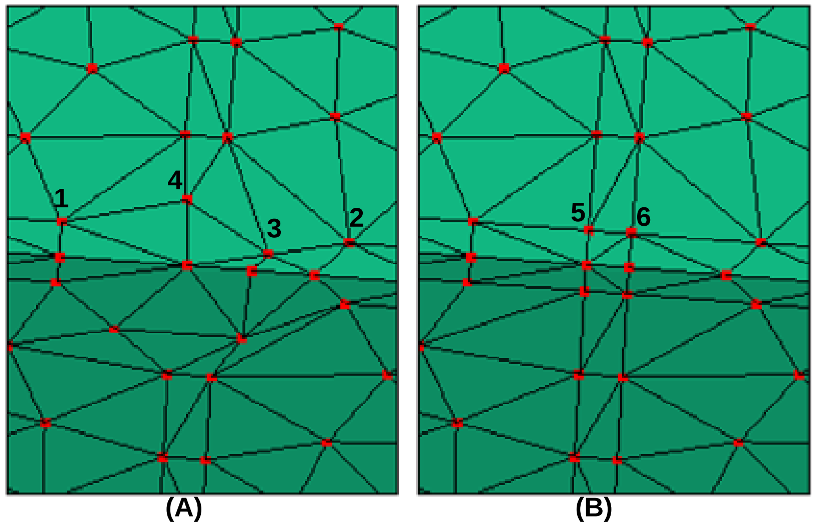


Figure 23: Align nodes

To Merge a node

1. Press the F5 key, or click the **Nodal Mesh Tools** icon  in the toolbox.

   1.1. Select **Merge node**.

2. Refer to Figure 24A.

   2.1. Click on the node labeled 1.

   ➤ This is the node to be kept.

   2.2. Click on the node labeled 2.

   2.3. Click **Apply**.

   - Node 2 is deleted and so is the extra element. The mesh should look like Figure 24B.

3. Click  icon to save the study.

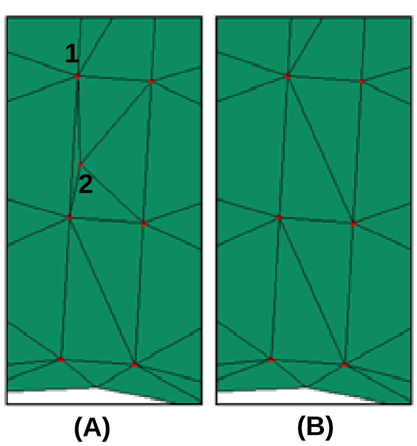


Figure 24: Merging a node

Cleanup layer Fix 2

In this fix session, a node will be inserted then moved to create better matching to a feature on the other side of the part. Elements will also be swapped for the same reason.

To prepare the model

1. Right click on the **Fix 2** layer in the layer pane.
2. Select **Hide all other layers** from the context menu.
3. Click the **Front View** icon , or enter the rotation **0, 0, 0** into the **Enter rotation angles** field of the viewpoint toolbar.
4. Click the **Fit to Window** icon  to scale the visible entities.

To insert a node

1. Press the F9 key, or click the **Nodal Mesh Tools** icon  in the toolbox.

   1.1. Select **Insert Nodes**.

2. Refer to Figure 25A.

   2.1. Click on the node labeled 1.

   2.2. Click on the node labeled 2.

   2.3. Click **Apply**.

   - A node is inserted labeled 3 in Figure 25B.

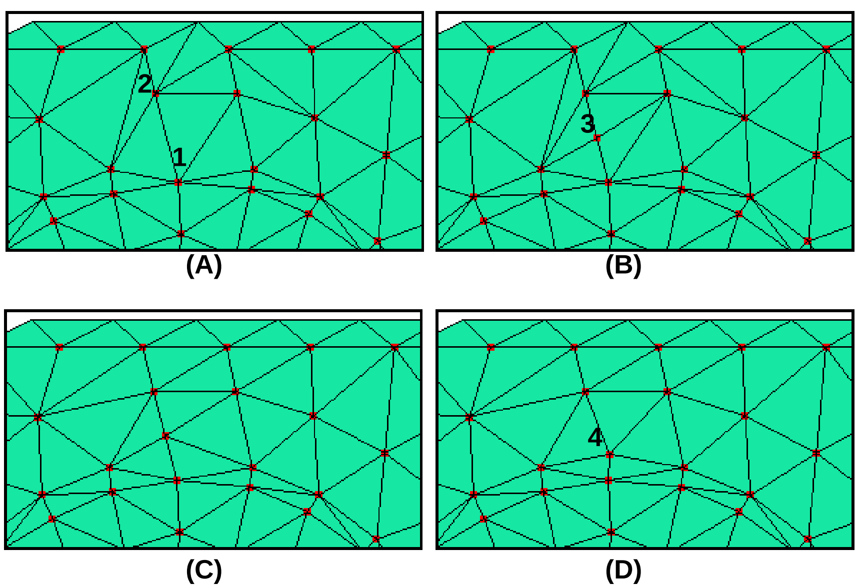


Figure 25: Inserting and moving nodes and swapping elements.

To Swap elements

1. Press the F6 key, or click the **Edge Mesh Tools** icon in the toolbox.

   1.1. Select **Swap edge**.

2. Refer to Figure 25B and C.
3. Click on pairs of elements to swap edges so the mesh looks like Figure 25C.

To move a node

1. Press the F10 key, or click the **Nodal Mesh Tools** icon in the toolbox.

   1.1. Select **Move node**.

2. Refer to Figure 25B.

   2.1. Click and drag the node labeled 3 in Figure 25B and drag it to the position labeled 4, shown in Figure 25D.

   2.2. Click **Apply**.

3. Click icon to save the study.

Cleanup layer Fix 3

In the fix 3 area of the part, there are free edges with a small gap between two sets of nodes. This is most often caused by a poorly constructed 3D model from the CAD package being read in. Most of the time, free edges like this are fixed by the Mesh Repair Wizard. If you did not fix the free edges with the wizard, the Stitch command can be used. Swap edge will also be used to fix a high aspect ratio problem.

To prepare the model

1. Right-click on the **Fix 3** layer in the layer pane.
2. Select **Hide all other layers** from the context menu.
3. Click the **Fit to Window** icon to scale the visible entities.
4. Click the **Front View** icon , or enter the rotation **0, 0, 0** into the **Enter rotation angles** field of the viewpoint toolbar.

To display the free edge

1. Click the **Mesh diagnostic** icon in the toolbox.

   1.1. Select **Free edges diagnostic**.

   1.2. Click **Show**.

   - Notice the red lines being displayed indicating the free edges.

2. Zoom up on the free edges near the bottom edge of the free edges until you can see individual nodes.
3. Right-click in the display area.

   3.1. Select **Measure**.
4. Click on two nodes to determine the gap between the nodes.
   - Refer to Figure 26. The gap is rather narrow at 0.09 mm. The stitch edges tool will be used to close the gap.
5. Close the measure tool dialog.
6. Click the **Select** icon  on the viewer toolbar.

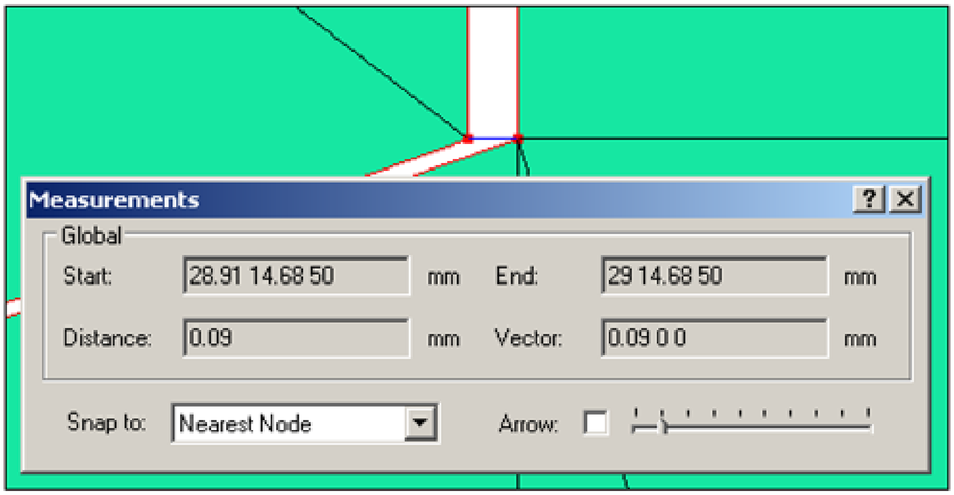


Figure 26: Measuring the gap between nodes

To stitch the edge

1. Click the **Fit to Window** icon  to scale the visible entities.
2. Click the **Edge Mesh Tools** icon  in the toolbox.

   2.1. Click the down arrow key in the combo box.

   2.2. Select **Stitch Free Edges**.
3. Click in the display window.

   3.1. Press **Ctrl + A**.

   ➤ This selects all the nodes visible on the screen and populates the layer.
4. Click **Apply**.
   - The free edges were eliminated because the gap between the nodes was less than the default value of 0.1 mm.
   - If the gap was wider than this tolerance, the tolerance could be specified.
5. Click in the display window.

6. Press **Ctrl + D**.
   - This will toggle off the Free Edge diagnostic.
7. Use the **Swap Edge** tool to fix a long thin element below the area just fixed by the stitching.
   - When done, the Fix 3 area should look like Figure 27.
8. Click  icon to save the study.

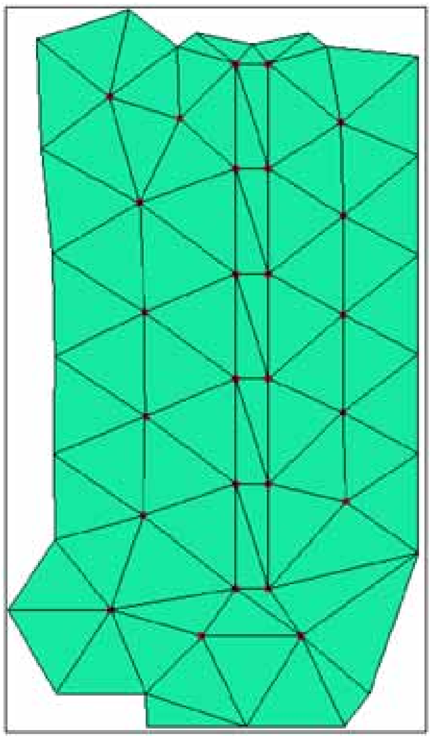


Figure 27: Completed Fix 3 area

Cleanup layer Fix 4

In the Fix 4 area of the part, there is a large hole. Hole are rarely found on a part unless the import geometry has a hole or one was created in the process of cleaning up the model. In this case, a hole was created for you to fill, using the fill holes tool.

To prepare the model

1. Right-click on the **Fix 4** layer in the layer pane.
2. Select **Hide all other layers** from the context menu.
3. Click the **Fit to Window** icon  to scale the visible entities.
4. Click the **Front View** icon , or enter the rotation 0, 0, 0 into the **Enter rotation angles** field of the viewpoint toolbar.

To fill a hole

1. Click the **Edge Mesh Tools** icon  in the toolbox.

   1.1. Click the down arrow key in the combo box.

   1.2. Select **Fill Hole**.

2. Click on any node on the boundary of the hole.

3. Click the **Search** button on the fill hole tool.

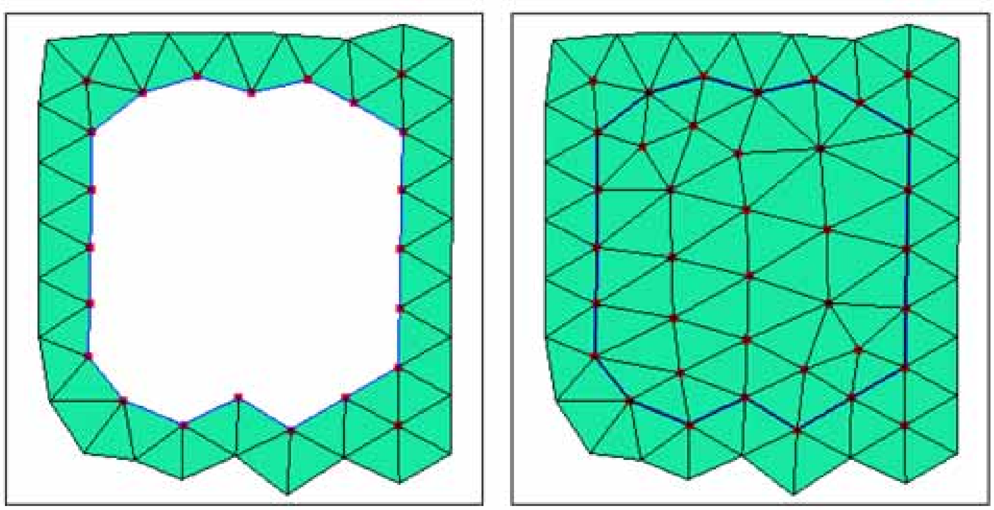
   - A blue outline will connect all the nodes around the hole, similar to Figure 28A. Ensure the boundary is correct.

4. Click **Apply**.

   - The hole is filled in with about the same density as the mesh surrounding the hole, as shown in Figure 28B.

5. Click  icon to save the study.

> The fill hole command may not correctly fill in a hole that goes around corners or on a highly curved surface. Check the mesh carefully once the hole has been filled.



(A) (B)

Figure 28: Filling a hole

Cleanup layer Fix 5

In the fix 5 area of the part, there are some unoriented elements. Rarely do unoriented elements occur. If they do, the command **Mesh ➡ Orient all** normally fixes the problem. Sometimes the orientation of elements must be done manually. This is what is being done here to get some practice.

To prepare the model

1. Right-click on the **Fix 5** layer in the layer pane.
2. Select **Hide all other layers** from the context menu.
3. Click the **Fit to Window** icon to scale the visible entities.
4. Click the **Front View** icon, or enter the rotation 0, 0, 0 into the **Enter rotation angles** field of the viewpoint toolbar.

To display the unoriented elements

1. Click the **Mesh diagnostic** icon in the toolbox.

   1.1. Select **Orientation diagnostic**.

   1.2. Click **Show**.

   - The diagnostic display should look like Figure 29A. There are a few elements that are red, the rest are blue. The red elements have the incorrect orientation. They will be picked manually and flipped.

To orient elements

1. Click the **Global Mesh Tools** icon in the toolbox.

   1.1. Select **Orient element**.

2. Refer to Figure 29A.

   2.1. Ensure the **Elements to edit** field is active.

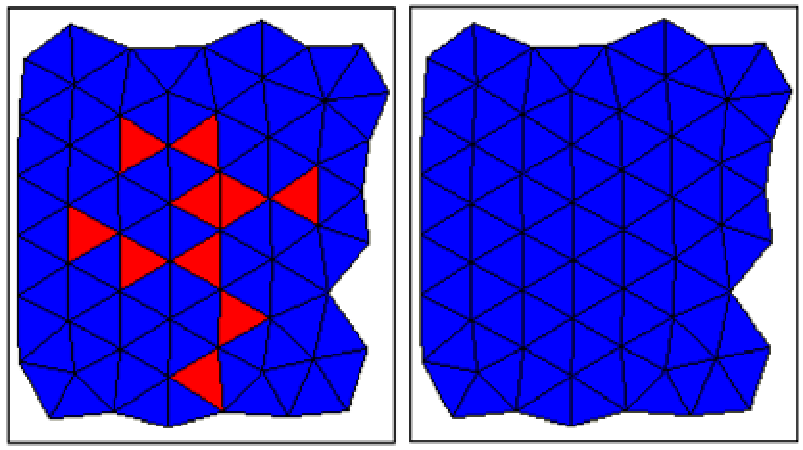
   2.2. Select the elements that are red by holding the **Ctrl** key and clicking on each element.

   2.3. Click the **Flip orientation** radio button.

   2.4. Click **Apply**.

   - The color of all the elements should be blue, indicating the orientation is correct and consistent, as shown in Figure 29B.

3. Click icon to save the study.



(A)
Elements not oriented

(B)
Elements oriented

Figure 29: Orienting elements

Cleanup layer Fix 6

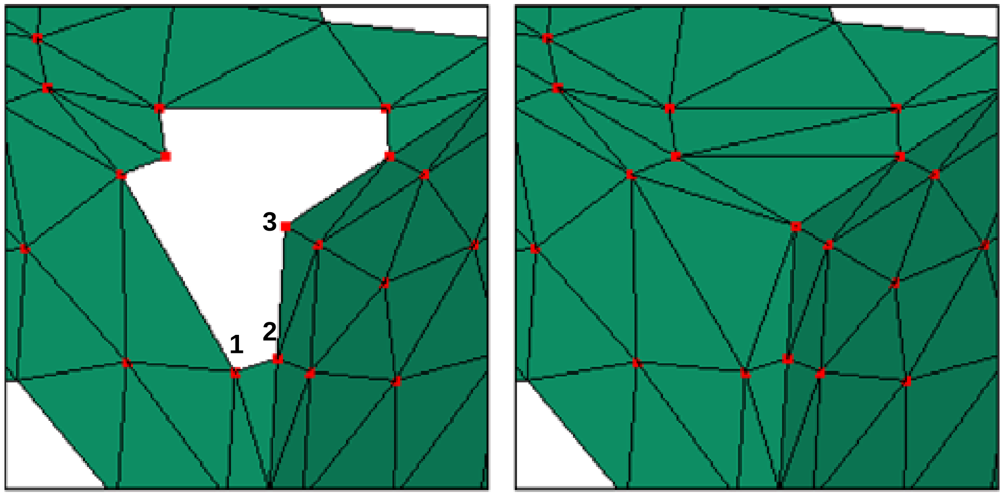
Area Fix 6 has a hole. Unlike area Fix 4, this hole goes around a 90 degree corner, so the fill hole tool will not work. These types of holes are rare. They can be caused by poor input geometry or a hole is create in the process of fixing problems, often intersections and overlaps. For this hole, elements will be created manually.

To prepare the model

1. Right-click on the **Fix 6** layer in the layer pane.
2. Select **Hide all other layers** from the context menu.
3. Enter the rotation **-45, -30, -30** into the **Enter rotation angles** field of the viewpoint toolbar.
4. Click the **Fit to Window** icon  to scale the visible entities.

To fill the hole

1. Click the **Create Beam/Tri/Tetra Tool** icon  in the toolbox.
   1.1. Select **Create Triangle**.
2. Select the nodes 1, 2, and 3, as indicated in Figure 30A.
3. Click **Apply**.
4. Repeat the process to create the remaining triangles so the mesh looks like Figure 30B.
5. Click  icon to save the study.



(A) (B)

Figure 30: Fill a hole by creating elements

Cleanup layer Fix 7

In the Fix 7 area, the Global merge tool will be used to fix a couple of areas. The first area (the runner) is NOT connected to the part. However at first glance it does look connected. The node at the end of the runner is not the same node as the part. The second area are free edges much like the stitch edges command does.

To prepare the model

1. Right-click on the **Fix 7** layer in the layer pane.
2. Select **Hide all other layers** from the context menu.
3. Enter the rotation **-145, -130, 40** into the **Enter rotation angles** field of the viewpoint toolbar.
4. Click the **Fit to Window** icon  to scale the visible entities.

To display a connectivity problem

1. Click the **Mesh diagnostic** icon  in the toolbox.

   1.1. Select **Connectivity diagnostic**.

   1.2. Click in the **Start connectivity check from entity** field.

   1.3. Click on the beam element furthest from the part.

   1.4. Click **Show**.

   - The diagnostic display shows that the runner and part are not connected because the part is red and the runner is blue. Refer to Figure 31.

   1.5. Press **Ctrl + D** to turn off the diagnostic.

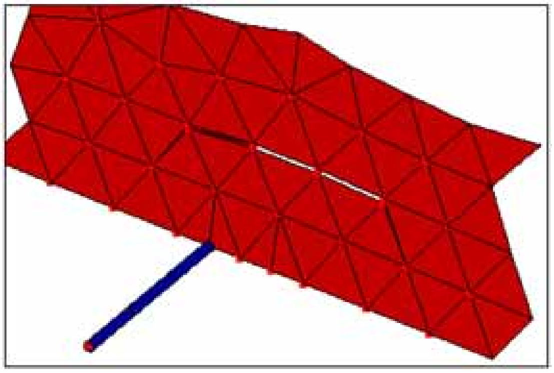


Figure 31: Connectivity diagnostic, runner is not connected to the part

2. Highlight the **Fix 7** layer in the layer pane.

   2.1. Click the **Layer Display** icon on the layer pane.

   2.2. Ensure the Entity type is **Beam element**.

   2.3. Select the Show as setting the **Transparent + Element Edges**.

   2.4. Click **Close**.

   - This makes the runner transparent so you will be able to see the connectivity problem.

3. Click the icon then click on the end of the runner nearest the part.
4. Use Dynamic zoom, to magnify this region of the part until you can see two nodes, one defining the end of the runner, and the other on the part.

To merge nodes at the runner

1. Click the **Global Mesh Tools** icon in the toolbox.

   1.1. Select **Global Merge**.

   1.2. Click **Apply**.

   - Notice how the two nodes at the end of the runner and part are merged. The program reports that 1 node has been merged, this information is listed in the status bar (bottom of the user interface).

To fill the rectangular hole

1. Zoom out so the rectangular hole can be seen.
2. Use the measure tool to determine the spacing between the nodes.

   - Notice the spacing is 0.4 mm and the merge tolerance is 0.1 mm.

3. Set the **Merge tolerance** to **0.45 mm**.

   3.1. Ensure the **Preserve Fusion** box is checked.

   3.2. Click **Apply**.

   - Notice how 2 Nodes were merged in the status line in the bottom left corner of Synergy. Synergy did not collapse all the nodes, only the two that were connected by thin elements in the gap, as shown in Figure 32B.

4. Uncheck **Preserve Fusion**.

   4.1. Click **Apply**.

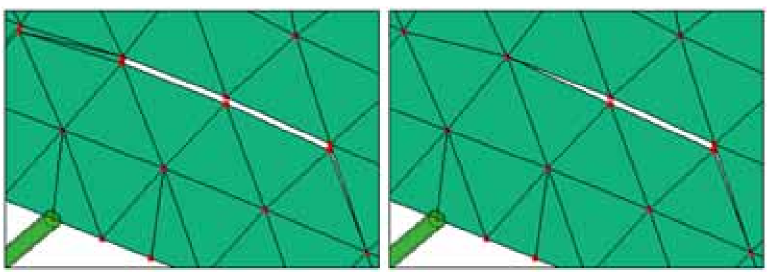
   - Notice that many nodes were merged. Based on what is visible on the screen, you would have expected 2 nodes to be merged. This means that there are 20 nodes in the part that were merged. You should assume this action has corrupted the model.

5. Click the Undo icon  to correct the problem.

6. Use the Merge nodes tool to manually fix the problem.

   - The **Preserve Fusion** box must be unchecked.

7. Click the Save icon  to save the study.



(A) Rectangular hole   (B) Rectangular hole

Figure 32: Merging nodes to fix the hole

Delete unused nodes

In the process of editing models and fixing meshes, sometimes there are nodes unused by any elements. These nodes should be deleted from the model. To delete these unused nodes the purge nodes command is used.

To purge unused nodes from the model

1. Click on the **Tools** tab on the panel.
2. Click the **Nodal Mesh Tools** icon  in the toolbox.
3. Select **Purge nodes**.
4. Click **Apply**.

   - Notice in the status area of Synergy, over 700 nodes were deleted.

Cover

For the cover model, you will:

- Open an existing project.
- Import in the cover IGES model.
- Mesh the model at different mesh settings.
- Diagnose problems with the model at the different settings.
- Clean up one of the models.

Importing

To open a project

1. Click the file open icon or **File** ➡ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Translation_cleanup**.
2. Double click the project file **Translation_cleanup.mpi**.
3. Click **File** ➡ **Preferences** and ensure that **System Units** are set to **Metric**.

To import the cover model

1. Click or **File** ➡ **Import**.

   1.1. Navigate to the folder **translation_cleanup.**

   - Depending on the settings in **Preferences**, the import dialog will open in the correct location.

2. Click on the file **Cover.IGS.**

   2.1. Click **Open**.

3. On the Import dialog:

   3.1. Set the mesh type to **Fusion**.

   3.2. Click **OK**.

4. In the **Project View** pane, rename the study.

   4.1. To rename, **right-click** over the study name.

   4.2. On the **context menu**, select **Rename.**

   4.3. Enter **Cover iges only** as the name and press **Enter**.

5. Click or **View** ➡ **Default Display.**

   5.1. Set Surface, Region, and STL Facet to **Net**.

   - These settings can be Solid, Transparent, Net, or a combination.
   - The Net setting allows you to see the actual IGES surfaces.

6. Rotate the model, inspect it and familiarize yourself with the geometry.

Meshing

The cover will be meshed at 12 different combinations of mesh settings so they can be compared. For each setting, you will open the **Cover iges only** study rename it then mesh it. The settings for mesh the cover is listed in Table 4 on page 107. This table is also used to record the results of the meshing.

To mesh the model

1. Click **File** ➨ **Save study as** and enter **Cover (Case no)**,
   - Where (Case no) the case listed in Table 4 on page 107.
   - The first time the part is meshed the study name is **Cover 1**.
2. Open the **Generate mesh** dialog by one of the methods:
   - Click the icon  on the mesh manipulation toolbar.
   - Double click the icon  in the study tasks list.
   - Click **Mesh** ➨ **Generate Mesh**.
3. Set the mesh options for the case being meshed according to Table 4 on page 107.
   - For example, for Case 1, the global edge length is set to 8 and the Chord height is off.
   - To access some of the settings the Mesh Control button must be clicked.
4. Click the **Mesh Now** button.

To review the results

1. Right-click on the **New Triangles** layer and select **Hide All Other Layers**.
2. Click the icon  on the Mesh manipulation toolbar or **Mesh** ➨ **Mesh statistics**.
3. Record the following information in Table 4 on page 107, from the mesh statistics for the case being meshed:
   - Surface triangles, (No of elements).
   - Maximum aspect ratio.
   - Average Aspect ratio.
   - Match percentage, (Match ratio).
4. Close the Mesh Statistics dialog.

5. Click the **Mesh diagnostic** icon  in the toolbox, on the **Tools** panel.
   5.1. Select **Aspect ratio diagnostic**.
   5.2. Click **Show**.
   5.3. Record the number of elements that are above the aspect ratio of 6:1 in Table 4 on page 107.
   - ➤ The number is listed on the status line at the bottom left corner of the Synergy window.
6. Rotate the part around to find the location of the high aspect ratio elements. In particular look at:
   - A lip on the under side of the part visible at a rotation of 130 45 30.
   - The boss near the center of the part.
   - Also note how well meshed the boss is.
7. **Save** the study.

Meshing Cases 2 to 12

The cases 2 to 11 just change settings on the Generate Mesh tool. Case 12 requires you to set a local mesh density. This is described below.

To mesh the cover with different mesh options

1. Follow the previous two tasks for each case.
2. Be sure to start with the Iges file each time and you re-name the study before you mesh it.
3. Record the results for each case.

To mesh case 12

1. Rotate the part to **130 45 30.**
2. Zoom up on the lip. See Figure 33.
3. Select the thin edge as shown.
4. Right-click and select **Define Mesh Density**.
5. Select the Surface in the list.
   - There should only be one surface listed.
6. Uncheck **Use global mesh density**.
7. Set the Target edge length to 4.25 mm.
8. Click **Apply**.
9. Continue to set up the mesh options like all the other cases.

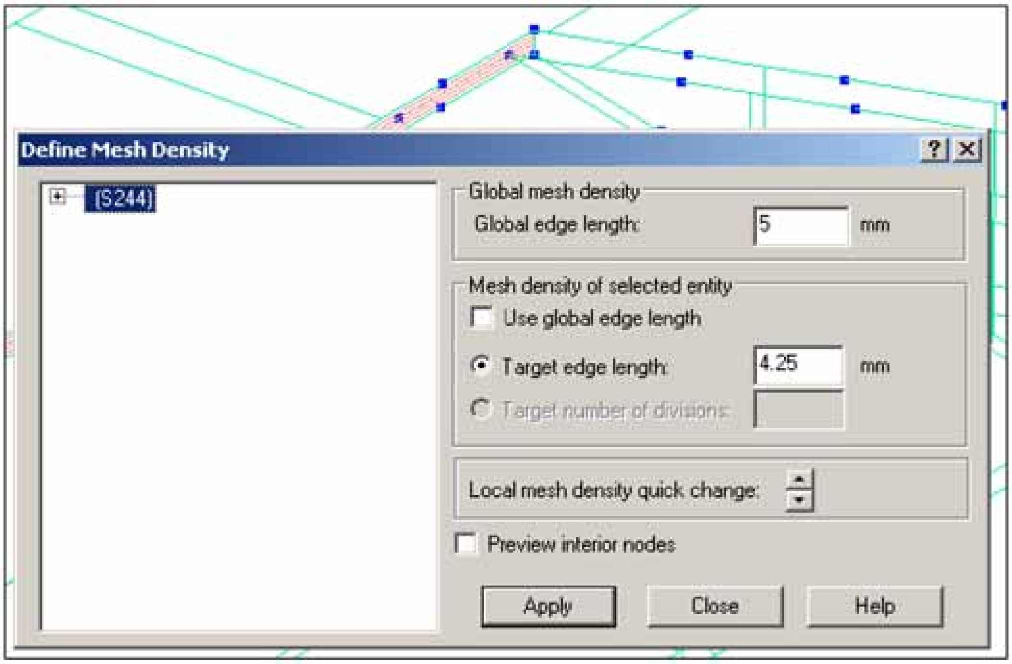


Figure 33: Setting the local mesh density

Table 4: Cover mesh settings and outcomes

If a mesh setting is not listed, the default value is used. They are:

- Chord height, 0.1.
- Surface curvature control, Off.
- Proximity control, Off.
- Match mesh, On.

A bolded field indicates the change in the setting from the previous case.

Case No.	Mesh Settings	No. Elements	Aspect Ratio Max	Aspect Ratio Avg	Match Ratio	No. elements with Aspect ratio > 6:1
1	Global edge length 8.0 Chord height Off					
2	**Global edge length 7.0** Chord height Off					
3	**Global edge length 6.0** Chord height Off					
4	**Global edge length 5.0** Chord height Off					
5	**Global edge length 4.0** Chord height Off					
6	**Global edge length 5.0** **Chord height 0.1**					
7	Global edge length 5.0 **Chord height 0.2**					
8	Global edge length 5.0 **Chord height 0.05**					
9	Global edge length 5.0 **Chord height 0.1** **Curvature control On**					
10	Global edge length 5.0 Chord height 0.1 Curvature control On **Proximity control On**					
11	Global edge length 5.0 Chord height 0.1 Curvature control On **Match Mesh Off**					
12	Global edge length 5.0 Chord height 0.1 Curvature control On **Local mesh density 4.25**					

A discussion about the results of the meshing is on page 109.

Cleanup the mesh for the cover case 12

To prepare the study for cleanup

1. Close all the studies except **Cover 12**.
2. Click **File** ➡ **Save study as** and enter **Cover Cleanup**.
3. Highlight the Layer **New Nodes**.

   3.1. Click the Layer Display Icon  .

   3.2. Set the color of nodes to **Red**.

   3.3. Click **Close**.

Cleaning the mesh

The Mesh Repair Wizard will be used to fix most problem areas.

To check the mesh

1. Click **Mesh** ➡ **Mesh Statistics**.
2. Confirm the only problem is with aspect ratio.

To use the Mesh Repair Wizard

1. Click **Mesh** ➡ **Mesh Repair Wizard**.
2. Click **Next** on the wizard until you get to the Degenerate Elements page.
3. Set the Tolerance to **0.4 mm**.
4. Check **Show diagnostics**.
5. Use the Diagnostic navigator to review the problem areas. You will need to rotate and zoom the model to see the problems.
6. Click **Fix**.
7. In the Aspect ratio page click **Skip**.
8. Click **Next** until the wizard closes.

To verify the mesh cleanup

1. Click **Mesh** ➡ **Mesh Statistics**.
2. Confirm the only problem is with aspect ratio (over 6).
3. **Save** the study.
4. Run an **Aspect ratio diagnostic**.

To fix the model

1. Use the **Diagnostic** navigator to find the problem areas.
2. Use what ever tool you need to do fix the problem.
3. Run the statistics to verify the mesh is clean.

4. **Save** the study.

Discussion on the cover meshing

Global edge length

By comparing Cases 1 to 5, it is clear to see that by decreasing the global edge length from 8 mm to 4 mm the number of elements goes up significantly, the average aspect ratio drops continually, the match ratio improves, and the number of elements with an aspect ratio higher than 6 drops significantly. Notice that the Maximum aspect ratio fluctuates up and down. It is not directly tied to the number of elements. This is caused mostly by the matching algorithm. With the global edge length of 5 then to 4, there is 2000 element increase and the number of elements over 5 goes way down from 45 to 5. This occurs because a thin rim has elements with a 5 mm global edge length, has elements over 6:1, but when the edge length drops to 4, the aspect ratio goes under 6:1.

Chord height

When comparing changes in chord height with cases 6 to 8, you can see that with a smaller chord height the number of elements goes up. The rest of the numbers are all about the same. You can see the difference when you look at the mesh on the boss in the center of the part. With the smaller chord heights, the elements are smaller, defining better the feature. When using chord height, use the preview button and try out different chord heights before meshing the part.

Curvature control

When curvature control is added, as in case 9, the mesher uses both the chord height and curvature control to mesh the curved features better. The best mesh on the boss is when curvature control is added. Curvature control may not always be appropriate for Fusion models. The match ratio may go down because of the mesh placed on the curved surface. For the cover, the match ratio with curvature control is good because it is above 90%, but it is slightly lower than cases 6 to 8. Curvature control is a very good option for 3D models, but for models that are going to stay as Fusion, it may be better to have the option off.

Proximity control

Proximity control was used in case 10 and will mesh an area finer when the distance between two edges is less than the global edge length. For this part, the nominal wall is about 2 mm, and the global edge length was 5.0. Therefore, all the edges, as some other detail falls below 5 mm so it is meshed finer. Turning this option on for this part was inappropriate. The number of elements for all other cases was under 8000, and for case 10, the number of elements is over 36000. The transition between the very fine mesh and relatively course mesh for this part is too extreme. Probably, for most Fusion parts, this option should be off. For 3D parts, this option may be useful for the “chunkier” parts where there many be some smaller features that need to be meshed finer and this will help.

Mesh matching

Case 11 turns off the mesh matching. Compared to case 9, the maximum aspect ratio is much lower. The post-meshing step of mesh matching causes most of the high aspect ratio problems. If the part is a 3D mesh, turning off the match ratio is a good idea. However, for Fusion models, you must have it on, to get the higher match ratio. Notice in case 11, the match ratio is under 81% while case 9 the match ratio is over 91%.

Local mesh density

In case 12, a local mesh density was applied to the thin wall that created most of the high aspect ratio elements. This is the main reason the number of elements with an aspect ratio higher than 6 goes from 53 to 15 between cases 9 and 12.

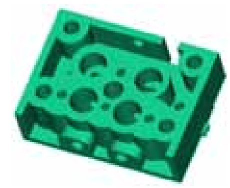
Table 5: Cover mesh results

Case No.	Mesh Settings	No. Elements	Aspect Ratio		Match Ratio	No. elements with Aspect ratio > 6:1
			Max	Avg		
1	Global edge length 8.0 Chord height Off	2466	10.74	2.66	86.2	191
2	**Global edge length 7.0** Chord height Off	3042	13.60	2.49	86.1	103
3	**Global edge length 6.0** Chord height Off	3746	54.82	2.25	89.5	51
4	**Global edge length 5.0** Chord height Off	5442	44.57	2.07	92.2	45
5	**Global edge length 4.0** Chord height Off	7490	9.38	1.77	92.6	5
6	**Global edge length 5.0** **Chord height 0.1**	5750	44.56	2.09	92.0	62
7	Global edge length 5.0 **Chord height 0.2**	5568	44.57	2.07	91.9	55
8	Global edge length 5.0 **Chord height 0.05**	5960	62.27	2.13	91.9	52
9	Global edge length 5.0 **Chord height 0.1** **Curvature control On**	6782	44.57	1.98	91.4	53
10	Global edge length 5.0 Chord height 0.1 Curvature control On **Proximity control On**	36060	63.78	1.67	91.8	191
11	Global edge length 5.0 Chord height 0.1 Curvature control On **Match Mesh Off**	6160	8.93	1.66	80.7	42
12	Global edge length 5.0 Chord height 0.1 Curvature control On **Local mesh density 4.25**	6842	44.56	1.96	91.2	15

Table 5: Cover mesh results

Case No.	Mesh Settings	No. Elements	Aspect Ratio		Match Ratio	No. elements with Aspect ratio > 6:1
			Max	Avg		
The results in the table above were obtained using MPI 6.0 Rev 1 build 06074. Different builds and revisions may have different results						

Manifold



In this exercise, you will do the following:

1. Read in an IGES file of a manifold as a Fusion mesh.
2. Mesh the part at several different mesh setting options.
3. Clean up the Fusion mesh to prepare it for conversion to a 3D mesh.
4. Mesh the part with 3 different numbers of element layers.
5. Review the statistics on the meshes.
6. Look at the mesh through the part.
7. Repair the 3D mesh.

Importing

To open a project

1. Click the file open icon or **File** ➨ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Translation_cleanup**.
2. Double click the project file **Translation_cleanup.mpi**.
3. Click **File** ➨ **Preferences** and ensure that **System Units** are set to **Metric**.

To import the manifold model

1. Click or **File** ➨ **Import**.

   1.1. Navigate to the folder **translation_cleanup.**

   - Depending on the settings in **Preferences**, the import dialog will open in the correct location.
2. Click on the file **Manifold.IGS**

   2.1. Click **Open**.
3. On the Import dialog:

   3.1. Set the mesh type to **Fusion**.

   3.2. Click **OK**.
4. In the **Project View** pane, rename the study.

   4.1. To rename, **right-click** over the study name.

   4.2. On the **context menu**, select **Rename**.

   4.3. Enter **Manifold iges only** as the name and press **Enter**.

5. Click  or **View ➡ Default Display**.

   5.1. Set Surface, Region, and STL Facet to **Net**.

   - These settings can be Solid, Transparent, Net, or a combination.
   - The Net setting allows you to see the actual IGES surfaces.

6. Rotate the model, inspect it and familiarize yourself with the geometry.

Meshing

The manifold will be meshed at 11 different combinations of mesh settings so they can be compared. For each setting, you will open the **Manifold iges only** study rename it then mesh it. The settings for mesh the manifold is listed in Table 6 on page 116. This table is also used to record the results of the meshing.

To mesh the model

1. Click **File ➡ Save study as** and enter **Manifold (Case no)**,
   - Where (Case no) the case listed in Table 6 on page 116.
   - The first time the part is meshed the study name is **Manifold 1**.
2. Open the **Generate mesh** dialog by one of the methods:
   - Click the icon  on the mesh manipulation toolbar.
   - Double click the icon  in the study tasks list.
   - Click **Mesh ➡ Generate Mesh**.
3. Set the mesh options for the case being meshed according to Table 6 on page 116.
   - For example, for Case 1, the global edge length is set to 4 and the Chord height is off.
   - To access some of the settings the Mesh Control button must be clicked.
4. Click the **Mesh Now** button.

To review the results

1. Right-click on the **New Triangles** layer and select **Hide All Other Layers**.
2. Click the icon  on the Mesh manipulation toolbar or **Mesh ➡ Mesh statistics**.
3. Record the following information in Table 6 on page 116, from the mesh statistics for the case being meshed:
   - Surface triangles, (No of elements).
   - Maximum aspect ratio.
   - Average Aspect ratio.
   - Match percentage, (Match ratio).
   - Other mesh problems, such as edge or intersection problems.

4. Close the Mesh Statistics dialog.

5. Click the **Mesh diagnostic** icon in the toolbox, on the **Tools** panel.

   5.1. Select **Aspect ratio diagnostic**.

   5.2. Click **Show**.

   5.3. Record the number of elements that are above the aspect ratio of 6:1 in Table 6 on page 116.

   - The number is listed on the status line at the bottom left corner of the Synergy window.

6. Save the study.

Meshing Cases 2 to 11

The cases 2 to 11 just change settings on the Generate Mesh tool.

To mesh the manifold with different mesh options

1. Follow the previous two tasks for each case.
2. Be sure to start with the Iges file each time and you re-name the study before you mesh it.
3. Record the results for each case.

Table 6: Manifold mesh settings and outcomes

If a mesh setting is not listed, the default value is used. They are:

- Chord height, 0.1.
- Surface Curvature control, Off.
- Proximity control, Off.
- Match mesh, On.

A bolded parameter indicates the change in the setting from the previous case.

Case No.	Mesh Settings	No. Elements	Aspect Ratio Max	Aspect Ratio Avg	Match Ratio	No. elements with Aspect ratio > 6:1	Other Problems List
1	Global edge length 4.0 Chord height Off						
2	**Global edge length 3.0** Chord height Off						
3	**Global edge length 2.5** Chord height Off						
4	**Global edge length 2.0** Chord height Off						
5	**Global edge length 1.0** Chord height Off						
6	**Global edge length 2.5** **Chord height 0.1**						
7	Global edge length 2.5 **Chord height 0.2**						
8	Global edge length 2.5 **Chord height 0.05**						
9	Global edge length 2.5 **Chord height 0.1** **Curvature control On**						
10	Global edge length 2.5 Chord height 0.1 Curvature control On **Proximity control On**						
11	Global edge length 2.5 Chord height 0.1 Curvature control On **Match Mesh Off**						

A discussion about the results of the meshing is on page 118.

Cleanup the mesh for the manifold case 11

To prepare the study for cleanup

1. Close all the studies except **Manifold 11**.
2. Click **File** ➡ **Save study as** and enter **Manifold Cleanup**.
3. Turn on the New Nodes layer.
4. Highlight the Layer **New Nodes**.

   4.1. Click the Layer Display Icon .

   4.2. Set the color of nodes to **Red**.

   4.3. Click **Close**.

Cleaning the mesh

The Mesh Repair Wizard will be used to fix most problem areas.

To check the mesh

1. Click **Mesh** ➡ **Mesh Statistics**.

You should find that there are several hundred elements that are not oriented, the maximum aspect ratio is above 6:1, and the match ratio is low. The match ratio is not an issue because the model will be converted to a 3D mesh. The aspect ratio is low enough to go to 3D, but it will be lowered to 6:1 for some practice.

To use the Mesh Repair Wizard

1. Click **Mesh** ➡ **Mesh Repair Wizard**.
2. Click **Next** on the wizard until you get to the Degenerate Elements page.
3. Set the Tolerance to **0.2 mm**.
4. Check **Show diagnostics**.
5. Use the Diagnostic navigator to review the problem areas. You will need to rotate and zoom the model to see the problems.
6. Click **Fix**.
7. Click **Next** until the wizard closes.
   - Clicking Next on the Orientation page will fix the orientation problems.

To verify the mesh cleanup

1. Click **Mesh** ➡ **Mesh Statistics**.
2. Confirm the only problem is with aspect ratio (over 6).
3. **Save** the study.
4. Run an Aspect ratio diagnostic.

To fix the model

1. Use the Diagnostic navigator to find the problem areas.
2. Use what ever tool you need to do fix the problem.
3. Run the statistics to verify the mesh is clean.
4. **Save** the study.

Discussion on the manifold meshing

Global edge length

The manifold has many important round features. Cases 1 to 5 have the chord height control off, so the ability of capturing the round features is solely based on the global edge length. Even with a global edge length of 2.0 mm, there are still some of the smaller blind holes that are not meshed correctly. Only when the edge length is 1.0 mm creating over 37,000 elements makes the mesh look reasonable on most of the part.

Chord height

Comparing case 3 and 6 the only difference is the Chord height. Case 3 has the chord height off and Case 6 has the chord height set at the default of 0.1 mm. With the chord height on, the mesh around the curved features is much better. This prevented the non-manifold edges from forming as did in Case 3. The number of elements in Case 6 is about 141% of the number of Case 3. Compare that to the number of elements in Case 5 which is 620% of the number of Case 3. The circular feature definition in Case 5 and 6 is very similar, even though the mesh is much coarser in Case 6.

Comparing Case 6 to 8 shows how the finer chord height forces a finer definition of the circular features. When using chord height, use the preview button and try out different chord heights before meshing the part.

Curvature control

When curvature control is added, as in Case 9, the mesher uses both the chord height and curvature control to mesh the curved features better. The best mesh on the curved features is when curvature control is added. Curvature control is a very good option for 3D models, but care must be used to choose a good chord height so the mesh is not too coarse or too fine in particular in the transition between the curved features and the rest of the part.

Proximity control

Proximity control was used in case 10 and will mesh an area finer when the distance between two edges is less than the global edge length. For this part, the global edge length of 2.5 mm is less than many of the features thickness. Therefore, not too much of the part is affected. For 3D parts, this option may be useful for the “chunkier” parts where there many be some smaller features that need to be meshed finer and this will help. This is the case for the manifold.

Mesh matching

Case 11 turns off the mesh matching. Compared to case 9, the maximum aspect ratio is much lower and there are far fewer elements with an aspect ratio above 6:1. The post-meshing step of mesh matching causes most of the high aspect ratio problems. For 3D meshes, turning off the match ratio is a good idea.

Table 7: Manifold mesh settings and results

If a mesh setting is not listed, the default value is used. They are:

- Chord height, 0.1.
- Surface Curvature control, Off.
- Proximity control, Off.
- Match mesh, On.

Case No.	Mesh Settings	No. Elements	Aspect Ratio		Match Ratio	No. elements with Aspect ratio > 6:1	Other Problems List
			Max	Avg			
1	Global edge length 4.0 Chord height Off	2682	17.0	2.1	70.9	21	5 Non-Man 22 overlap 3 unoriented
2	**Global edge length 3.0** Chord height Off	4316	33.4	1.8	73.6	17	3 Non-Man 6 unoriented
3	**Global edge length 2.5** Chord height Off	6048	12.6	1.7	79.0	14	9 Non-Man 16 unoriented
4	**Global edge length 2.0** Chord height Off	9422	14.0	1.6	80.4	17	3 Non-Man
5	**Global edge length 1.0** Chord height Off	37790	18.6	1.4	82.0	22	none
6	**Global edge length 2.5** **Chord height 0.1**	8606	21.8	1.9	75.5	42	none
7	Global edge length 2.5 **Chord height 0.2**	7026	30.8	1.9	77.2	43	None
8	Global edge length 2.5 **Chord height 0.05**	12360	37.9	2.1	72.0	96	None
9	Global edge length 2.5 **Chord height 0.1** **Curvature control On**	18766	32.9	1.6	73.6	47	None
10	Global edge length 2.5 Chord height 0.1 Curvature control On **Proximity control On**	21204	24.9	1.6	74.6	39	None
11	Global edge length 2.5 Chord height 0.1 Curvature control On **Match Mesh Off**	18340	8.1	1.5	68.5	7	>700 unoriented

The results in the table above were obtained using MPI 6.0 Rev 1 build 06074. Different builds and revisions may have different results

Create a 3D Mesh

To create a 3D mesh

1. Click **File** ➡ **Save Study as** and enter the name **Manifold_3D**.
2. Click the **Tasks** tab.
3. Right click the **Fusion Mesh** icon and select **Set Mesh type** and pick **3D**.
4. Click **Mesh** ➡ **Generate mesh**.
5. Click **Tetra refinement**.
   - Review the settings for the 3D mesh.
   - None will be changed.
6. Click **Mesh Now**.

Using the 3D Mesh Repair Wizard

To check the 3D mesh

1. Click **Mesh** ➡ **Mesh Repair Wizard**.
   - The first page of the wizard shows all the diagnostics that can be run for a 3D mesh. You can select which ones you want to be checked. By default all are checked.

> Moldflow recommends you check all diagnostics.

2. Click **Next** to view the summary page.
3. Click **Next** to view the first diagnostic.
   - On each page of the diagnostic:
     - If a diagnostic indicates a problem, click **Fix** to try to correct the problem. **Fix** can be clicked repeatedly to correct problems.
     - Click the **Skip** button to go to the next diagnostic without trying to fix the problem.
     - Click **Next** to correct a mesh problem indicated by the current page of the wizard. By clicking **Next**, you can't see if the problem was resolved.

> Moldflow recommends using the **Fix** and **Skip** buttons unless there is no problem to solve, then the **Next** button can be used.

4. On the **Internal long edges** page, reduce the ratio to 2.5 by clicking **Fix** as many times as necessary.
5. On the **Tetras with large volume** page, reduce the ratio to 20 by clicking **Fix** as many times as necessary.

6. On the **Aspect ratio (Tetras)** page:

   6.1. Change the ratio from 50 to 40.

   6.2. Click **Update**.

   6.3. Reduce the aspect ratio to 40 by clicking **Fix** as many times as necessary.

7. On the **Included angles of tetras** page, reduce the number of elements with a small included angle by clicking **Fix** as many times as necessary, or until no changes are made when Fix is clicked.

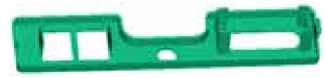
8. Click **Close** on the summary page.

9. If repairs are made, re-run the **Mesh Repair Wizard** to ensure problems were not created during the fix.

10. **Save** the study.

Snap Cover

Chord Height and Radii



In this exercise, you will do the following:

1. Read in an IGES file of the snap cover that has small radii in it.
2. Choose your own global edge length and chord height.
3. Clean up the Fusion mesh.

To open a project

1. Click the file open icon or **File** ➨ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Translation_cleanup**.
2. Double click the project file **Translation_cleanup.mpi**.
3. Click **File** ➨ **Preferences** and ensure that **System Units** are set to **Metric**.

To import the Snap cover model

1. Click or **File** ➨ **Import**.

   1.1. Navigate to the folder **translation_cleanup.**

   - Depending on the settings in **Preferences**, the import dialog will open in the correct location.
2. Click on the file **Snap_Cover_rad.IGS**

   2.1. Click **Open**.
3. On the Import dialog:

   3.1. Set the mesh type to **Fusion**.

   3.2. Click **OK**.
4. Click or **View** ➨ **Default Display**.

   4.1. Set Surface, Region, and STL Facet to **Net**.

   - These settings can be Solid, Transparent, Net, or a combination.
   - The Net setting allows you to see the actual IGES surfaces.
5. Rotate the model, inspect it and familiarize yourself with the geometry.

To mesh the snap cover

1. Double click the icon in the study tasks list, or any other method to open the Generate Mesh dialog.
2. Click the **Preview** button.

3. Review the preview nodes on the part.

   3.1. Decide if you like the density.

   3.2. Concentrate on the:

   - Round hollow bosses.
   - Other curved features.
   - Radiused corners.

   3.3. Change the global edge length and/or chord height value if you want to.

4. Mesh the part.

To evaluate the mesh

1. Click **Mesh ➡ Mesh Statistics**.
2. Enter in Table 8 the information about the mesh.
3. Decide if you like the mesh.
   - Normally, you should try several mesh densities before making up your mind.
   - Consider setting a local mesh density on the two round bosses.
4. Remesh the part as necessary.

   4.1. Delete the current mesh first. If you wish, save the current mesh then save as to create a new study before deleting the mesh.

> To set a local mesh density, highlight the geometry you want to change the density of, right click and select **Define mesh density.**

Table 8: Snap cover with radii mesh statistics

Edge Length	No. of Tri's	Aspect Ratio		Match Ratio%	No. Tri's above 6:1 Aspect ratio
		Avg.	Max		

To cleanup the mesh

1. Use the Mesh Repair Wizard first.

   1.1. Take advantage of the degenerate elements page. Set the tolerance high enough to remove most of the high aspect ratio elements.

2. Use the mesh tools to finish fixing aspect ratio problems.
3. Check the match ratio using the diagnostic.
    - You should aim to have a match ratio above 90%. Higher than 85% is acceptable.
4. Check the thickness of the part using the diagnostic.
    - Repair any mistakes in thickness if you find them.
    - Select the problem elements and either:
        - Create a new property with the correct thickness.
        - Re-define the current property with the correct thickness.

Mesh Results Discussion

The chord height MUST be set to ensure the round boss is properly meshed. The small radii on the inside and outside corners should NOT be included as part of the IGES model. They are shown in Figure 34. They get meshed as a facet, which is not acceptable. The aspect ratios are high and they will not be matched to anything. The Degenerate Element page of the Mesh Repair Wizard might remove the radii, depending on the chord height tolerance used to mesh the part. At the default 0.1 mm, the small boss gets meshed so the nodal distance on some of the nodes in the boss is less than the distance between nodes due to the edge radii. With a chord height of 0.2, the boss' node spacing is greater than the nodal spacing of the radii, so the degenerate element too will remove the radii. However, the corner is not straight, it is slightly tapered. If the curvature control was used to mesh this part, the element count goes from about 6000 elements to over 92,000 elements with the same 3.0 mm global edge length. The small radii are meshed very fine and then the mesh transitions out to the 3.0 mm size. In some places the transition is not good, as shown in Figure 34.

The best solution would be to make sure the radii are not in the IGES file to begin with.

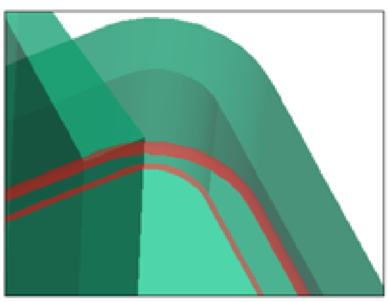


Figure 34: Snap Cover Radii

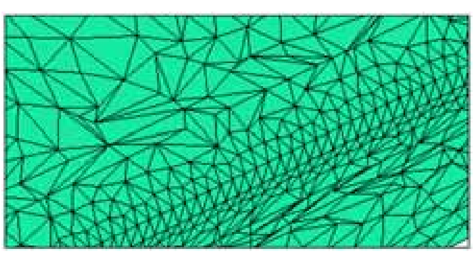


Figure 35: Edge radius of snap cover meshed with curvature control

Different File Formats

The **Snap cover** is also available in the following formats:

- STL.
- STEP.
- Parasolid.
- Pro/ENGINEER.

The STL format can be imported into Synergy without a special license. The other formats require an MDL license. Import the snap cover in whatever formats you can. Experiment with different mesh densities. If you would like the mesh, clean up the versions you have imported and meshed.

All the models are located in the **translation_cleanup** folder.

Dustpan

The following steps are for importing a Dustpan STL model. Once you have imported it, you will convert the model into a Midplane model.

To import the STL Dustpan model

1. Click (**File ➡ Import)**.

   1.1. Navigate to the folder **translation_cleanup.**

   - Depending on the settings in **Preferences**, the import dialog will open in the correct location.

2. Click on the file **Dustpan.stl.**
3. Click **Open**.
4. On the Import dialog, accept the defaults for:

   4.1. The mesh type as **Fusion**.

   4.2. The units as **millimeters**.

   4.3. Click **OK**.

To Mesh the Dustpan

1. Double click the icon in the study tasks list, or any other method to open the Generate Mesh dialog.
2. Click the **Preview** button.
3. Review the preview nodes on the part.

   3.1. Decide if you like the density.

   3.2. Change the global edge length value if you want to.

4. Mesh the part.

To evaluate the mesh

1. Click (**Mesh ➡ Mesh Statistics)**.
2. Decide if you like the mesh.
   - Normally, you should try several mesh densities before making up your mind.
3. Remesh the part as necessary.
   - Delete the current mesh first.
   - If you wish, save the current mesh then save as to create a new study before deleting the mesh.

Create a Midplane Dustpan Model

A midplane model is created using the Midplane Generator. This requires a separate license. The Midplane generator is automatically used when you convert the mesh type to Midplane then remesh the part.

To generate a Midplane Dustpan model

1. Right click over the icon in the study tasks list.
2. Select the context menu item **Set Mesh type**.
3. Select **Midplane** on the sub context menu.
4. Open the **Generate Mesh** dialog.
5. Check the **Remesh already meshed parts of the model** box.
6. Click **Mesh Now**.

Reviewing the Mesh

The Midplane Generator uses the matched elements in the model and collapses them to form a Midplane model. In areas of a part that are thick and “chunky” the Midplane generator may not work well. In the case of the dustpan, you will not have to do much cleaning up – you will only need to delete a couple of elements.

Watch for:

- High aspect ratio problems.
- Free edges and non-manifold edges.
  - In this case, free and non-manifold edges are allowed in the model. However, you need to make sure they are in the correct locations.
  - Use the free and non-manifold edge diagnostic. You may find it useful to have the model set to transparent to see the free and non-manifold edges easily.
- Thickness.
  - Check to make sure the thickness of the part is properly represented. In the case of the dustpan, the wall thickness is uniform except for the front edge.
  - Consider manually setting the part thickness.

To clean up the Midplane model

1. Use the mesh diagnostics to identify problem areas.
2. Use the mesh cleanup tools to fix any problem areas on the part.

Housing

The housing model is in STEP format. This requires an MDL license. MDL has the ability to import geometry, a mesh, or both. The mesh generator built into MDL is different from Synergy's.

For the housing, Import the STEP file using different input options. Mesh it with MDL settings and Synergy. Compare the meshes using the mesh statistics.

Figure 36: Housing

Competency check - Model Translation and Cleanup

1. What does changing the **Global edge length** do when meshing a translated model?
2. If the mesh density is not adequate, how can the density be changed, in addition to the global edge length?
3. Generally, what should the procedure be when fixing a mesh?
4. When can the model be regarded as ready for analysis?

Evaluation Sheet - Model Translation and Cleanup

1. What does changing the Global edge length do when meshing a translated model?

Answer:

Changing the global edge length influences the mesh density of the part. Increasing the edge length decreases the mesh density. You can change the global edge length, which should give you a mesh density that is detailed enough to then run an analysis.

2. If the mesh density is not adequate, how can the density be changed, in addition to the global edge length?

Answer:

In addition to the global edge length, you can define a local edge length on the geometry. Also, the chord height control is useful for adjusting the mesh density on circular features.

3. Generally, what should the procedure be when fixing a mesh?

Answer:

After using the mesh statistics and thickness diagnostics to determine what **problems** exist, use the Mesh Repair Wizard, if the statistics and thickness diagnostic suggests it's OK to proceed. Then use the Mesh tools to fix any remaining issues.

4. When can the model be regarded as ready for analysis?

Answer:

When you have fixed all problems with the mesh and when the aspect ratio is as low as practical. It must be below 6:1 if going on to Cooling and Warpage analysis.

CHAPTER 10

Modeling Tools

Aim

To learn about the modeling tools available inside Synergy. Learn how to use them to create geometry and/or features in the model to be studied such as part features or gates.

Why do it

Even though MPI is not a CAD program (it was not developed to design parts), it is sometimes necessary and convenient to create or add geometry to the model inside MPI for running an analysis. Changing the geometry inside MPI is a quick and convenient way to have the changes made in the model.

Overview

In this chapter, you will be learn about mesh elements and practice modeling within Synergy. The concepts that you will review are:

- Terminology.
- Properties.
- Features likely to be modeled within Synergy.

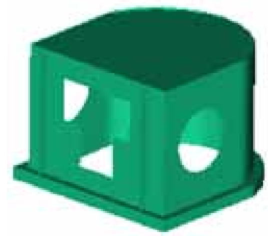
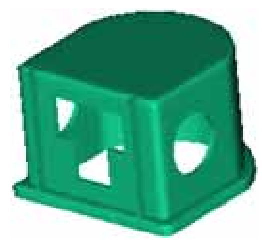
After you determine that a feature needs to be added to the model you will need to do the following:

1. Decide if the feature should be designed as a midplane or fusion mesh.
2. Create lines (or nodes) that will be used to define the features.
3. Set the property type depending on the model type, midplane or Fusion.
4. Create the regions.
5. Mesh the regions.
6. Check the mesh quality, and if needed do the adjustments.

The geometry creation steps will be discussed in detail in this unit.

Practice - Modeling Tools

This chapter has the practice split up by mesh type. For midplane mesh users, there are practices on two different parts. For Fusion and 3D mesh users, the practice involves one part. Pick the practice you want to work on.

Description	Model
Midplane mesh users	
Speedo: starts on page 139 The midplane version of the speedo is incomplete. You will finish creating the part model. This gives you practice at curve and region construction. As a reference, you will have the STL model. Then this model is used to practice using local coordinate systems and modeling plane.	
Dustpan: starts on page 155 A midplane model of a dustpan will be completely modeled using the tools in Synergy. This is additional practice at using curve and region construction.	
Fusion and 3D mesh users:	
Speedo: starts on page 163 The Fusion version of the speedo is complete. This model is used to practice using local coordinate systems and modeling plane. These techniques are used for both Fusion and 3D meshes.	

Speedo (Midplane)

Design Criteria

You will practice using the modeling tools by completing the speedo model. Modeling entities already constructed have been placed on specific layers to aid in the quick creation of the part model. In this exercise, you will:

- Construct a single-surface model of a part, in this case the outer surface dimensions.
- Create surface regions.
- Assigning the thickness to the model.
- Use local coordinate systems (LCS) and modeling planes to aid in the creation of runners and cooling circuits layouts.

Setup

To open a project

1. Click the file open icon or **File ➡ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Modeling_Tools**.
2. Double click the project file **Modeling_Tools.mpi**.
3. Click **File ➡ Preferences** and ensure that **System Units** are set to **Metric**.
4. Click **File ➡ Preferences ➡ Default Display tab**.

   4.1. Set the Region display to **Transparent**.

   4.2. Click **OK**.

   - Regions can be displayed several different ways including:
     - Solid.
     - Transparent.
     - Net.
     - Solid & net.
     - Transparent & net.
   - Figure 37 shows the display of regions in transparent versus net.

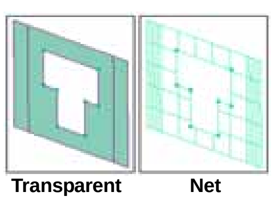


Figure 37: Transparent and net display methods for regions

The tasks below access commands by the menu. The toolbox can also be used.

To review the model

1. Open the study **speedo_MD**.
2. Investigate the model geometry using the **model manipulation** tools.
3. Turn on and off the layers.
   - Notice the various modeling entities are placed on specific layers to be used in the following steps.
   - The STL layer shows the actual part shape see Figure 38.

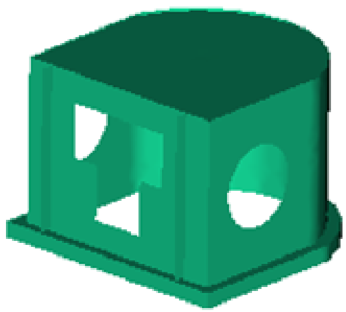


Figure 38: STL representation of the Speedo model

Finishing the Speedo

To create a region by boundary

1. Turn off all layers except for the one labeled **Region by Boundary**.
2. Rotate the model so the geometry can be seen. A reasonable rotation is **-60 -20 -15**.
3. Click **Modeling ➧ Create Region ➧ By Boundary.**
4. Select the curves one by one (hold down Ctrl) to form a rectangle.

   4.1. Pick the curves in order going around (clockwise or counterclockwise) for the region to be created.

   ➤ Be sure to include for each region any curve(s) that is common to two regions.

   4.2. Click **Apply** and the region is created.
5. **Repeat** step 3 to create the other two regions.
   - There are three different rectangular regions when you complete the task as shown in Figure 39.

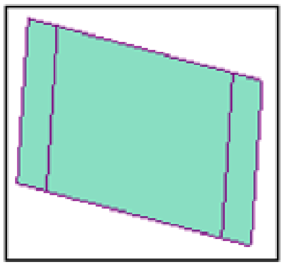


Figure 39: Create regions by boundary

To create a region by nodes

1. Turn off all layers except for the one labeled **Region by Nodes**.
2. Rotate the model so the geometry can be seen. A reasonable rotation is **-60 -40 -20**.
3. Click **Modeling ➡ Create Region ➡ By Nodes.**
4. Click the **STL Representation** layer briefly to see the way the regions need to be created. The surfaces are vertical (two) parallel to each other.
5. Hold down Ctrl key and select each node one by one, in a clockwise or counterclockwise direction.
6. Click **Apply** to create the region.
7. **Repeat** steps 4 and 5 to create the second rib. The regions create should look like Figure 40.

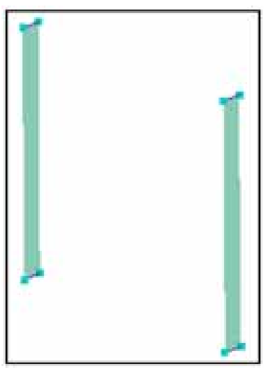


Figure 40: Regions created by nodes

To create a region by ruling

1. Turn off all layers except for the one labeled **Region by Ruling**.
2. Rotate the model so the geometry can be seen. A reasonable rotation is **-20 10 5**.
3. Click **Modeling ➡ Create Region ➡ By Ruling**.

4. Click **First Curve** in the dropdown beside the first curve entry.

Selection Lists saves selected model entities and assign them a specific name. These selection lists can be referenced for most functions. The selection lists in this exercise were previously saved using the **Save Selection List** icon from the Selection Toolbar. The benefit of the selection list is that it does not alter the entities original layer assignment.

5. Click **Second Curve** for the second curve entry.
   - Notice these two curves are parallel.
6. Click **Apply** to create the region.
   - The geometry should look like Figure 41.
7. **Continue** using the command create region by ruling by picking the curves manually until the curved surface is fully defined.

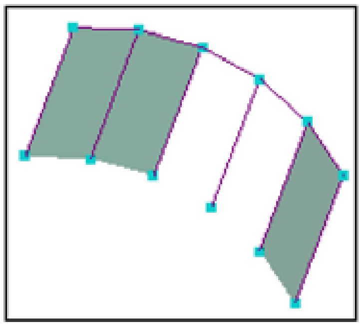


Figure 41: Create regions by ruling

To create a local coordinate system at the global location

1. Turn off all layers except for the one labeled **Region by Extrusion**.
2. Click **Modeling** ➡ **Local Coordinate System /Modeling Plane** ➡ **Define**.
3. Enter **0 0 0**, in the first coordinate field.
4. Click **Apply**.
5. Click the **STL Representation** layer to turn it on.
6. **Rotate** the model to identify the global location of the LCS.
7. Click the **STL Representation** layer to turn it off.

To create a region by extrusion

1. Rotate the model so the geometry can be seen. A reasonable rotation is **-50 -50 -30**.
2. Click **Modeling** ➡ **Create Region** ➡ **By Extrusion**.
3. Select the curve and enter the Extrude vector: **0, 0, -60.2.**
4. Click **Apply** to create the region.
   - Notice that the line was extruded in the negative Z axis of the global coordinate system, as shown in Figure 42.

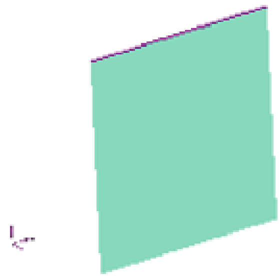


Figure 42: Region created by extrusion

To create a hole by boundary

1. Turn off all layers except for the one labeled **Region by Extrusion** and the one labeled **Hole by Boundary**.
2. Make the Hole by Boundary layer **Active** as well.
3. Click **Modeling** ➨ **Create Holes** ➨ **By Boundary**.
4. Select the **region** that is visible.
5. Select the **curve** that forms the hole.
6. Click **Apply** to create a hole in the region, as shown in Figure 43.

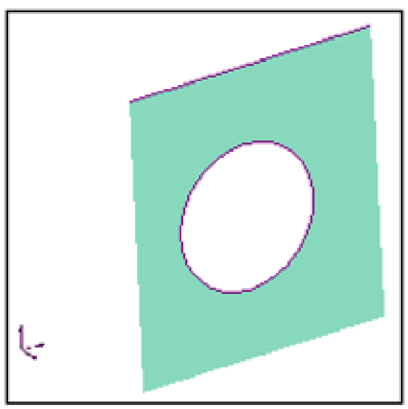


Figure 43: Create a hole by boundary

To create a hole by nodes

1. Turn off all layers except the one labeled **Region by Boundary** and the one labeled **Hole by Nodes.**
2. Make the Hole by Nodes layer **Active** as well.
3. Click **Modeling** ➨ **Create Holes** ➨ **By Nodes**.
4. Select the region that is visible.
5. Hold down the **Ctrl** and select each of the nodes that form the hole, in a clockwise or counterclockwise direction.
6. Click **Apply**, notice that the hole is created, as shown in Figure 44.

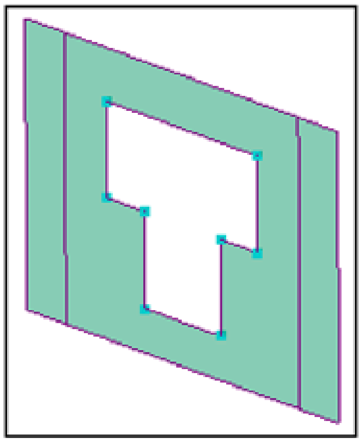


Figure 44: Hole by nodes

To finish creating the regions

1. Toggle on all the layers except for the STL layer.
   - Notice that some regions have been created for you previously which are located in the layer called Regions Remaining.
2. Create a curve starting at the center of the front surface to the top back center node. Use End of curve as the filter.
   - The new curve should look like Figure 45.

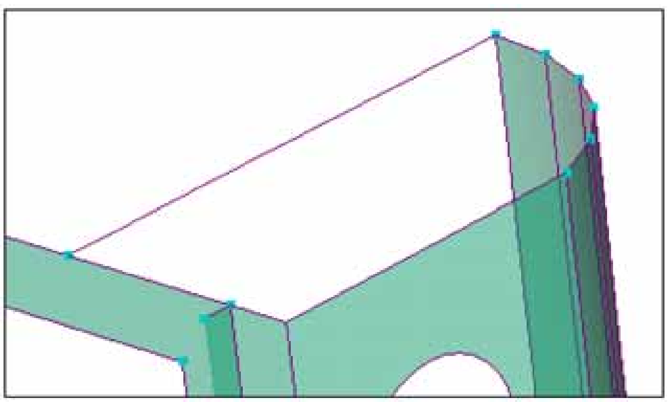


Figure 45: Top region center curve

3. Create the (half) top region using any of the commands described previously. The region should look like:

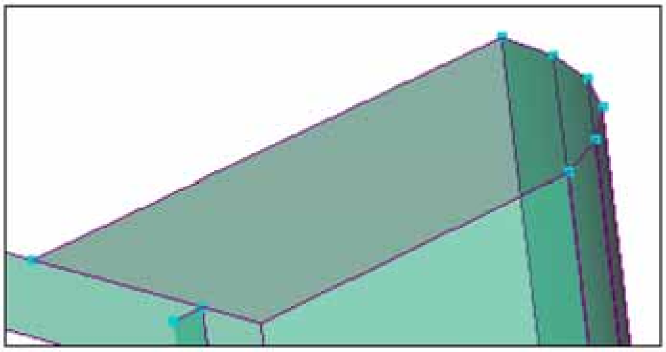


Figure 46: Top half region

4. Click **Modeling** ➡ **Move/Copy** ➡ **Reflect**.

4.1. Select all the regions that only create half of the part, as shown in Figure 47.

4.2. Pick **YZ** as the mirror plane.

4.3. Click **Copy**.

4.4. Check **Attempt connection to existing model**.

4.5. Click **Apply**.

- The model should now resemble the STL model but as a single surface form, as shown in Figure 48.

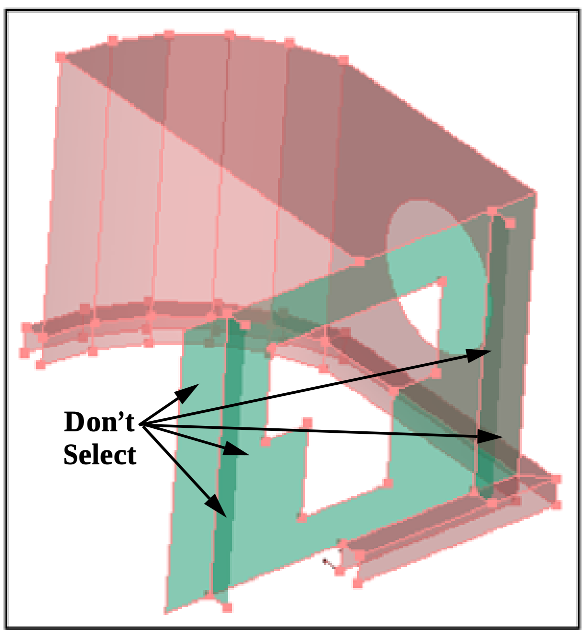


Figure 47: Entities selected for reflecting

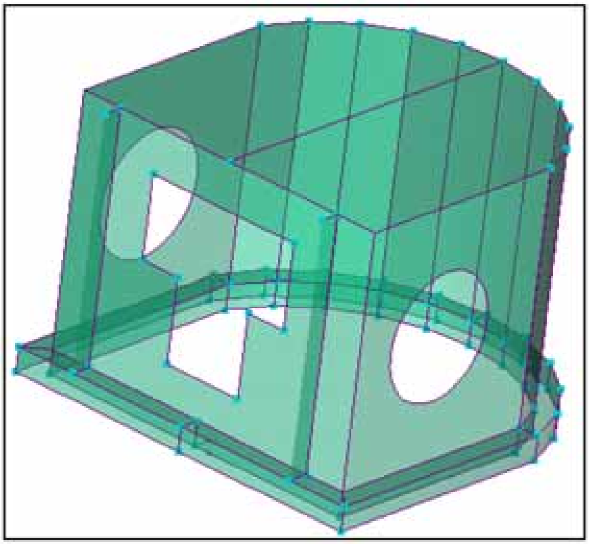


Figure 48: Completed model

Notice how the curved surface is a tessellation of many planar surfaces. This was done intentionally since MPI does not support the creation of non-planar regions. This technique is appropriate since the triangle mesh itself is also planar making the curved surface tessellated when it is meshed anyway.

To define thickness of the main regions

1. Toggle all the **layers on**.
2. Click **Edit ➡ Select By ➡ Properties**
   2.1. Select **Region**.
   2.2. Click **OK**.
3. Click **Edit ➡ Assign Property.**
   3.1. Click **New.**
   3.2. Select **Part Surface (Midplane)**.
   3.3. Set the thickness to **2.5** mm.
   3.4. Enter **Part surface 2.5 mm thickness** in the name field.
   3.5. Click **OK** on the part surface (midplane) dialog.
4. Click **OK** on the Assign property dialog.

To define thickness of the rib regions

1. Select the **regions** for the two ribs that extend from the part, as shown in Figure 49.
2. Click **Edit ➡ Assign Property.**
   2.1. Click **New.**
   2.2. Select **Part Surface (Midplane).**
   2.3. Set the thickness to **0.7**.
   2.4. Enter in the name field, **Part surface 0.7 mm thickness**.
   2.5. Click **OK** on the part surface (midplane) dialog.
3. Click **OK** on the Assign property dialog.

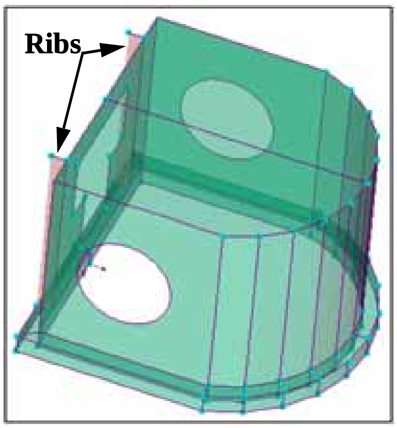


Figure 49: Ribs selected

To mesh the regions and verify mesh quality

1. Click **Mesh ➡ Generate Mesh ➡ Mesh Now.**
2. Click **Mesh ➡ Mesh Statistics.**
3. Ensure there are no problems with the mesh the mesh.

Creating runners using a LCS

You have completed the creation of the part model, you may now proceed to create the runner system with the help of a Local Coordinate System.

To create a Local Coordinate System (LCS)

1. The injection location should be located in the center of the top surface; in order to ensure this location you will use an LCS.
2. Click **Modeling ➡ Local Coordinate System / Modeling Plane ➡ Define.**
3. For the first coordinate, enter: **0, 43.2, 66**.
4. Click **Apply**.
5. Zoom into the area where the new LCS has been created (center top surface).

6. Create a new layer.

   6.1. Click the new layer icon  in the layers pane.

   6.2. Name the layer **LCS**.

   6.3. Select the LCS icon on the model.

   6.4. Click the Assign to layer icon  to put the LCS on the new layer.

To move a node for the new injection location

1. Click **Mesh** ➨ **Mesh Tools** ➨ **Nodal Tools** ➨ **Move Nodes**.
2. Ensure the **New Nodes** layer is on.
3. Select the node that is closest to the LCS (the coordinate **0, 43.2, 66**).
4. Enter the absolute coordinate of **0, 43.2, 66.**
5. Click **Apply** then **Close**.
   - The LCS was created in this task so you could see where you needed to move the node.

To specify an injection location

1. Turn off the LCS layer.
   - You may not be able to assign the injection location with the LCS displayed.
2. Click **Analysis** ➨ **Set Injection Location**.
3. **Select** the same node that you just moved.

To generate the runner system using the Runner System Wizard

1. Click **Modeling** ➨ **Runner System Wizard.**
2. On page one enter the following:

   2.1. Enter the sprue position of **X=0** and **Y= -25.4.**

   2.2. Check on the box for a **I would like to use a hot runner system**.

   2.3. Enter top runner plane distance of **152**.

   2.4. Click **Next**.
3. On page two enter the following:

   3.1. Enter **10** for orifice diameter.

   3.2. Enter **0** for the included angle.

   3.3. Enter **102** for the length.

   3.4. Enter **10** for the runner diameter.

   3.5. Enter **10** for the drop diameter.

   3.6. Enter **0** for the included angle.

   3.7. Click **Next**.

4. On page three enter the following:
   - **4.1.** Enter **2.5** for the top gate start diameter.
   - **4.2.** Enter **2.5** for the end diameter.
   - **4.3.** Enter **5** for the length.
   - **4.4.** Click **Finish**.

> The Runner System Wizard used the global coordinate system. You have identified its location previously when you created the first LCS. Until now you have used the LCS as a reference since it has not been activated.

> You will learn all about the Runner System Wizard in the Runner and Gate Design unit.

You will learn all about the Cooling Circuit Wizard in the Modeling components unit of the MPI/Advanced Cool training. Now you will use the wizard to create a basic cooling circuit layout.

To create a cooling line system using the Cooling Circuit Wizard

1. Click **Modeling ➡ Cooling Circuit Wizard.**
2. On page one enter the following:
   - **2.1.** Enter **10** mm as the channel diameter.
   - **2.2.** Enter **25** as the distance above and below the part.
   - **2.3.** Enter **Y** as the direction of circuit alignment.
   - **2.4.** Click **Next**.
3. On page two enter:
   - **3.1.** Enter **2** as the number of channels.
   - **3.2.** Enter **64** as the distance between channels.
   - **3.3.** Enter **25** as the distance beyond part.
   - **3.4.** Click **Finish**.

To create and activate a Local Coordinate System (LCS)

The last cooling channel needs to be created manually. A LCS will be created at the coolant entrance to speed up the circuit creation. The channel starting position is at **0, - 30.5, - 51**.

1. Click **Modeling ➡ Local Coordinate System/ Modeling Plane ➡ Define.**
2. Enter the following information for the coordinates:
   - First: **0, - 30.5, -51.**
   - Second: **0, 25, -51**.
   - Third: **-25, -30.5, -51**.
3. Click **Apply** then **Close**.

4. **Select** the LCS.
5. Right-click and select **Activate as LCS**.
    - The color turns to red when active.

To create the cooling channel manually

1. Create a new Layer called Channel 3 and ensure it is active.
2. Click **Modeling ➡ Create Curves** and create the following curves using absolute coordinates:
    - 0, 0, 0 to 68.5, 0, 0
    - 68.5, 0, 0 to 71, 0, 102
    - 71, 0, 102 to 74, 0, 0
    - 74, 0, 0 to 203, 0, 0
        - ➤ The model should look like Figure 50.
3. Click **Apply** to create each line individually.
4. Click **Close** when finish creating the lines.

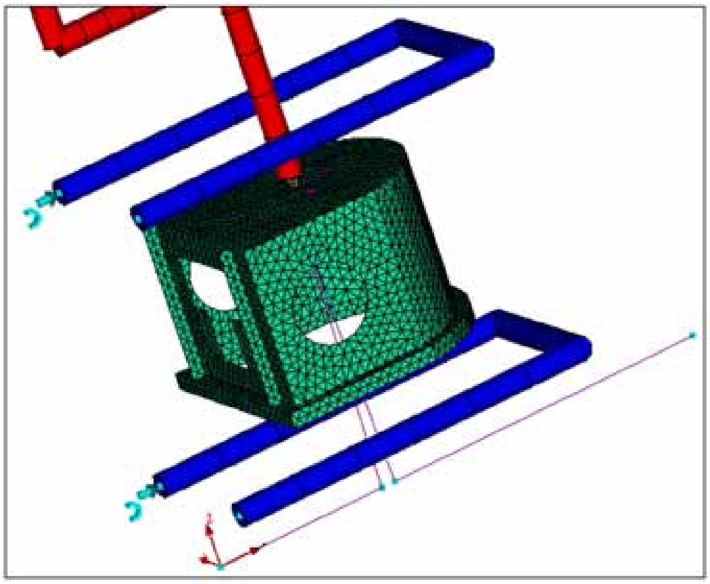


Figure 50: Curves for the bottom cooling line

To set the properties of the new cooling channel section just created

1. Select the two vertical lines (along the z axis).
2. Click **Edit** ➡ **Assign Property**.
   - **2.1.** Select **New**.
   - **2.2.** Select **Baffle**.
   - **2.3.** Enter **15** as the diameter.
   - **2.4.** Click **OK** on the Baffle dialog.
   - **2.5.** Click **OK** on the Assign property dialog.
3. Select the two horizontal lines.
4. Click **Edit** ➡ **Assign Property** ➡ **New** ➡ **Channel.**
5. Click **Edit** ➡ **Assign Property**.
   - **5.1.** Select **New**.
   - **5.2.** Select **Channel**.
   - **5.3.** Enter **10** as the diameter.
   - **5.4.** Click **OK** on the channel dialog.
   - **5.5.** Click **OK** on the Assign property dialog.

To mesh the new cooling channel

1. Click **Mesh** ➡ **Generate Mesh.**
2. Enter **25** (2.5 x channel diameter) as the global edge length.
3. Check **Place mesh in active layer**.
4. Click **Mesh Now**.

We would like to modify the cooling channel end points so they extend past the hot runner. In the next task you will learn how you can modify it easily using the Modeling plane.

To set up modeling plane preferences

1. Click **File** ➡ **Preferences** ➡ **General**.
2. Under Modeling Grid:
   - **2.1.** Enter **76** for the grid size
   - **2.2.** Check the box for snap to grid.
3. Press **OK**.

To use a modeling plane

1. Click **Mesh** ➡ **Mesh Tools** ➡ **Nodal Tools** ➡ **Move Nodes**.
2. Pick the node to move as one of the five endpoints.
   - Notice how a modeling grid appears and the node that was selected lines up with one of the grid points.
   - Make sure that the nodes that you need to move are visible, if they are not you will need to turn on the layer to display the node(s).
3. Drag the node until the crosshairs reaches one grid line in the extension direction.
4. Let go the left mouse button.
   - The program will display a dark blue point at the location where the node will be moved. See this blue point in Figure 51 highlighted by a circle.

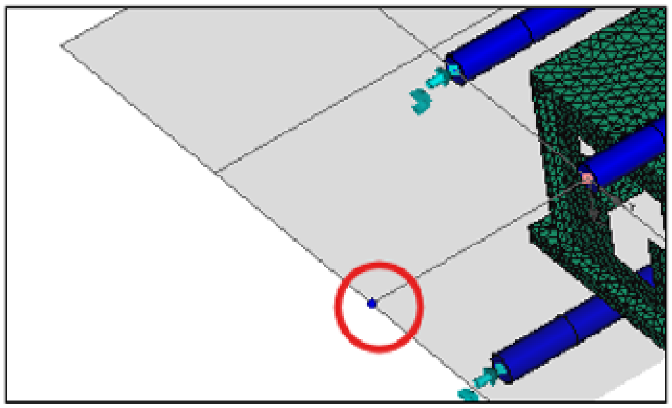


Figure 51: Point where to move the end cooling channel node

5. Click **Apply** and the node will be moved and the mesh density will remain intact.
6. **Repeat** the necessary steps to move the end nodes of the other channels.
   - The final mesh model should look like Figure 52.

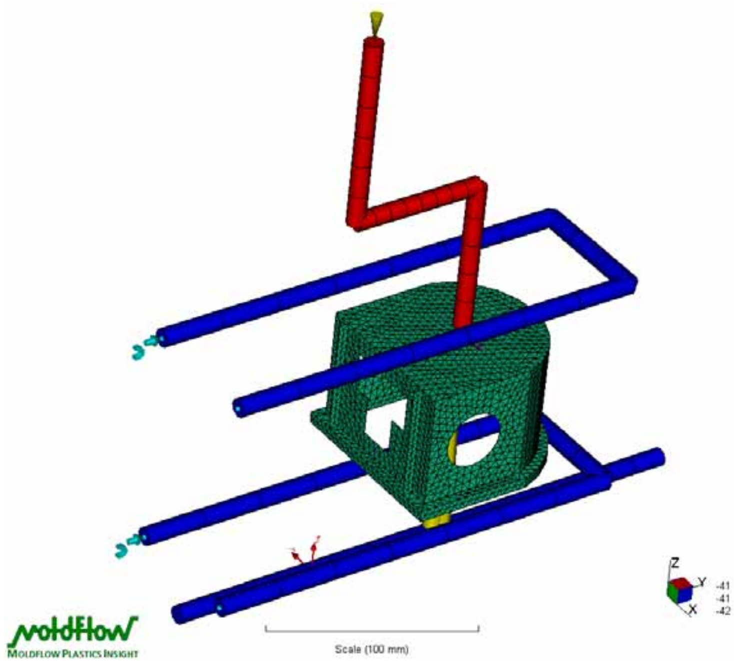


Figure 52: Completed Speedo model

In this example you learned how modeling planes could be used to edit existing entities. Modeling planes can also be used effectively in assisting the creation of geometry, especially runner and cooling systems. After creating a LCS, right-click and select **Activate as Modeling Plane**. Additionally, each of the geometry tools has a filter that can be used with the Modeling Plane.

There are several additional steps to finally prepare this model for simulation, but this is the end of the focus for this exercise. If there is still extra time before moving on, try the next exercise.

This example gives only a brief idea of the capabilities of the LCS. Consider the ability to create entities at defined angles or to define a local modeling plane. Look at the description of the LCS and modeling plane in the on-line help to get further guidance.

Dustpan (Midplane)

Design Criteria

For the dustpan model you will create a midplane model of a dustpan with a curved handle using many of MPI's modeling tools. In this exercise, you will:

- Construct a single-surface model of a part at the outer surface dimensions.
- Create surface regions.
- Mesh the model.

Setup

To open a project

1. Click the file open icon or **File** ➨ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Modeling_Tools**.
2. Double click the project file **Modeling_Tools.mpi**.
3. Click **File** ➨ **Preferences** and ensure that **System Units** are set to **Metric**.

To create a new study

1. Click **File** ➨ **New** ➨ **Study.**
2. Click the right mouse button and select **Rename**.
3. Type **pan**.
    - This creates a new study, called pan.

Creating the dustpan

To create horizontal curve for the basic shape

1. Click **Modeling** ➨ **Create Curves** ➨ **Line**.
2. In the **Filter** box, ensure that **Any Item** is selected.
3. In the **Create As** box, ensure that **Modeling Entity** is selected.
4. Ensure that **Absolute** is selected.
    - Enter **0** in the First Coordinate box.
    - Enter **60** in the Second Coordinate box.
5. Click **Apply**.
    - Notice that a horizontal line is drawn.

To create vertical curves for the basic shape

1. Continue in the **Create Curves** ➨ **Line** pane.
2. Select **End of curve**, in the **Filter** field.

3. Click in the First Coordinate box.
4. **Band-select** the **left** end of the line.
   - Notice that in the pane, First Coordinate is set to **0 0 0**.
5. Click the **Relative** button.
6. In the Second Coordinate box, enter **0 40 0**.
7. Click **Apply**.
8. **Repeat** steps 4 to 7 (selecting the right end of the horizontal line) to **create** a third line on your own, so that the model looks Figure 53.

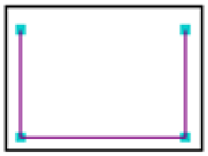


Figure 53: Dustpan first three curves

To create the reference node for an arc

1. Click **Modeling** ➡ **Create Nodes** ➡ **By Coordinate**.
2. In the Coordinate box, enter **30 50 0**.
3. Click **Apply**.

To create curve for arc

1. Click **Modeling** ➡ **Create Curves** ➡ **Arc by Points**.
2. Ensure **Modeling Entity** is selected in the **Create As** box.
3. Select **End of curve**, in the **Filter** box.
4. **Band-select** the top corner of the **left vertical** line.
   - Notice that in the pane, First Coordinate changes to **0 40 0**.
5. Click in the **Third Coordinate** box.
6. **Band-select** the top corner of the **right vertical** line.
   - Notice that in the pane, Third Coordinate changes to **60 40 0**.
7. Click in the **Second Coordinate** box.
8. Set Filter to **Node**.
9. **Rubber band** the node (at 30 50 0) to select it.
10. Ensure that **Arc** is selected.
11. Click **Apply**.

To copy by translating

1. Click **Modeling** ➡ **Move/Copy** ➡ **Translate**.
2. Ensure **Any ite**m is the **Filter**.

3. **Band-select** the entire model you have created so far.
   - Notice that in the pane, the Select box is set to N4 N1 N2 N3 N5 C1 C2 C4 C3.
4. In the **Vector** box enter: **0 0 10**.
5. Click **Copy**.
6. Enter **1** in the Number of Copies box.
7. Click **Apply** then **Close**.
8. Rotate the model to see the created curves as shown in Figure 54.

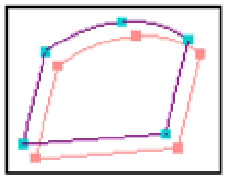


Figure 54: Curves created by translation

To connect the original shape to the copy

1. Rotate the part to **-50 -20 -5**.
2. Click **Modeling** ➡ **Create Curves** ➡ **Line**.
3. Select **End of curve** as the **Filter**.
4. Click in the **First Coordinate** box.
5. **Band-select** the location labeled 1 in Figure 55.
   - Notice that in the pane, First Coordinate changes to **0 40 0**.
6. **Click** in the Second Coordinate box.
7. **Band-select** the location labeled 2 in Figure 55.
   Notice that in the pane, Second Coordinate changes to **0 40 10**.
8. Click **Apply**.
   - You have created one new line connecting the top and bottom geometries.
9. Follow the procedure used in the previous steps to create **three additional corner lines**.
   - Remember to rubber band-select a corner, rather than a single line, to correctly specify an end point.
10. Click **Close** when finished modeling.
    - Your model should look like Figure 56.

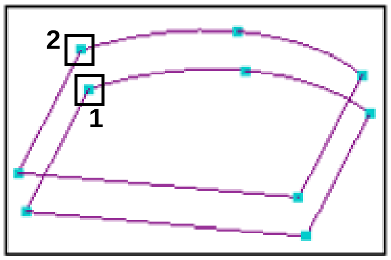


Figure 55: Band selecting the end of a curve to define coordinate

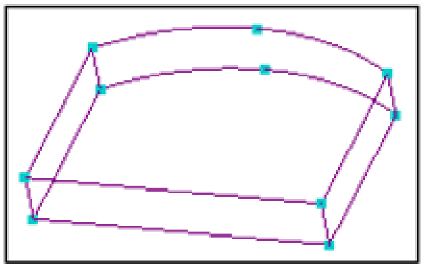


Figure 56: Connecting lines top and bottom geometry

To delete one arc and node

1. Rotate the part to **-50 -20 -5**.
2. Select the upper arc and the center node on the arc.
3. **Right-click** the mouse and click **Delete**.

   3.1. Ensure that both Node and Curve are selected, on the Select entity types dialog.

   3.2. Click **OK**.

   - Notice that the node and curve are deleted.

To create 2 regions to form the sides of the pan

1. Click **Modeling** ➧ **Create Regions** ➧ **Region by Boundary.**
2. Select **Any Item** as the **Filter**.
3. Hold down the **Ctrl** key and **select** all four curves that form the boundary of the left side of the dust pan, as shown in Figure 57.

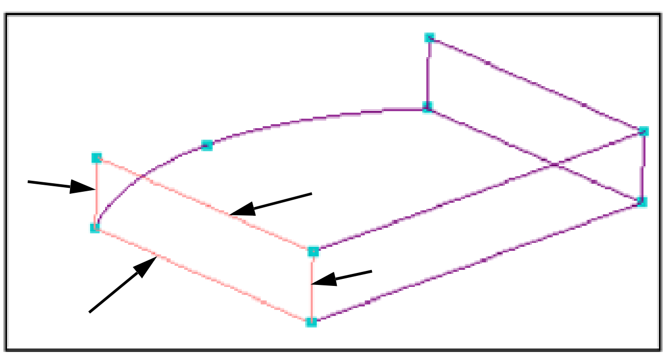


Figure 57: Selecting curves to define a region

4. Click **Apply**.
5. **Create** a region for the right side of the pan as you did for the left side.
6. Click **Close**.

To divide the back of the pan for preparing to model the handle

1. Click **Modeling ➡ Create Curves ➡ Line**.
2. Select **Middle of curve** in the **Filter** field.
3. Create a center curve.
   - **3.1. Click** the top line at the back of the pan.
   - **3.2. Click** in the **Second Coordinate** box.
   - **3.3. Click** the bottom line.
     - ➤ Parallel to the one selected previously.
   - **3.4.** Ensure the box **Automatically create nodes at end points of curve** is checked.
   - **3.5.** Click **Apply**.
     - ➤ The new line is created, dividing the back of the pan in half.
4. Divide the curves on the back of the dust pan.
   - **4.1.** Select **Break Curve** in the combo box in the tools pane, or click **Modeling ➡ Create Curves ➡ Break curves**.
   - **4.2.** Ensure that the **First curve** box is highlighted.
   - **4.3. Select** the top line at the back of the pan, as shown in Figure 58.
   - **4.4.** Ensure that the **Second curve** box is highlighted.
   - **4.5. Select** the middle line, just created.

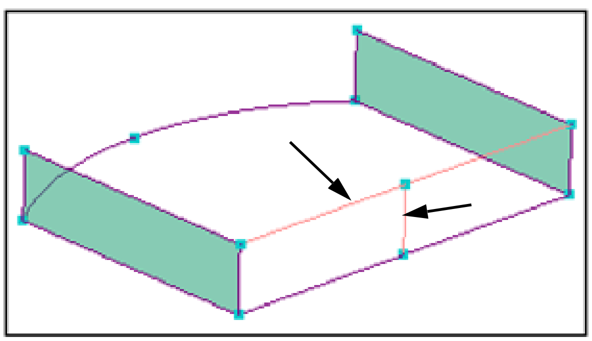


Figure 58: Curves to break

   - **4.6.** Click **Apply** to break the top line.
   - **4.7. Break** the bottom line at the back of the pan, just as you broke the top line.
5. Click **Close**.

To create offset nodes for the handle

1. Click **Modeling ➡ Create Nodes ➡ By Offset.**
2. Set the Base Coordinate box to the midpoint of the bottom back of the pan (30 0 0) for each node created.

> You may need to close the measurements dialog box several times while completing this task.

3. Create nodes, using these vectors:
   - 0 –5 3.
   - 0 –10 3.5.
   - 0 –15 3.5.
   - 0 –18 3.
   - 0 –25 4.
   - 0 –25 8.
   - 0 –15 11.
4. Click **Close**.

To create curves as spline for the handle

1. Click **Modeling ➡ Create Curves ➡ Spline**.
2. Click each node, in the **same order** they were created.
3. Click the node at **30 0 10**, at the top of the dustpan as the last node in the sequence.
4. Click **Apply**.
   - Notice that the spline is created. The model should look like Figure 59.

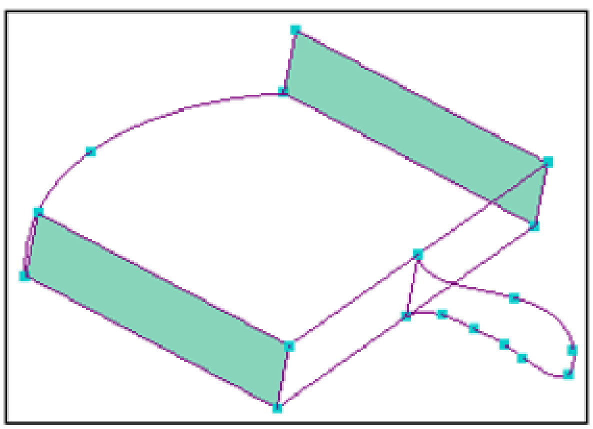


Figure 59: Dust pan geometry model

To create the rest of the regions

1. Create a curve dividing the bottom of the pan.
   - It should appear as indicated in Figure 60.

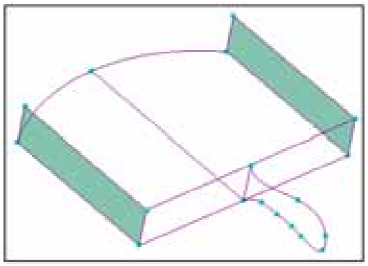


Figure 60: Bottom split

2. **Break** the arc where it intersects the line you just created.
3. Create five new regions, as indicated in Figure 61.

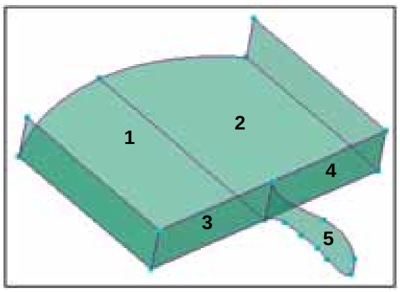


Figure 61: Regions for dustpan finished

To mesh the model

1. Click **Mesh ➨ Generate Mesh.**
2. Enter **2.5** as the Global edge length.
3. Enter **0.5** as the chord height.
4. Click **Mesh Now**.
5. Click **Mesh ➨ Mesh Statistics** and evaluate the mesh.

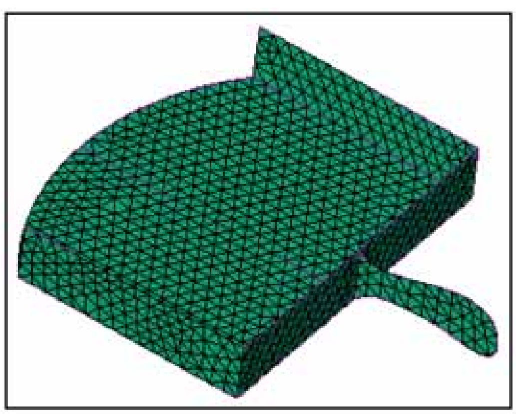


Figure 62: Dust pan mesh model

6. The model is ready for you to set an injection location, select the material, and run an analysis.

Speedo (Fusion)

Design Criteria

If you use Fusion or 3D models in your analysis this will be the practice that you are interested in completing. In this exercise, you will:

- Use local coordinate systems (LCS) and modeling planes to aid in the creation of runners and cooling circuits layouts.

Setup

To open a project

1. Click the file open icon or **File** ➡ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Modeling_Tools**.
2. Double click the project file **Modeling_Tools.mpi**.
3. Click **File** ➡ **Preferences** and ensure that **System Units** are set to **Metric**.

To review the model

1. Open the study **speedo_Fusion**.
2. Investigate the model geometry using the **model manipulation** tools.
3. Click **File** ➡ **Preferences** and ensure the units are set to **Metric**.
4. Turn on and off the layers.
   - Notice the STL layer shows the actual part shape see Figure 63.

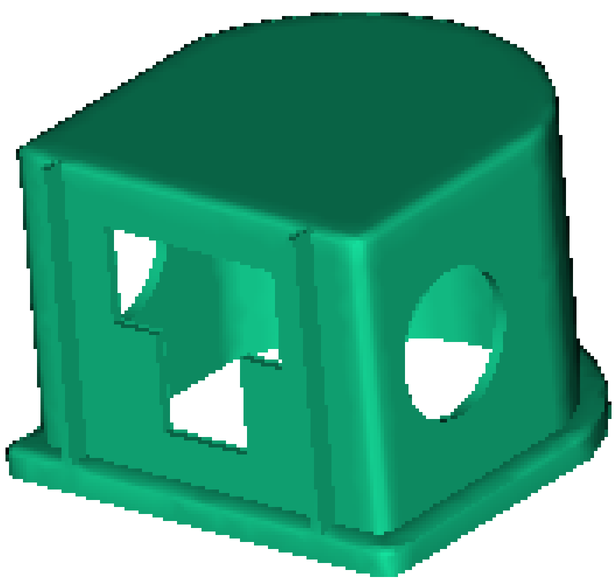


Figure 63: STL representation of the Speedo model

Creating runners using a LCS

Proceed to create the runner system with the help of a Local Coordinate System.

To create a Local Coordinate System (LCS)

1. The injection location should be located in the center of the top surface; in order to ensure this location you will use an LCS.
2. Click **Modeling ➡ Local Coordinate System / Modeling Plane ➡ Define.**
3. For the first coordinate, enter: **0, 43.2, 66.25**.
4. Click **Apply**.
5. Zoom into the area where the new LCS has been created (center top surface).
6. Create a new layer.

   6.1. Click the new layer icon  in the layers pane.

   6.2. Name the layer **LCS**.

   6.3. Select the LCS icon on the model.

   6.4. Click the Assign to layer icon  to put the LCS on the new layer.

To move a node for the new injection location

1. Click **Mesh ➡ Mesh Tools ➡ Nodal Tools ➡ Move Nodes**.
2. Ensure the **New Nodes** layer is on.
3. Select the node that is closest to the LCS (the coordinate **0, 43.2, 66.25**).
4. Enter the absolute coordinate of **0, 43.2, 66.25.**
5. Click **Apply** then **Close**.
   - The LCS was created in this task so you could see where you needed to move the node.

To specify an injection location

1. Turn off the LCS layer.
   - You may not be able to assign the injection location with the LCS displayed.
2. Click **Analysis ➡ Set Injection Location**.
3. **Select** the same node that you just moved.

To generate the runner system using the Runner System Wizard

1. Click **Modeling ➡ Runner System Wizard.**
2. On page one enter the following:
   - **2.1.** Enter the sprue position of **X=0** and **Y= -25.4.**
   - **2.2.** Check on the box for a **I would like to use a hot runner system**.
   - **2.3.** Enter top runner plane distance of **152**.
   - **2.4.** Click **Next**.
3. On page two enter the following:
   - **3.1.** Enter **10** for orifice diameter.
   - **3.2.** Enter **0** for the included angle.
   - **3.3.** Enter **102** for the length.
   - **3.4.** Enter **10** for the runner diameter.
   - **3.5.** Enter **10** for the drop diameter.
   - **3.6.** Enter **0** for the included angle.
   - **3.7.** Click **Next**.
4. On page three enter the following:
   - **4.1.** Enter **2.5** for the top gate start diameter.
   - **4.2.** Enter **2.5** for the end diameter.
   - **4.3.** Enter **5** for the length.
   - **4.4.** Click **Finish**.

> The Runner System Wizard used the global coordinate system. You have identified its location previously when you created the first LCS. Until now you have used the LCS as a reference since it has not been activated.

> You will learn all about the Runner System Wizard in the Runner and Gate Design unit.

You will learn all about the Cooling Circuit Wizard in the Modeling components unit of the MPI/Advanced Cool training. Now you will use the wizard to create a basic cooling circuit layout.

To create a cooling line system using the Cooling Circuit Wizard

1. Click **Modeling ➡ Cooling Circuit Wizard.**
2. On page one enter the following:
   - **2.1.** Enter **10** mm as the channel diameter.
   - **2.2.** Enter **25** as the distance above and below the part.
   - **2.3.** Enter **Y** as the direction of circuit alignment.
   - **2.4.** Click **Next**.

3. On page two enter:

   3.1. Enter **2** as the number of channels.

   3.2. Enter **64** as the distance between channels.

   3.3. Enter **25** as the distance beyond part.

   3.4. Click **Finish**.

To create and activate a Local Coordinate System (LCS)

The last cooling channel needs to be created manually. A LCS will be created at the coolant entrance to speed up the circuit creation. The channel starting position is at **0, - 30.5, - 51**.

1. Click **Modeling ➡ Local Coordinate System/ Modeling Plane ➡ Define.**
2. Enter the following information for the coordinates:
   - First: **0, - 30.5, -51.**
   - Second: **0, 25, -51**.
   - Third: **-25, -30.5, -51**.
3. Click **Apply** then **Close**.
4. **Select** the LCS.
5. Right-click and select **Activate as LCS**.
   - The color turns to red when active.

To create the cooling channel manually

1. Create a new Layer called Channel 3 and ensure it is active.
2. Click **Modeling ➡ Create Curves ➡ Line** and create the following curves using absolute coordinates:
   - 0, 0, 0 to 68.5, 0, 0
   - 68.5, 0, 0 to 71, 0, 102
   - 71, 0, 102 to 74, 0, 0
   - 74, 0, 0 to 203, 0, 0
     - ➤ The model should look like Figure 64.
3. Click **Apply** to create each line individually.
4. Click **Close** when finish creating the lines.

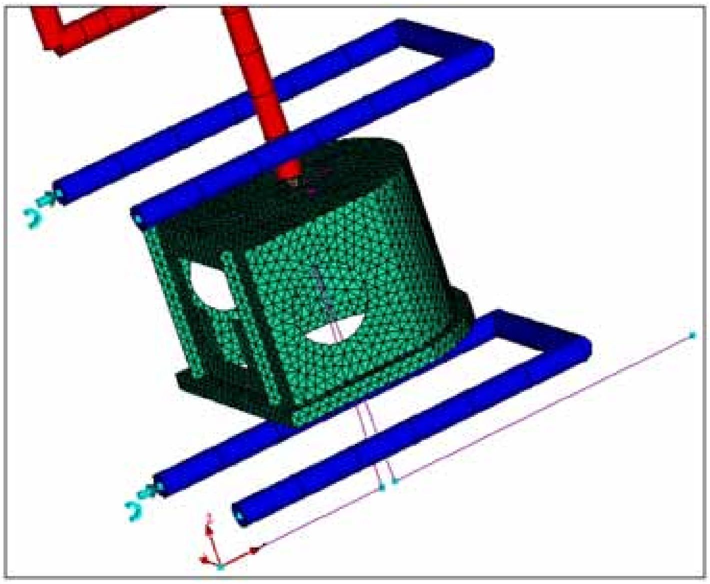


Figure 64: Curves for the bottom cooling line

To set the properties of the new cooling channel section just created

1. Select the two vertical lines (along the z axis).
2. Click **Edit** ➨ **Assign Property**.
   - **2.1.** Select **New**.
   - **2.2.** Select **Baffle**.
   - **2.3.** Enter **15** as the diameter.
   - **2.4.** Click **OK** on the Baffle dialog.
   - **2.5.** Click **OK** on the Assign property dialog.
3. Select the two horizontal lines.
4. Click **Edit** ➨ **Assign Property** ➨ **New** ➨ **Channel.**
5. Click **Edit** ➨ **Assign Property**.
   - **5.1.** Select **New**.
   - **5.2.** Select **Channel**.
   - **5.3.** Enter **10** as the diameter.
   - **5.4.** Click **OK** on the channel dialog.
   - **5.5.** Click **OK** on the Assign property dialog.

To mesh the new cooling channel

1. Click **Mesh** ➨ **Generate Mesh.**
2. Enter **25** (2.5 x channel diameter) as the global edge length.
3. Check **Place mesh in active layer**.
4. Click **Mesh Now**.

We would like to modify the cooling channel end points so they extend past the hot runner. In the next task you will learn how you can modify it easily using the Modeling plane.

To set up modeling plane preferences

1. Click **File** ➡ **Preferences** ➡ **General**.

2. Under Modeling Grid:

  2.1. Enter **76** for the grid size

  2.2. Check the box for snap to grid.

3. Press **OK**.

To use a modeling plane

1. Click **Mesh** ➡ **Mesh Tools** ➡ **Nodal Tools** ➡ **Move Nodes**.

2. Pick the node to move as one of the five endpoints.

- Notice how a modeling grid appears and the node that was selected lines up with one of the grid points.
- Make sure that the nodes that you need to move are visible, if they are not you will need to turn on the layer to display the node(s).

3. Drag the node until the crosshairs reaches one grid line in the extension direction.

4. Let go the left mouse button.

- The program will display a dark blue point at the location where the node will be moved. See this blue point in Figure 65 highlighted by a circle.

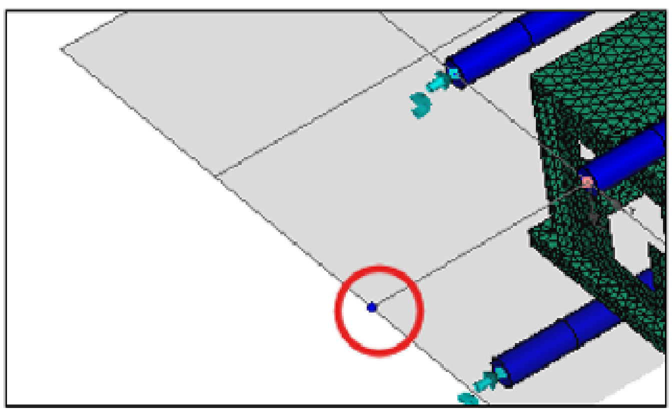


Figure 65: Point where to move the end cooling channel node

5. Click **Apply** and the node will be moved and the mesh density will remain intact.

6. **Repeat** the necessary steps to move the end nodes of the other channels.

- The final mesh model should look like Figure 66.

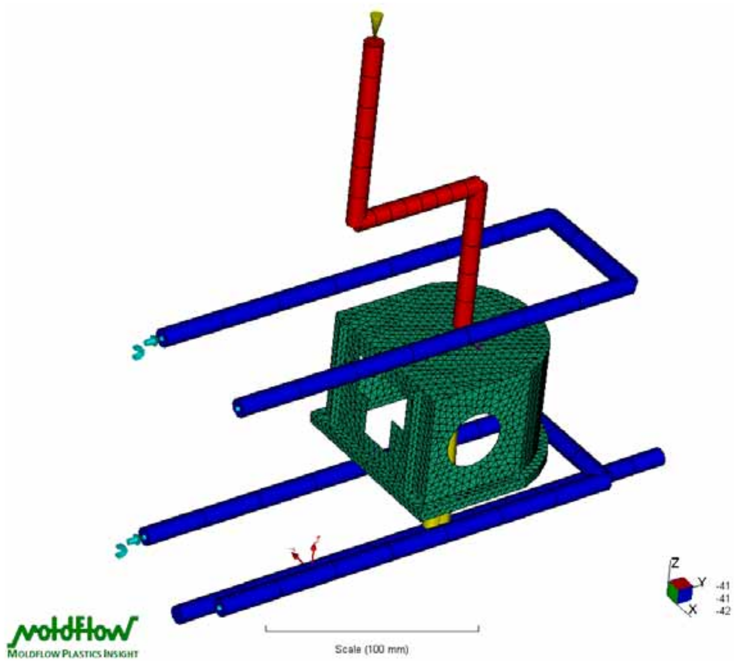


Figure 66: Completed Speedo model

In this example you learned how modeling planes could be used to edit existing entities. Modeling planes can also be used effectively in assisting the creation of geometry, especially runner and cooling systems. After creating a LCS, right-click and select **Activate as Modeling Plane**. Additionally, each of the geometry tools has a filter that can be used with the Modeling Plane.

There are several additional steps to finally prepare this model for simulation, but this is the end of the focus for this exercise. If there is still extra time before moving on, try the next exercise.

This example gives only a brief idea of the capabilities of the LCS. Consider the ability to create entities at defined angles or to define a local modeling plane. Look at the description of the LCS and modeling plane in the on-line help to get further guidance.

Competency check - Modeling Tools

1. What kind of entities are you likely to model inside MPI?
2. Why do you use filters for?
3. What does LCS stand for?
4. Is there a limit of LCS that you can have in a model?
5. When are LCS used mainly?

Evaluation Sheet - Modeling Tools

1. What kind of entities are you likely to model inside MPI? Runner, gates and sprues, cooling line circuits and even fan gates. The geometries are based on nodes, curves and or regions. In general, the geometries are features that are straightforward to create inside the interface.
2. Why do you use filters for? To narrow down the selection. Very useful when many elements of different kind are on the screen close to each other.
3. What does LCS stand for? Local coordinate system.
4. Is there a limit of LCS that you can have in a model? No, there is no limit.
5. When are LCS used mainly? LCS are used for modeling aid and as for results interpretation.

CHAPTER 11

Introduction to Moldflow Magics STL Expert

Aim

Learn how Moldflow Magics STL Expert can be used to fix & optimized the original CAD file.

Why do it

When importing a model inside MPI previously optimized by Moldflow Magics STL Expert the cleanup time process inside MPI is reduced.

Overview

In this chapter, you will be introduced and practice using Moldflow Magics STL Expert. You will learn about:

- Program overview.
- Supported Models.
- Licensing & Hardware Support.

You will practice with the program features:

- STL Fix Wizard.
- Manual Fixing Tools.
- Optimization Tools.
- Measurement Tools.

Practice - Introduction to Moldflow Magics STL Expert

In this practice session, you complete different exercises designed to familiarize you with the usage of Moldflow Magics STL Expert. The exercises include:

- Exercise 1: Fix Wizard on page 178.
- Exercise 2: Normals on page 180.
- Exercise 3: Stitch on page 181.
- Exercise 4: Stitch + Normals on page 183.
- Exercise 5: Holes on page 184.
- Exercise 6: Shells on page 188.
- Exercise 7: Overlaps on page 190.
- Exercise 8: Summary on page 191.
- Exercise 9: Triangles on page 192.
- Exercise 10: Importing an IGES file on page 196.

Exercise 1: Fix Wizard

This exercise introduces you to the Fix Wizard inside Moldflow Magics STL Expert which fixes automatically the model for you.

To open the model

1. Click **File ➨ Load Part.**
2. **Browse…** and navigate to the **My MPI 6.0 Projects\MPI_Fundamentals\Magics_STLs** folder.
3. Select the file **01 FixWizard.stl.**
4. Click **Open**.
   - The model for this exercise is shown in Figure 67.

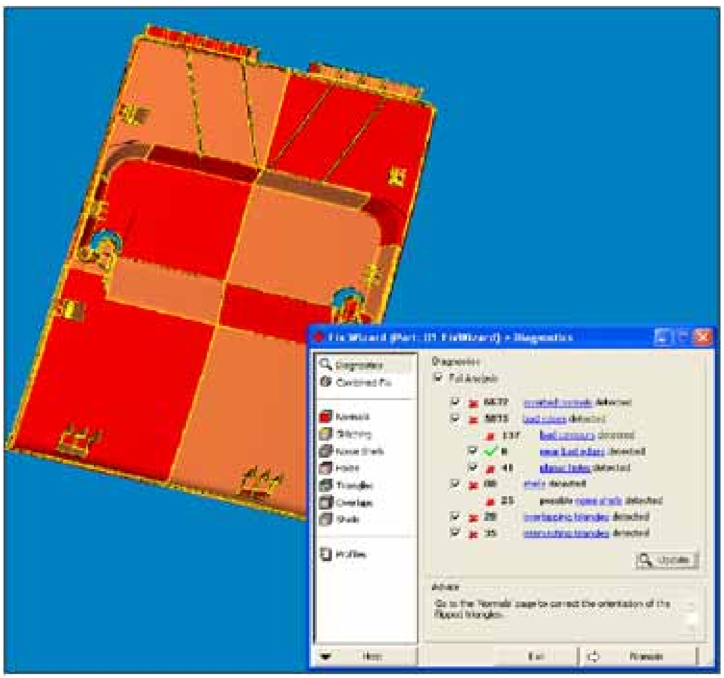


Figure 67: Model to fix - 01 FixWizard.stl

To fix automatically the model

1. If the **Fix Wizard** does not launch automatically, select **Tools ➨ Fix Wizard (Ctrl+F).**
2. Click **Update**.
   - A list of various errors is shown. The Advice window at the bottom provides problem-specific advice on the correct course of action. The advice window suggests going to the **Normals** page.
3. Click **Normals** to go to the normal fixing page.
4. Click **Apply** to automatically fix STL facets whose normals are not properly oriented.

5. After the automatic fixing is completed (should only take a second or two), click **Update** to check how many normals remain to be fixed. In this case, all the normals are fixed in the first step.
6. Click **Diagnostics** to return back to the diagnostics page.
7. Click **Update** to recalculate the list of issues.
    - Based on the revised list of issues, the program will provide Advice to target the problems.

> The above sequences of steps are very commonly repeated in Moldflow Magics STL Expert. The steps are:
>
> 1. Follow the Advice to go to a specific fixing page.
> 2. Use the automatic fixing tool to fix as many problems as possible.
> 3. Use the manual tools to correct the remainder.
> 4. After each fixing step, select **Update** to get a count of the number of issues remaining.
> 5. After all issues are fixed, select **Diagnostics** to confirm the change.

8. Repeat the above steps to fix **Noise Shells**.
9. Go to the **Diagnostics** page and check on the **Full Analysis** option.
10. Click **Update**.
    - This will reveal that there are still overlapping & intersecting triangles remaining. Follow the wizard to solve them automatically.

Exercise 2: Normals

In this exercise you will use Moldflow Magics STL Expert to fix the normals' orientation of the part automatically.

To open the model

1. Click **File ➡ Load Part.**
2. **Browse…** and navigate to the **My MPI 6.0 Projects\MPI_Fundamentals\Magics_STLs** folder.
3. Select the file **02 normals.stl**.
4. Click **Open**.
    - The model for this exercise is shown in Figure 68.

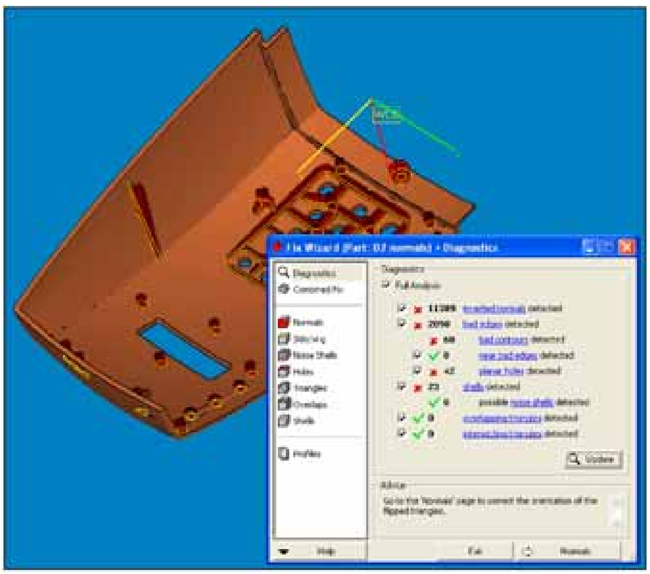


Figure 68: Model to fix - 02 normals.stl

To fix automatically the model

1. If the **Fix Wizard** does not launch automatically, select **Tools ➡ Fix Wizard (Ctrl+F).**
2. Follow the wizard.
    - Notice that in just one step all errors are solved.

Exercise 3: Stitch

This part is similar to the one above but targets different errors.

To open the model

1. Click **File** ➡ **Load Part**.
2. **Browse…** and navigate to the **My MPI 6.0 Projects\MPI_Fundamentals\Magics_STLs** folder.
3. Select the file **03 stitch.stl**.
4. Click **Open**.
   - The model for this exercise is shown Figure 69.

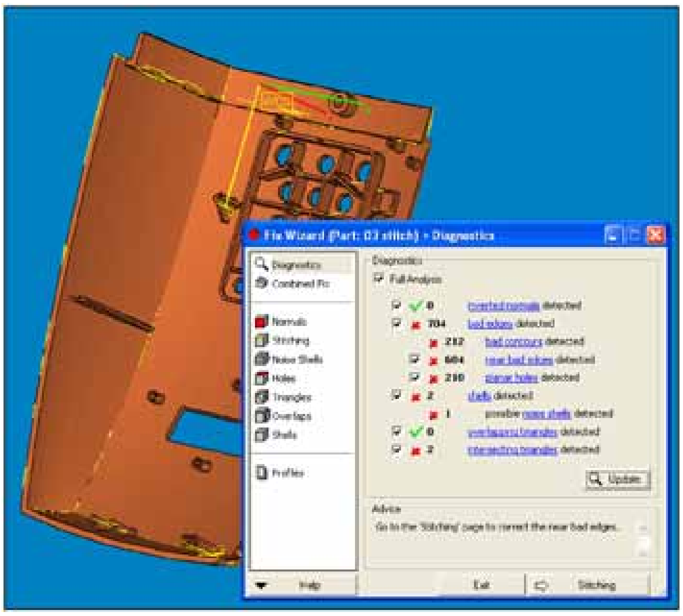


Figure 69: Model to fix – 03 stitch.stl

To fix the model

1. If the **Fix Wizard** does not launch automatically, select **Tools** ➡ **Fix Wizard (Ctrl+F).**
2. Click **Update**.
3. This time the wizard guides you to the stitching page, where you will need to run (Apply) the **Automatic fix twice** to correct all the remaining bad edges.

> As stitching redraws existing triangles, only small gaps (hence the term “near bad edges”) are corrected.

4. Click **Diagnostics** to return back to the diagnostics page.

5. Click **Update** to recalculate the list of issues.
   - Based on the revised list of issues, the program will provide Advice to target the problems.
6. Repeat the above steps to fix **Noise Shells**.
7. Go to the **Diagnostics** page and check on the **Full Analysis** option.
8. Click **Update**.

Exercise 4: Stitch + Normals

In this exercise you will used the “Combined Fix” page which makes the fixing process easier.

To open the model

1. Click **File ➔ Load Part.**
2. **Browse…** and navigate to the **My MPI 6.0 Projects\MPI_Fundamentals\Magics_STLs** folder.
3. Select the file **04 stitch + normals.stl.**
4. Click **Open**.
   - The model for this exercise is shown Figure 70.

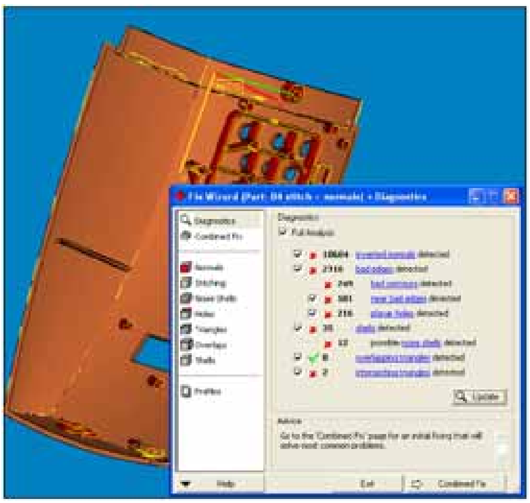


Figure 70: Model to fix – 04 stitch + normals.stl

Does the part look familiar? Yes, it is the same part, all errors together. But the fixing becomes even easier.

To fix the model

1. If the **Fix Wizard** does not launch automatically, select **Tools ➔ Fix Wizard (Ctrl+F).**
2. Click **Update**.
3. Select **Combined Fix** on the left pane.
4. Click **Apply**.
5. Go to the **Diagnostics** page and check on the **Full Analysis** option.
6. Click **Update**.

The “Combined Fix” page takes care of all errors at the same time. In almost all cases, the first thing you will do is a “Combined Fix”. Remaining errors often need manual correction.

Exercise 5: Holes

For this part, the automatic "apply" will fail to fill some holes correctly. You will notice that the depending on the complexity of the holes geometry it may be necessary to fix the holes manually instead of using the automatic tools.

To open the model

1. Click **File ➡ Load Part**.
2. **Browse…** and navigate to the **My MPI 6.0 Projects\MPI_Fundamentals\Magics_STLs** folder.
3. Select the file **05 holes.stl**.
4. Click **Open**.
   - The model for this exercise is shown Figure 71.

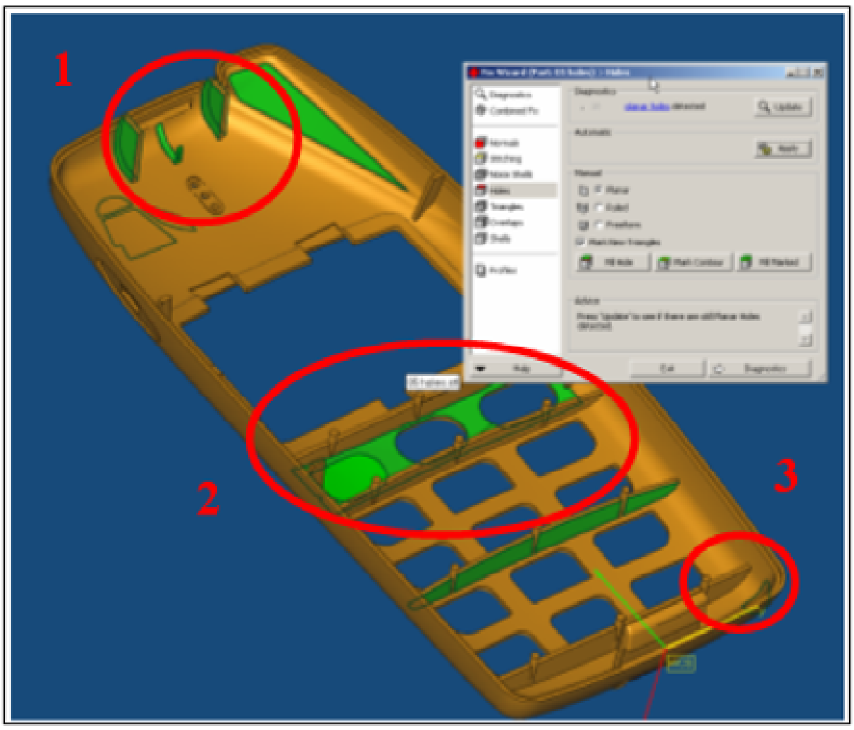


Figure 71: Model to fix - 05holes.stl

Simple planar holes existing out of one contour pose no problem. If you ran the automatic fixing, press the "Undo" icon in the main toolbar and proceed to fill holes 1, 2 and 3 manually.

To fix manually hole 1

The section of the model indicated as **Hole 1** needs a "*Ruled Hole Fill*".

1. Select **Ruled** from the Manual section.
2. **Select** the **X** direction for creating triangles.

3. Press **Fill Hole** button.

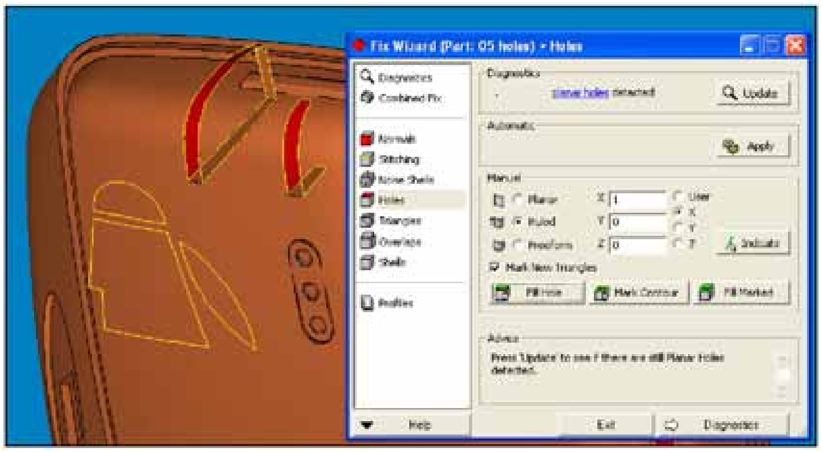


Figure 72: Close view of problem area 1

4. Click the Left Mouse Button (LMB) near the hole on the model.
5. Continue selecting the red surfaces (or the yellow edges) until the red or yellow sections have disappeared as indication that the holes have been filled.

You may click Update in the Diagnostics sections to check on the count down of holes pending to be fixed.

To fix manually hole 2

The section of the model indicated as **Hole 2** needs a "*Planar Hole Fill*". The difficulty here is that it exists out of **4 contours**.

1. Select **Planar** from the Manual section.
2. Select **Mark Contour**.
3. Select all 4 contours (they should all be marked in green).
4. Press **Fill Marked** to create the triangles in the gap.

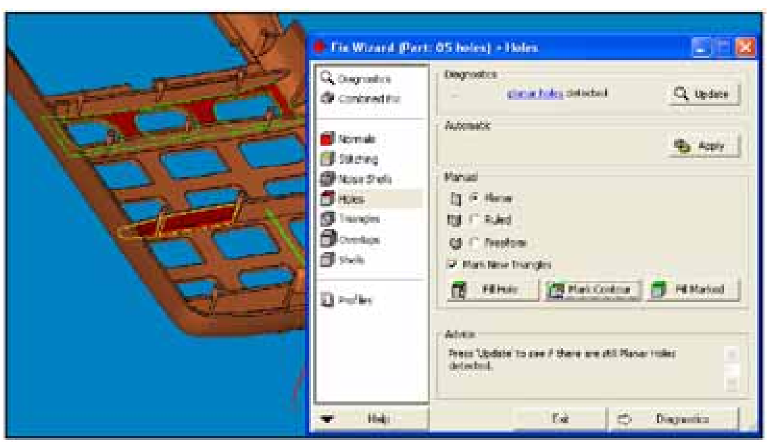


Figure 73: Close view of problem area 2

5. Press **Fill Hole** button.
6. Click the Left Mouse Button (LMB) near the hole on the model.
   - If the program displays the message “Magics detected another Contour that could belong to the same Hole. Use the extra contour? Click **Yes**.

To fix manually hole 3

The section of the model indicated as **Hole 3** needs a “*Freeform Hole Fill*” which is tangential to the existing surfaces.

1. Select **Freeform** from the Manual section.
2. Check on the box for **Tangent.**
3. The best result is obtained when choosing a grid size close to the triangles of the surrounding surface (0.3 mm). Enter **0.3** mm for the grid size.
4. Click **Fill Hole**.

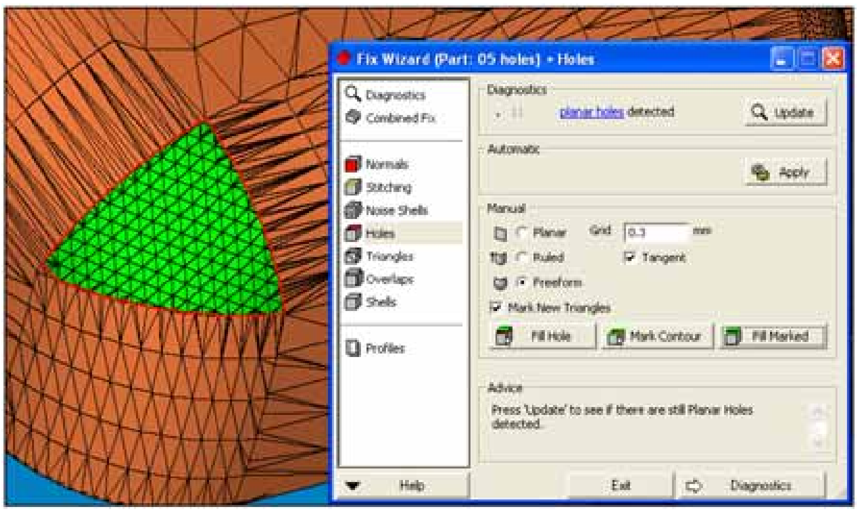


Figure 74: Close view of problem area 3

5. Click on the yellow edge.
6. Click on the red edge.
   - Notice that the model is now all gray in the corner as an indication that the holes have been closed.
7. Once they are correctly filled, use the Automatic “Apply” to correct the rest of the holes.
8. Click **Diagnostics** to return back to the diagnostics page.
9. Check on the **Full Analysis** option.
10. Click **Update**.
    - Based on the revised list of issues, the program will provide Advice to target the problems.

11. Follow the suggested steps until you completely fixed the model.

- Go to the Triangles page and use the automatic tool for fixing the remaining problems.

Exercise 6: Shells

In this exercise you will learn how to unite disconnected objects in your model.

To open the model

1. Click **File ➡ Load Part**.
2. **Browse…** and navigate to the **My MPI 6.0 Projects\MPI_Fundamentals\Magics_STLs** folder.
3. Select the file **06 shells.stl**.
4. Click **Open**.
   - The model for this exercise is shown Figure 75.

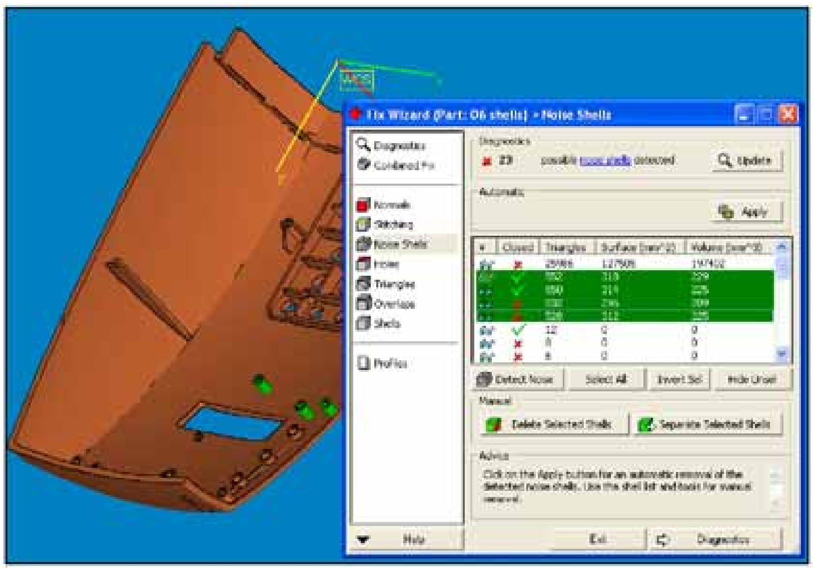


Figure 75: Model to fix - 06 shells.stl

To identify the problems

1. If the **Fix Wizard** does not launch automatically, select **Tools ➡ Fix Wizard (Ctrl+F).**
2. Click **Update**.
   - The advice prompts you to first delete the noise shells.

As you have noticed by now, noise shells are nothing more then a small collection of triangles not connected to the main body & not representing any geometry. They can be safely deleted.

However, there are five shells that are not noise shells. Four of these shells represent small pins which were designed as separate objects and not united to the main body.

In case you do not need these small features for your simulation, you can mark them via the shells list and hit “Delete Selected shells”

To fix the separated objects and the model

1. Click on **Noise Shells**.
2. Expand the window to view the list better.

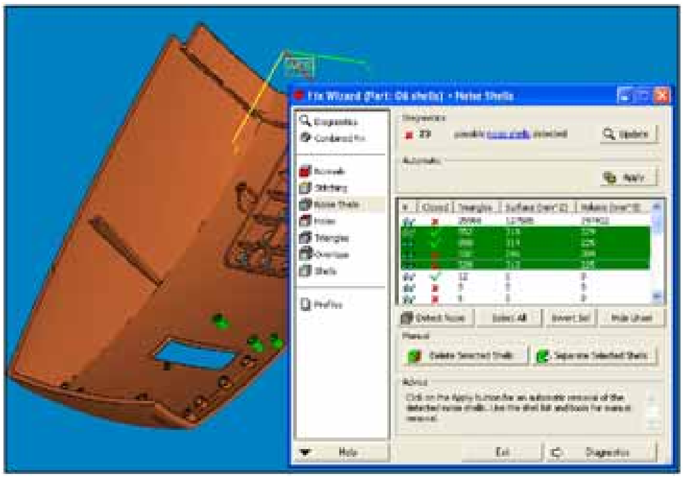


Figure 76: Noise Shells page

3. Holding the **Ctrl** key down select from the list the triangles with the IDs **552, 550, 532** & **528**.
   - The program will highlight the items on the model.

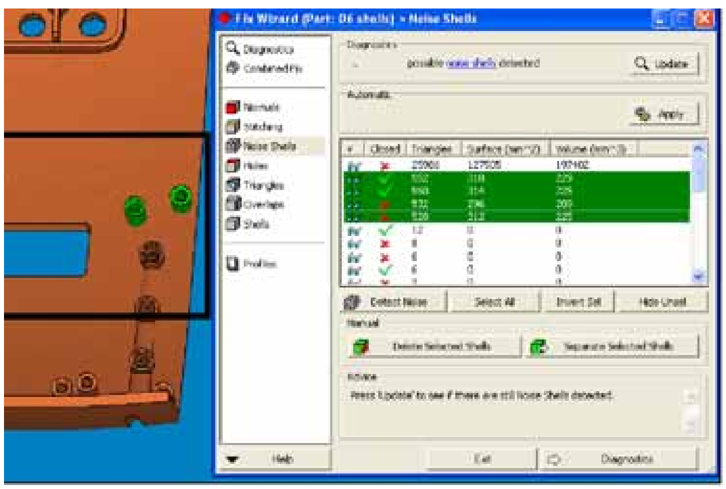


Figure 77: Separated items on the model

4. Select **Shells** on the left pane to go to the Shells page.
5. Check that the items are still highlighted. Click **Unify**.
   - The program will merge the four small shells to the main body.
6. Click **Yes** to the message on the screen.
7. Go back to the **Diagnostics** page, run a **Full Analysis** and follow the recommendations to finish fixing the model.

Exercise 7: Overlaps

In this exercise you will use the tools to fix overlapping elements on the model.

To open the model

1. Click **File ➡ Load Part.**
2. **Browse…** and navigate to the **My MPI 6.0 Projects\MPI_Fundamentals\Magics_STLs** folder.
3. Select the file **07 overlaps.stl**.
4. Click **Open**.
   - The model for this exercise is shown Figure 78.

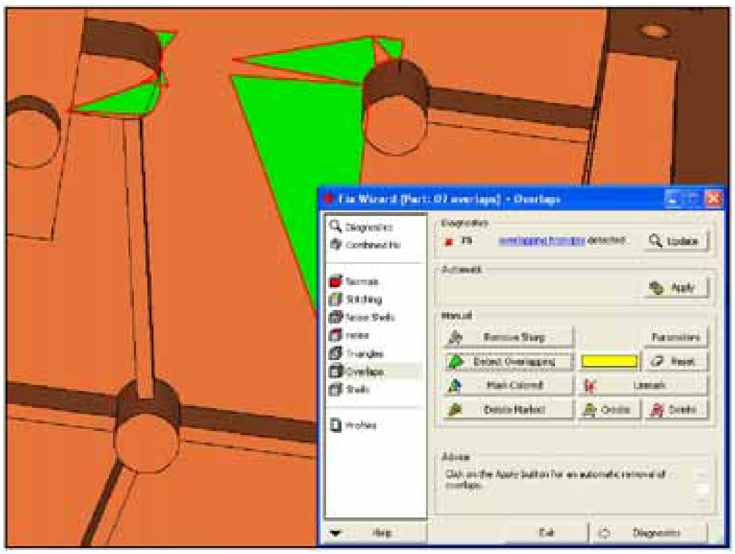


Figure 78: Model to fix - 07 overlaps.stl

To identify and fix the problems

1. If the **Fix Wizard** does not launch automatically, select **Tools ➡ Fix Wizard (Ctrl+F).**

A **Full Analysis** will reveal that there are double surfaces & intersecting triangles.

2. Use the wizard to fix automatically the problems.
3. In some cases manual work will be needed, the way to proceed then is:
4. Go to the Overlaps page.
5. Analyze the number of overlapping triangles.
6. Mark them by clicking on **Detect Overlapping.**
7. Delete triangles where needed (note that a hole may be created by deleting triangles).
8. Use the **Wizard** to correct the holes & Near bad edges.

The manual correction of intersecting triangles follows the same philosophy.

Exercise 8: Summary

In this exercise you will practice all what you have learned up to now.

To open the model

1. Click **File ➡ Load Part.**
2. **Browse…** and navigate to the **My MPI 6.0 Projects\MPI_Fundamentals\Magics_STLs** folder.
3. Select the file **08 Fixing_test.stl.**
4. Click **Open**.
    - The model for this exercise is shown Figure 79.

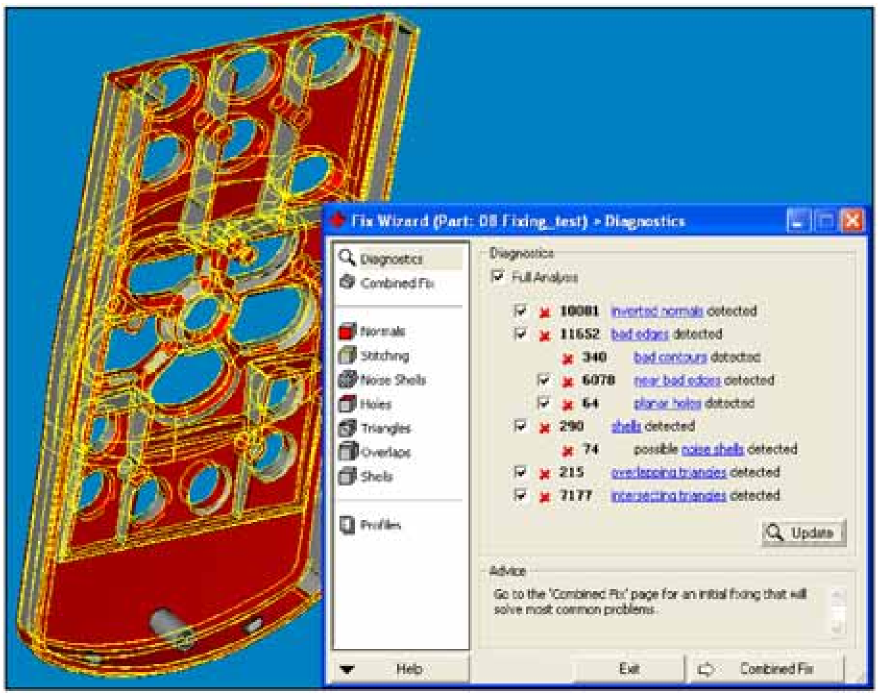


Figure 79: Model to fix -08 Fixing_test.stl

To fix the model

1. Apply all that you have learned so far to fix this model.

> You might need to delete overlaps and fix the holes left because of the overlaps removal.

Exercise 9: Triangles

In this exercise you will learn how to use Moldflow Magics STL Expert to reduce the size of the file which will reduce the analysis time in our simulation products.

To open the model

1. Click **File ➨ Load Part.**
2. **Browse…** and navigate to the **My MPI 6.0 Projects\MPI_Fundamentals\Magics_STLs** folder.
3. Select the file **09 ChildCarSeat.stl**.
4. Click **Open**.
   - The model for this exercise is shown Figure 80.

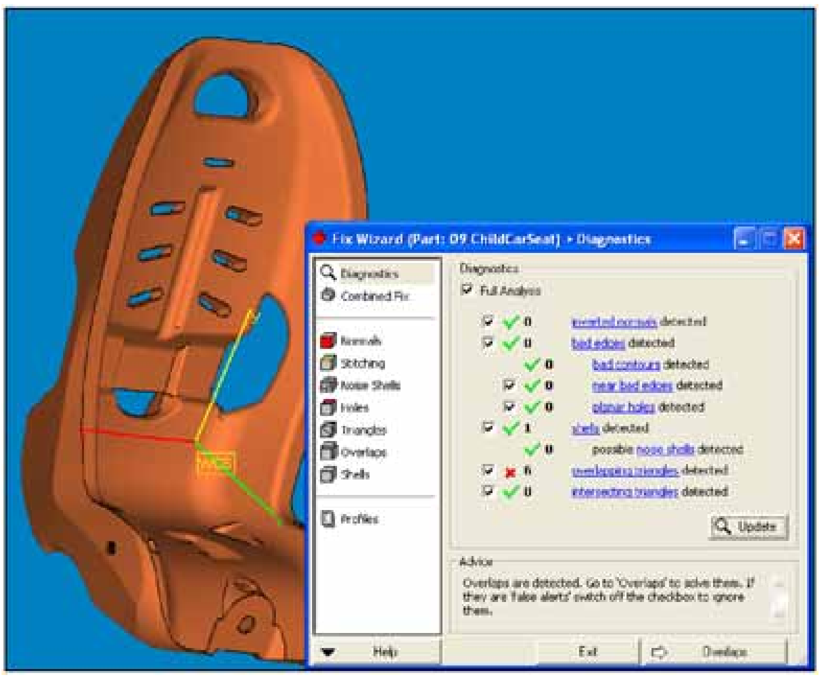


Figure 80: Model to fix - 09 ChildCarSeat.stl

To check model properties

1. Press the “i” icon to display the properties of the model.
   - The program will display the information shown in Figure 81.

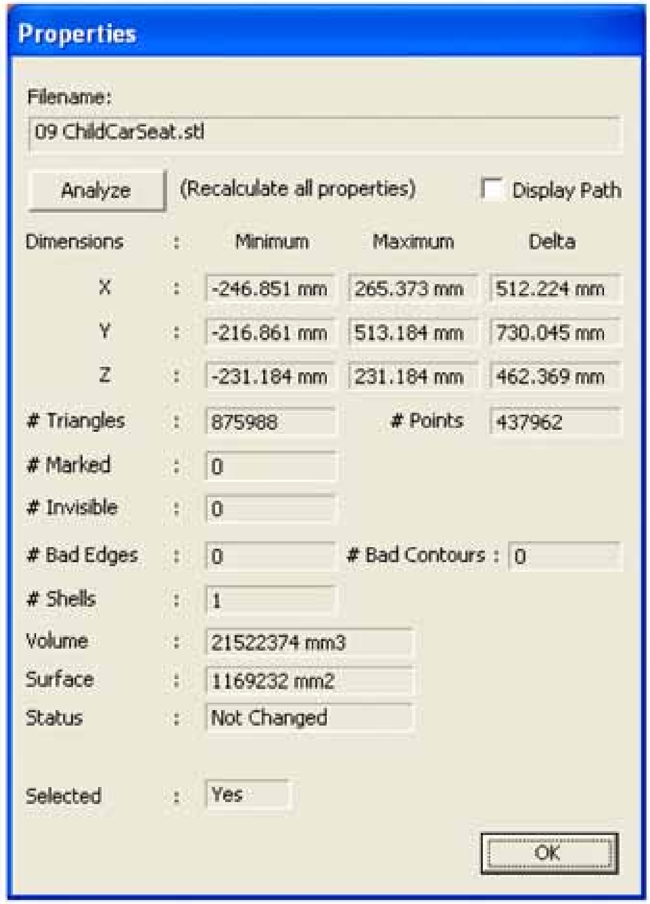


Figure 81: Model properties dialog

2. Review the number of triangles listed in the section below:

The model contains a large number of triangles, the following tasks will step you thorough the process of decreasing this value.

To reduce the number of triangle

1. Select **Tools** ➡ **Triangle reduction (Ctrl + T)**.

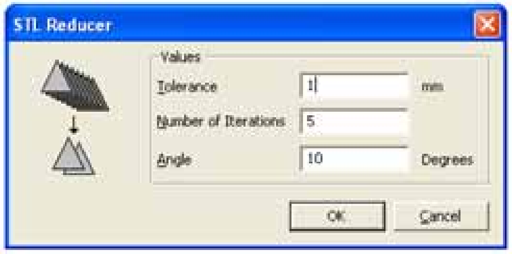
2. Specify the following:

2.1. Tolerance **1** mm.

- ➤ You can increase the tolerance from 0.1 (default) to 1 since it is a large model with no little features.

2.2. Number of iterations **5**.

2.3. Angle **10** degrees.



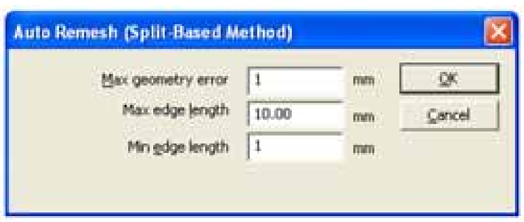
3. Click **OK**.
4. Check the properties of the model again.

5. The triangle count was reduced from **875988** to **110296** triangles.

It is recommended to perform the Triangle Reduction **BEFORE** running the Fixing Wizard.

To remesh the model

1. Select **Tools** ➡ **Remesh.**
2. Specify the following:
   2.1. Max geometry error **1** mm.
   2.2. Max. edge length **10.00** mm.
   2.3. Min. edge length **1** mm.



3. Click **OK**.
4. Check the properties of the model again.

5. The triangle count was reduced further; it went from **110296** to **64110** triangles.

Since these are tasks which require lengthy calculations, the model result is provided for you to review. The file name is **ChildCarSeat_result.stl.**

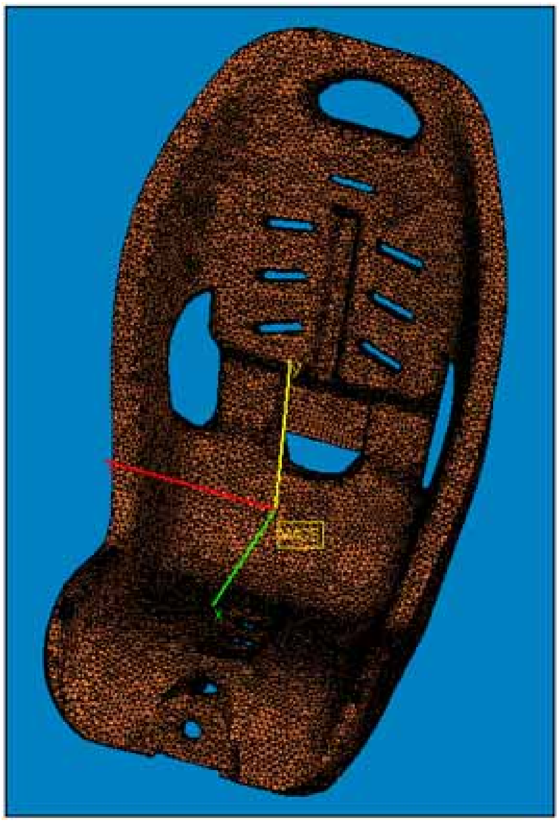


Figure 82: Child Car Seat with number of triangles reduced

Exercise 10: Importing an IGES file

In this exercise you will import and IGES file into Moldflow Magics STL Expert and learn about the different options available for basic fixing operations.

To import the model

1. Click **File** ➡ **Import** ➡ **IGES...**
2. **Browse…** and navigate to the **My MPI 6.0 Projects\MPI_Fundamentals\Magics_STLs** folder.
3. Select the file **Q-base.igs**.
4. Click **Open**.
   - When importing an *.IGES file, the accuracy of conversion (the triangle count) can be set by the user. Additionally some basic fixing operations (normals & stitching) can be performed directly.
5. Specify the following:

   5.1. Accuracy **0.1** mm.

   5.2. Check the boxes for **Visible only** and **Stitch**.

   5.3. Tolerance **0.1** mm.

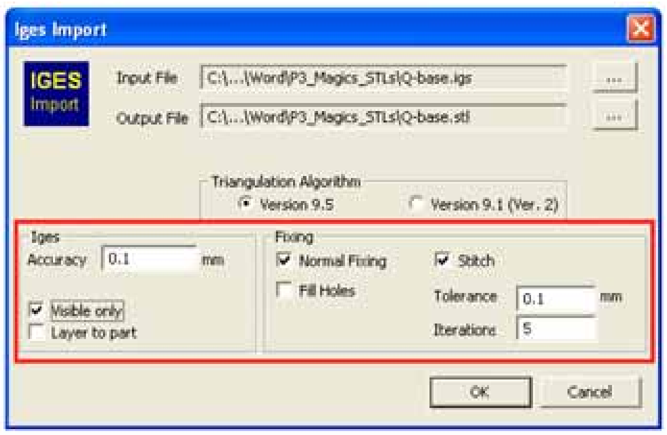


Figure 83: IGES import dialog

6. Click **OK**.

Competency check - Introduction to Moldflow Magics STL Expert

1. **What does** Moldflow Magics STL Expert help you with?
2. **What are the default model format files you can import into Moldflow Magics STL Expert?**
3. **What other file formats could you import using MDL?**
4. **What are the file formats exported by Moldflow Magics STL Expert?**

Evaluation Sheet - Introduction to Moldflow Magics STL Expert

1. What does Moldflow Magics STL Expert help you with? • Viewing. • Measuring. • Correcting. • Optimizing Stereolithography (STL) and solid surface models.
2. What are the default model format files you can import into Moldflow Magics STL Expert? • STL. • IGES. • MGX (Materialise compressed STL format).
3. What other file formats could you import using MDL? • Parasolid. • SolidWorks. • Pro/ENGINEER. • CATIA V5. • STEP.
4. What are the file formats exported by Moldflow Magics STL Expert? Corrected and optimized STL models can be output in Moldflow's proprietary UDM (Unified Data Model) format, which can then be imported into MPA and MPI.

CHAPTER 12

Material Searching and Comparing

Aim

The aim is to learn how to search for and compare different thermoplastic materials in the material database.

Why do it

To run an analysis in MPI, you need to select a material. The choice of material may be predetermined, or a material can be selected based on the results of a database search. To help in the selection of the material, one material can be compared to one or more materials to see what the variation between the materials will be.

Overview

There are several ways a material can be selected for an analysis, depending on the available information. If the manufacturer and trade name are known, you can simply select it from a list. If the material to be used is not known, a material search can be performed to find a suitable material. Having selected a material, you can compare it to other materials, or view a report on the quality of the material data.

Note the answers are based on MPI 6.0 build 05495. If a different build of software is used, the answers may be slightly different.

Practice - Material Searching and Comparing

Below are several examples of using the material selection and searching capabilities of MPI.

Setup

To open a project

1. Click icon (**File ➨ Open Project**), and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Material_Searching**.
2. Double click the project file **Material_Searching.mpi**.
3. Click **File ➨ Preferences** and ensure that **System Units** are set to **Metric**.

To open the cover model

1. Double click the **Cover** study in the Project View pane.
2. Rotate the model to review the geometry.

Find a material with a known manufacturer and trade name

How you check to see if a material is on the database depends on the information you are given about the material. If the material's manufacturer and trade name are known, the best place to find the material is to scroll through the Manufacturer and Trade name drop-down menus on the Select Material dialog, shown in Figure 84.

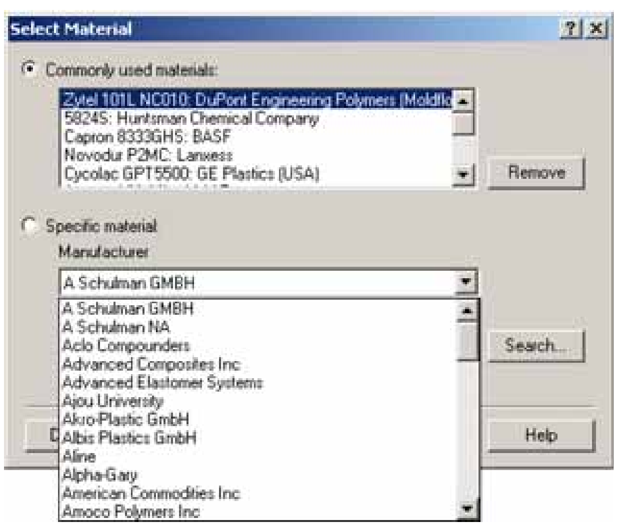


Figure 84: Select material dialog

Navigating through list is easily done by:

- The scroll bar.
- Typing the first few letters of the name you are looking for.
- The arrow keys.

> The **tab** key can be used to navigate between the **Manufacturer** and **Trade Name** drop-down menus.

To find the material Eastman Chemical Tenite LDPE 1870.

1. Double-click (**Material** icon) in the Study Tasks pane.
2. Click the **Manufacturer** drop-down menu.
3. Scroll through the list until you find **Eastman Chemical Products**.
   - As an alternative, type **east**.
4. Click the **Trade name** drop-down menu.
5. Scroll through the list until you find **Tenite LDPE 1870,** or type **ten**.

   5.1. Use the arrow keys to find Tenite LDPE 1870.

   - Results should look like Figure 85.

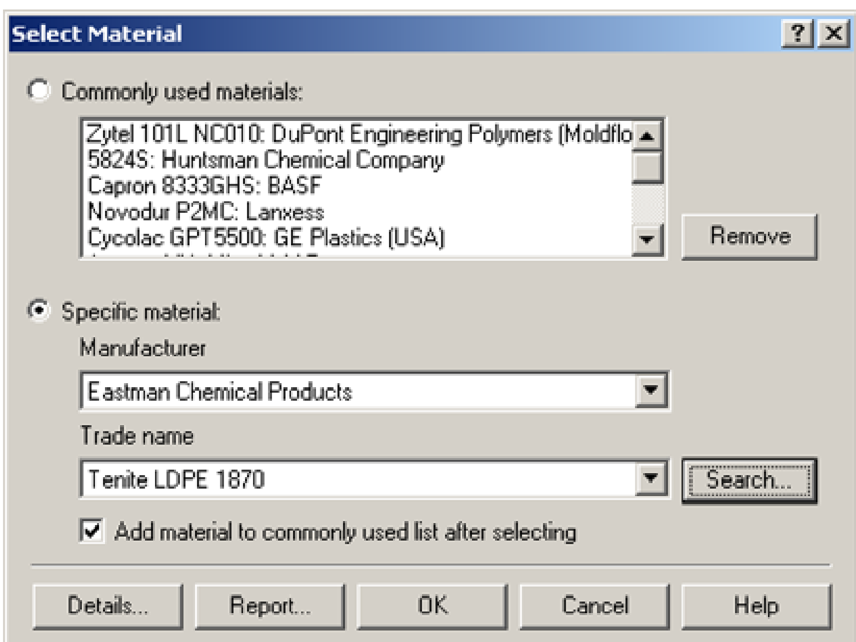


Figure 85: Find Tenite LDPE 1870

Using search

Sometimes the manufacturer of the material is not known, but the trade name is. The **Search Criteria** dialog is useful for finding materials when the manufacturer and/or the trade name are not known.

The **Search Criteria** dialog shows a default list of search fields each time it is opened. Other fields can be added at any time. Once you have set up a search you like, you can save it so you can retrieve it quickly.

To find the material Lexan 141

1. Click the **Search** button on the **Select Material** dialog.
2. Select **Trade name** in the Search Fields list.
3. In the **Substring** text box on the right, enter **lexan 141**, as shown in Figure 86.
   - Ensure there is 1 space between lexan and 141.
4. Click the diskette icon.

   4.1. Enter the name **Lexan**.

   ➤ This will save the search criteria for later use.

   4.2. Click the **Save** button.
5. Click the **Search** button.

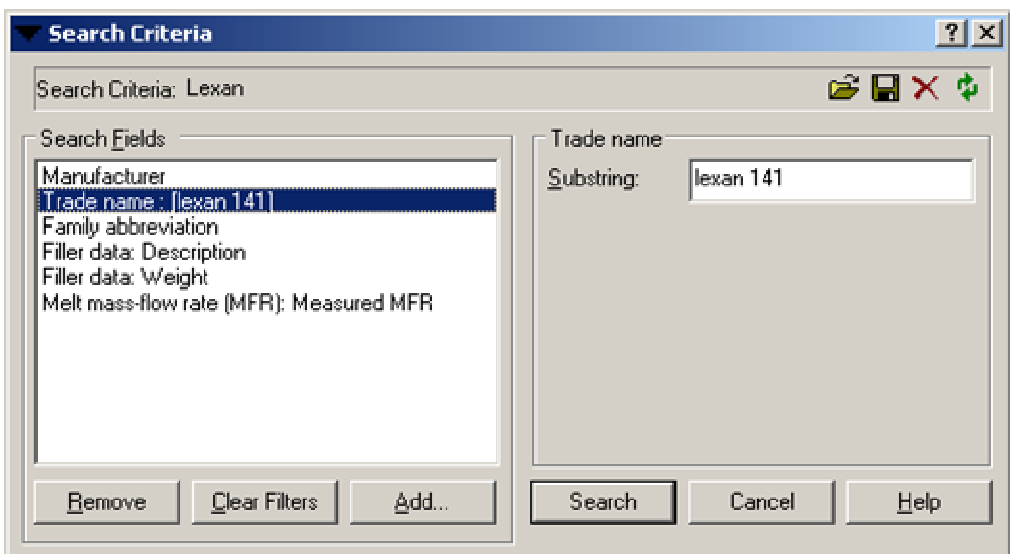


Figure 86: Search Criteria dialog

A new dialog called **Select Thermoplastics material** will appear with the search results, as shown in Figure 87. The list shows there are two grades of Lexan 141, one is from a European division, the other from a US division of the same company. The other 4 materials have Lexan 141 as a base and have been further modified.

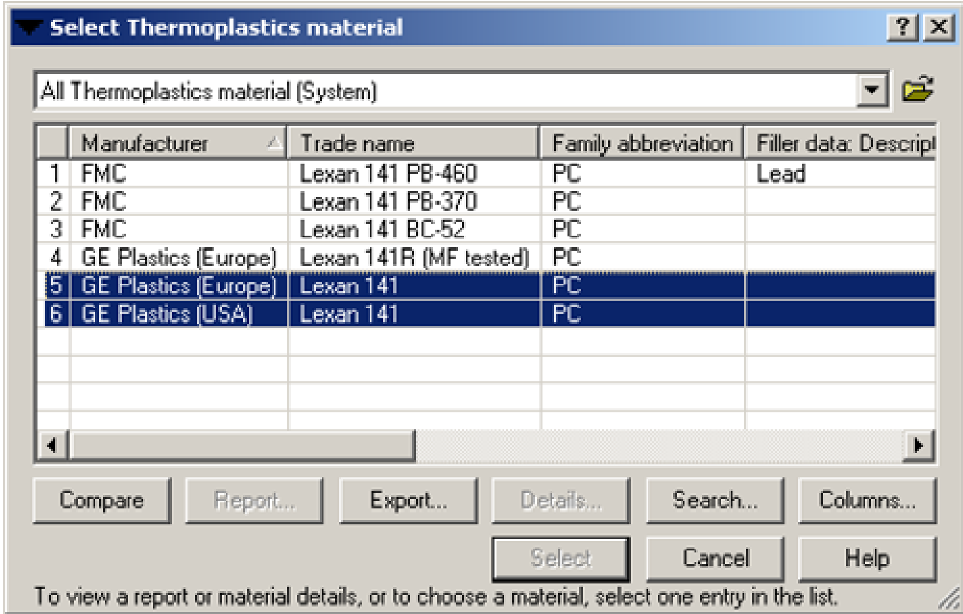


Figure 87: Lexan search results

To compare the two Lexan 141 materials

1. Select one of the Lexan 141 grades, then hold down the Ctrl key and click on a second grade.

- Both grades should now be highlighted.

2. Click **Compare**.

3. Scroll through the report to view the differences between the materials.

4. Click  (Rheology graph icons) to see the data compared graphically.

4.1. Click  (Plot properties icon).

4.2. Remove the middle two temperatures from the list.

➤ This will reduce the clutter on the graph.

4.3. Click **OK**.

Find a similar material

To find similar materials in the MPI material database, use specific search criteria to narrow your search. You can use the following popular search criteria including;

- Manufacturer.
- Trade name.
- Family abbreviation.
- Filler data: Description.
- Filler data: Weight.
- Moldflow viscosity index.
- Data source.
- Any property stored in the database.

Once a search has been performed, you can:

- Sort a column by clicking on the column heading in the **Select Thermoplastics material** dialog.
- Select a material and click **Details** to view the properties of that material.
- Plot viscosity or PVT curves for the material after selecting **Details**.
- Compare multiple materials, as previously described.

All this information can be used to find and compare similar materials. By setting good search criteria, you can narrow down the list of materials before using the compare command.

There are several reasons why you might search for similar materials in the database:

- To find a suitable substitute material for a material that is not on the database.
- To find the best choice of material, amongst similar grades, with which to mold the part.
- To see the effect different materials have on the part's wall thickness.

Find all polycarbonates with 20% glass filler

You will find the material using a stepwise procedure. This is a common way to search for a material. Start with one search field and add criteria one at a time.

To find all PC's

1. Click **Search** on the **Select Material** dialog.
2. Click **Clear Filters**.
3. Select the criterion **Family abbreviation** and enter **pc** in the substring text box.
4. Click **Search**.
5. Scroll down to the bottom of the **Select Thermoplastic material** dialog.
    - Note that the search results contain about 1000 materials.

6. Look in the **Family Abbreviation** column.
   - Many of the materials listed are PC blends.
7. Click the **Family Abbreviation** column heading.
   - This will sort the column.
8. Scroll up and down the list.
   - Note that all **PC's** are now listed together and that there are about 640 of them.

The search will look for the substring you entered in the search field. You can't limit the search to an exact string.

The **Select Thermoplastics material** dialog can be resized so more information can be seen at one time.

To add glass to the search

1. Click **Search** on the **Select Thermoplastics material** dialog.
2. Select the criterion **Filler data: Description**

   2.1. Enter **glass** in the Substring text box.
3. Click **Search**.
4. Scroll down to the bottom of the **Select Thermoplastic Material** dialog to determine the number of materials found.
   - Note that the search results now contain 176 materials.
5. Look in the **Family Abbreviation** column.
   - Many of the materials listed are PC blends.
6. Click the **Family Abbreviation** column heading to sort by this column.
7. Scroll up and down the list.
   - Note that all **PC's** are now listed together and that there are 137 of them.
8. Click the **Filler Data: Weight** column heading.
   - Note the range of glass filler is from about 5% to 50%. There are several 20% glass materials.

: To add weight to the search

1. Click **Search** to return to the **Search Criteria** dialog.
2. Select the criterion **Filler data: Weight** and enter **20** in the Minimum and Maximum text boxes.
3. Click **Search**.
4. Scroll down to the bottom of the **Select Thermoplastic material** dialog.
   - Note that the search results now contain 45 materials.
5. Look in the **Family Abbreviation** column.
   - Some of the materials listed are PC blends.

6. Click the **Family Abbreviation** column heading.
   - This will sort the column.
7. Scroll up and down the list.
   - Note that all **PC's** are now listed together and that there are 37 of them.

All other fields can be searched in the same way. A combination of using different search fields and sorting the columns can be used to narrow down the search results.

Using Moldflow Viscosity Index

Many times the viscosity index is used in conjunction with the search criteria family abbreviation to separate materials that are blended. Often, the viscosity index of blends is based on a different temperature so you can use this information to identify blends.

To find all ABS's

1. Click **Search** on the **Select Material** dialog.
2. Click **Clear Filters**.
3. Select the criterion **Family abbreviation**, enter **abs** in the substring text box.
4. Click **Search**.
5. Scroll down to the bottom of the **Select Thermoplastic material** dialog.
   - Note that the search results contain about 700 materials.
6. Click the **Family Abbreviation** column heading to sort the ABS from the blends.
7. Scroll up and down the list.
   - Note that all **ABS'** are now listed together and that there are about 480 of them.

Add viscosity index to the search

1. Click **Search** to return to the **Search Criteria** dialog.
2. Click **Add** to open the **Add Search Fields** dialog.

   2.1. Find and select **Moldflow Viscosity Index** in the list of search fields.

   2.2. Click **Add** to add it to the Search Fields list.
3. Highlight the **Moldflow Viscosity Index** field, enter **240**.
4. Click **Search**.
5. Scroll down to the bottom of the **Select Thermoplastic material** dialog.
   - Note that the search results now contain about 490 materials. Most of the blends were removed from the search.
6. Look in the **Family Abbreviation** column.
   - Some of the materials listed are ABS blends.
7. Click the **Family Abbreviation** column heading.
8. Scroll up and down the list.
   - Note that all **ABS'** are now listed together and that there are about 480 of them.

9. Click and drag the **Moldflow Viscosity Index** column to the right of Family abbreviation.
   - The column will stay in that position for the remainder of the searches.
10. Click the **Moldflow Viscosity Index** column heading.
   - This will sort the Viscosity Index values. The higher the number, the more viscous the material. This is a handy way to do a quick comparison on the viscosity between materials.
   - You could further refine the viscosity index search by putting in a longer string for the viscosity index such as **(240)01**. This would search for all viscosity indexes in the range 100 - 199.

Plotting viscosity

Once you have filtered out a suitable list of materials, you can get further information about a material by highlighting it and clicking **Details**. This allows you to view all the properties stored for the material.

To plot the viscosity curve for a material

1. Select a material on the **Select Thermoplastic material** dialog.
2. Click **Details**.
3. Click the **Rheological Properties** tab.
4. Click **Plot Viscosity,** as shown in Figure 88.
   - Once the viscosity plot is created, there are several buttons in the bottom left corner of the plot window that you can use.
   - The  (query icon) shows the shear rate and viscosity values as you click on different locations between the viscosity curves.
   - The  (plot properties icon) allows you to scale the X and Y-axes and to specify the temperatures to be plotted.
   - The  (save icon) allows you to save the plot data to a tab-delimited text file.
   - The  (save image icon) lets you save the plot to an image file which you can later incorporate into a report.
   - The  (print icon) opens the Print dialog from where you can send the plot to a selected printer.

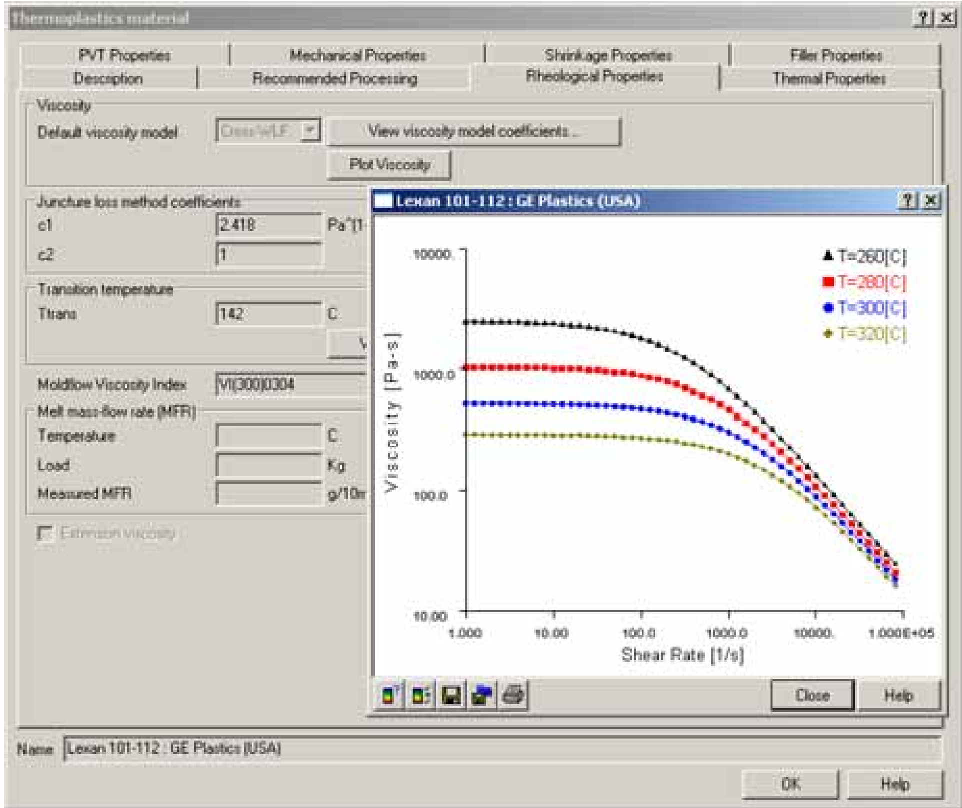


Figure 88: Material details

Material Data Method Report

On the **Select Material** and **Select Thermoplastic material** (search results) dialog, there is a **Report** button. This button can be used to show a report for the selected material. This will highlight the quality of the data and give advice for its use, as shown in Figure 89.

```
Material Data Method Report:

Manufacturer              GE Plastics (USA)
Trade Name                Lexan 101
Family Abbreviation       PC
Fibers / Fillers          Unfilled
Moldflow Material ID      52234
Moldflow Grade Code       GE1005

The material was tested by the manufacturer. Data was last updated on
AUG-12-2005.
This data is Non-Confidential.


SUMMARY:
Data Type             Date                Method
-------------------------------------------------------------------
Rheology              29-JUN-01           Unknown
Thermal               01-JUN-90           Unknown
Specific Heat         01-JUN-90           Unknown
pvT                   Unknown             Supplemental
Shrinkage             10-NOV-2005         Uncorrected Residual Stress


RHEOLOGY:
The material's rheological behavior was tested by the manufacturer by
an unspecified test method. Data was last updated on 29-JUN-01.

THERMAL:
The material's thermal conductivity was tested by the manufacturer by
an unspecified method. Data was last updated on 01-JUN-90. Thermal
conductivity was established at a single temperature.

SPECIFIC HEAT:
The material's specific heat was tested by the manufacturer by an
unspecified method. Data was last updated on 01-JUN-90. Specific heat
was established at a single temperature.

pvT:
The material's pvT data was not measured.
Supplemental pvT data is provided to allow for simulations. However,
supplemental pvT data can show specific volume variations from
specific grade pvT data by more than 25%.
More reliable simulation results may be possible by obtaining
measured pvT data on this material.

SHRINKAGE (MIDPLANE & FUSION MODELS ONLY):
For this material, shrinkage and warpage predictions will be based on
an uncorrected residual stress model with generic estimates of
material mechanical properties. Data was last updated on 10-NOV-2005.
You can achieve more accurate shrinkage predictions by using measured
shrinkage data and Moldflow's CRIMS model.


DISCLAIMER:
The information contained in this report has been prepared by
Moldflow Corporation based on data and other information received
from third parties. No representations or warranties are provided
regarding this report or its conclusions and Moldflow Corporation
specifically disclaims any liability that may result from reliance on
these results.

Copyright 2005 Moldflow Corporation
```

Figure 89: Material data method report

Competency Check - Material Searching and Comparing

Answer the following questions in the space provided, using the material searching functions in MPI to obtain your answers. Each question part builds on the previous part.

1. Find the information for Magnum 3504

- Manufacturer ____________________
- Trade name ____________________
- Who tested the viscosity? ____________________

2. How many HDPE grades are there on the database? ____________________

- How many HDPE grades are from BP Chemicals? ____________________
- What is the highest Viscosity index (VI) for a HDPE from BP ____________________
- What is the trade name of the highest VI HDPE from BP ____________________

3. What is the total number of PA66 grades on the database with a VI of VI(290), excluding blends? ____________________

- What is the number of glass fiber filled PA66 grades with a VI of VI(290), excluding blends? ____________________
- What is the highest glass content of the PA66 grades with a VI of VI(290), excluding blends? ____________________
- What is the trade name of the lowest Viscosity index Zytel that is 33% glass filled? ____________________

4. What is the total number of PBT grades on the database? ____________________

- What is the number of 30% glass filled PBT grades on the database? ____________________
- What is the trade name for the Ticona material with 30% glass and the lowest viscosity index? ____________________
- What is the viscosity index for the Ticona material? ____________________
- Who tested the PVT data for the Ticona material? ____________________

5. How many grades of PEEK are on the database? ____________________

- How many PEEK grades are from LNP? ____________________
- How many LNP PEEK grades have carbon fillers? ____________________
- Who tested the viscosity data for the carbon fiber LNP PEEK grades? ____________________

Evaluation Sheet - Material Searching and Comparing

1. Find the information for Magnum 3504	
• Manufacturer	Dow Chemical Europe
• Trade name	Magnum 3504
• Who tested the viscosity?	Other
2. How many HDPE grades are there on the database?	189
• How many HDPE grades are from BP Chemicals?	17
• What is the highest Viscosity index (VI) for a HDPE from BP	VI(230)0429
• What is the trade name of the highest VI HDPE from BP	Rigidex HM5420XP
3. What is the total number of PA66 grades on the database with a VI of VI(290), excluding blends?	475
• What is the number of glass fiber filled PA66 grades with a VI of VI(290), excluding blends?	253
• What is the highest glass content of the PA66 grades with a VI of VI(290), excluding blends?	60%
• What is the trade name of the lowest Viscosity index Zytel that is 33% glass filled?	Zytel FE5422 BK275
4. What is the total number of PBT grades on the database?	529
• What is the number of 30% glass filled PBT grades on the database?	69
• What is the trade name for the Ticona material with 30% glass and the lowest viscosity index?	Celanex JKX-1081
• What is the viscosity index for the Ticona material?	VI(250)0071
• Who tested the PVT data for the Ticona material?	Moldflow
5. How many grades of PEEK are on the database?	26
• How many PEEK grades are from LNP?	7
• How many LNP PEEK grades have carbon fillers?	3
• Who tested the viscosity data for the carbon fiber LNP PEEK grades?	Manufacturer

Note the answers are based on MPI 6.0 build 05495. If a different build of software is used, the answers may be slightly different.

CHAPTER 13

Gate Placement

Aim

The aim of this chapter is to review the design guidelines for placing gates and to run the gate location analysis for the cover model and other parts.

Why do it

Placing a gate correctly can be one of the most critical factors in determining the final quality of the part. The location of the gate may have many requirements and restrictions including part design, usage, aesthetics, and tool construction.

Overview

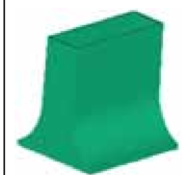
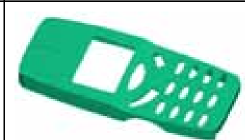
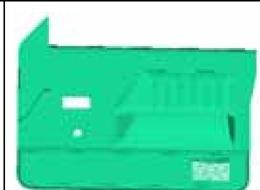
A review of design guidelines will be done to give a background for determining the gate location. The gate location analysis will be run on several different parts and the results will be compared to the requirements of the part. The gate location analysis can be done on Midplane and Fusion models. 3D flow simulation does not support this type of analysis.

Once a gate location is chosen, a fast flow analysis can be run to determine if the gate location is acceptable.

Practice - Gate Placement

This chapter has several models that are used for practice and are described below. At least the first one should be analyzed. Do the others as time permits.

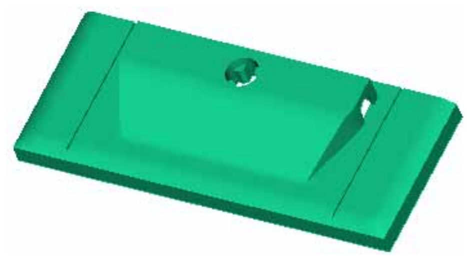
Table 9: Models used for gate placement

Description	Model
Cover: starts on page 221	
The cover is the same part that is used in the translation and cleanup chapter. It is given to you already meshed and cleaned up ready for analysis. This part will be produced in a two plate tool with a sub gate.	
Paper holder: starts on page 225	
The paper holder will be produced in a single cavity 2-plate tool with a cold runner.	
Phone cover: starts on page 229	
The phone has been designed using a 2-plate tool with a hot runner system. The hot drops can be directly on the part or feed a cold runner. The results of the analysis will determine the layout of the runner system.	
Door panel: starts on page 233	
The door panel has been designed using a 2-plate tool with hot runners feeding a cold runner system. The gates will be edge gates.	

Gate location analysis on the cover

In this section, you will determine the optimum gate location for the cover model. You will run a Fast Fill analysis with the gate location you have chosen for the cover.

Design criteria



The cover has been designed using a 2-plate tool with cold runners and tunnel gates. This will limit the gate locations to somewhere along the sidewall of the part.

Setup

To open a project

1. Click the file open icon or **File ➨ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Gate_Placement**.
2. Double click the project file **Gate_Placement.mpi**.
3. Click **File ➨ Preferences** and ensure that **System Units** are set to **Metric**.

To open the cover model

1. Double click the **Cover** study in the Project View pane.
2. Rotate the model to review the geometry.

Analysis

To set up the analysis

1. Change the analysis sequence to gate location by either:
   - Double click on the analysis sequence icon in the Study Tasks list, and select **Gate Location**.
   - Click **Analysis ➨ Set Analysis Sequence ➨ Gate Location**.
2. Double-click the icon selection material in the Study Tasks pane.
3. Select the manufacturer **BASF** and the trade name **Capron 8333GHS**.
4. Save the model.
5. Click the **Analyze Now** icon .
   - The analysis should only take about a minute to finish.

Results interpretation

To view gate location results

1. Click the **Best gate location** plot in the Study Tasks list.
2. Rotate the model to view all sides and then pick a good rotation where you can see the blue area(s), as shown in Figure 90.

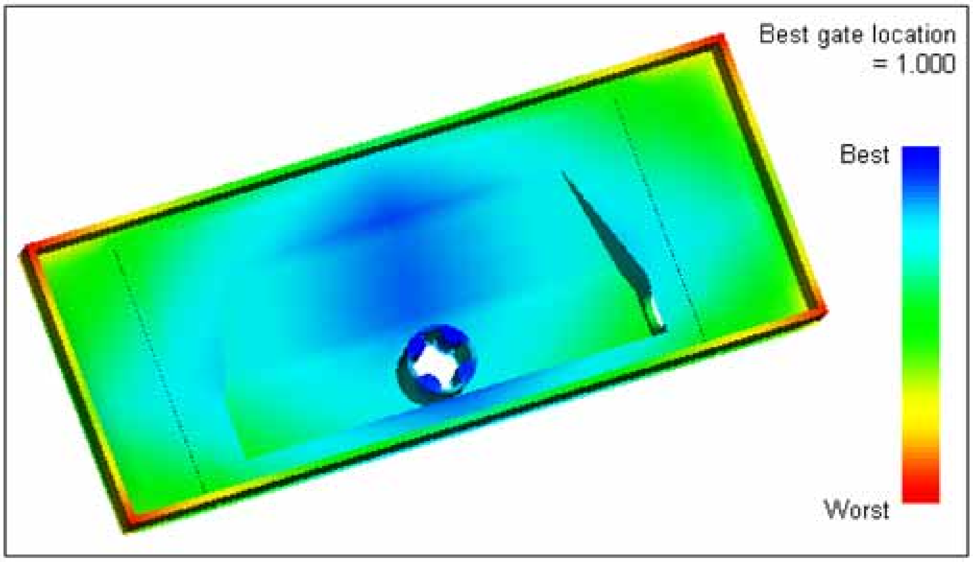


Figure 90: Best gate location result

In Figure 90, the properties of the results were set to have a smooth rather than a banded color transition. Sometimes banded is useful for highlighting the best area.

To set the banded color transition, highlight the **Best gate** location result, in the Study Tasks list, right-click, select **Properties**, go to the **Optional Settings** tab, select **Banded** and click **OK**.

Interpreting the results

A best gate location plot has a scale from about 0.1 (Worst) to 1.0 (Best), which corresponds with a color scale ranging from red to blue. The higher the number (blue), the better a location is for placing a gate, according to the gate location analysis. Notice the blue area of the plot is near the center of the part. This is due to the flow resistance criteria used in the analysis. The best area is on the heavy sections of the boss on the core side of the part. This is due to the thickness criteria. The gate location will favor heavy areas so parts can be packed out better.

Keeping in mind the design criteria, this best gate location will not work because it can't be reached with a cold runner and tunnel gate. Currently, there is no way the gate location analysis can take into account the design criteria of where you can or cannot place a gate. The closest gate location to the best gate location that will work is shown in Figure 91 and is the center of the long side. Either long side will have about the same results.

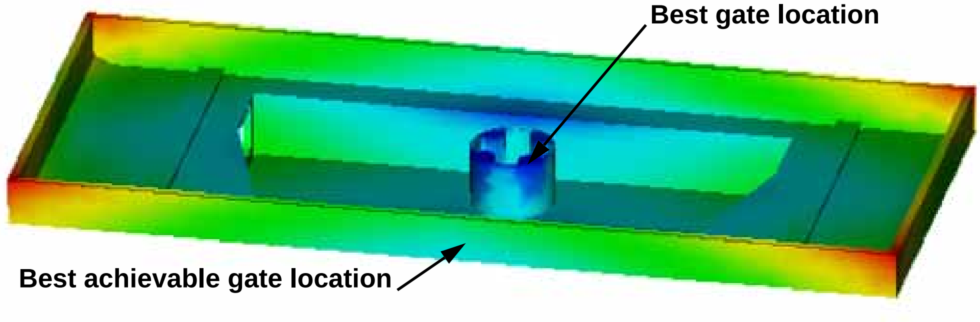


Figure 91: Best location for the gate

Gate location validation

Now that the gate location analysis is finished and those results were compared to the design criteria, a flow analysis is run to look at the filling pattern from the gate location. A normal multi-laminate fill analysis can be done, but if the primary purpose is to validate the gate location by looking at the filling pattern a fast fill analysis will work very well. It runs much faster than a regular fill analysis and the results will allow you to determine if the gate location is acceptable with regards to the filling pattern.

To set up the fast fill analysis

1. Save the **Cover** model using the command, **File ➡ Save Study as**.
2. Enter the new name **Cover Fast Fill**.
3. Change the analysis sequence to **Fast Fill** by either:
   - Double-click on the analysis sequence icon in the Study Tasks list and choose **Fast Fill**.
   - Click, **Analysis ➡ Set Analysis Sequence ➡ Fast Fill**.
4. Double-click the **Set Injection Locations** icon in the study tasks list. Place the injection location at the best achievable gate location as shown in Figure 91.
5. Save the model.
6. Double-click the icon **Analyze now** in the study tasks list.

To review the fast filling results

1. Click **Fill time** in the study tasks list.
2. Click on the Animate icon on the animation toolbar to animate the filling results.
3. Click on each of the results to review them.

Filling pattern

With the gate location in the center of the long side of the part, the filling of the part is predominantly radial and balanced. The two opposite corners fill last and at the same time.

Pressure at V/P switchover

The pressure at V/P switchover is well below the pressure limit of 70 MPa (10,000 psi) for the part by itself, so pressure will not limit the filling of the part.

Temperature at flow front

The temperature of the flow front has a small variation, about 1ºC. This would indicate that the automatic injection time is in a good range. The gate location will have little effect on the temperature distribution because the part wall thickness is mostly uniform.

Time to freeze

The majority of the part has a cooling time in the range of 4.5 to 5 seconds. The only area that is significantly different is the boss on the part. This area has both the thinnest and thickest areas on the part. As a result the cooling time has a wide variation here.

Weld Lines and Air Traps

The weld lines are unavoidable but should not be a significant problem. None of the air traps are in locations that require any special attention.

Paper holder

Design criteria

This part has been designed using a single cavity 2-plate tool with a cold runner system. The results of the analysis will determine the layout of the runner system.

Setup

To open a project

1. Click the file open icon or **File ➨ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Gate_Placement**.
2. Double click the project file **Gate_Placement.mpi**.
3. Click **File ➨ Preferences** and ensure that **System Units** are set to **Metric**.

To open the paper holder model

1. Double click the **Paper Holder** study in the Project View pane.
2. Rotate the model to review the geometry.

Analysis

To set up the analysis

1. Change the analysis sequence to gate location by either:
   - Double click on the analysis sequence icon in the Study Tasks list, and select **Gate Location**.
   - Click **Analysis ➨ Set Analysis Sequence ➨ Gate Location**.
2. Double-click the icon selection material in the Study Tasks pane.
3. Select the manufacturer **Lanxess** and the trade name **Lustran SAN 29**.
4. Click the **Analyze Now** icon .
   - The analysis should only take a few minutes to finish.

Displaying the results

To view gate location results

1. Click the **Best gate location** plot in the Study Tasks list.
2. Rotate the model to view all sides and then pick a good rotation where you can see the blue area(s) well. A good rotation is shown in Figure 92.

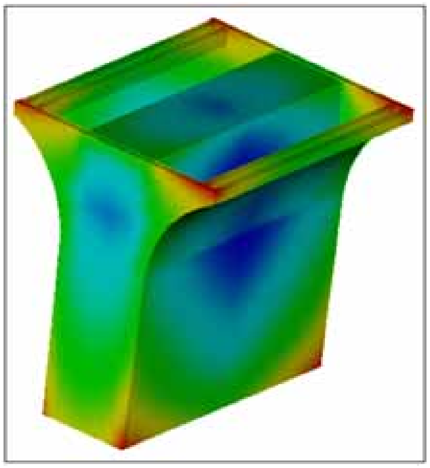


Figure 92: Paper holder - best gate

Results discussion

The best gate location plot indicates thicker regions of the part as the best locations for a gate. However, it would not be practical to place a gate in these areas due to the type of tool. The center of the bottom is blue so it is ranked high on the scale. The center of the bottom is also a good gate location for the criteria of the mold, a single cavity 2 plate tool. The gate can be a cold sprue gate or a hot sprue.

Gate location validation

If time permits, run a Fast Fill analysis with the gate in the center of the bottom of the part. Review the problems with this gate location and part design. How would you change the gate location or part design?

To set up the fast fill analysis

1. Save the **Paper holder** model using the command, **File ➡ Save Study as**.

1.1. Enter the new name **Paper holder Fast Fill**.

2. Double-click on the analysis sequence icon in the Study Tasks list and choose **Fast Fill**.

3. Double-click the **Set Injection Locations** icon in the study tasks list. Place the injection location in the center of the bottom of the part.

4. Double-click the icon **Analyze now** in the study tasks list.

To review the fast filling results

1. Click **Fill time** in the study tasks list.

2. Click on the Animate icon on the animation toolbar to animate the filling results.

3. Click on each of the results to review them.

Phone cover

Design criteria

This part has been designed using a 2-plate tool with a hot runner system. The hot drops can be directly on the part or feed a cold runner. The results of the analysis will determine the layout of the runner system.

Setup

To open a project

1. Click the file open icon or **File ➨ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Gate_Placement**.
2. Double click the project file **Gate_Placement.mpi**.
3. Click **File ➨ Preferences** and ensure that **System Units** are set to **Metric**.

To open the phone model

1. Double click the **Phone** study in the Project View pane.
2. Rotate the model to review the geometry.

Analysis

To set up the analysis

1. Change the analysis sequence to gate location by either:
   - Double click on the analysis sequence icon in the Study Tasks list, and select **Gate Location**.
   - Click **Analysis ➨ Set Analysis Sequence ➨ Gate Location**.
2. Double-click the icon selection material in the Study Tasks pane.
3. Select the manufacturer **Bayer MaterialScience** and the trade name **Bayblend FR 2000**.
4. Click the **Analyze Now** icon .
   - The analysis should only take a few minutes to finish.

Displaying the results

To view gate location results

1. Click the **Best gate location** plot in the Study Tasks list.

2. Rotate the model to view all sides and then pick a good rotation where you can see the blue area(s) well. A good rotation is shown in Figure 93. The results are scaled 0.8 to 1.0 and unscaled, in Figure 93 to better highlight the best gate location.

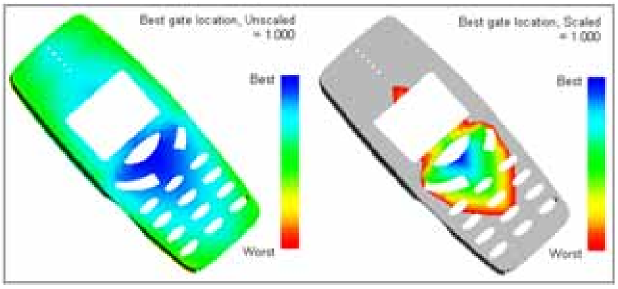


Figure 93: Phone - best gate

Results discussion

The best gate location is on the bottom edge of the large oval button hole. This gate location may or may not be practical for the part and will depend on the aesthetic qualities of the part and if a hot runner drop can be placed there. An alternative location could be on the edge of the large square opening. That is not far from the “Best” gate location. Gate blush will be a major concern for this part. There was not enough information given about this part to know if the gate location is practical. This gate location should be investigated further to see if this location is possible.

Gate location validation

If time permits, run a Fast Fill analysis with the gate in the center of the bottom of the part. Review the problems with this gate location and part design. How would you change the gate location or part design?

To set up the fast fill analysis

1. Save the **Phone** model using the command, **File ➡ Save Study as**.

   1.1. Enter the new name **Phone Fast Fill**.

2. Double-click on the analysis sequence icon in the Study Tasks list and choose **Fast Fill**.

3. Double-click the **Set Injection Locations** icon in the study tasks list. Place the injection location in the center of the bottom of the part.

4. Double-click the icon **Analyze now** in the study tasks list.

To review the fast filling results

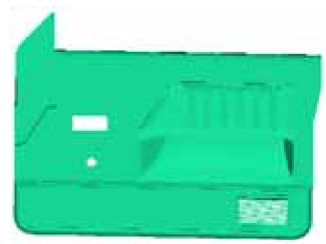
1. Click **Fill time** in the study tasks list.

2. Click on the Animate icon on the animation toolbar to animate the filling results.

3. Click on each of the results to review them.

Door panel

Design criteria



This part has been designed using a 2-plate tool with hot runners feeding a cold runner system. The gates will be edge gates. There can be no gates directly on the face part due to the requirements of a high quality textured finish on the door panel.

Setup

To open a project

1. Click the file open icon or **File** ➡ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Gate_Placement**.
2. Double click the project file **Gate_Placement.mpi**.
3. Click **File** ➡ **Preferences** and ensure that **System Units** are set to **Metric**.

To open the door panel model

1. Double click the **Door panel** study in the Project View pane.
2. Rotate the model to review the geometry.

Analysis

To set up the analysis

1. Double click on the analysis sequence icon in the Study Tasks list, and select **Gate Location**.
2. Double-click the icon selection material in the Study Tasks pane.
3. Select the manufacturer **Lanxess** and the trade name **Novodur P2MC**.
4. Click the **Analyze Now** icon .
   - The analysis should only take a few minutes to finish.

Displaying the results

To view gate location results

1. Click the **Best gate location** plot in the Study Tasks list.
2. Rotate the model to view all sides and then pick a good rotation where you can see the blue area(s) well. A good rotation is shown in Figure 94.

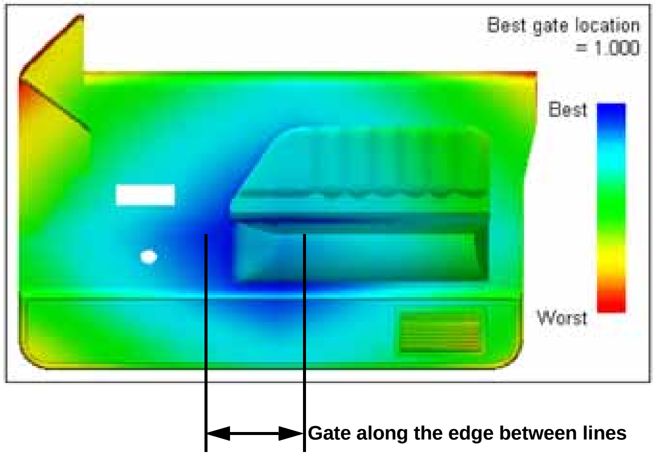


Figure 94: Door panel - best gate

Results discussion

The best gate location is on a vertical wall on the arm rest near the center of the part. The design criteria dictate that you must use edge gates, so the best gate location from the analysis can't be used. However, with the gate location analysis, you cannot input the type of gate or gate location restrictions. The gate location analysis in this case is of very limited use. The closest thing it can show is that the gate should be located on the center of the edge somewhere below the best gate location area shown in Figure 94.

Gate location validation

If time permits, run a Fast Fill analysis with the gate in the center of the bottom of the part, somewhere in the area highlighted in Figure 94. Review the problems with this gate location. How would you change the gate location?

To set up the fast fill analysis

1. Save the **Door panel** model using the command, **File** ➡ **Save Study as**.

   1.1. Enter the new name **Door panel Fast Fill**.

2. Double-click on the analysis sequence icon in the Study Tasks list and choose **Fast Fill**.

3. Double-click the **Set Injection Locations** icon in the study tasks list. Place the injection location in the center of the bottom of the part.

4. Double-click the icon **Analyze now** in the study tasks list.

To review the fast filling results

1. Click **Fill time** in the study tasks list.

2. Click on the Animate icon on the animation toolbar to animate the filling results.

3. Click on each of the results to review them.

Competency check - Gate Placement

1. How can you use the gate location analysis if you cannot select the exact gate location determined by the analysis?

Evaluation sheet - Gate Placement

1. **How can you use the gate location analysis if you cannot select the exact gate location determined by the analysis?**

   Answer: Often the restrictions to where you can place a gate on a part, prevent the **best** possible gate location. If this is the case, position the actual gate as close to the **best** gate location as possible. Running a regular flow analysis will help determine if the chosen gate will be acceptable.

CHAPTER 14

Molding Window Analysis

Aim

The aim of this chapter is to find the optimum molding conditions and widest possible molding window for the parts provided.

Why do it?

The optimum molding conditions for the part, i.e. mold temperature, melt temperature and injection time, is a good starting point of a finite element flow analysis. A molding window helps determine how much variation you can have in mold and melt temperatures, and the injection time and still make a good part. In addition, several gating locations can be evaluated with a molding window analysis. You can determine if a particular gating location is possible, or which one gives a wider molding window.

Overview

Molding conditions are evaluated on a part by running a quick analysis on the part. The analysis is done with a simple model that represents the thickness, flow length and volume of the part. This model is automatically built on the midplane or Fusion model. To run this analysis, a gate location and material must be defined. This simple model can be analyzed at a small fraction of the time of the midplane or fusion model, saving a significant amount of time.

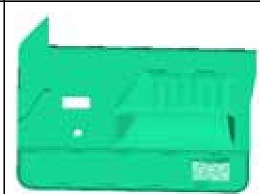
The output of this quick analysis includes an “optimum” set of conditions, plots indicating the size of the molding window and plots showing how part quality, injection pressure, shear stress, flow front temperature, cooling time and shear rate are influenced by changes in processing conditions.

Results from the analysis can be reviewed to determine if the optimum conditions should be used, or if there should be changes in the molding conditions. If several different gate locations are to be compared, results from the different gate locations need to be evaluated to determine which gate location is best.

Practice - Molding Window Analysis

This chapter has several models that are used for practice and are described below. At least the first one should be analyzed. Do the others as time permits.

Table 10: Models used for molding window analysis

Description	**Model**
Cover: starts on page 245	
The cover is the same part that is used in the translation and cleanup chapter. It is given to you already meshed and cleaned up ready for analysis. A molding window analysis will be run to determine the size of the molding window.	
Cell Phone cover: starts on page 253	
For the cell phone, two different gate locations will be investigated to determine which gate location has the biggest molding window.	
Door panel: starts on page 257	
For the door panel, two different grades of material are to be investigated to determine which one has the best molding window.	

Molding Window Analysis on the Cover

In this section, you will determine the optimum processing conditions for the cover model.

Design Criteria

This part will be created using a 2-plate tool. The gate location is defined in the center of the long side to be used with tunnel gates. Optimum conditions for the given gate location need to be found.

Setup

To open a project

1. Click the file open icon or **File ➡ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Molding_Window**.
2. Double click the project file **Molding_Window.mpi**.
3. Click **File ➡ Preferences** and ensure that **System Units** are set to **Metric**.

To open the cover model

1. Double click the **Cover** study in the Project View pane.
2. Rotate the model to review the geometry.

Running a Molding Window Analysis

To set up and run the analysis

1. Change the analysis sequence to molding window by either:
   - Double click on the analysis sequence icon in the Study Tasks list, and select **Molding window**.
   - Click **Analysis ➡ Set Analysis Sequence ➡ Molding window**.
2. Double-click the icon selection material in the Study Tasks pane.

   2.1. Select the manufacturer **BASF** and the trade name **Capron 8333GHS**.
3. Double-click the Injection location icon in the Study Tasks pane.

   3.1. Set the injection location in the center of the long side, half way up the wall, as indicated in Figure 95.

4. Double click the Process Settings icon in the Study Tasks pane.
   4.1. Click the **Edit** button in the Injection molding machine frame.
   4.2. Click the **Hydraulic Unit** tab.
   4.3. Set the maximum machine injection pressure to **140 MPa**.
   4.4. Click **OK** to exit the dialog.
5. Click the **Advanced options** button.
   5.1. Set the pressure factor to **0.8** in the **Feasible** molding window.
   5.2. Set the pressure factor to **0.5** in the **Preferred** molding window.
   5.3. Set the Flow front temp. Maximum drop to **20ºC**.
   5.4. Set the Flow front temp. rise limit to **2ºC**
   5.5. Click **OK** to exit advanced options.
   5.6. Click **OK** to exit the Process Settings Wizard.
6. Save the model.
7. Click the **Analyze Now** icon .
   - The analysis should only take a minute to finish.

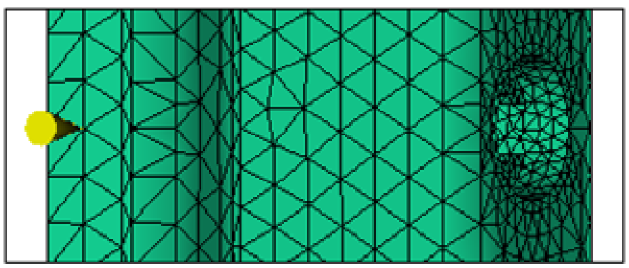


Figure 95: Cover injection location

Interpreting the Results

To view the Screen Output Log

1. Check the **Logs** box, if necessary, to view the log files.
2. Click on the **Screen Output** tab.
3. Compare the recommended conditions vs. the ranges analyzed. The output will look similar to Figure 96.
4. Record the recommended conditions, on line 1 of Table 11 on page 250.
5. Uncheck the Logs box.

```
   Mold temperature range to analyze                 =   Automatic
      from mold temperature                          =        80.0 C
      to   mold temperature                          =        95.0 C
   Melt temperature range to analyze                 =   Automatic
      from melt temperature                          =       270.0 C
      to   melt temperature                          =       295.0 C
   Injection time range to analyze                   =   Automatic
   Limits for calculation of feasible molding window
      Shear rate limit                               =         Off
      Shear stress limit                             =         Off
      Flow front temperature drop limit              =         Off
      Flow front temperature rise limit              =         Off
      Injection pressure limit factor                 =        0.80
      Clamp force limit                              =         Off
   Limits for calculation of preferred molding window
      Shear rate limit factor                        =        1.00
      Shear stress limit factor                      =        1.00
      Flow front temperature drop limit              =       20.00 C
      Flow front temperature rise limit              =        2.00 C
      Injection pressure limit factor                =        0.50
      Clamp force limit factor                       =        0.80
Maximum Design Clamp Force     7000.22 tonne
Maximum Design Injection Pressure    :      140.00 MPa
Recommended Mold Temperature   :       90.00 C
Recommended Melt Temperature   :      290.45 C
Recommended Injection Time     :      0.4384 s
```

Figure 96: Sample screen output from a molding window analysis

To view the Quality XY plot

1. Click the **Quality XY** plot in the Study Tasks pane.
2. Open plot properties by one of 3 ways:
   - Right-click on the result in the study tasks list and click **Properties**.
   - Click **Results** ➨ **Plot Properties**.
   - Click the Plot Properties icon on the Results toolbar.

   2.1. Click the **Injection time** box to set the graph X axis to Injection time.

   2.2. Scroll the mold and melt temperature sliders to the recommended values from the screen output.

   ➤ As the sliders are moved, notice how the maximum quality changes.
3. Decide on the mold and melt temperature to use.
   - Generally, if the quality does not change much by adjusting the mold temperature slider, choose a mid-range mold temperature.
   - The same is true for melt temperature.
   - This will give you more flexibility when running a fill or flow analysis and would like to adjust the temperatures.
   - If the recommended values are at the extremes of the usable range, there will be little room to make adjustments.

4. Record the new recommended processing conditions (mold and melt temperature) on line 2 of Table 11 on page 250.

To prevent the Y-axis scale of the quality plot from changing,

1. Click the plot properties button on the Explore Solution Space dialog.
2. Click the **XY Plot Properties (2)** tab.
3. Click **Manual** in the Y range frame.
4. Enter the **Min.** and **Max** values as needed.

To find the Molding window: injection time

1. Set the mold and melt temperature sliders to the new recommended processing conditions (if not already set).
2. Click the **query** tool icon,  to select the **highest quality** point on the XY graph and note the x value. An example is shown in Figure 97.
   - This represents the optimum injection time.
   - With the example in Figure 97, the new recommended conditions include a mold temperature of 90º C. a melt temperature of ~283º C. and 0.5 seconds fill time.
3. Record the injection time on line 2 of Table 11 on page 250.

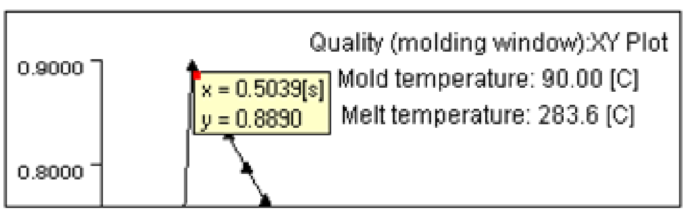


Figure 97: Query on quality plot

To view the Molding window:2D zone plot

1. Click the **Zone (molding window):2D Slice Plot** in the Study tasks list.
2. Open plot properties. One way is to click the Plot Properties icon .

   2.1. Set the Cut axis to **Mold temperature**.

   2.2. Set the Cut position to **90ºC**.

   2.3. Click **OK**.

3. Ensure the Add XY plot icon is depressed.
   3.1. Hold the left mouse button down.
   3.2. Drag the mouse up and down to move the cut axis position.
   3.3. Watch how the preferred molding window changes.
   3.4. Reset the cut position back to 90ºC.

4. Click the Query icon
   4.1. Click at the bottom yellow / green interface at the temperature of ~283.
   - It may take several clicks to get to the correct location.
   - Note the y value (injection time).

   4.2. Click at the top yellow / green interface at the temperature of ~283.
   - It may take several clicks to get to the correct location.
   - Note the y value (injection time).

   - These times will correlate to the molding window defined by the temperature plot.
   - The total size of the green area represents the size of the molding window for the mold temperature of 90º C.

To plot the injection pressure

1. Click the **Injection pressure XY** plot in the Study Tasks pane.

2. Open plot properties. One way is to click the Plot Properties icon .
   2.1. Click the **Injection time** box to set the graph X axis to Injection time.
   2.2. Scroll the mold and melt temperature sliders to the new recommended values that you recorded on line 2 of Table 11 on page 250.

3. Click the Query icon , to find the recommended time on the pressure curve.
   - The magnitude of the pressure should be below 70 MPa, (10,000 psi). The lower the better.
   - Preferably, it should be at the bottom of the pressure curve. This rarely happens.

To plot minimum flow front temperature

1. Click the **Minimum flow front temperature XY** plot in the Study Tasks pane.

2. Open plot properties. One way is to click the Plot Properties icon .

   2.1. Click the **Injection time** box to set the graph X axis to Injection time.

   2.2. Scroll the mold and melt temperature sliders to the new recommended values that you recorded on line 2 of Table 11 on page 250.

3. Use the query tool to find where the temperature is 0º C, 10º C. (18º F), and 20º C. (36º F) below the melt temperature.

   3.1. Record the values found in Table 11, lines 3, 4, and 5.

   - A 0º C drop in temperature defines the highest quality.
   - A 10º C (18º F) drop in temperature in most cases is a very acceptable amount of drop.
   - A 20º C (36º F) defines the limit of the preferred molding window, assuming it was set a maximum temperature drop of 20ºC (36ºF) in the advanced options.
   - Finding at what times these temperatures occur will give you another way to get a sense for the size of the molding window as the flow front temperature is generally the limiting factor in the molding window. The times you find here should correlate to the times you found by looking at the 2D zone plot.

Table 11: Record molding conditions for the cover

	Mold Temp	Melt Temp	Injection Time
1. Recommended in the summary file.			
2. New recommended molding conditions. Based on quality plot analysis.			
3. Injection time with 0º C Flow Front Temp Drop.			
4. Injection time with 10º C Flow Front Temp Drop.			
5. Injection time with 20º C Flow Front Temp Drop.			

To plot maximum shear stress

1. Click the **Maximum Shear stress XY** plot in the Study Tasks pane.

2. Open plot properties. One way is to click the Plot Properties icon .

   2.1. Click the **Injection time** box to set the graph X axis to Injection time.

   2.2. Scroll the mold and melt temperature sliders to the new recommended values that you recorded on line 2 of Table 11.

3. Make sure the shear stress is below the limit of the material as found in the material database.

> To check the shear stress limit, right click on the material icon and select details.

To plot maximum cooling time

1. Click the **Molding window: Maximum cooling time XY** plot in the Study Tasks pane.
2. Scroll the melt temperature and injection time sliders to the new recommended values that you recorded on line 2 of Table 11.
3. Check to make sure the cooling time is acceptable.

To plot maximum shear rate

1. Click the **Maximum shear rate XY** plot in the Study Tasks pane.
2. Open plot properties. One way is to click the Plot Properties icon .
   - **2.1.** Click the **Injection time** box to set the graph X axis to Injection time.
   - **2.2.** Scroll the mold and melt temperature sliders to the new recommended values that you recorded on line 2 of Table 11.
3. Make sure the shear rate values are well below the material limit.
   - This should NOT ever be a problem.

Optional work

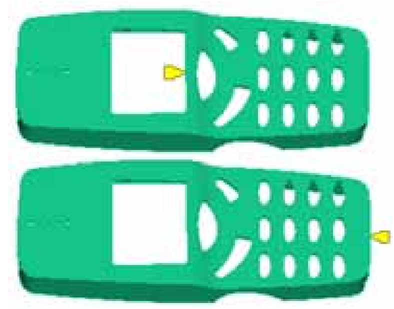
Any of the results can be created as a 2D slice plot. To do this, create a new plot and select the result then plot type. The 2D slice plots are another interesting way to view the results.

Results discussion

The results for the Cover are discussed on page 261.

Cell phone cover

Design Criteria



Two different gate locations are to be considered for this project. One is a center-gating location, and the second one is an end gate. The molding window analysis will be used to compare both gate locations. The material used for each is Bayer Material Science, Bayblend FR 2000.

Setup

To open a project

1. Click the file open icon or **File ➡ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Molding_Window**.
2. Double click the project file **Molding_Window.mpi**.
3. Click **File ➡ Preferences** and ensure that **System Units** are set to **Metric**.

To open the phone model with a center gate

The model used is provided with the gate location defined.

1. Double click the **Phone MW Cent** study in the Project View pane.
2. Rotate the model around to review the geometry.

Running a Molding Window Analysis

To set up and run the analysis

1. Change the analysis sequence to gate location by either:
   - Double click on the analysis sequence icon in the Study Tasks list, and select **Molding window**.
   - Click **Analysis ➡ Set Analysis Sequence ➡ Molding window**.
2. Double-click the icon selection material in the Study Tasks pane.

   2.1. Select the manufacturer **Bayer Material Science** and the trade name **Bayblend FR 2000**.

3. Double click the Process Settings icon in the Study Tasks pane.

   3.1. Click the **Edit** button in the Injection molding machine frame.

   3.2. Click the **Hydraulic Unit** tab.

   3.3. Set the maximum machine injection pressure to **140 MPa**.

   3.4. Click **OK** to exit the dialog.

4. Click the **Advanced options** button.

   4.1. Set the pressure factor to **0.8** in the **Feasible** molding window

   4.2. Set the pressure factor to **0.5** in the **Preferred** molding window

   4.3. Set the Flow front temp. Maximum drop to **20ºC**.

   4.4. Set the Flow front temp. rise limit to **2ºC**

   4.5. Click **OK** to exit advanced options.

   4.6. Click **OK** to exit the Process Settings Wizard.

5. Click the **Analyze Now** icon .

To open the phone model with a End gate

The model used will be provided with the gate location defined.

1. Double click the **Phone MW End** study in the Project View pane.
2. Rotate the model around to review the geometry.
3. Change the analysis sequence to Molding window by either:
   - Double click on the analysis sequence icon in the Study Tasks list, and select **Molding window**.
   - Click **Analysis ➨ Set Analysis Sequence ➨ Molding window**.
4. Double-click the icon selection material in the Study Tasks pane.

   4.1. Select the manufacturer **Bayer Material Science** and the trade name **Bayblend FR 2000**.

5. Double click the Process Settings icon in the Study Tasks pane.

   5.1. Click the **Edit** button in the Injection molding machine frame.

   5.2. Click the **Hydraulic Unit** tab.

   5.3. Set the maximum machine injection pressure to **140 MPa**.

   5.4. Click **OK** to exit the dialog.

6. Click the **Advanced options** button.

6.1. Set the pressure factor to **0.8** in the **Feasible** molding window

6.2. Set the pressure factor to **0.5** in the **Preferred** molding window

6.3. Set the Flow front temp. Maximum drop to **20ºC**.

6.4. Set the Flow front temp. rise limit to **2ºC**

6.5. Click **OK** to exit advanced options.

6.6. Click **OK** to exit the Process Settings Wizard.

7. Save the model.

8. Click the **Analyze Now** icon.

- The analysis should only take a minute to finish.

To compare the analyses

1. Use the General Interpretation Procedure on page 259 in the Theory and Concepts for MPI 6.0 manual and to view the results of both molding window analyses.

2. Fill out Table 12 on page 256 below with the results.

3. Determine which gate location is best and why.

Table 12: Molding window work sheet for the phone

	Mold Temperature Deg. C	Melt Temperature Deg. C	Injection Time Sec.
Bayblend, Minimum Temperatures analyzed			
Bayblend, Maximum Temperatures analyzed			
Phone **Center** gate location, Recommended in the Screen Output log			
Phone **End** gate location, Recommended in the Screen Output log			
Phone **Center** gate location, Selected after interpretation of the quality plot			
Phone **End** gate location, Selected after interpretation of the quality plot			
	Pressure MPa	Min. Flow Front Temp Deg. C.	Shear Stress MPa
Phone **Center** gate location, Selected after interpretation of the quality plot			
Phone **End** gate location, Selected after interpretation of the quality plot			

What gate location did you select and why?

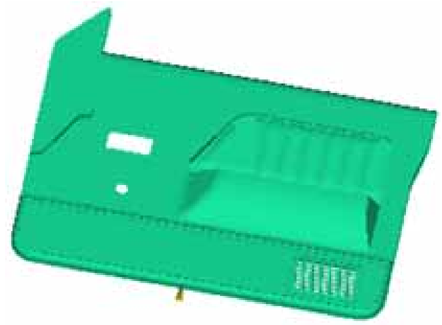
Results discussion

The results for the cell phone are discussed on page 262.

Door Panel

Design Criteria

The initial gate location to be investigated on this part is shown to the side. Two different materials are under consideration; Chi Mei Corporation, Polylac PA-727 and Asahi Kasei Corporation, Stylac 190F. A molding window analysis will be run to determine which one to use.



Setup

To open a project

1. Click the file open icon or **File ➡ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Molding_Window**.
2. Double click the project file **Molding_Window.mpi**.
3. Click **File ➡ Preferences** and ensure that **System Units** are set to **Metric**.

To prepare the door panel models

- The models used will be provided with the gate location defined.

1. Highlight the **Door Panel MW** study in the Project View pane.
2. Press the **F2** key.

   2.1. Name the study **Door Panel Asahi**.
3. Double click the **Door Panel Asahi** to open it.
4. Click **File ➡ Save Study as**.

   4.1. Name the study **Door Panel Chi Mei**.
5. Rotate the model around to review the geometry.

Running a Molding Window Analysis

To set up and run the analysis with the Chi Mei material

1. Change the analysis sequence to Molding window by either:

   - Double click on the analysis sequence icon in the Study Tasks list, and select **Molding window**.
   - Click **Analysis ➡ Set Analysis Sequence ➡ Molding window**.

2. Double-click the icon selection material in the Study Tasks pane.

   2.1. Select the manufacturer **Chi Mei Corporation.**

   2.2. 7 Select the trade name **Polylac PA-72**.

3. Double click the Process Settings icon in the Study Tasks pane.
   3.1. Click the **Edit** button in the Injection molding machine frame.
   3.2. Click the **Hydraulic Unit** tab.
   3.3. Set the maximum machine injection pressure to **140 MPa**.
   3.4. Click **OK** to exit the dialog.
4. Click on the **Melt temperature range to analyze** field arrow.
   4.1. Pick **Specified**.
   4.2. Click the **Edit range** button.
   4.3. Enter **235** in the Minimum field.
   4.4. Enter **260** in the Maximum field.
   4.5. Click **OK**.
5. Click on the **Injection time range to analyze** field arrow.
   5.1. Pick **Specified**.
   5.2. Click the **Edit range** button.
   5.3. Enter **1** in the Lower limit field.
   5.4. Enter **8** in the Upper limit field.
   5.5. Click **OK**.
6. Click the **Advanced options** button.
   6.1. Set the pressure factor to **0.8** in the **Feasible** molding window
   6.2. Set the pressure factor to **0.5** in the **Preferred** molding window
   6.3. Set the Flow front temp. Maximum drop to **20ºC**.
   6.4. Set the Flow front temp. rise limit to **2ºC**
   6.5. Click **OK** to exit advanced options.
7. Click the **Analyze Now** icon .

To set up and run the analysis with the Asahi material

1. Double click the study **Door Panel Asahi** to open it.
2. Change the analysis sequence to Molding window by either:
   - Double click on the analysis sequence icon in the Study Tasks list, and select **Molding window**.
   - Click **Analysis** ➨ **Set Analysis Sequence** ➨ **Molding window**.
3. Double-click the icon selection material in the Study Tasks pane.
   3.1. Select the manufacturer **Asahi Kasei Corporation.**
   3.2. 7 Select the trade name **Stylac 190F**.

4. Double click the Process Settings icon in the Study Tasks pane.
   4.1. Click the **Edit** button in the Injection molding machine frame.
   4.2. Click the **Hydraulic Unit** tab.
   4.3. Set the maximum machine injection pressure to **140 MPa**.
   4.4. Click **OK** to exit the dialog.
5. Click on the **Melt temperature range to analyze** field arrow.
   5.1. Pick **Specified**.
   5.2. Click the **Edit range** button.
   5.3. Enter **235** in the Minimum field.
   5.4. Enter **260** in the Maximum field.
   5.5. Click **OK**.
6. Click on the **Injection time range to analyze** field arrow.
   6.1. Pick **Specified**.
   6.2. Click the **Edit range** button.
   6.3. Enter **1** in the Lower limit field.
   6.4. Enter **8** in the Upper limit field.
   6.5. Click **OK**.
7. Click the **Advanced options** button.
   7.1. Set the pressure factor to **0.8** in the **Feasible** molding window
   7.2. Set the pressure factor to **0.5** in the **Preferred** molding window
   7.3. Set the Flow front temp. Maximum drop to **20ºC**.
   7.4. Set the Flow front temp. rise limit to **2ºC**
   7.5. Click **OK** to exit advanced options.
8. Click the **Analyze Now** icon.

To compare the analyses

1. Use the General Interpretation Procedure on page 259 in the Theory and Concepts for MPI 6.0 manual and to view the results of both molding window analyses.
2. Fill out Table 14 below with the results.
3. Determine which material is best and why.
   3.1. To help evaluate the results, find one set of conditions that works for both materials and compare the results.

Table 13: Molding window work sheet for the door panel

	Mold Temperature Deg. C	Melt Temperature Deg. C	Injection Time Sec.
Minimum Temperatures analyzed	40	235	
Maximum Temperatures analyzed	80	260	
Chi Mei, Recommended in the Screen Output log			
Asahi, Recommended in the Screen Output log			
Common set of conditions			
	Pressure MPa	Min. Flow Front Temp Deg. C.	Shear Stress MPa
Chi Mei at common conditions			
Asahi at common conditions			

What material did you select and why?

Results discussion

The results for the door panel are discussed on page 263.

Cover Molding window: results discussion

Table 14: Molding conditions for the cover

	Mold Temp	Melt Temp	Injection Time
1. Recommended in the summary file.	90	290	0.44
2. New recommended molding conditions. Based on quality plot analysis.	90	285	0.5
3. Injection time with 0º C Flow Front Temp Drop.			0.5
4. Injection time with 10º C Flow Front Temp Drop.			1.0
5. Injection time with 20º C Flow Front Temp Drop.			1.6

The temperatures and injection times listed in Table 14 above are approximate values. Realistically, they can be rounded to the nearest 5º, and 0.1 seconds.

The zone plot indicated a large molding window with injection time being the most restricted variable, as shown in Figure 98. This is normally the case. The quality plot indicates that the maximum quality for any combination of mold and melt temperature does not change much. For mold and melt temperatures, values closer to the center of the range were picked. Because the full range of mold or melt temperatures is in the preferred window, the values are not critical.

The recommended injection time of 0.5 seconds represents the highest quality for the mold and melt temperature picked. It is also the time required so the minimum flow front temperature is approximately equal to the melt temperature. The 1.0 second injection time represents a drop of about 10º C and is in the center of the molding window. The 1.6 second injection time represents the slow end of the molding window with a temperature drop of about 20º C.

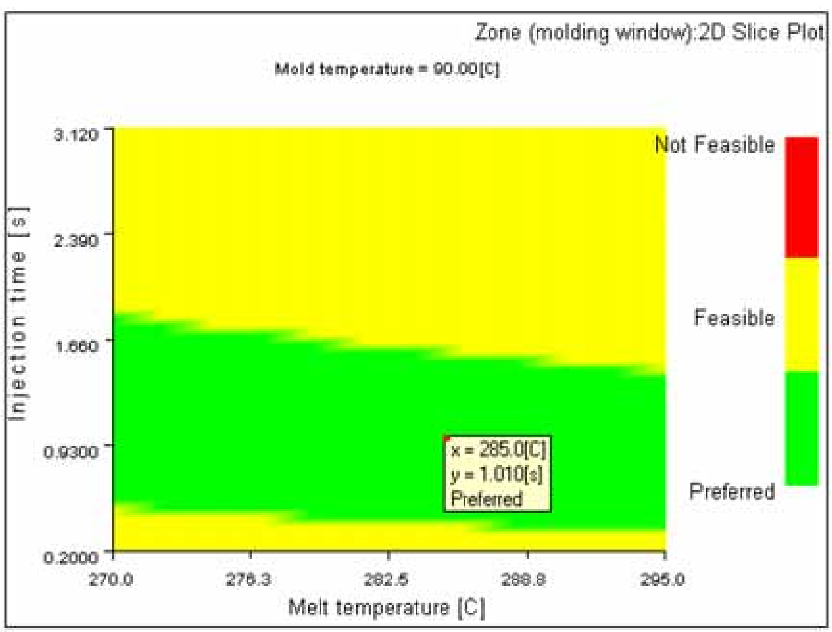


Figure 98: 2D Zone plot for the cover

Cell phone results discussion

Table 15: Molding window results for the phone

	Mold Temperature Deg. C	Melt Temperature Deg. C	Injection Time Sec.
Bayblend, Minimum Temperatures analyzed	70	240	
Bayblend, Maximum Temperatures analyzed	89	290	
Phone **Center** gate location, Recommended in the Screen Output log	89	288	0.35
Phone **End** gate location, Recommended in the Screen Output log	87	288	0.5
Phone **Center** gate location, Selected after interpretation of the quality plot	80	270	0.5
Phone **End** gate location, Selected after interpretation of the quality plot	80	275	0.6
	Pressure MPa	**Min. Flow Front Temp Deg. C.**	**Shear Stress MPa**
Phone **Center** gate location, Selected after interpretation of the quality plot	32	267	0.26
Phone **End** gate location, Selected after interpretation of the quality plot	23	275	0.22

The temperatures and injection times listed in Table 15 above are approximate values. Realistically, they can be rounded to the nearest 5°, and 0.1 seconds.

What gate location did you pick and why?

Both gate locations have a reasonably sized molding window, as shown in Figure 99. The pressure from the end gate is still well below the limit at recommended processing conditions. The shear stress in both parts is below the limit. The end gate does have more unidirectional flow. For this reason this location is the best.

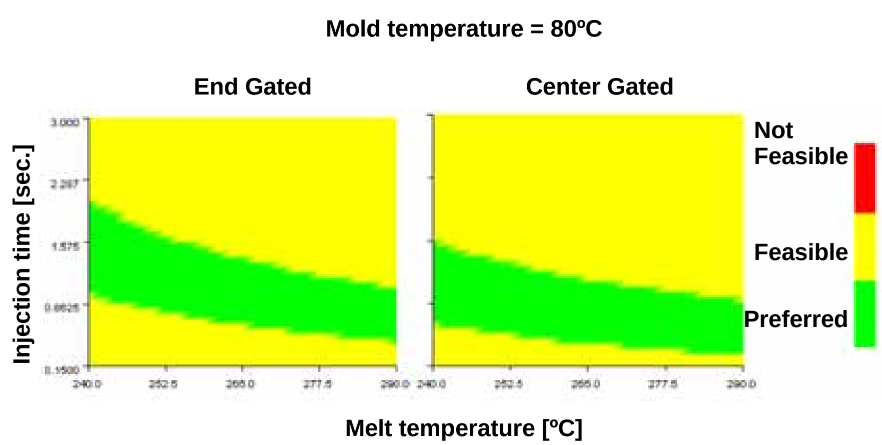


Figure 99: 2D Zone plots for the phone with different gate locations

Door panel results discussion

Table 16: Molding window results for the door panel

	Mold Temperature Deg. C	Melt Temperature Deg. C	Injection Time Sec.
Minimum Temperatures analyzed	40	235	
Maximum Temperatures analyzed	80	260	
Chi Mei, Recommended in the Screen Output log	80	257	3.6
Asahi, Recommended in the Screen Output log	67	260	2.3
Common set of conditions	75	257	4.5
	Pressure MPa	Min. Flow Front Temp Deg. C.	Shear Stress MPa
Chi Mei at common conditions	68	248	0.1
Asahi at common conditions	47	239	0.15

The temperatures and injection times listed in Table 16 above are approximate values. Realistically, they can be rounded to the nearest 5º, and 0.1 seconds.

What material did you pick and why?

The best material for the door panel considering only processing conditions is with no question the Asahi material. The zone plot very clearly shows the molding window for the Asahi material is much larger than the Chi Mei material. The viscosity is much lower for the Asahi material. The pressure and shear stress are also lower with the Asahi material. The injection times required to fill the part are also a bit different for the materials. This is primarily due to the differences in thermal properties for the materials.

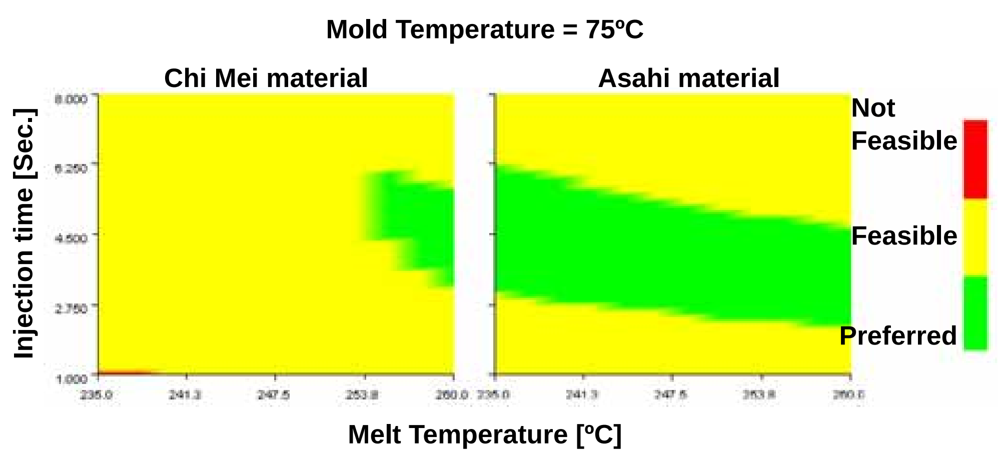


Figure 100: 2D zone plots for the door panel with different materials

Competency Check -Molding Window Analysis

Once an acceptable mold and melt temperature are found by using the quality plot, how can the injection time be found?

Evaluation Sheet - Molding Window Analysis

Once an acceptable mold and melt temperature are found by using the quality plot, how can the injection time be found?

Answer:

The best injection time may not always be at the point where the quality plot is the highest. Review the pressure and temperature plots to see the effect of injection time. As a reference start with the time with the highest quality. Compare the highest quality time with the change in pressure over time. A relatively low pressure is desirable. Compare the minimum flow front temperature with the highest quality time. For instance a slightly slower injection time may give a much lower pressure with only a small drop in temperature.

CHAPTER 15

Fiber Flow Analysis

Aim

The aim of this chapter is to be introduced to the MPI/Fiber analysis program. This includes background theory about fibers, why the fiber-orientation analysis is important, and how to run an analysis and interpret the results.

Why do it

A fiber flow analysis is a standard flow analysis plus an algorithm that calculates the distribution of short fibers in a polymer matrix. For conducting a warpage or stress analysis of a filled material, it is critical that the distribution of the fibers be considered as they dominate the physical properties of the material.

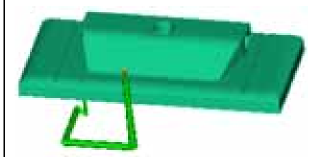
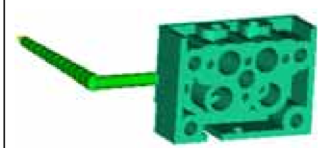
Overview

Running a fiber flow analysis is a simple task. If the material is a fiber filled material, the database contains all of the additional data necessary to run this material. By default, a fiber flow analysis is run unless you indicate that the fiber-orientation flow analysis should not be run. In this chapter, you will review fiber flow analysis results on a Fusion or Midplane part.

Practice - Fiber Flow Analysis

This chapter has two models to choose from and are described below. Pick one to work on.

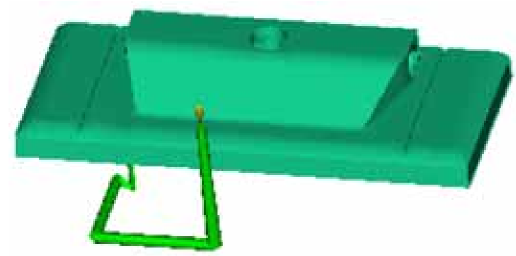
Table 17: Models used for Fiber flow analysis

Description	Model
Cover: starts on page 273	
The cover is the same part that is used in other chapters, however in this version; the feed system has been added to the model. This is a Fusion model. Running a fiber analysis and looking at fiber results is the same for both midplane and Fusion mesh types. If you primarily use midplane models, use this example.	
Manifold: starts on page 277	
The manifold is a 3D model. This is the same model used for the translation chapter, however a feed system has been added.	

To save time, the results are provided for both models.

Cover Model

Design Criteria



For the cover, only the initial fiber flow analysis will be run, and the results will be viewed. In this practice, you are only working on reviewing the fiber results. The whole project involves solving the warpage problems with this part.

The cover will be assembled with other components along the bottom edge of the part. To meet assembly requirements, the bottom edge needs to be flat within the warpage specifications. If changes need to be made to reduce the warpage, the part design cannot change but the gate location and processing conditions can be modified. The current gate location is through a pin on the underside of the part. Subgates can also be used in the side of the part.

Project/Design Parameters:	
Model Type	Fusion
Material	33% Glass Filled Nylon
Minimum warpage criteria	Part bottom edge must be flat within 1.0 mm

For the cover, you will run a fiber flow analysis and look at the fiber results and form an opinion on whether the part will warp due to the fiber orientation.

Setup

To open a project

1. Click the file open icon or **File ➡ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Fiber_Flow**.
2. Double click the project file **Fiber_Flow.mpi**.
3. Click **File ➡ Preferences** and ensure that **System Units** are set to **Metric**.

To open the cover model

1. Double click the **Cover_Fiber** study in the Project View pane.
2. Rotate the model to review the geometry.

Viewing Results

To save time, the flow analysis has been run for you.

To view the fill time results

1. Click **Fill time** in the study tasks list.
2. Open plot properties by one of 3 ways:
    - Right-click on the result in the study tasks list and click **Properties**.
    - Click **Results ➡ Plot Properties**.
    - Click the Plot Properties icon on the Results toolbar.

    2.1. Click on the **Methods** tab.

    2.2. Select the **Contour** radio button.

    2.3. Click **OK**.
3. Use the Animate tools to watch the filling pattern.
    - Notice where the flow front is primarily radial and where it is mostly straight.

To view the Pressure result

1. Click **Pressure** in the study tasks list
2. Open the **Plot Properties**.

    2.1. Click the **Optional Settings** tab.

    2.2. Select **Banded** in the Color field.

    2.3. Click **OK**.
3. Use Animate tools and watch how the pressure changes over time.

Interpreting fiber orientation results

There are two results that show the fiber orientation; **Average fiber orientation** and **Fiber orientation tensor**. Both by default are displayed with tensors.

Tensor plots

- The tensor by default is plotted by **First Principal value**. This means that the highest probability percentage of the fibers is in the direction indicated by the tensor axis, as shown in Figure 101.
- The scale for the first principal value goes from ~0.5 (blue) to ~1.0 (red).
- The blue values indicate that the fibers are randomly oriented in plane. A value of 0.5 would mean that all the fibers are oriented in the plane of the element. Below 0.5 indicates some alignment in the element thickness direction.
- The red value indicates that fibers are highly aligned. A value of 1.0 indicates that all fibers are aligned in one direction.

- The tensor axis, (two lines crossing) will have the legs at the same length with a random distribution, and a very short secondary leg when the tensor value is high.

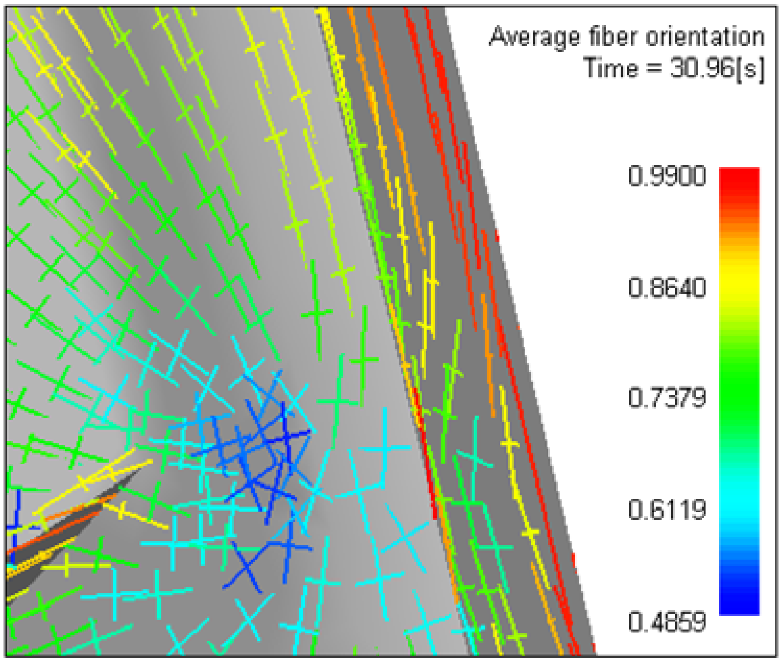


Figure 101: Fiber orientation tensors

Average fiber orientation

- Average fiber orientation shows how the fibers are oriented on average through the thickness of the part over time.
- Animation is done through time.

Fiber orientation tensor

- This plot represents the fiber orientation at a specific location in the cross section at the end of the analysis.
- For a Flow analysis, the time is at the end of the cycle. If this were a fill analysis, it would represent the results at the end of fill.
- Animation is done by Normalized thickness or through the cross section. A normalized thickness of 0.0 is the center line, with 1.0 and -1.0 at the mold wall.

To plot average fiber orientation

1. Click **Average fiber orientation** in the study tasks list.
2. Animate the fiber orientation plot.
   - Notice how the orientation of an area changes over time. Most of the change goes from less aligned to more aligned.

3. Change the scale of the model. Animate so the whole model can be seen then zoom up to concentrate on an area of interest.

- What can you conclude about the orientation? Is the changing orientation good or bad?

To make fiber orientation results easier to see

1. Change the glyph size.

1.1. Click the Plot Properties icon on the Results toolbar.

1.2. Click the **Tensor** tab.

1.3. Enter **0.15** in the Glyph Size Scale Factor.

1.4. Click **Apply**.

➤ If desired, change the glyph to other sizes and see the effect on the model.

2. Change the Animation value range.

2.1. Click on the **Animation** tab on the Plot properties dialog.

2.2. Set the **Animate result over** to **Single Dataset**.

2.3. Set the **time** to the maximum.

2.4. Click the **Value range** to **0.1**.

2.5. Click **Current frame only**.

2.6. Click **Apply**.

3. Change the scale.

3.1. Click Specified.

3.2. Enter **0.4** as the min.

3.3. Enter **1.0** as the Max.

3.4. Click **OK**.

4. Animate the result one frame at a time with the icon .

To plot fiber orientation tensor

1. Click **Fiber orientation tensor** in the study tasks list.

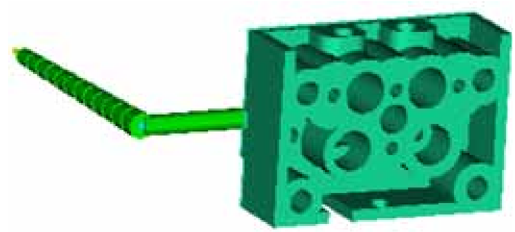
2. Step through the animation one frame at a time.

- Watch how the orientation changes through the thickness. What does this correlate to?
- What conclusions can be made about the fiber orientation? What is your opinion on fiber orientation's affect on warpage?

> The same modification can be done with the fiber orientation tensor that was done with the average fiber orientation. For both plots, the Animate result at value can be changed to see the influence at different times, or locations within the cross section.

Manifold Model

Design Criteria



For the manifold, only the initial fiber flow analysis results will be viewed. In this practice, you are only working on reviewing the fiber results. The whole project involves solving the warpage problems with this part.

The manifold will be assembled with other components along the bottom face of the part. To meet assembly requirements, the bottom face needs to be flat within the warpage specifications so the seal to be used will work properly. If changes need to be made to reduce the warpage, the part design cannot change but the gate location and processing conditions can. The current gate location is an edge gate through the side of the part.

Project/Design Parameters:	
Model Type	3D Tetrahedral mesh
Material	33% Glass Filled Nylon
Minimum warpage criteria	Part bottom edge must be flat within 0.2 mm

Setup

To open a project

1. Click the file open icon or **File ➡ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Fiber_Flow**.
2. Double click the project file **Fiber_Flow.mpi**.
3. Click **File ➡ Preferences** and ensure that **System Units** are set to **Metric**.

To open the manifold model

1. Double click the **Manifold_Fiber** study in the Project View pane.
2. Rotate the model to review the geometry.
3. Turn on and off the layers.
   - There is only one layer for the tetrahedral mesh of the part. To see the mesh inside the part, create a new layer select and assign elements to the new layer.

Viewing Results

To save time, the flow analysis has been run for you.

To view the fill time results

1. Click **Fill time** in the study tasks list.
2. Use the Animate tools to watch the filling pattern.
    - Notice how the flow front is initially mostly radial then straightens out.

To view the Pressure result

1. Click **Pressure** in the study tasks list
2. Use Animate tools and watch how the pressure changes over time.

Interpreting fiber orientation results

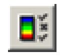
- Fiber orientation tensor (3D) indicates how the fibers are oriented in an element at the end of the cycle.
- The tensor by default is plotted by **First Principal value**. This means that the highest value will be in the direction where most of the fibers are oriented.
- The scale for the first principal value goes from 0.33 (blue) to 1.0 (red).
- A value of 1.0 (red) would indicate all the fibers are oriented in one direction.
- Planar random orientation means a11 and a22 are equal to 0.5.
- Three dimensional random orientation means a11 = a22 = a33 = 1/3.
- Zero in any component means there is no percentage of fibers in this direction.
- The tensor symbol, (three lines crossing) will have the legs at the same length with a random distribution, and very short secondary legs when the tensor value is high.
- The symbol used to show the fiber orientation is called a glyph. The default size is based on the element. Changing its size will help with the orientation.

Viewing Fiber Results

To plot fiber orientation tensor (3D)

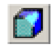
1. Click **Fiber orientation tensor (3D)** in the study tasks list.
2. Review to Figure 102 on page 281.
    - Note how very busy the plot looks. It needs to be modified to make it easier to interpret.

To make fiber orientation results easier to see

1. Change the glyph size.
   - **1.1.** Open plot properties by one of 3 ways:
     - Right-click on the result in the study tasks list and click **Properties**.
     - Click **Results ➡ Plot Properties**.
     - Click the Plot Properties icon  on the Results toolbar.
   - **1.2.** Click the **Tensor** tab.
   - **1.3.** Enter **0.15** in the Glyph Size Scale Factor.
   - **1.4.** Click **Apply**.
     - If desired, change the glyph to other sizes and see the effect on the model.
2. Change the Animation value range.
   - **2.1.** Click on the **Animation** tab on the Plot properties dialog.
   - **2.2.** Click the **Value range** to **0.1**.
   - **2.3.** Click **Current frame only**.
   - **2.4.** Click **Apply**.
3. Change the scale.
   - **3.1.** Click Specified.
   - **3.2.** Enter **0.3** as the min.
   - **3.3.** Enter **1.0** as the Max.
   - **3.4.** Click **OK**.
4. Animate the result one frame at a time with the icon .

5. Define cutting planes to reduce the amount of the part shown.

5.1. Click the top view button on the viewpoint toolbar to rotate the part to 90, 0, 0.

5.2. Click the edit cutting plane icon  on the viewer toolbar.

5.3. Click the **New** button to create a cutting plane with the screen being the plane.

- ➤ The **Move Cutting Plane** dialog will open.

5.4. Ensure the **Show active plane** box is checked.

5.5. Move the cutting plane by holding the left mouse button and moving the mouse up and down.

- ➤ Stop the plane so it has cut the part, but not too close to the center of the part.

5.6. Click the edit cutting plane icon on the viewer toolbar.

5.7. Select the ZX plane to activate it.

- Depending on where you left the plane you created, the part may not be visible. If the part is not visible, flip the ZX plane.

5.8. Make either plane active so you have a thin slice of the part.

- Changing the size of the glyph will make each element's glyph easier to see, but it is still very difficult to see how the fiber orientation is distributed.
- By setting the animation value, you are controlling the increment of the animation from a random distribution to the highly aligned distribution. When the scale is set from 0.3 to 1.0, the animation is done in rounded increments, i.e. 0.3 to 0.4, etc.
- Adding cutting planes allow you focus on a specific area of the part and look at the orientation in this small area. See Figure 103.
- What can you conclude about the orientation?
- Is the changing orientation good or bad?

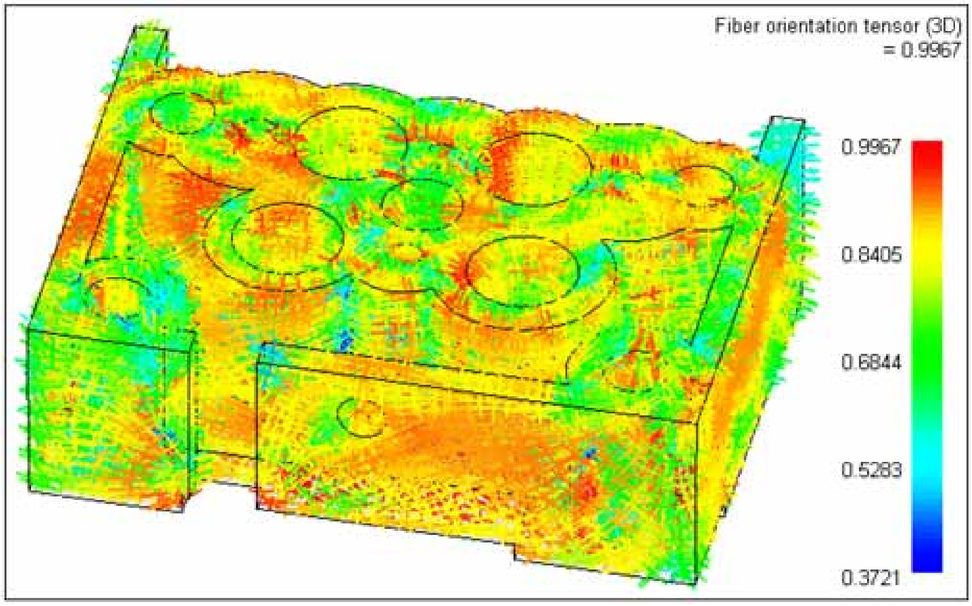


Figure 102: Fiber orientation tensor on a 3D part with default settings

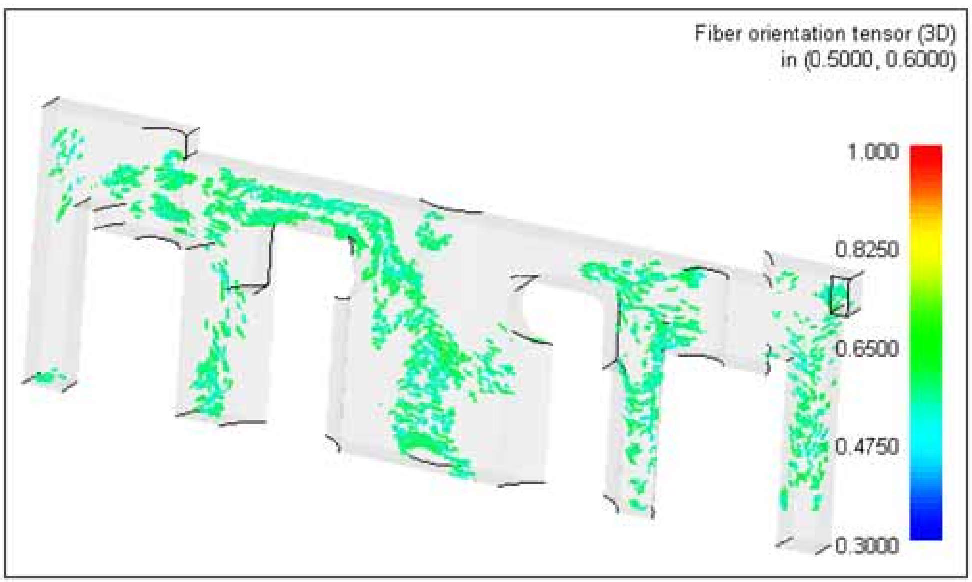


Figure 103: Fiber orientation tensor slice through the part

Competency Check - Fiber Flow Analysis

1. **What 3 things need to happen for a fiber analysis to run?**

-
-
-

2. **What is the definition of the value of 0.5 on an Average fiber orientation plot with the tensor set to First principal, on a Fusion or midplane model?**

-

Evaluation Sheet - Fiber Flow Analysis

1. What 3 things need to happen for a fiber analysis to run?

- A Flow and Fiber license have to be available.
- A fiber filled material must be selected.
- The Fiber orientation check box must be checked on the Process settings wizard.

2. What is the definition of the value of 0.5 on an Average fiber orientation plot with the tensor set to First principal, on a Fusion or midplane model?

- A value of 0.5 would indicate the fibers are randomly oriented in the plane of the element.

CHAPTER 16

Results Interpretation

Aim

In this chapter you will review the types of results available from a flow analysis, how they can be manipulated and a basic interpretation of the result.

Why do it

Understanding the results from a flow analysis is one of the most important skills an analyst must possess to determine and solve problems using simulation.

Overview

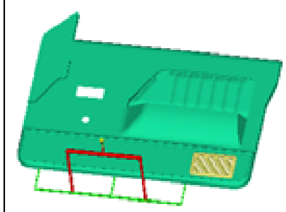
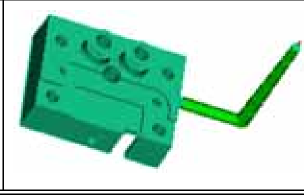
We will first look at the types of results that are available. Each type of result has a default way the result can be plotted and will have different options for manipulating the results. The properties for the result allow us to change how the result can be manipulated. This will be reviewed. Finally, common results from a flow analysis will be discussed to understand its interpretation.

Review of Fusion and midplane models is very similar. 3D results provide considerable more flexibility in how the results are displayed. Examples of viewing Fusion midplane and 3D results will be discussed.

Practice - Results Interpretation

This chapter has two models to choose from and are described below. Pick one to work on.

Table 18: Models used for results interpretation

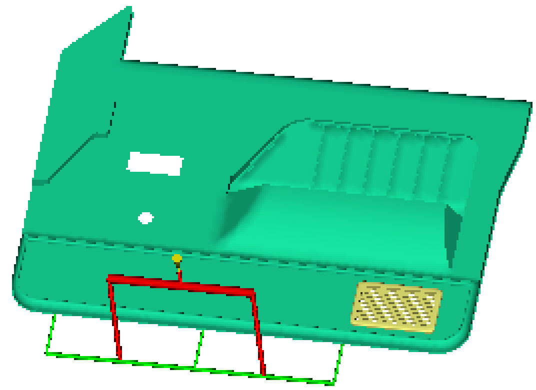
Description	Model
Door panel: starts on page 291	
The door panel is a Fusion model. Review these results to determine if the gating location is acceptable and if the molding conditions should be improved.	
Manifold: starts on page 301	
The manifold is a 3D model. Review the results to determine any problems with packing related issues, any venting issues.	

To save time, the results are provided for both models.

Door panel

Design criteria

The results are of the proposed gate locations. You are to determine the gate's impact of filling and packing. You would like to have the part meet the following requirements:



- Balanced fill.
- Minimum number of weld lines.
- Below the Shear stress limit for the material.
- Uniform volumetric shrinkage.
- Below the clamp force limit of 3000 tons.
- Recommendations are needed on how to proceed.

Setup

To open a project

1. Click the file open icon  or **File** ➡ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Result_Interpretation**.
2. Double click the project file **Result_Interpretation.mpi**.
3. Click **File** ➡ **Preferences** and ensure that **System Units** are set to **Metric**.

To familiarize yourself with the model

1. Double click the **Door Panel** study in the Project View pane.
2. Use the following tools to familiarize yourself with the model.
   - Click the **Rotate** icon .
   - Click the Dynamic Zoom icon .
   - **Navigate** to the **Layers** box. Turn on and off the layers.
   - Click the **Measure** icon  and click twice in different two locations of the model.
   - Use the tools on the **Viewer**, **Viewpoint**, and **Precision view** tool bars, shown in Figure 104, as necessary to aid in viewing the results.

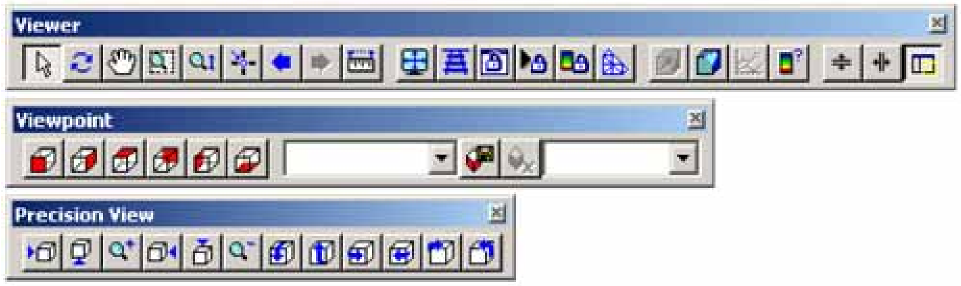


Figure 104: Toolbars used for manipulation

Viewing results

Not all results that are produced are automatically added to the **Study Tasks List**. If a new result is needed, it by one of the following ways:

- Click **Results ➡ Create New Plot**.
- Click the New result plot icon  on the results toolbar.
- Right-click in the Study Tasks list on the Results or folder icons and select New Plot, as shown in Figure 105.

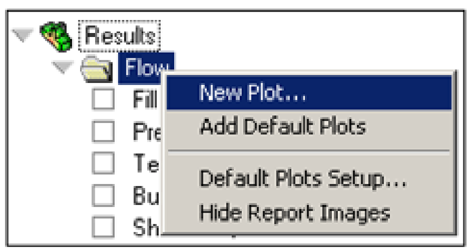


Figure 105: Create a new plot from the study tasks list

To view the material used

1. Right-click on the **Material** icon  in the **Study Tasks** pane.
2. Select **Details** from the context menu.
3. View the following on the **Description** tab.
   - Trade name.
   - Manufacturer.
   - Family abbreviation.
4. View the Maximum shear stress on the Recommended processing tab.
5. Record the following information in Table 19, “Door Panel Worksheet,” on page 298.

To view the process settings

1. Double-click the **Processing Settings** icon in the Study Tasks pane.
2. Record the following information in Table 19, “Door Panel Worksheet,” on page 298.
   - Mold temperature.
   - Melt temperature.
   - Injection time.
   - Switchover method and value.
   - Pack profile.
   - Cooling time.
3. Click **Cancel** on the Process Settings wizard when done.

To view the screen output log file

1. Click the **Logs** box in the Study tasks list.
2. Click the Screen Output tab (if necessary to display the log).
3. Record the following information in Table 19, “Door Panel Worksheet,” on page 298.
   - V/P Switchover time [Sec.].
     - Found in the filling phase progress table, heading shown in Figure 106, with a **V/P** in the status column.
   - End of fill time [Sec].
     - Found in the filling phase progress table, with **Filled** in the status column.
   - End of pack time [Sec].
     - Found in the packing phase progress table, with **Pressure released** in the status column.
   - End of Cooling time [Sec].
     - Found in the packing phase progress table, the time on the last line.
   - Maximum pressure [MPa].
     - Normally the highest pressure is at the V/P switchover. As a result, it can be found on the filling phase progress table, with a **V/P** in the status column. It is specifically listed in the **Filling phase results summary** section.
   - Pack pressure [MPa].
     - Found in the filling phase progress table and packing phase progress table. It is the pressure value that is constant for most of the times listed after the V/P switchover and before the pressure is released.
   - Clamp force maximum [tonnes].
     - Found in the Packing phase results summary section.

4. Click the Logs box to close the log files.

```
 Filling phase:    Status: V  = Velocity control
                           P  = Pressure control
                           V/P= Velocity/pressure switch-over
|--------------------------------------------------------------|
|  Time  | Volume|  Pressure   | Clamp force|Flow rate|Status |
|  (s)   |  (%)  |    (MPa)    |  (tonne)   |(cm^3/s) |       |
|--------------------------------------------------------------|
```

Figure 106: Heading for the filling phase progress table in the screen output file

To display the fill time

1. Ensure the following layers are on:

- Main body.
- Grill.
- Runners.
- Hot Drops.

2. Click the **Fill time** plot from the study tasks list.

3. **Right-click** the plot name and select **Properties**.

3.1. Click the **Method tab.**

3.2. Click the radio button **Contour.**

3.3. Click **OK**.

- The display changes from a shaded image to contour lines so changes in velocity are easier to see.

4. Animate the results.

- If you do not have the Animation tools displayed, click **View ➨ Toolbars ➨ Animation**.

4.1. Manipulate the results as necessary to understand the results.

5. Record information in Table 19, “Door Panel Worksheet,” on page 298 regarding the filling of the part.

To display the Pressure at V/P switchover

1. Click **Pressure at V/P switchover** in the results list.

- Is the V/P switchover balanced?
- Record information in Table 19, “Door Panel Worksheet,” on page 298 regarding the V/P switchover pressure of the part.

To display the Pressure at end of fill

1. Click **Pressure at end of fill** in the results list.
   - Is the V/P switchover balanced?
   - Record information in Table 19, “Door Panel Worksheet,” on page 298 regarding the end of fill pressure of the part.

To display the pressure

1. Click **Pressure** in the results list.
2. Animate the results.
   - What problems do you notice?
   - Record your observations in Table 19, “Door Panel Worksheet,” on page 298.

To display the bulk temperature at end of fill

1. Click **Bulk temperature at end of fill** in the results list.
2. Animate the results.
3. What problems do you notice?
4. Record your observations in Table 19, “Door Panel Worksheet,” on page 298.
5. Turn off the **Grill** layer.
   - The results will automatically re-scale based on the visible layers.
6. Record your observations in Table 19, “Door Panel Worksheet,” on page 298.

To display weld lines

1. Click **Weld Lines** in the results list.
1. Record your observations in Table 19, “Door Panel Worksheet,” on page 298.

To display air traps

1. Click **Air traps** in the results list.
2. Record your observations in Table 19, “Door Panel Worksheet,” on page 298.

To display frozen layer fraction at end of fill

Click Frozen layer friction at end of fill in the results list.

3. Record your observations in Table 19, “Door Panel Worksheet,” on page 298.

To display frozen layer fraction

Click Frozen layer friction in the results list.

4. **Right-click** the plot name and select **Properties**.
   - **4.1.** Click the **Optional settings tab.**
   - **4.2.** Uncheck **Nodal averaged.**
   - **4.3.** Click **OK**.
5. Zoom up on one of the gates.
6. Animate the results.
7. Watch the time when the gate is frozen, (Frozen layer fraction is 1.0).
8. Does the gate freeze before or after packing pressure is released?
9. Record your observations in Table 19, "Door Panel Worksheet," on page 298.

To display time to freeze

1. Ensure the Runners and Hot drop layers are off.
2. Click **Time to Freeze** in the results list.
   - Is the part frozen at the end of the cycle?
   - Are there any problems? What?
3. Record your observations in Table 19, "Door Panel Worksheet," on page 298.

To display shear stress at wall

1. Click **Shear stress at wall** in the results list.
   - Note the maximum shear stress value.
   - Is it above the shear stress limit?
2. Check the shear stress limit from the details for the material.
3. **Right-click** the plot name and select **Properties**.
   - **3.1.** Click the **Scaling** tab.
   - **3.2.** Click **Specified**.
   - **3.3.** Enter the shear stress limit from the database in the **Min.** field.
   - **3.4.** Uncheck **Extended color**.
   - **3.5.** Click the **Mesh Display** tab.
   - **3.6.** Click **Transparent,** as the element surface display.
   - **3.7.** Click the **Optional settings tab.**
   - **3.8.** Uncheck **Nodal averaged.**
   - **3.9.** Click **OK**.
4. Animate the display.
5. Is the shear stress during fill, the highest value during cycle?

6. Where is the area of highest stress during fill?
7. Is it a major problem?
8. Record your observations in Table 19, “Door Panel Worksheet,” on page 298.

To display clamp force

1. Click **Clamp force: XY Plot** in the results list.
2. Use the query tool to:
   - Determine when the max clamp force was reached.
   - Is the maximum clamp tonnage above the machine limit?

To display volumetric shrinkage at ejection

1. Click **Volumetric shrinkage (at ejection)** in the results list.
2. Display the plot with and without the grill layer.
3. Is the majority of the part uniform?
4. Why is there higher shrinkage by the gates than further in the part?
5. Record your observations in Table 19, “Door Panel Worksheet,” on page 298.

Door panel results worksheet

Table 19: Door Panel Worksheet

Location /variable	Observations
Select Material	
Family abbreviation	
Manufacturer	
Trade Name	
Shear Stress limit [MPa]	
Process Settings	
Mold Temperature [°C]	
Melt Temperature [°C]	
Injection Time [Sec.]	
Switchover	
Pack Profile	
Cooling time	
Screen Output	
V/P Switchover time [Sec.]	
End of Fill time [Sec.]	
End of Pack time [Sec.]	
End of Cooling time [Sec.]	
Maximum Pressure [MPa]	
Pack Pressure [MPa]	
Clamp force Maximum [tonnes]	

Table 19: Door Panel Worksheet

Location /variable	Observations
Study Tasks List	
Fill Time [Sec.] Balanced? Y / N, Why? Other problems?	
Pressure at V / P Switchover [MPa], Balanced? Y / N, Why?	
Pressure at End of Fill, Balanced? Y / N, Why?	
Pressure, animated through time, Problems?	
Bulk Temperature at EOF [ºC] Range min. / max with grill.	
Bulk temperature at EOF [ºC] Range min. / max without grill, Problems?	
Weld line locations & Air traps, Where?	
Frozen layer Fraction EOF Range min. / max	
Frozen layer Fraction When do the gates freeze? Is this before the pressure is released? Problems?	
Time to Freeze [Sec.] range min. / max, (without runners) Problems?	

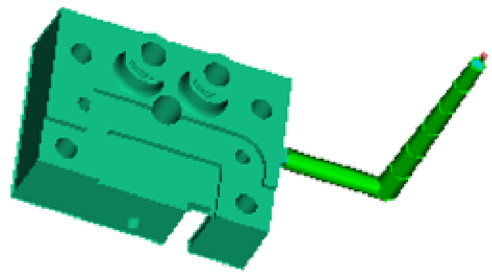
Table 19: Door Panel Worksheet

Location /variable	Observations
Shear Stress at wall [MPa] Areas above limit during fill? Where?, Problems?	
Clamp Force XY Plot, Max clamp force? Does the max force occur during fill or pack?	
Volumetric shrinkage (at ejection), range min./max with grill, Problems?	
Summary of what to correct	

Answers to this worksheet can be found in Table 21 on page 316.

Manifold

Design criteria



The analysis is to determine if there are any weld lines or air traps in critical areas and determine the most likely location for sinks or voids for the part. You would like to have the part meet the following requirements:

- Balanced fill.
- Minimum number of weld lines.
- Below the shear stress limit for the material.
- Uniform volumetric shrinkage.
- No voids.
- Air traps in ventable areas.

Recommendations are needed on how to proceed.

Setup

To open a project

1. Click the file open icon  or **File** ➨ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Result_Interpretation**.
2. Double click the project file **Result_Interpretation.mpi**.
3. Click **File** ➨ **Preferences** and ensure that **System Units** are set to **Metric**.

To familiarize yourself with the model

1. Double click the **Manifold** study in the Project View pane.
2. Use the following tools to familiarize yourself with the model.

   - Click the **Rotate** icon .
   - Click the Dynamic Zoom icon .
   - **Navigate** to the **Layers** box. Turn on and off the layers.
   - Click the **Measure** icon  and click twice in different two locations of the model.
   - Use the tools on the **Viewer**, **Viewpoint**, and **Precision view** tool bars, shown in Figure 107, as necessary to aid in viewing the results.

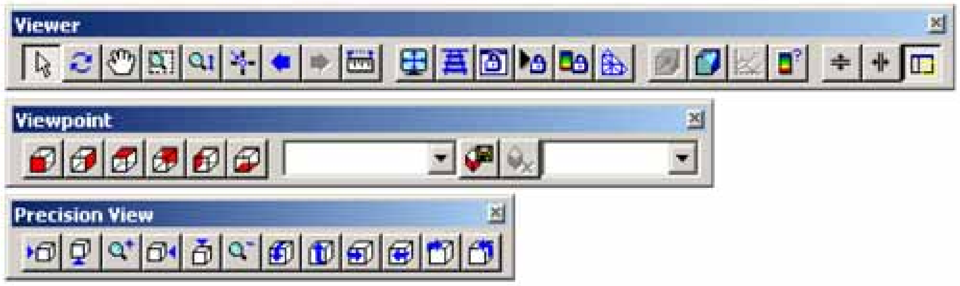


Figure 107: Toolbars used for manipulation

Viewing results

Not all results that are produced are automatically added to the **Study Tasks List**. If a new result is needed, it by one of the following ways:

- Click **Results** ➡ **Create New Plot**.
- Click the New result plot icon  on the results toolbar.
- Right-click in the Study Tasks list on the Results or folder icons and select New Plot, as shown in Figure 108.

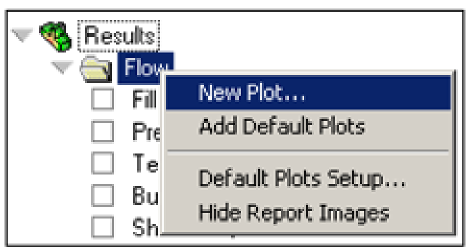


Figure 108: Create a new plot from the study tasks list

To view the material used

1. Right-click on the **Material** icon  in the **Study Tasks** pane.
2. Select **Details** from the context menu.
3. View the following on the **Description** tab.
   - Trade name.
   - Manufacturer.
   - Family abbreviation.
4. Record the following information in Table 20, “Manifold worksheet,” on page 313.

To view the process settings

1. Double-click the **Processing Settings** icon in the Study Tasks pane.
2. Record the following information in Table 20, “Manifold worksheet,” on page 313.
    - Mold temperature.
    - Melt temperature.
    - Injection time.
    - Switchover method and value.
    - Pack profile.
    - Cooling time.
3. Click **Cancel** on the Process Settings wizard when done.

To view the screen output log file

1. Click the **Logs** box in the Study tasks list.
2. Click the Screen Output tab (if necessary to display the log).
3. Record the following information in Table 20, “Manifold worksheet,” on page 313.
    - V/P Switchover time [Sec].
        - Found in the filling phase progress table, heading shown in Figure 109, with a **V/P** in the status column.
    - End of fill time [Sec].
        - Found in the filling phase progress table, with **Filled** in the status column.
    - End of pack time [Sec].
        - Found in the packing phase progress table, when the pressure goes to zero.
    - End of Cooling time [Sec].
        - Found in the packing phase progress table, the time on the last line.
    - Maximum pressure [MPa].
        - Normally the highest pressure is at the V/P switchover. As a result, it can be found on the filling phase progress table, with a **V/P** in the status column. It is specifically listed in the **End of filling phase results summary** section.
    - Pack pressure [MPa].
        - Found in the filling phase progress table and packing phase progress table. It is the pressure value that is constant for most of the times listed after the V/P switchover and before the pressure is released.
4. Click the Logs box to close the log files.

```
Filling Phase:          Status: V   = Velocity control
=============                   V/P = Velocity/pressure switch-over
                                P   = Pressure control
|------------------------------------------------------------------|
| Time   | Fill Vol| Inj Press | Clamp F | Flow Rate| Frozen |Status|
|  (s)   |   (%)   |   (MPa)   |(tonne)  | (cm^3/s) | Vol (%)|      |
|------------------------------------------------------------------|
```

Figure 109: Heading for the filling phase progress table for 3D in the screen output file

To display the fill time

1. Ensure the following layers are on:
   - Part.
   - Runners.
2. Click the **Fill time** plot from the study tasks list.
3. **Right-click** the plot name and select **Properties**.

   3.1. Click the **Optional settings tab.**

   3.2. Select **Banded** in the Color frame.

   3.3. Click **OK**.

   - The display changes from a smooth shaded image to banded colors so changes in velocity are easier to see.
4. Animate the results.
   - If you do not have the Animation tools displayed, click **View ➨ Toolbars ➨ Animation**.

   4.1. Manipulate the results as necessary to understand the results.

   - Look carefully for the formation of weld lines or air traps.

> To get more resolution in a particular area, the plot properties can be manipulated in two ways.
>
> The number of frames can be increased on the Animation page.
>
> - The scale can be reduced so the Min. and Max are just large enough to see the area of interest.
>
> The second method is best because less ram is used.

5. Record information in Table 20, “Manifold worksheet,” on page 313 regarding the filling of the part.

To display air traps

1. Click **Air traps (3D)** in the study tasks list.
2. Carefully view the air trap results.
3. Zoom in on specific areas as necessary to understand the location and any problems related with the air traps.
   - Can they all be vented?
4. Record information about any problems related to air traps, in Table 20, “Manifold worksheet,” on page 313:

To display the pressure

1. Click on **Pressure** in the study tasks list.
2. Animate the plot.
   - The result can be scaled to better interpret the result.
3. Click on **Pressure at injection location:XY Plot**.
   - Use this plot to look at the pressure gradient in the part and feed system.
4. Record information about the pressure, in Table 20, “Manifold worksheet,” on page 313.

To display temperature at flow front

1. Click on **Temperature at flow front** in the study tasks list.
2. Animate the plot.
3. Determine what is the temperature of the polymer entering the part.

   3.1. Step through the animation frames one at a time until the gate is plotted.

   3.2. Note the time when the gate is plotted.
   - How much of the part has a flow front temperature above the temperature entering the part?
   - What does this indicate about the injection time?

> For more resolution on the time, the scale can be reduced centered around the temperature you think the gate is at. For instance, with the default settings you estimate the temperature at the gate to be 300ºC, set the scale to 295ºC to 305ºC and have 20 frames. The resolution will then be 0.5ºC.

To display freeze time

1. Click on **Freeze time** in the study tasks list.
2. Click on the **Edit cutting plane** icon .

   2.1. Check the **Plane ZX**.

   2.2. Flip the plane as necessary so the part missing is the **–Y’ve** side of the part.

3. Animate the plot.

- The plot will show a hole for parts of the cross section that do NOT have results.
- This indicates the part is NOT frozen.

4. Manually animate the plot until the time is ~10 seconds.

5. Click the **Move Cutting Plane** icon .

5.1. Click off the **Show active plane** box.

5.2. Enter **1** in the **Distance** field.

5.3. Check **Animation**.

5.4. Watch the animation.

5.5. Uncheck the Animation box to stop the animation.

5.6. Manually animate the plot until the time is ~16 seconds.

5.7. Animate the cutting plane again and notice how much of the part is still not frozen.

5.8. When done, stop animating the cutting plane and turn it off so the whole part can be seen.

6. Open the plot properties for the Freeze time plot.

6.1. Click the **Methods** tab.

6.2. Click **Contour** radio button.

6.3. Click **Single contour.**

6.4. Click **OK**.

6.5. Animate the plot.

6.6. The area plotted indicates the time at any given step.

6.7. This can be used to see which area(s) are the last to freeze.

6.8. Record information about the Freeze time, in Table 20, “Manifold worksheet,” on page 313.

To display shear rate (3D)

1. Click on **Shear Rate (3D)** in the study tasks list.

2. Click on the **Edit cutting plane** icon .

2.1. Check the **Plane ZX**.

2.2. Flip the plane as necessary so the part missing is the **–Y’ve** side of the part.

3. Animate the plot.

- The highest shear rate in the gate, and is very localized. The shear rate in the part is well below the maximum value.

4. Manually animate the plot until the time is ~0.49 seconds.

5. Click the Move Cutting Plane icon.

5.1. Uncheck **Show active plane**.

5.2. Enter **0.5** in the **Distance** field.

5.3. Check **Animation**.

5.4. Watch the animation.

5.5. Stop the animation by un-checking the Animation box.

- Virtually all the part is dark blue because the scale is controlled by a very high local shear rate in the gate.

6. Open the plot properties for the shear rate, maximum plot.

6.1. Click the **Scaling** tab.

6.2. Click the **Specified** radio button.

6.3. Enter **500** in the Max field.

6.4. Click **OK**

6.5. Animate the cutting plane.

➤ The plot will show in red all areas of the part that have a shear rate of 500 1/sec. or higher.

- Stop the cutting plane animation.
- Move the cutting plane to a location of interest.

7. Animate the shear rate plot.

- When done, turn off the cutting plane.

8. Open the **plot properties** for the shear rate plot.

8.1. Click the **Methods** tab.

8.2. Click the **Contour** radio button.

8.3. Click **Single** contour.

8.4. Enter **500** for the contour value.

8.5. Click the **Scaling** tab.

8.6. Click the **All Frames** radio button.

8.7. Click **OK**

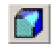
9. Animate the plot.

- The area plotted indicates the location where the shear rate gets to 500 1/sec. shear rate or higher.
- Record information about the Shear rate, in Table 20, "Manifold worksheet," on page 313.

The same type of result manipulation done for shear rate, maximum can be done for **Shear rate, maximum** also.

To display temperature (3D)

1. Click the **Temperature (3D)** plot in the study tasks list.

2. Click on the **Edit cutting plane** icon .

   2.1. Check the **Plane ZX**.

   2.2. Flip the plane as necessary so the part missing is the **–Y've** side of the part.

3. Animate the plot.
   - Watch how the temperature falls in the cross section.
   - When done stop the animation.

4. Open the plot properties for the temperature (3D) plot.

   4.1. Click the **Optional Settings** tab.

   4.2. Uncheck the **Cut with capping** box.

   4.3. Click **OK**.

   - This will display only the cross section and not information behind it. In some cases this aids in the interpretation because the cross section is easily seen.

5. Click the Move Cutting Plane icon.

   5.1. Click off the **Show active plane** box.

   5.2. Enter **0.5** in the **Distance** field.

   5.3. Click the **Animation** box.

   5.4. Watch the animation.

   5.5. Uncheck the Animation box, to stop the animation.

   5.6. Turn off the cutting plane when done.

6. Right-click on the material icon  in the study tasks list and select details.

   6.1. Click on the **Rheological Properties** tab.

   6.2. Note the Transition temperature. ______________________

   6.3. Click on the **Recommended Processing** tab.

   6.4. Note the Ejection temperature. ______________________

   6.5. Click **OK**

7. Open the plot properties for the **Temperature (3D)** plot.

   7.1. Click the **Methods** tab.

   7.2. Click **Contour** radio button

   7.3. Click **Single contour**.

   7.4. Enter the transition temperature for the contour value.

   7.5. Click **OK.**

8. Animate the result.
   - This plot shows the areas of the part that are above the transition temperature.
   - What is the first time step when there are no contours in the part?
   - What time step does the part and runner get separated from each other, indicating the feed system cross section has frozen off?
   - How does this relate to the packing time?
9. Set the single contour to the ejection temperature.
   - Animate the result.
   - Does all of the part get to the ejection temperature at the end of the cycle? If so when?
10. Record information about the Temperature (3D), in Table 20, “Manifold worksheet,” on page 313.

To display velocity (3D)

1. Click on **Velocity (3D)** in the study tasks list.
2. Animate the plot.
   - The maximum value on the scale is dominated by a small localized area.
   - The scale should be set.
3. Open the plot properties for the Velocity (3D) plot.

   3.1. Click the **Scaling** tab.

   3.2. Click the **Specified** radio button.

   3.3. Enter **50** in the Max field.

   3.4. Click **OK**

   ➤ Any value at 50 of higher is plotted in Red with the Extended color on.
4. Animate the result.
   - Now that the scale is smaller, the differences in velocity are more apparent.
   - If desired, change the scale to a lower maximum value and plot it again.
5. Turn on two cutting planes.

   5.1. Click the bottom view icon on the viewpoint toolbar.

   ➤ This will rotate the part to -90 0 0.

   5.2. Click the **Edit Cutting Plane** icon .

   5.3. Check the **Plane ZX**.

   5.4. Click the **New** button.

   ➤ This will create a new cutting plane with the screen being the plane. Since the rotation is -90, 0, 0, the new plane will be parallel to the ZX plane.

   ➤ The part may have disappeared when the second plane was created.

6. Move the new Cutting Plane.

   6.1. Ensure the **Show active plane** box is checked.

   6.2. Rotate the part using one of the following methods:

   - Rotate command programmed to one of the mouse buttons.
   - The rotate buttons on the Precision view toolbar.

- This will allow you to keep the Move Cutting plane dialog on the screen.

   6.3. Click and drag the left mouse up and down to change the location of the second plane. Drag the plane so it has a narrow slice in the part, similar to Figure 110.

   - In many cases, having two parallel cutting planes will make looking at velocity easier.

   6.4. Animate the Velocity (3D) plot on the narrow slice.

7. Record information about the Freeze time, in Table 20, “Manifold worksheet,” on page 313.

8. Click the **Edit Cutting Plane** icon .

   8.1. Uncheck all the planes so the full part can be seen.

   8.2. Click **Close**.

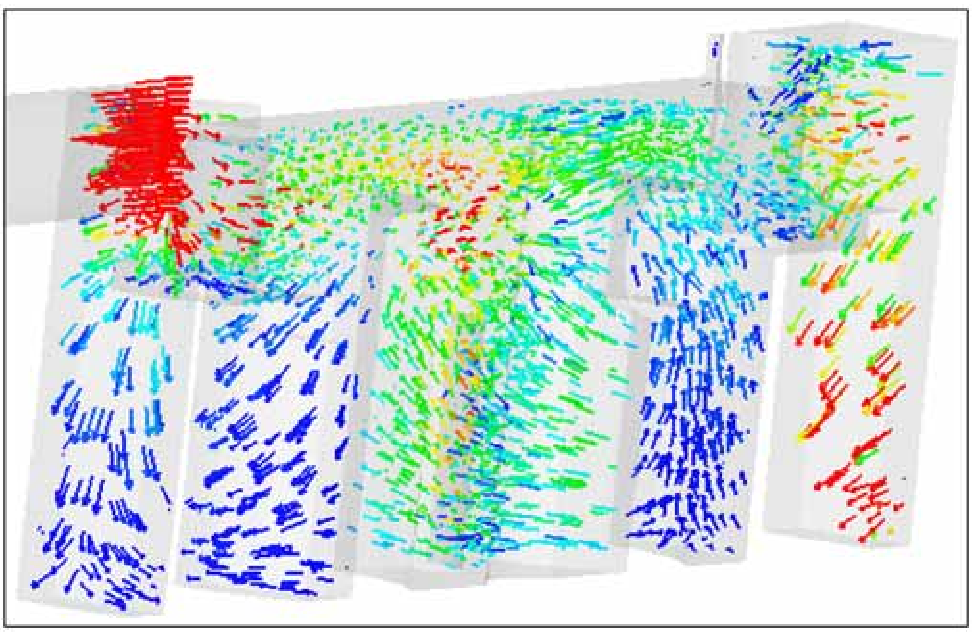


Figure 110: Velocity plot with two cutting planes

To display volumetric shrinkage (3D)

1. Click on **Volumetric Shrinkage (3D)** plot in the study tasks list.
2. Open the plot properties for the **Volumetric Shrinkage (3D)** plot.

   2.1. Click the **Scaling** tab.

   2.2. Click the **Specified** radio button.

   2.3. Enter **17** in the Max field.

   2.4. Click **OK**.

   - The only area that is above 17% is the sprue.
   - Click the **Methods** tab.

   2.5. Click **Contour** radio button

   2.6. Click **Single contour**.

   2.7. Enter **16** for the contour value.

   2.8. Click **OK**.

   - The plot shows the isolated areas that have shrinkage higher than 16%.
3. Open the plot properties for the **Volumetric Shrinkage (3D)** plot.
   - Click the **Methods** tab.
   - Click the **Shaded** radio button
   - Click **OK**.
4. Click on the **Edit cutting plane** icon .

   4.1. Check the **Plane ZX**.

   4.2. Flip the plane as necessary so the part missing is the **–Y've** side of the part.

   4.3. Ensure the **Plane ZX** is highlighted and click **Make Active**.
5. Move the cutting plane.

   5.1. Slide the cutting plane through the part to see the locations of the high volumetric shrinkage.

   5.2. Click the **Edit Cutting Plane** icon .

   5.3. Uncheck all the planes so the full part can be seen.

   5.4. Click **Close**.

6. Open the plot properties for the **Volumetric Shrinkage (3D)** plot.

6.1. Click on the **Animation** tab.

6.2. Set **Single dataset** in the Animate result over frame.

6.3. Set the time to the last one in the list for the Animate result at frame.

6.4. Click the **Value range** radio button, and set the value to **1**.

6.5. Click **Current frame only** radio button.

6.6. Click the **Scaling** tab.

6.7. Click the **Specified** radio button.

6.8. Enter **3** in the **Min.** field, and **17** in the **Max** field.

6.9. Click **OK**.

7. Click the **Step forward** icon  on the animation toolbar and go through all the steps.

- This plot method is similar to the single contour, but it allows you to animate through the range of values.
- It only works for single dataset results so the plot type and time had to be set.
- The value range was set to a round number, and the scale was set to a multiple of the value range so the display would be in round numbers.
- Interpret the results.
- Notice how most of the part is near the low end of the volumetric shrinkage scale.
- The highest shrinkages are in the center of the cross sections.
- How do you think the high volumetric shrinkage relates to possible void locations?

8. Record information about the volumetric shrinkage, in Table 20, “Manifold worksheet,” on page 313.

Manifold results worksheet

Table 20: Manifold worksheet

Location /variable	Observations
Select Material	
Material abbreviation	
Material Manufacturer	
Material Trade Name	
Process Settings	
Mold Temperature [°C]	
Melt Temperature [°C]	
Injection Time [Sec.]	
Switchover	
Pack Profile	
Cooling time	
Screen Output	
V/P Switchover time [Sec.]	
End of Fill time [Sec.]	
End of Pack time [Sec.]	
End of Cooling time [Sec.]	
Maximum Pressure [MPa]	
Pack Pressure [MPa]	

Table 20: Manifold worksheet

Location /variable	Observations
Study Tasks List	
Fill Time [Sec.] Balanced? Weld lines? Other problems?	
Air traps? Problems?	
Pressure [MPa] Problems?	
Temperature at flow front [ºC] range min./max, Problems?	
Freeze Time, [Sec.] Problems?	
Shear Rate (3D), [1/sec.] Problems?	
Temperature (3D), [°C] Range, Problems?	
Velocity (3D), [cm/s] Problems?	
Volumetric shrinkage (3D) Range min./max Problems?	

Table 20: Manifold worksheet

Location /variable	Observations
Summary of what to correct	

Answers to this worksheet can be found in Table 22 on page 319.

Door panel worksheet answers

Table 21: Door Panel worksheet answers

Location /variable	Observations
Select Material	
Family abbreviation	ABS
Manufacturer	Lanxess
Trade Name	Lustran ABS 648
Shear Stress limit [MPa]	0.3 MPa
Process Settings	
Mold Temperature [°C]	70ºC
Melt Temperature [°C]	260ºC
Injection Time [Sec.]	4.5 Sec.
Switchover	By % volume filled, 95%
Pack Profile	0.5 80% 15.0 80% 4.5 0%
Cooling time	20 Seconds
Screen Output	
V/P Switchover time [Sec.]	~4.7 Sec.
End of Fill time [Sec.]	~5.9 Sec.
End of Pack time [Sec.]	~24.9 Sec.
End of Cooling time [Sec.]	~44.9
Maximum Pressure [MPa]	~100.3 MPa
Pack Pressure [MPa]	~80.3 MPa

Table 21: Door Panel worksheet answers

Location /variable	Observations
Clamp force Maximum [tonnes]	~3100 Tonnes
Study Tasks List	
Fill Time [Sec.] Balanced? Y / N, Why? Other problems?	The door panel is nearly balanced, about 96% by volume filled. There are major weld lines in the grill due to material racing around the grill.
Pressure at V / P Switchover [MPa], Balanced? Y / N, Why?	The pressure at switchover is ~100 MPa and is balanced. With the part rotated at 0 0 0, the upper left (ear) and right corners still need to be filled.
Pressure at End of Fill, Balanced? Y / N, Why?	The pressure at the end of fill is the packing pressure and is ~80 MPa. The pressure is NOT balanced because the ear is the last place to fill. The upper right corner has pressurized to ~36 MPa.
Pressure, animated through time, Problems?	Some of the part is above zero pressure when the pressure has been released (24.9 sec.). This is an indication that the packing may be too short. This is confirmed with the frozen layer fraction plot.
Bulk Temperature at EOF [ºC] Range min. / max with grill.	~98ºC to ~264ºC
Bulk temperature at EOF [ºC] Range min. / max without grill, Problems?	~211ºC to ~264ºC, The grill gets very cold due to the thin walls and being close to the gate. The band of hotter material from the gate to the end of fill shows the dominant flow path. The cold areas between the gate indicate little plastic movement between the gates.
Weld line locations & Air traps, Where?	Most of the air traps are in the grill. There are also weld lines on the back side of the hole.
Frozen layer Fraction EOF Range min. / max	0 to 0.36. The grill is freezing off quickly. Outside the grill, the maximum frozen layer fraction is only 0.05.
Frozen layer Fraction When do the gates freeze? Is this before the pressure is released? Problems?	The gate freezes between 33 and 35 seconds. The pressure is released at 24.9 seconds so the packing is too short and back flow has occurred. This will lead to inconsistent part weights.
Time to Freeze [Sec.] range min. / max, (without runners) Problems?	The range is between 5.2 sec. in the grill, most of the part is between 40 and 50 seconds, and some thick areas take 100 seconds to cool.

Table 21: Door Panel worksheet answers

Location /variable	Observations
Shear Stress at wall [MPa] Areas above limit during fill? Where? Problems?	Only limited areas in the grill are above the shear stress limit during filling. During packing both the grill and areas near the gates are higher than the limit, but typically for a short period of time and not too much above the limit.
Clamp Force XY Plot, Max clamp force? Does the max force occur during fill or pack?	The maximum clamp force is ~3100 tonnes and occurs at ~8 seconds into the cycle so it is during the packing phase.
Volumetric shrinkage (at ejection), range min./max with grill, Problems?	The range of shrinkage is from ~0.5% to ~7.4%. The highest shrinkage is in the thick area above the arm rest. The shrinkage is typically higher at the end of fill and lower as it gets close to the gate. The shrinkage goes up close to the gate due to the short packing time relative to gate freeze.
Summary of what to correct	The injection time may be a bit fast due to the increase in flow front temperature away from the gate. A slower injection time may help lower the fill pressure although it is not too high. Lowering the packing pressure and increasing the packing time may be warranted as well. Lower the pack pressure to get the clamp force down, and increase the pack time so the gates freeze under pressure. The packing profile should not be constant. It should decay during the fill to prevent overpacking near the gate. Possibly try to adjust the gate locations to make the filling more balanced, and the grill to fill all from one direction.

Manifold worksheet answers

Table 22: Manifold worksheet answers

Location /variable	Observations
Select Material	
Material abbreviation	PA66
Material Manufacturer	DuPont Engineering Polymers (Moldflow Verified)
Material Trade Name	Zytel 3189 HSL NC010
Process Settings	
Mold Temperature [°C]	70ºC
Melt Temperature [°C]	290ºC
Injection Time [Sec.]	0.5 Sec
Switchover	By % volume filled at 98%
Pack Profile	0 100% 15 100% 5 0%
Cooling time	46 Sec
Screen Output	
V/P Switchover time [Sec.]	~0.49 Sec.
End of Fill time [Sec.]	~0.51 Sec.
End of Pack time [Sec.]	~20.49 Sec.
End of Cooling time [Sec.]	~66.49 Sec.
Maximum Pressure [MPa]	~25.8 MPa
Pack Pressure [MPa]	~25.8 MPa

Table 22: Manifold worksheet answers

Location /variable	Observations
Study Tasks List	
Fill Time [Sec.] Balanced? Weld lines? Other problems?	The filling of the part is well balanced. There are weld lines around all the holes but none should cause a particular problem.
Air traps? Problems?	All the air traps are in locations that can be vented at the parting line, on the sides of cores or on slides.
Pressure [MPa] Problems?	The pressure to fill is very low @ ~25 MPa. The pressure to fill just the part is only about 5 MPa due to the thick walls in the part. The feed system has a high drop due to the relatively restrictive area and fast injection time.
Temperature at flow front [ºC] range min./max, Problems?	Range is ~237ºC to ~312ºC. The temperature entering the part is about 296ºC. Only very limited areas are below this temperature and is no problem. The part gains temperature due to the fast fill time. Possibly, slightly fast.
Freeze Time, [Sec.] Problems?	Much of the part is frozen in under 15 seconds, however, there are many thick areas that take up to 30 seconds to cool and limited areas over 60 seconds. This part is too Chunky.
Shear Rate (3D), [1/sec.] Problems?	The maximum shear rate is a very limited area in the gate. The vast majority of the part has a shear rate below 500 1/sec.
Temperature (3D), [°C] Range, Problems?	Due to chunky wall thickness in the part, there are many areas that are not below the transition temperature at the end of the packing. Due to the long cycle time, the part is completely below the ejection temperature, However, most of the part is below the ejection temperature at half the cycle time.
Velocity (3D), [cm/s] Problems?	No problems highlighted. It does show however where material is moving quickly towards the last place to fill.
Volumetric shrinkage (3D) Range min./max Problems?	The maximum volumetric shrinkage is ~17%. The volumetric shrinkage is lowest at the mold wall and highest in the center of the cross section. Some of the thinnest walls have a maximum shrinkage of about ~9%. There are significant areas that have shrinkage above 13%.

Table 22: Manifold worksheet answers

Location /variable	Observations
Summary of what to correct	The primary problem to fix is the volumetric shrinkage. This must be fixed by increasing the packing pressure. The packing pressure is currently equal to the fill pressure, but due to the low pressure drop in the part, caused by the thick walls, the pack pressure is too low. The results also suggest the re-designing the part to get the walls more uniform in thickness may be warranted.

Competency check - Results Interpretation

1. What type of result in the Fusion result called “Temperature”? How can the type be determined?
2. What is the best result to determine when the gate is no longer open so the cavity can’t be packed any more, for a Fusion part? For a 3D part?
3. What is the interpretation of Normalized thickness at the following values: **0.0 on a Fusion model.** ____ **1.0 on a Fusion model**. ____ **-1.0 on a Fusion model.** ____ **1.0 on a midplane model.** ____ **1.0 on a midplane model.** ____

Evaluation sheet - Results Interpretation

1. **What type of result in the Fusion result called "Temperature"? How can the type be determined?**

   The Fusion result temperature is an intermediate profiled result. This is determined by looking at the Animate result over options on the Animation tab of the plot properties.

2. **What is the best result to determine when the gate is no longer open so the cavity can't be packed any more, for a Fusion part? For a 3D part?**

   For Fusion, look at the result Frozen layer fraction. Set the Nodal averaging off. Animate the result and determine the time when the gate goes to a value of 1.0

   For 3D, use the temperature plot. Set the method to single contour with the value the material's transition temperature. Animate the result to find the time when the contour separates between the runner and the part.

   NOTE for both techniques, the resolution of the result is based on the number of intermediate results specified for the analysis.

3. **What is the interpretation of Normalized thickness at the following values:**

   0.0 on a Fusion model. This is the center of the plastic cross section.

   1.0 on a Fusion model. This is the element being viewed or picked.

   -1.0 on a Fusion model. This is the matched element from the one being viewed or picked.

   1.0 on a midplane model. This is the top (red) side of the element.

   1.0 on a midplane model. This is the bottom (blue) side of the element.

CHAPTER 17

Gate & Runner Design

Aim

The aim of this chapter is to review the types of gate and runner designs, and to model multiple cavity tools using two different techniques. Finally, an automatic runner balance will be preformed to size the runners, achieving a good balance between the cavities.

Why do it

It is important to understand the different types of gates and runner systems that are available and how to model them. The two methods of creating runner systems will be used so you can see how they differ, and to learn how to do both.

Runner balancing, and sizing are important when optimizing the tool. If a runner is not sized and balanced, there are generally many problems associated with the lack of balance. Problems can include, wasting material, flash, short shots, and warpage etc.

Overview

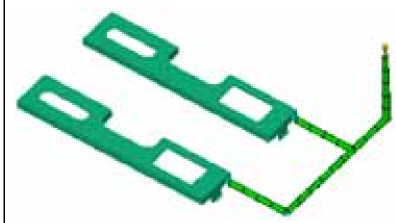
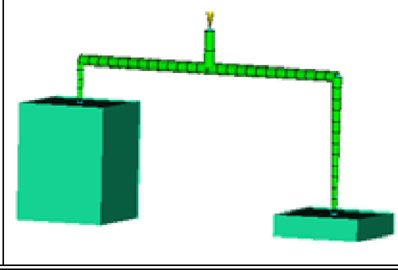
Various gate types and how they are modeled followed by runners will discussed. You will create an eight-cavity tool using the Cavity Duplication Wizard, and manual runner creation techniques.

You will create a second runner system to represent a 2-cavity family tool using the Runner Creation Wizard. You will then size the runners using the runner balance analysis.

Practice - Gate & Runner Design

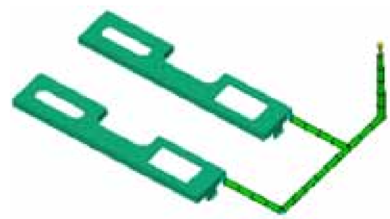
This chapter has two models that will be used in this chapter and are described below.

Table 23: Models used for gate and runner design

Description	Model
Snap cover: starts on page 331	
You will use the cavity duplication wizard to create the multiple parts then create the runners for the snap cover by manual methods.	
Box & Lid family tool: starts on page 341	
You will use the runner system wizard to create the runners for this family tool. Given predetermined input, you will size the runners using the runner balance analysis.	

Snap Cover

Design Criteria



You will create a two cavity model that represents an eight-cavity layout by using occurrence numbers. The gate detail is shown in Figure 111 and the cavity layout is shown in Figure 112. Refer to these drawings as you create the runner system. You are given the study **Snap Cover Runner Modeling** as the starting point. The basic steps you need to do are the following:

1. Use the cavity duplication wizard to create an 8 cavity layout.
2. Delete the cavities that are not needed.
3. Set the occurrence number of the part.
4. Create and mesh the gates.
5. Create and mesh the secondary runners.
6. Create primary runner.
7. Create sprue.
8. Set injection location.

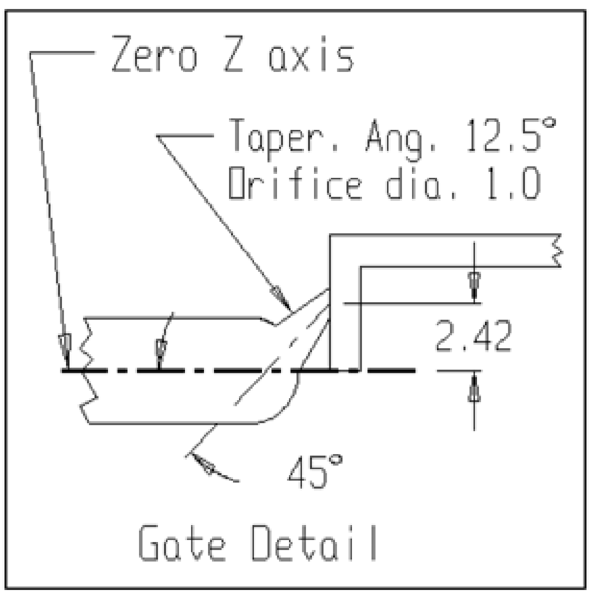


Figure 111: Snap cover gate detail

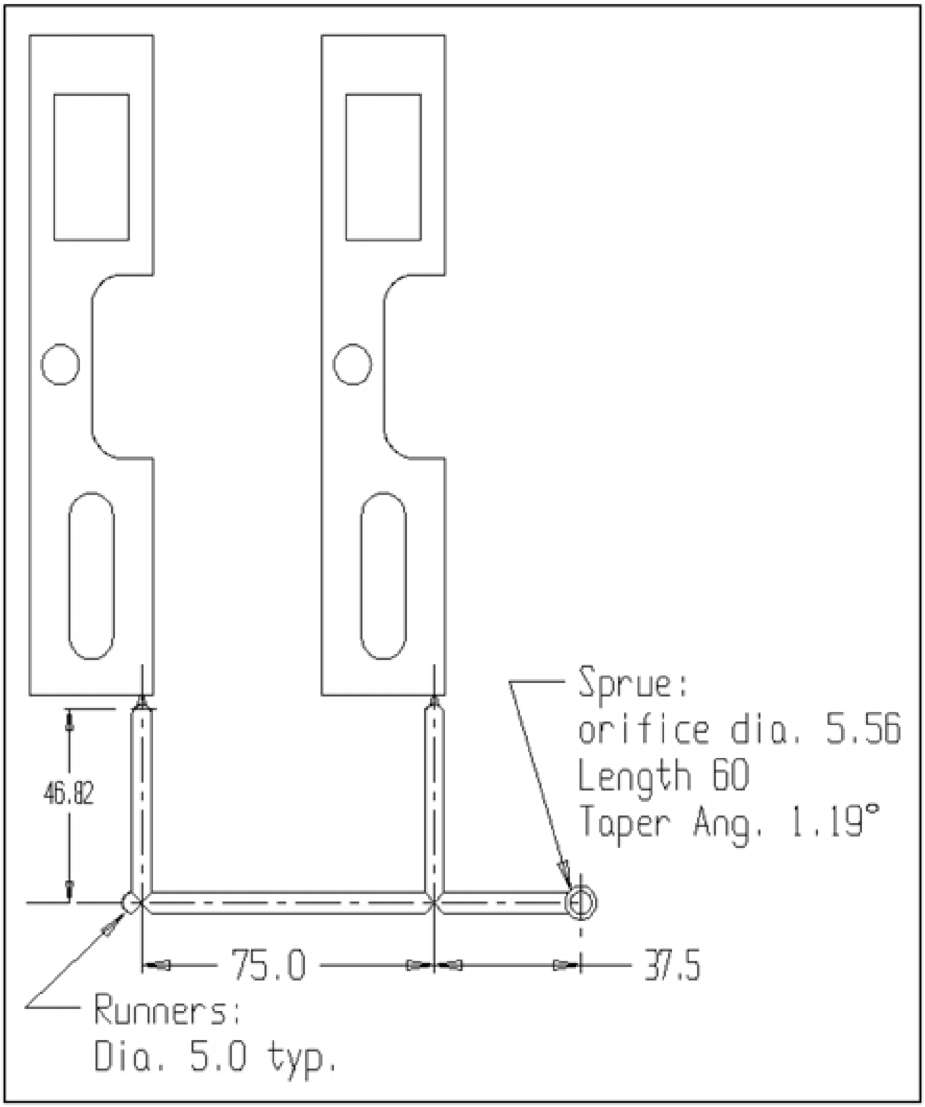


Figure 112: Snap cover runner detail

Setup

To open a project

1. Click the file open icon or **File ➨ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Gate_Run_Design**.
2. Double click the project file **Gate_Run_Design.mpi**.
3. Click **File ➨ Preferences** and ensure that **System Units** are set to **Metric**.

To open the snap cover study

1. Double click the **Snap Cover Runner Modeling** study in the Project View pane.
2. Rotate the model to review the geometry.

To duplicate the cavities

1. Click **Modeling ➨ Cavity Duplication Wizard**.
2. Enter **8** as the number of cavities.
3. Click the **Rows** radio button.
4. Enter **75** as the column spacing.
5. Enter **260** as the Row spacing.
6. Ensure the box **Offset cavities to align gates** is checked.
7. Click the **Preview** button to see the layout.
8. Click **Finish** if the preview looks like Figure 113.

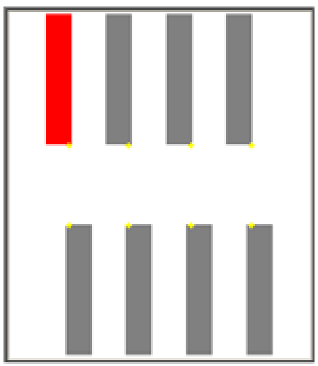


Figure 113: Cavity duplication preview

The column and row spacing are based on center to center dimensions.

The cavity duplication wizard was run to show its use and to show the layout of the eight cavity tool. Six of the eight cavities are going to be deleted. Occurrence numbers will then be used to represent the 6 cavities that were deleted.

To delete the cavities not needed

1. Click the Front view icon  on the viewpoint tool bar to rotate the model to 0 0 0.
2. Click the fit to window icon  to ensure all the cavities are shown on the screen.
3. Right-click on the layers pane and select **Show All Layers**.
4. Band select around all 4 cavities in the bottom row.
5. Hold the **CTRL** key and band-select the right 2 cavities of the upper row.
   - Your selection should look like Figure 114.
6. Click the **Delete** button, on the keyboard.

7. Click **OK.**
   - This accepts the selection of all 4 entity types.
8. Right-click **New Triangles** in the Layer pane and select **Hide all other layers**.

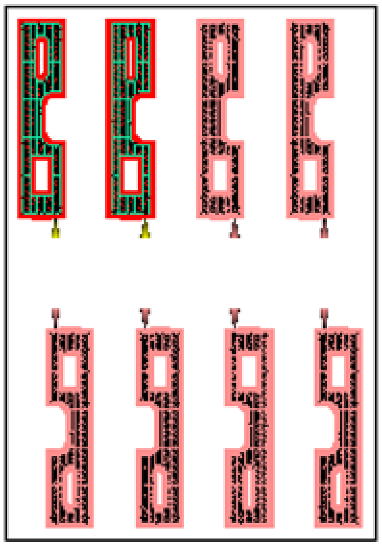


Figure 114: Selected entities

To assign occurrence numbers for part symmetry

1. Band-select both parts.
2. Right-click and select **Properties**.
3. Select all the properties in the **Select Properties** dialog by holding down the shift key and click the last property in the list, and click **OK**.
4. Enter **4** into the **Occurrence number** field.
   - This will both parts' occurrence to 4.
5. Click **OK**.

To prepare for gate modeling

1. Ensure that the **Nodes** and **Part Triangles** layers are checked in the **Layers** dialog.
2. Click the Bottom view button to rotate the part to -90, 0, 0.
3. Zoom in on the injection location of the right part select it and then delete it. Make a mental note of where it was located, you will need this reference later in the exercise.
4. Zoom on the left injection location to delete it. Make a mental note of where it was located, you will need this reference later in the exercise.
   - The injection location was used to mark the location of the gate but is **not** needed when modeling the actual gate.

5. Create a new layer.

5.1. Click the new layer icon  in the Layer pane to create a layer.

5.2. Right click on the new layer and rename it **Gates**.

5.3. Make sure it is the active layer by verifying it is the layer in bold in the layers pane.

> When modeling the gates and runners, refer to the drawings in Figure 111 and Figure 112 to see where the dimensions are derived.

To manually create the gate curve

1. Select the **Tools** pane.

1.1. Select the Create curves icon  in the toolbox.

1.2. Select **Create line**.

2. Set the **Filter** to **Node**.

3. Ensure **Coordinates(x,y,z), First** is active (yellow).

4. Click on the node where the injection location was located.

- Notice the values in the **Coordinates(x,y,z), First** field update with your selected node.

5. Select the **Relative** radio button.

6. Enter 0 –2.42 –2.42 (0 *space* -2.42 *space* -2.42) into the **Coordinates(x,y,z), second** field.

7. Set the gate properties.

7.1. Click the icon .

7.2. Click the **New** button.

7.3. Click **Cold Gate**.

7.4. Select the Cross-section as **Circular**.

7.5. Select the Shape as **tapered (by angle)**.

7.6. Click **Edit dimensions**.

7.7. Enter **1 mm** in the Start diameter field.

7.8. Enter **12.5**° in the Tapered angle field.

7.9. Click **OK**.

7.10. Set the **Occurrence number** to **4**.

7.11. Change the property name to **Cold gate 1mm x 25 deg. Inc**.

7.12. Click **OK** twice.

8. Click **Apply**.
    - Rotate the part as necessary to see the curve.
    - The curve should look like Figure 115.
9. Click **Close**.

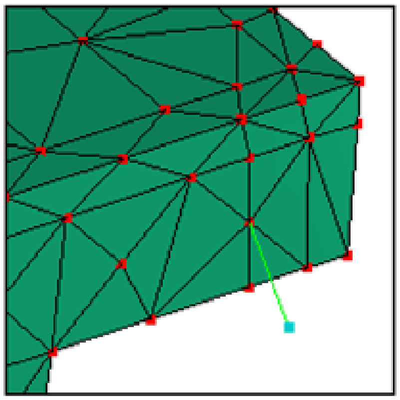


Figure 115: Gate curve created

To copy the gate curve

1. Band select the newly create curve and node at the end of the curve.
    - Ensure there are only two entities in the selection field.
2. Select the Move/copy icon  in the toolbox.

    2.1. Select **translate**.
3. Click in the **Vector** field.
    - The Measurements dialog opens.

    3.1. Click on the gate node, (on the part) for the part that has the curve.

    3.2. Pan or zoom as necessary and find the gate node on the other part.

    3.3. Click on the second gate node.
    - The vector field should have filled in with a value of **75 0 0**.
    - If not, re-select the nodes or manually enter the value.
4. Click the **Copy** radio button.
5. Click **Apply**.
    - The gate curve and node are copied from the original part to the second part.

To generate the mesh

1. Click **Mesh** ➡ **Generate mesh**.
    - You can also click on the tasks tab and double-click on the Fusion mesh icon.
2. Set the **Global edge length** to **0.5 mm**.
3. Click the **Preview** button to see the mesh density.

4. Check **Place mesh on active laye**r box.
5. Click **Mesh Now**.
    - The gates should look like Figure 116.
6. Uncheck the **Logs** box to close the log files.

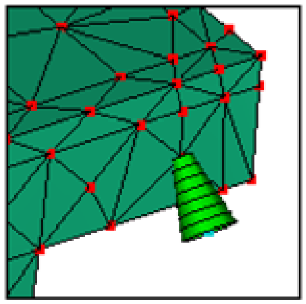


Figure 116: Gate with a mesh

To manually create the runner

1. Create a new layer.
    1.1. Click the new layer icon  in the Layer pane to create a layer.
    1.2. Right click on the new layer and rename it **Runners**.
    1.3. Make sure it is the active layer by verifying it is the layer in bold in the layers pane.
2. Zoom up on one of the gates so you can easily see the node at the end of the gate.
3. Select the **Tools** pane.
    3.1. Select the Create beams/Tris/Tetras icon  in the toolbox.
    3.2. Select **Create Beams**.
4. Set the **Filter** to **Node**.
5. Select the node at the end of the gate, (not touching the part), to fill in the **Coordinates(x,y,z), First** field.
6. Select the same node again to fill in the **Coordinates(x,y,z), Second** field.
7. Change the **Y** value of the **Coordinates(x,y,z)**, **Second** field to **0**.
    - Depending on the part you are working on, the Coordinates(x,y,z), **Second** field value should be either:
        - -112.5 0 0
        - -37.5 0 0
8. Specify **4** beam elements for the runner.

9. Set the Runner properties.

    9.1. Click the icon .
    9.2. Click the **New** button.
    9.3. Click **Cold Runner**.
    9.4. Select the Cross-section as **Circular**.
    9.5. Select the Shape as **Non-tapered**.
    9.6. Click **Edit dimensions**.
    9.7. Enter a diameter of **5 mm**.
    9.8. Click **OK**.
    9.9. Set the **Occurrence number** to **4**.
    9.10. Change the property name to **Cold Runner 5 mm 4 oc**.
    9.11. Click **OK** twice.

10. Click **Apply.**
11. Zoom up on the other gate and build the runner off this gate with the same dimensions as the runner just created.

> Beams can be created 2 ways, by first creating regions then meshing, or by creating an end node(s) then creating the beams directly. The first method is best for tapered geometry so each element will be the correct size

To create the primary runners

1. Select the Create beams/Tris/Tetras icon in the toolbox.
    1.1. Select **Create Beams**.
2. Set the Runner properties.

    2.1. Click the icon .
    2.2. Click the **New** button.
    2.3. Click **Cold Runner**.
    2.4. Select the Cross-section as **Circular**.
    2.5. Select the Shape as **Non-tapered**.
    2.6. Click **Edit dimensions**.
    2.7. Enter a diameter of **5 mm**.
    2.8. Click **OK**.
    2.9. Set the **Occurrence number** to **2**.
    2.10. Change the property name to **Cold Runner 5 mm 2 oc**.
    2.11. Click **OK** twice.

3. Set the **Filter** to **Node**.

4. Rotate the part to about -70 -20 -10. and zoom so you can see both runner ends.
5. Create the primary runners between the secondaries.
   5.1. Click in the **Coordinates(x,y,z), First** field to make it active.
   5.2. Click on the node at the end of the secondary runner that is left most on the screen.
   5.3. It's coordinate should be -112.5 0 0.
   5.4. Click on the node at the end of the secondary runner that is right most on the screen.
   5.5. Its coordinate should be -37.5 0 0.
   5.6. Enter **6** for the **Number of beams** field.
   5.7. Click **Apply**.
6. Create the remaining section of the primary runner.
   6.1. Click in the Coordinates(x,y,z), **Second** field and enter 0 0 0.
   6.2. Enter **3** for the **Number of beams** field.
   6.3. Click **Apply**.
   - The runner system should look like

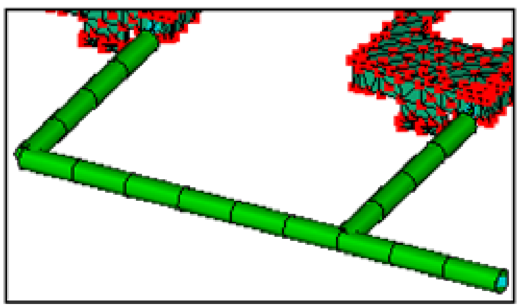


Figure 117: Runners created

Sprues, and other tapered beam sections are created with curves then meshed so each element will have a different diameter. For the sprue, we know the orifice diameter and the taper angle. To model this correctly, we need to create the curve from the top of the sprue to the intersection with the runner.

To create the sprue

1. Select the Create curves icon  in the toolbox.
   1.1. Select **Create line**.
2. Set the **Filter** to **Node**.
3. Ensure **Coordinates(x,y,z), First** is active (yellow).
4. Enter in **Coordinates(x,y,z), First**, **0 0 60**.
   - This is top of the sprue.

5. Set the filter to **Node**.
6. Highlight the **Coordinates(x,y,z), second** field.
7. Select the node at the end of the primary runner.
8. Set the Sprue properties.

   8.1. Click the icon .

   8.2. Click the **Select** button.

   8.3. Select **Cold Sprue.**

   8.4. Click **Cold sprue (7/32”, 1/2” taper per foot (included angle))**.

   8.5. Click **Select**.

   8.6. Click **OK**.
9. Click **Apply** to create the curve.
10. Click **Mesh ➡ Generate mesh**.
    - You can also click on the tasks tab and double-click on the Fusion mesh icon.

    10.1. Set the **Global edge length** to **12 mm**.

    10.2. Click the **Place mesh on active layer** box.

    10.3. Click **Mesh Now.**
    - The finished model should look like Figure 118.
11. Place the injection location.

    11.1. Double click on the Set injection locations in the study tasks list.

    11.2. Select the top of the sprue to place the injection location.

    11.3. Right-click in the display area and select, **Finish set injection locations.**
12. Click the icon  to save the study.

> There are databases for various geometries. The database for the sprue is handy as it has many standard sprue sizes in it. It will generally be faster to set the cross section from a defined database rather than doing it manually.

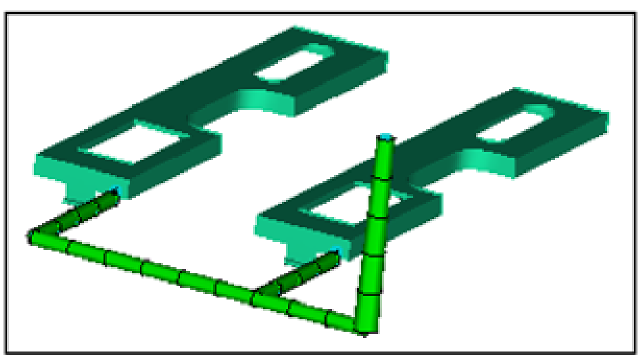
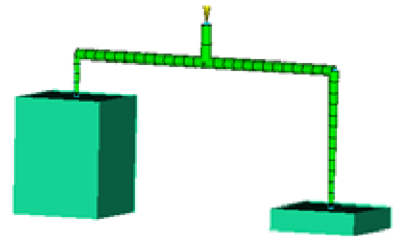


Figure 118: Completed runner system

Box & Lid family tool



You will use the Runner System Wizard to create the feed system for a two cavity family tool. You are given a study with the two parts already in it. Finally, you will balance the runners of the family tool so the parts fill at about the same time. The basic steps you need to balance a runner system are the following:

1. Optimize the part.

- ➤ This is done for you. The gate location and processing conditions are provided. All filling issues for the parts must be done individually for each part.
- ➤ The processing conditions used for each part must be the same. A molding window analysis for both parts was run and the results were compared and one set of conditions were chosen that would work for both parts.

2. Model the runner system.

- ➤ The study given to you has both parts in it. The command **File ➡ Add** was used to get both parts in the same study.

3. Run a fill analysis with the runner system.

- ➤ Based on the optimum fill time of the parts and the volume of the parts, a flow rate is used as input into the analysis.

4. Determine the target pressure.

5. Run the balance analysis.

6. Review the results.

7. Revise the target pressure as necessary and re-run.

8. Round the runner sizes if desired.

Model the Runner System

To open the study

1. Double click on the study **Box Lid**.

2. Use the model manipulation tools in the **Viewer** toolbar to investigate the model geometry.

To build the runner system

1. Click **Modeling ➡ Runner System Wizard**.

- Refer to the drawings shown in Figure 119, and Figure 120 when using the wizard.

2. Click **Center of Gates** to get the sprue position to be **0 X**, and **0 Y**.

3. Enter **85** for the **Top runner plane Z**.

4. Click **Next>**.

5. Enter the **sprue** information

   5.1. Orifice Diameter, **5.56** mm

   5.2. Included angle **2.38º**

   5.3. Length **20** mm

6. Enter the **runner diameter** of **6** mm.

   - The runner for this tool is trapezoidal but is analyzed as round. Once the final sizes are determined, the size of the trapezoidal runner is determined as shown in Figure 120.

7. Enter the **Drop** information.

   7.1. Bottom diameter, **3** mm.

   7.2. Included angle **2.38º.**

8. Click **Next>**.

9. Enter the **Top Gates** information.

   9.1. Start diameter **3** mm.

   9.2. End diameter **1** mm.

   9.3. Length **2** mm.

10. Click **Finish**.

    - The runner system should look like Figure 121.

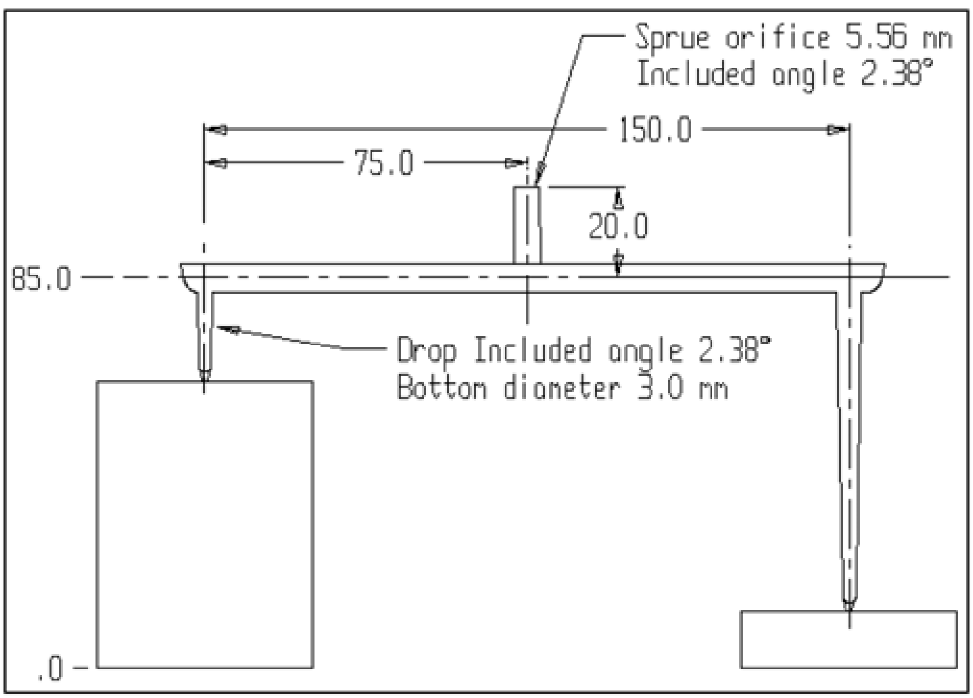


Figure 119: Runner drawing for the box and lid

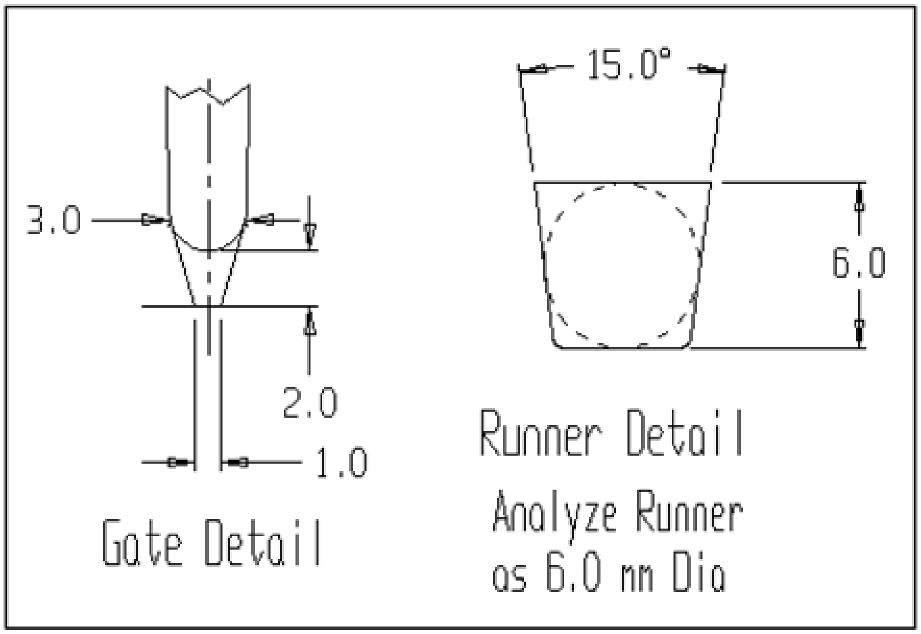


Figure 120: Gate and runner detail for the box and lid

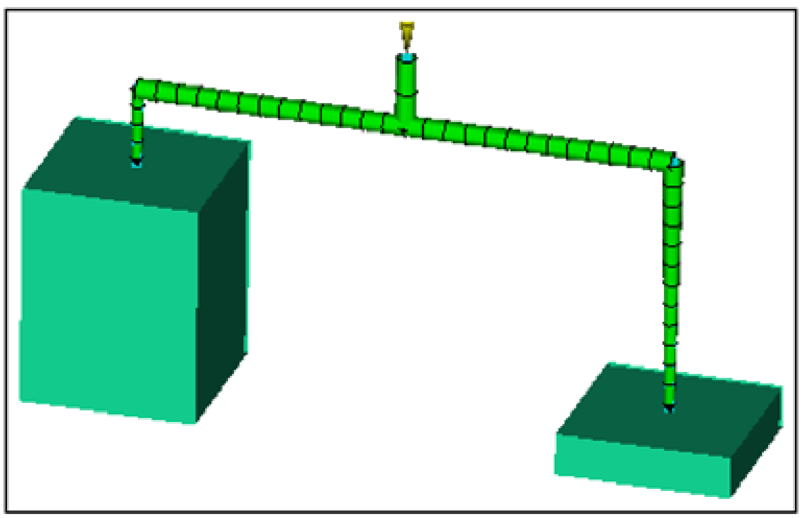


Figure 121: Completed runner system

The drops in the runner system are set to unconstrained. Tapered features of the runner system should not be changed during a runner balance. To prevent the change, set the runner balance constraints to **Fixed** on the drops.

To fix the drop dimensions

1. Click the bottom view icon to rotate the part to -90 0 0.
2. Zoom up on one of the drops.
3. Band-select around all the elements of one of the tapered drops.

4. Right-Click and select **Properties** from the context menu.
5. Select all the properties in the list and click **OK**.
6. Select **Edit dimensions** on the cold runner dialog.
7. Select **Edit runner balancing constraints** on the cross-sectional dimensions dialog.
8. Select **Fixed** in the list box.
9. Click **OK** on the 3 dialogs to accept the change.
10. Repeat the steps 2 to 9 on the other drop.

Run a fill analysis

A flow rate should be used as the filling control when setting up for a runner balance analysis. This will ensure the parts will fill in the time determined as optimum. The flow rate is calculated by:

$$56.7 = \frac{28.35}{0.5}$$

$$\begin{aligned} 28.35 &= \text{The volume of the two parts [cm\^{}3]} \\ 0.5 &= \text{The optimum injection time [sec]} \\ 56.7 &= \text{Flow rate [cm\^{}3/sec]} \end{aligned}$$

To run a Fill analysis

1. Select the material.
   1.1. Huntsman Chemical Company as the Manufacturer.
   1.2. Austran SAN23 as the Trade name.
2. Double-click the **Process Settings Wizard**.
3. Set the process settings to the values listed in the table below.

Parameter	Value
Mold surface temperature	60º
Melt temperature	225º
Filling Control	Flow rate
Flow rate	56.7
Velocity/pressure switch-over	By % volume filled
% volume filled	100 %
Pack/holding Control profile	0 100 10 100

4. Click **Analyze Now**.

Determine the target pressure

The fill time and pressure plots should be reviewed before determining the balance pressure. This will give you a sense for the amount of imbalance.

To plot the fill time result

1. Click **Fill Time** in the study tasks list.
2. Animate the result.
3. Stop the animation.
4. Step the animation from the start of fill until the Lid is just full.
   - Note how much there is left to fill on the Box.

To plot the pressure result

1. Click **Pressure** in the study tasks list.
   - Pressure is very sensitive to the runner balance. Notice how the pressure of the lid is very high compared to the box.
2. Animate the result.
3. Stop the animation.
4. Step the animation from the start of fill until the Lid is just full and the lid is mostly dark blue.
   - Note the time compared to the time at the end of fill.

To plot the pressure at injection location result

1. Click **Pressure at injection location:XY plot** in the study tasks list.
   - This result shows the pressure gradient at the injection location.
   - The pressure spike at about 0.5 seconds corresponds to the lid filling and the box has yet to fill out. See Figure 122.
   - In this case, the target pressure you will use is 70 MPa which is about equal to the pressure at the end of fill. This will be a good starting point for this tool.

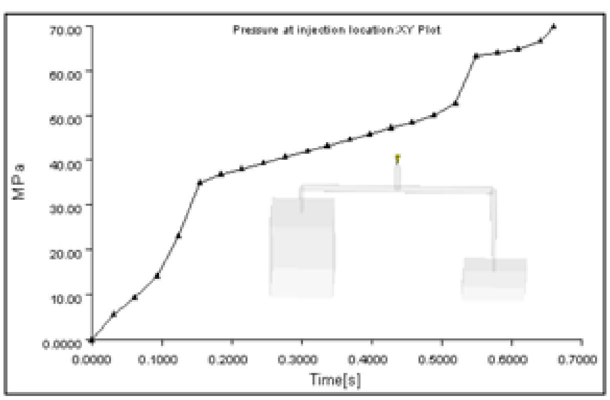


Figure 122: Pressure XY graph at the injection location

The higher the balance pressure, the smaller the runner diameters which will save material. However if the runner diameters are too small, the parts with the small runner may not pack out well.

Run the balance analysis

To balance the runners

1. Double-click the analysis sequence icon in the study tasks list.
2. Select **Runner Balance.**
3. Click **OK**.
    - If Runner balance is not in the list, click the **More** button and choose runner balance from the full list of analysis sequences.
4. Double-click the Process Settings Wizard.

    4.1. Ensure the filling parameters are correct.

    4.2. Click **Next**.

    4.3. Set the target pressure to **70** MPa.

    4.4. Click **Finish**.
5. Double click **Continue analysis**.

The analysis will start where the filling analysis left off. The screen output will have an iteration table indicating the progress of the analysis. It will finish when all 3 tolerances are met, or the iteration limit is reached.

Review the results

To view the screen output results

1. Open the screen output if necessary.
2. Review the iteration table.
    - The last iteration should be below all three tolerances.
    - If it reached the iteration limit, one more analysis is run with the iteration that had the lowest time imbalance.
    - In general, the time and pressure imbalance tolerances may osculate up and down a bit, but the section imbalance should just go down.
    - If the section imbalance goes up and down, this would indicate a balance that is difficult to converge.
    - This may be an indication of a balance pressure that is not correct.

To review volume change

1. Click on the volume change result in the **Box Lid** study.
2. Click the **Query result** icon.
   - **2.1.** Click on the runner leading to the box.
   - **2.2.** Hold the CTRL key and click on the runner leading to the lid.
     - ➤ Holding down the CTRL key will keep the previous label on the screen.
3. Notice the distribution of volume change.
   - Both the runners have a negative volume change. This indicates the runners got smaller. The lid has a much higher volume change. This was necessary to achieve a balance.
   - The drops are zero because their size is fixed.

> The runner diameters in this model have not changed. The Study Box Lid (Runner Balance) has the revised runner sizes in it.

To view the thickness changes

1. Double-click on the **Box Lid (Runner Balance)** study to open it.
2. Rotate the model as necessary to see the runners and parts. A good rotation is -70 -25 -10.
3. Plot the thickness diagnostic.
   - **3.1.** Click the Tools tab.
   - **3.2.** Click on the Diagnostics tool .
   - **3.3.** Select **Thickness Diagnostic**.
   - **3.4.** Click **Show**.
4. Click the icon  to query the runners diameters.
   - Notice how both are less than 6 mm, (the original sizes), hence the negative volume in the volume change plot.

To plot fill time

1. Click the **Fill time** result.
2. Animate fill time.
3. Notice how the flow fronts are very balanced compared to the first analysis.

To plot pressure

1. Click the **Pressure** result.
   - Notice how the Lid now fills last.

2. Step back the result one frame at a time until the pressure in the box is zero.
    - Notice how the time is ~0.61 seconds. The end of fill time is ~0.63 seconds. There is essentially no difference in the time.

To plot time to freeze

3. Click the **Time to Freeze** result.
4. Query the runners, drops, and the part.
5. Compare the cooling times.
    - The freeze times for the runners are still well above the cooling time of the parts, but are below the cooling time for the sprue and drops.
    - The freeze times for the sprue is high, indicating the sprue could be made smaller. The drop to the lid is also high, but this is difficult to address because of the drop length and the included angle of drop is already small.

To review other results

1. Plot other results to see the influence of the balance.
2. If desired, tile the windows and lock the studies:
    - Box Lid.
    - Box Lid (Runner Balance).
3. Compare the filling results between the two analyses.

Revise the target pressure as necessary and re-run

For the box and lid family tool, the runner sizes are about right. The runner leading to the lid has reduced in size a bit. It is possible to make it smaller, because the time to freeze is still about 3 times that of the part. However, the runner to the lid is larger than the drop entrance. While this is possible, it is not normally recommended. The runner leading toward the box has reduced in size from it’s original dimensions and is still larger than its drop opening. If you don’t mind if a runner feeding a drop is smaller than the drop diameter, the runner can be decreased in size.

If you would like to re-run the analysis with a revised balance pressure, save the study with a new shorter name, select the runner balance analysis sequence, enter a new balance pressure, and launch another analysis.

Round the runner sizes if desired

If the balance results are acceptable, possibly the can be rounded. Any changes in runner diameter from the optimum will influence the balance. If the sized runners are close to a standard size, rounding may be acceptable. Any rounding that is done should be validated using a flow analysis to check the balance but also volumetric shrinkage.

To round a runner, select the elements that you want to round or standardize. Right-click and select **Properties**. Enter the new diameter you want and click OK. *****

Competency Check - Gate & Runner Design

1.	**What is the minimum number of elements there should be when defining an edge gate.**
2.	**When a polymer is filled, what shear rate limit should be used when sizing the gate if possible?**
3.	**What type of runner system layout does the runner wizard make?**
4.	**What type of gate is created in the runner wizard if there are side gates with a cold runner?**
5.	**How should sub gates, and sprues be created if they are created manually?**
6.	**What is the definition of occurrence numbers?**
7.	**What is the procedure for doing a runner balance once the model has been created and the processing conditions, i.e. mold and melt temperature, injection time, transfer method, and packing profile are defined?**

Evaluation Sheet - Gate & Runner Design

1. **What is the minimum number of elements there should be when defining an edge gate.**

   There should be at least 3 elements in a gate to properly define when the gate will freeze off.

2. **When a polymer is filled, what shear rate limit should be used when sizing the gate if possible?**

   The shear rate limit should be quite low, about 20,000 1/sec. If possible. Some gate geometries would prevent this low shear rate but the shear rate should be as low as possible.

3. **What type of runner system layout does the runner wizard make?**

   The runner system attempts to make a naturally balanced runner system.

4. **What type of gate is created in the runner wizard if there are side gates with a cold runner?**

   The runner creation wizard for side gates creates to sub gates, or gates with a round cross section. If an edge gate is required, set the parting line to the Z depth of the gate, set the gate included angle to 0 and enter the gate length as the land length of the gate.

5. **How should sub gates, and sprues be created if they are created manually?**

   Sprues and sub gates should be created with curves and then meshed. When done this way, each element is a different size. If beams are created directly, each element has exactly the same properties; therefore tapered sections cannot be produced.

6. **What is the definition of occurrence numbers?**

   Occurrence numbers account for symmetry within a flow path. They are used to simplify the modeling of multi cavity tools for doing a runner balance.

7. **What is the procedure for doing a runner balance once the model has been created and the processing conditions, i.e. mold and melt temperature, injection time, transfer method, and packing profile are defined?**

   A filling analysis is done. The pressure at the injection location is interpreted to determine the balance pressure. The runner balance is done. Fill time, Pressure, Time to Freeze, and runner thickness are viewed to determine if the results are good. If not, the runner balance is run again.

CHAPTER 18

Basic Packing

Aim

The aim of this chapter is to learn the procedures for running a packing analysis.

Why do it

How well a part is packed out is of primary importance when considering warpage, shrinkage, and defects such as sink marks. The main output of a packing analysis is volumetric shrinkage, and the distribution and magnitude of volumetric shrinkage, which plays a key role in part quality.

Overview

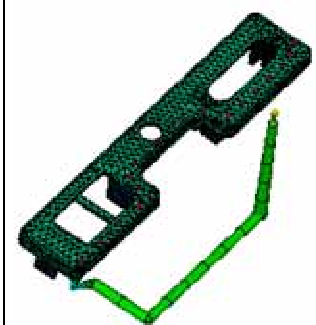
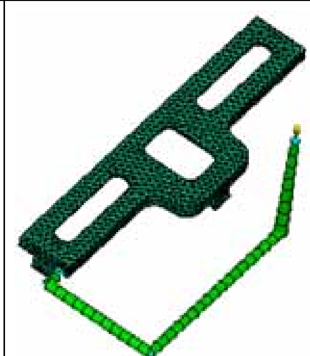
In this chapter you will review:

- When to run a packing analysis.
- Definitions of terms used for packing.
- Input parameters used for a packing analysis.
- Running a packing analysis.
- Reviewing results for all 3 mesh types.

Practice - Basic Packing

This chapter has two models to choose from and are described below. Pick one to work on.

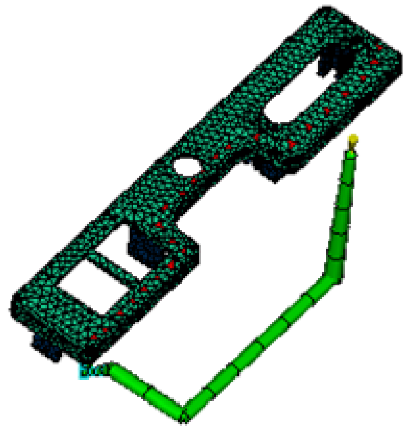
Table 24: Models used for the quick Cool-Flow-Warp analysis

Description	Model
Snap Cover: starts on page 357	
The **Snap cover** model uses a Fusion mesh. Use this model if you will primarily be using a mesh type of Fusion or midplane in your analysis work.	
3 Snap Cover: starts on page 367	
The **3 Snap cover** model uses a 3D mesh. Use this model if you will primarily be using a mesh type of 3D.	

Snap Cover

Design criteria

The criterion for this project is to run 2 packing analyses on the supplied part. The first analysis will use the given process settings. The switchover will be set at 99% volume filled, and with a packing profile, 10 seconds at 80% of the fill pressure. The second analysis will be done with a packing pressure half of the first analysis and with a packing time that is equal to the gate freeze of the first analysis.



The objective is to see how the volumetric shrinkage has changed as a result of the change in packing conditions, and to look at how the pressure decays in each of the parts.

Analysis setup

To open a project

1. Click the file open icon or **File** ➡ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Basic_Packing**.
2. Double click the project file **Basic_Packing.mpi**.
3. Click **File** ➡ **Preferences** and ensure that **System Units** are set to **Metric**.

To open the snap cover model

1. Double-click on the file **Snap Cover Packing** in the Project View pane.
2. Rotate the model around to review the geometry.
3. Turn on and off layers to see what they contain.

To setup the first packing analysis

1. Double-click the analysis sequence icon and set the sequence to **Flow**.
2. Double-click the Material icon .

   2.1. Select the material is **ABS HF380, LG Chemical**.
3. Double-click **Process Settings** icon and set the information in Table 25

.

Table 25: Snap cover first packing analysis input

Parameter	Value
Mold Temperature	60°C
Melt Temperature	240°C
Filling control	Injection time
Fill time	1.7 Sec.
Velocity/pressure switchover	%volume filled of 99%
Pack/holding control	%Filling pressure vs. time
Edit profile	0 80 15 80
Cooling time	Specified at 10 sec.

4. Ensure the **Intermediate results** are set:

4.1. Click the **Advanced options** button.

4.2. Click the **Solver parameters Edit** button in the frame.

4.3. Click the **Intermediate Output** tab.

4.4. Click the **Filling phase, Regular result Edit times** button.

4.5. Scroll through the list to see an intermediate result is listed at every 0.1 seconds up to 1.9 seconds.

4.6. Click **Cancel**.

4.7. Click the **Packing phase, Regular result Edit times** button.

4.8. Scroll through the list to see the intermediate results are written at varies times from 2 seconds to 22 seconds.

4.9. Click **Cancel** three times to exit Advanced options.

Normally, intermediate results are set up by specifying the number of steps for the filling and packing. The times were specified so the exact times would be used for both analyses used in this practice.

5. Click **OK** to exit the Process Settings Wizard.

To launch the analysis

1. Double-click **Analyze Now!** in the study tasks list to run the analysis.

- The analysis should take several minutes to complete.

2. Watch the progress in the Screen Output Log.

Review first analysis results

Volumetric shrinkage

To display the volumetric shrinkage results of the first analysis

1. Click **Volumetric shrinkage at ejection** in the results list.
2. Ensure the following layers are all turned on:
   - Body.
   - Detail.
   - Runners.
   - Path.
3. Record the minimum and maximum values in Table 27 on page 365.

   3.1. Round the volumetric shrinkages to 2 decimal places.
4. Click off the layers in the following order and watch the scale change:

   4.1. Runners.

   4.2. Detail.

   - This shows the range of volumetric shrinkage across the part. It is relatively high at the end of fill and relatively low near the gate.

   4.3. Record the minimum and maximum values in Table 27 on page 365 for the two remaining layers, Body and Path.

Click in the display window, and then click **F1** to get Result Interpretation help on each result.

To display volumetric shrinkage path plot

1. Click **New result plot** icon on the results tool bar.

   1.1. Select **Volumetric shrinkage** from the available results list.

   1.2. Select **Path plot** as the result type.

   1.3. Click the **Plot Properties** tab.

   1.4. Click the **Mesh Display** tab.

   1.5. Click **Off** in the edge display on undeformed part frame.

   1.6. Click **Solid** in the Filling Frame.

   1.7. Click **OK.**
2. Uncheck all layers except the Path layer.
   - Only the elements shown in Figure 123 should be used for the path plot.

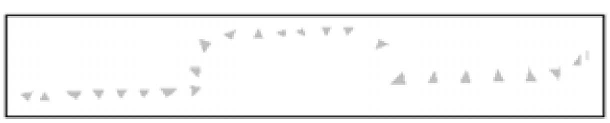


Figure 123: Path layer elements

3. Pick each element from the gate location to the end of fill.
   - Rotate and zoom the model as necessary.
4. Click **Plot properties** on the results tool bar.

   4.1. Click **Manual** in the **Y** range frame.
   - ➤ Set the Min. to **1**.
   - ➤ Set the Max to **7**.

   4.2. Click the **mesh display** tab.
   - ➤ Click **Feature lines** in the edge display on undeformed part frame.
   - ➤ Click **Transparent** in the filling frame.
   - ➤ Click **OK**.
5. Zoom and rotate the part as necessary to see the entire part and path plot.
6. Animate the result, one frame at a time.

Volumetric path plot interpretation

The path plot is a good way to look to see how the volumetric changes:

- Across the part.
- Through time.

The shrinkage at the end of fill and at ejection is about 4.8% and drops to about 1.2% by the gate. This is due to the long packing time at a constant pressure. Watch how the shrinkages starts out high everywhere, then quickly reduces. The end of fill stops shrinking first, then areas closer to the gate continues to shrink until the gate area freezes.

Supporting results

Other results are displayed to help understand the volumetric shrinkage plots, including:

- Frozen layer fraction.
- Pressure XY plot.
- Screen output.

To display the frozen layer fraction result

1. Turn on the layers:
   - Body.
   - Detail.
   - Path.
   - Runners.

2. Click **Frozen layer fraction** in the results list.

   2.1. Open the plot properties dialog.

   ➤ Right-click on the result name in the study tasks list and select **Properties.**

   2.2. Uncheck **Nodal averaged** on the Optional settings tab.

   ➤ This will show a different color for each individual element and is easier to see when the gate freezes.

   2.3. Click **OK**.

3. Animate the result manually one step at a time.

   - When the gate's value is 1.0, the gate is frozen.
   - Most of the part freezes before the gate.
   - Use your judgment on when the part is no longer being packed out effectively.

4. Record in Table 27 on page 365 the gate or part freeze time.

To plot pressure as an XY graph

1. Create a Pressure as an XY graph result.

   1.1. Open the Create new plot dialog by one of the following methods:

   ➤ Right-click the **Results** icon  and select **Create New Plot…**.

   ➤ Click the **New plot** icon .

   ➤ Click **Results** ➡ **New plot**.

   1.2. Select **Pressure** from the Available plot list.

   1.3. Select **XY Plot** as the Plot type.

   1.4. Click **OK**.

   1.5. Enter the following node numbers in the Entity Id's dialog.

   ➤ 2351.

   ➤ 2507.

   ➤ 2614.

   ➤ 2705

   ➤ 2809.

   1.6. Press the **Enter** key when done.

   - The XY graph will be created.
   - Notice how the pressure decays quite rapidly through the part.

2. Click **Query Result** icon .

  2.1. Select the maximum pressure for node 2809.

  - The pressure here is about half the packing pressure.

  2.2. Record the value in Table 27 on page 365.

Nodes can be added automatically by clicking on the model after you have created the result. Nodes were manually entered in this example so that the same nodes would be used in all cases

To find the pack pressure in the screen output log

1. Check the Logs box in the study tasks.
2. Click the Screen Output tab.
3. Locate the filling/packing analysis progress table.
4. Find the pressure at 2 seconds.
   - This is the first step in the packing phase.
   - This represents the pressure used to pack the part.
5. Record the pack pressure in Table 27 on page 365.

Running a second analysis

You will now run another filling and packing analysis, using a pack pressure half of the pressure used before.

To create a second study

1. Click the **Save** icon or click **File** ➡ **Save study**.
   - This saves the **Snap Cover Packing** study.

2. Click the **Save as** icon, or click **File** ➡ **Save Study As**.

   2.1. Enter **Snap Cover Half** as the file name.

   2.2. Click **Save**.

To run a second filling and packing analysis

1. Double-click **Process Settings** icon and set the information in Table 26.

Table 26: Snap cover second packing analysis input

Parameter	Value
Velocity/pressure switchover	%volume filled of 99%
Pack/holding control	Packing pressure vs time
Edit profile	.1 36 7.8 36 .1 0

- This will pack the part at about half the packing pressure as the first analysis and until the gate is frozen and the 0.1 second transitions gives time for the machine controls to respond, if this profile was entered into a molding machine.

2. Double-click analyze now to start the analysis.

Review the results

To compare volumetric shrinkage at ejection for both studies

1. Ensure there are only two windows open.

- **Snap cover packing**.
- **Snap cover packing half**.

1.1. Close and open windows as necessary to get just those two windows displayed.

2. Click **Window ➡ Tile Vertically**.

> If you don't like the size of the windows, try Tile Horizontally.

3. Use the View ➡ Lock commands, or the icons to Lock:

- Views.
- Animations.
- Plots with the **View ➡ Lock** commands.

4. Ensure that only the **Body** layer is on for each study.

5. Click the **Snap cover packing** study to make it the active window.

6. Plot **Volumetric shrinkage at ejection**.

- This should display the result in both studies because the results are locked.

7. Record the volumetric shrinkage for the second part in Table 27 on page 365.

8. Open the **Plot Properties**, on one of the studies.

   8.1. Click the **Scale** tab.

   8.2. Set the min. and max value that covers the range for both studies.

   8.3. Click **OK**.

   - Both studies will get there scale set.

9. Turn on the **Detail** layer for both studies.
10. Rotate the model as needed to evaluate the shrinkage differences.

To compare volumetric shrinkage as path plots

1. Ensure the **snap cover packing** study is active by clicking in its window.
2. Click the **Volumetric shrinkage:Path Plot** result.
   - This result is created for the second study because it did not exist.
3. Open the plot properties on one of the studies.

   3.1. Set the Y range values to:

   - Min. to 1.0.
   - Max to 7.0.

4. Animate the result.
   - The snap cover packing half study has a much higher shrinkage at the end of fill due to the lower packing pressure. The area near the gate is closer due to more time to pack the part.

To compare frozen layer fraction

1. Ensure the **snap cover packing** study is active by clicking in its window.
2. Click the **Frozen layer fraction** result.
3. Animate the result.

   3.1. Stop the animation.

   3.2. Step through the animation until the gate/part freeze off.

4. Record the gate freeze time for the second part in Table 27 on page 365.

To compare Pressure:XY Plots

1. Ensure the **snap cover packing** study is active by clicking in its window.
2. Click the **Pressure:XY Plot** result.
   - This result is created for the second study because it did not exist.
3. Click on the maximum pressure with the query tool on both studies.

   3.1. Record the maximum pressure for the second part in Table 27 on page 365.

   - Notice how the pressure for the **Snap Cover Packing half**, is about 12 MPa less than the other study, and drops to zero about 1.5 seconds sooner.

To compare pack pressure

1. Check the Logs box on the Snap Cover Packing Half analysis.
2. Click the Screen Output tab.
3. Locate the filling/packing analysis progress table.
4. Find the pressure at 2 seconds.
   - This is the first step in the packing phase.
   - This represents the pressure used to pack the part.
5. Record the pack pressure in Table 27.

Answers for the Packing analysis worksheet are on page 366.

Snap cover packing analysis worksheet

Table 27: Worksheet for the packing analyses results

Result	Snap Cover Packing	Snap Cover Packing half
Volumetric shrinkage at Ejection all layers on min./max [%]		
Body & path layers only min./max [%]		
Time for the part/gate to freeze [Sec.] (Frozen layer fraction 1.0)		
Peak pressure at N2809 [MPa]		
Pressure at the end of fill [MPa]		

Snap cover packing analysis answers

The table below contains the answers for the packing analysis.

Table 28: Worksheet answers for the packing analyses results

Result	Snap Cover Packing	Snap Cover Packing half
Volumetric shrinkage at Ejection all layers on min./max [%]	0.25%/ 5.13%	1.47% / 7.11%
Body & path layers only min./max [%]	0.75%/ 5.13%	1.97%/ 7.11%
Time for the part/gate to freeze [Sec.] (Frozen layer fraction 1.0)	9.75 Sec.	9.75 Sec.
Peak pressure at N2809 [MPa]	37.5 MPa	25.5 MPa
Pressure at the end of fill [MPa]	71.47 MPa	36 MPa

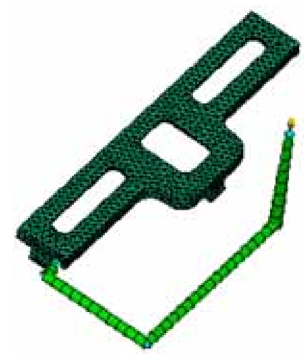
Results Discussion

The reduction in packing pressure by half makes a significant difference in the pressure and volumetric shrinkage results. The volumetric shrinkage gradient on the part is very similar between the parts, but the volumetric shrinkage is higher on **Snap Cover packing half** because of the lower packing pressure. You can see that the pressure at the end of fill with **Snap Cover packing** is about equal to the packing pressure of the second analysis.

The results were created using MPI 6.0 build 054495. If different process settings or software versions were used, the answers may be slightly different.

3 Snap Cover

Design criteria



The criteria for this project is to run 2 packing analyses on the supplied part. The first analysis will use the given process settings. The second analysis will be done with a packing pressure half of the first analysis and with a packing time that is equal to the gate freeze of the first analysis.

The objective is to see how the volumetric shrinkage has changed as a result of the change in packing conditions, and to look at how the pressure decays in each of the parts.

Analysis setup

To open a project

1. Click the file open icon or **File** ➡ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Basic_Packing**.
2. Double click the project file **Basic_Packing.mpi**.
3. Click **File** ➡ **Preferences** and ensure that **System Units** are set to **Metric**.

To open the 3 snap cover model

1. Double-click on the file **3D 3Snap Cover Packing** in the Project View pane.
2. Rotate the model around to review the geometry.

To setup the first packing analysis

1. Double-click the analysis sequence icon and set the sequence to **Flow**.
2. Double-click the Material icon .

   2.1. Select the material is **Cycolac 28818E, GE Plastics (USA)**.
3. Double-click **Process Settings** icon and set the information in Table 29.

.

Table 29: Snap cover first packing analysis input

Parameter	Value
Mold Temperature	60°C
Melt Temperature	250°C
Filling control	Injection time
Fill time	1.5 Sec.
Velocity/pressure switchover	%volume filled of 99%
Pack/holding control	%Filling pressure vs. time
Edit profile	0 90 10 90
Cooling time	Specified at 5 sec.

4. Ensure the **Intermediate results** are set:

4.1. Click the **Advanced options** button.

4.2. Click the **Solver parameters Edit** button in the frame.

4.3. Click the **Intermediate results, Edit intervals** button.

4.4. Ensure there are **10** filling phase and **30** packing phase intermediate results.

4.5. Click **Cancel** three times to exit Advanced options.

5. Click **OK** to exit the Process Settings Wizard.

To launch the analysis

1. Double-click **Analyze Now!** in the study tasks list to run the analysis.

- The analysis should take several minutes to complete.

2. Watch the progress in the Screen Output Log.

Review first analysis results

Volumetric shrinkage

The volumetric shrinkage for this part will be displayed 3 different ways so you can see how the three methods are set up. Depending on your part geometry and analysis objectives, you may find one method better than another.

To plot volumetric shrinkage as a shaded image

1. Ensure the **Part** and **Runner system** layers are on, and all others are **off**.

2. Click the result **Volumetric shrinkage (3D)** from the study tasks list.

3. Click the **Edit cutting planes** icon .

3.1. Check the **Plane ZX**.

3.2. Click **Close**.

4. Rotate the part so you can see the part's cross section on the cutting plane.

5. Click the **Plot properties** icon .
   5.1. Click the **Scaling** tab.
   5.2. Click **Per frame**.
   5.3. Click **OK**.

6. Click the **Move cutting plane** icon .
   6.1. Uncheck **Show active plane**.
   6.2. Click and hold the left mouse button and drag the cursor up and down to move the cutting plane.

7. Record the volumetric shrinkage range when both layers are on in Table 31 on page 377.

8. Turn off the Runner system layer.

9. Move the cutting plane over the part again.
   - Notice how the shrinkage is highest in the center of the cross section furthest from the gate. The shrinkage is lowest near the gate.

10. Record the volumetric shrinkage range for the part layer only in Table 31 on page 377.

To plot volumetric shrinkage as a single contour

1. Click the **Edit cutting planes** icon .
   1.1. Uncheck the **Plane ZX**.
   1.2. Click **Close**.

2. Click the **Plot properties** icon .
   2.1. Click the **Methods** tab.
   2.2. Click **Contour**.
   2.3. Check **Single contour**.
   2.4. Enter **1.25** in the **Contour value** field.
   2.5. Click the **Scaling** tab.
   2.6. Click **All frames**.
   2.7. Click **OK**.
   - This creates an iso-surface with a value of 1.25% volumetric shrinkage

3. Animate the result.
   - Watch how the size of the iso surface changes through time.

4. Click the **Plot properties** icon .
   4.1. Change the **Single contour** to:
      - 2.0.
      - 2.5
      - 2.75.
   4.2. Animate the results at each setting.
      - Notice the area that is plotted.

To plot volumetric shrinkage with a value range

1. Click the **Plot properties** icon .
   1.1. Click the **Methods** tab.
   1.2. Click **Shaded**.
   1.3. Click the **Animation** tab.
   1.4. Set **Animate result over** to **Single dataset**.
   1.5. Set the time to the last one in the pull down list.
   1.6. Click **Value range**.
   1.7. Set the range to **0.1**.
   1.8. Set the **Single dataset animation** to **Current frame only.**
   1.9. Click the Scaling tab.
   1.10. Click **Specified**.
   1.11. Enter **1** in the **Min.** field.
   1.12. Enter **3** in the **Max** field.
   1.13. Click **OK**.
2. Animate the result.
   2.1. Step through the frames one by one.
   2.2. Notice the range starts at 1.0 to 1.1 and each frame increments up by 0.1.
   - The range is a round number because the scale was set from 1 to 3%.

Supporting results

The freeze time for the gate with 3D results is done with the Temperature (3D) result. The transition temperature of the material must be known.

To determine the materials transition temperature

1. Right-click on the **Material** icon in the study tasks list.
2. Pick **Details** on the context menu.
3. Click the **Rheological Properties** tab.

4. Note the **Transition temperature**, then click **OK**. ____________________________

To plot temperature as a single contour

1. Ensure the **Part** and **Runner System** layers are on.
2. Click the **Temperature (3D)** result.
3. Click the **Plot properties** icon .
   - **3.1.** Click the **Methods** tab.
   - **3.2.** Click **Contour**.
   - **3.3.** Check **Single contour**.
   - **3.4.** Enter the material's transition temperature as the **Contour value**.
   - **3.5.** Click **OK**.
4. Animate the results.
   - **4.1.** Step through the animation one frame and a time. Determine when the part can no longer be packed out.
   - **4.2.** Enter the value in Table 31 on page 377.

To plot pressure as an XY graph

1. Create a Pressure as an XY graph result.
   - **1.1.** Open the Create new plot dialog by one of the following methods:
     - ➤ Right-click the **Results** icon and select **Create New Plot…**.
     - ➤ Click the **New plot** icon .
     - ➤ Click **Results** ➡ **New plot**.
   - **1.2.** Select **Pressure** from the Available plot list.
   - **1.3.** Select **XY Plot** as the Plot type.
   - **1.4.** Click **OK**.
   - **1.5.** Enter the following node numbers in the Entity Id's dialog.
     - ➤ 1648.
     - ➤ 83.
     - ➤ 3993.
     - ➤ 3125
     - ➤ 1621.
   - **1.6.** Press the **Enter** key when done.
   - The XY graph will be created.
   - Notice how the pressure decays quite rapidly through the part.

2. Click **Query Result** icon .

   2.1. Select the maximum pressure for node 1621.

   ➤ The pressure here is about 15 MPa less than the packing pressure.

   2.2. Record the value in Table 31 on page 377.

Nodes can be added automatically by clicking on the model after you have created the result. Nodes were manually entered in this example so that the same nodes would be used in all cases

To find the pack pressure in the screen output log

1. Check the Logs box in the study tasks.
2. Click the Screen Output tab.
3. Locate the filling/packing analysis progress table.
4. Find the pressure at 1.58 seconds.
   - This is the first step in the packing phase.
   - This represents the pressure used to pack the part.
5. Record the pack pressure in Table 31 on page 377.

Running a second analysis

You will now run another filling and packing analysis, using a pack pressure half of the pressure used before.

To create a second study

1. Click the **Save** icon or click **File** ➡ **Save study**.
   - This saves the **3D 3Snap Cover Packing** study.
2. Click the **Save as** icon, or click **File** ➡ **Save Study As**.

   2.1. Enter **3D 3Snap Cover Packing Half** as the file name.

   2.2. Click **Save**.

To run a second filling and packing analysis

1. Double-click **Process Settings** icon and set the information in Table 30.

Table 30: Snap cover second packing analysis input

Parameter	Value
Velocity/pressure switchover	%volume filled of 99%
Pack/holding control	Packing pressure vs time
Edit profile	0.1 34 6.0 34 0.1 0
Cooling Time	Specified, 7.5

- This will pack the part at about half the packing pressure as the first analysis and until the gate is frozen and the 0.1 second transitions gives time for the machine controls to respond, if this profile was entered into a molding machine.

2. Double-click analyze now to start the analysis.

Review the results

To plot volumetric shrinkage as a shaded image

1. Ensure the **Part** and **Runner system** layers are on, and all others are **off**.
2. Click the result **Volumetric shrinkage (3D)** from the study tasks list.
3. Click the **Edit cutting planes** icon .

   3.1. Check the **Plane ZX**.

   3.2. Click **Close**.
4. Rotate the part so you can see the part's cross section on the cutting plane.

5. Click the **Plot properties** icon .

 5.1. Click the **Scaling** tab.

 5.2. Click **Per frame**.

 5.3. Click **OK**.

6. Click the **Move cutting plane** icon .

 6.1. Uncheck **Show active plane**.

 6.2. Click and hold the left mouse button and drag the cursor up and down to move the cutting plane.

7. Record the volumetric shrinkage range when both layers are on in Table 31 on page 377.

8. Turn off the Runner system layer.

9. Move the cutting plane over the part again.

 - Notice how the shrinkage is highest in the center of the cross section furthest from the gate. The shrinkage is lowest near the gate.

10. Record the volumetric shrinkage range for the part layer only in Table 31 on page 377.

To plot volumetric shrinkage as a single contour

1. Click the **Edit cutting planes** icon .

 1.1. Uncheck the **Plane ZX**.

 1.2. Click **Close**.

2. Click the **Plot properties** icon .

 2.1. Click the **Methods** tab.

 2.2. Click **Contour**.

 2.3. Check **Single contour**.

 2.4. Enter **1.25** in the **Contour value** field.

 2.5. Click the **Scaling** tab.

 2.6. Click **All frames**.

 2.7. Click **OK**.

 - This creates an iso-surface with a value of 1.25% volumetric shrinkage

3. Animate the result.

 - Watch how the size of the iso surface changes through time.

4. Click the **Plot properties** icon .

   4.1. Change the **Single contour** to:

   - 2.0.
   - 2.5
   - 2.75.

   4.2. Animate the results at each setting.

   - Notice the area that is plotted.

To compare volumetric shrinkage for both studies

1. Ensure there are only two windows open.
   - **3D 3Snap cover packing**.
   - **3D 3Snap cover packing half**.

   1.1. Close and open windows as necessary to get just those two windows displayed.

2. Click **Window ➡ Tile Vertically**.

> If you don't like the size of the windows, try Tile Horizontally.

3. Use the View ➡ Lock commands, or the icons  to Lock:
   - Views.
   - Animations.
   - Plots with the **View ➡ Lock** commands.
4. Ensure that only the **Part** layer is on for each study.
5. Click the **3D 3Snap cover packing half** study to make it the active window.
6. Plot **Volumetric shrinkage**.
   - This should display the result in both studies because the results are locked.
   - The volumetric shrinkage should be the single contour at 2.75%.
   - Note the differences.

To plot temperature as a single contour

1. Ensure the **Part** and **Runner System** layers are on.
2. Ensure the **3D 3Snap cover packing** study is active by clicking in its window.
3. Click the **Temperature (3D)** result.
   - The result should be a single contour at the ejection temperature.
4. Click the unlock all animations icon  or **View ➡ Unlock ➡ All animations**.
5. Adjust the animation to the freeze time found for the **3D 3Snap cover packing** study.

6. Animate the study **3D 3Snap cover packing half** to find the gate freeze time.
7. Record the gate freeze time in Table 31 on page 377.

To compare Pressure:XY Plots

1. Ensure the **3D 3Snap cover packing** study is active by clicking in its window.
2. Click the **Pressure:XY Plot** result.
   - This result is created for the second study because it did not exist.
3. Click on the maximum pressure with the query tool on both studies.

   3.1. Record the maximum pressure for the second part in Table 31 on page 377.

   - Notice how the pressure for the **3D 3Snap Cover Packing half**, is about 13 MPa less than the other study, and drops to zero about 2 seconds sooner.

To compare pack pressure

1. Check the Logs box on the Snap Cover Packing Half analysis.
2. Click the Screen Output tab.
3. Locate the filling/packing analysis progress table.
4. Find the pressure at 1.66 seconds.
   - This represents the pressure used to pack the part.
5. Record the pack pressure in Table 31 on page 377.

> Answers for the Packing analysis worksheet are on page 378.

3 snap cover packing analysis worksheet

Table 31: Worksheet for the 3 snap packing analyses results

Result	3D 3Snap Cover Packing	3D 3Snap Cover Packing half
Volumetric shrinkage (3D), for part and runner system layers min./max [%]		
Volumetric shrinkage (3D), for Part layer only min./max [%]		
Time for the part/gate to freeze [Sec.] (Part/gate goes below transition temperature)		
Peak pressure at N1621 [MPa]		
Pressure at the end of fill [MPa]		

3 snap cover packing analysis worksheet answers

Table 32: Worksheet for the 3 snap packing analyses results

Result	3D 3Snap Cover Packing	3D 3Snap Cover Packing half
Volumetric shrinkage (3D), for part and runner system layers min./max [%]	0.98% / 12.18%	0.97% / 12.33%
Volumetric shrinkage (3D), for Part layer only min./max [%]	0.98% / 2.95%	0.97% / 2.95%
Time for the part/gate to freeze [Sec.] (Part/gate goes below transition temperature)	7.6 Sec.	7.4 Sec.
Peak pressure at N1621 [MPa]	53.3 MPa	40.0 MPa
Pressure at the end of fill [MPa]	68.2 MPa	34 MPa

Results Discussion

The reduction in packing pressure by half makes a significant difference in the pressure and volumetric shrinkage results. With the lower packing pressure in the second analysis, much more of the part has volumetric shrinkages that are on the higher end of the scale, compared to the first packing analysis with a higher pressure.

The results were created using MPI 6.0 build 054495. If different process settings or software versions were used, the answers may be slightly different.

Competency Check - Basic Packing

1.	**When increasing the packing time, why does it get to a point when increasing the packing time has no effect on the volumetric shrinkage of the part, or portion of a part?**
2.	**What effect does changing the packing pressure have on volumetric shrinkage and Why?**

Evaluation Sheet - Basic Packing

1.	**When increasing the packing time, why does it get to a point when increasing the packing time has no effect on the volumetric shrinkage of the part, or portion of a part?** Once the gate or some other geometry freezes off, any portion of the part past the point of freeze off will not be packed any longer, increasing the volumetric shrinkage.
2.	**What effect does changing the packing pressure have on volumetric shrinkage and Why?** As the pack pressure increases the volumetric shrinkage will decrease because the more pressure on the plastic the lower the shrinkage due to PVT characteristics of the material.

CHAPTER 19

Using Valve Gates

Aim

This chapter will introduce you to the modeling and analysis of valve gates.

Why do it

Valve gates are a common form of hot runner component. Valve gates create a positive shut off so the plastic flow front and packing can be controlled by opening and closing the valve gate as required. One common use of valve gates is to eliminate weld lines, which is sometimes referred to as sequential gating. One gate is open and a second gate is not opened until the flow front from the first gate has passed the location of the second. When the second gate is opened, no weld line is formed.

Overview

In this chapter, you are provided with a model of a tub that requires 3 gates. There is a requirement for no weld lines, so valve gates will be used. The runner system will be created using the Runner System Wizard, and then further editing of the runner and gate properties will be required.

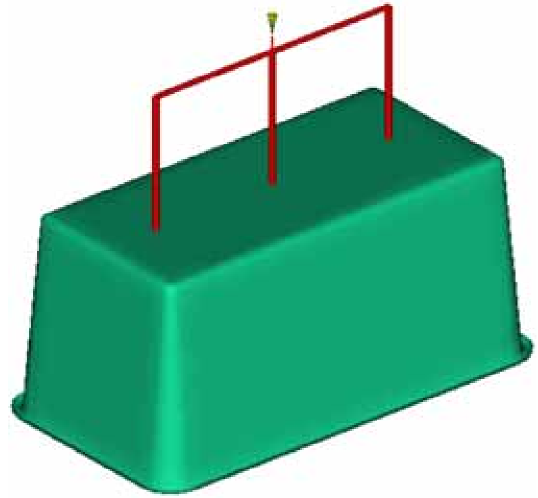
The gates will be modified to turn them into valve gates. This requires making the last element in the gate a valve gate controller. Each gate will have a different controller to provide the greatest flexibility. The center gate will have an initial state of open, and the outer two will have a state of flow front.

Practice - Using Valve Gates

Tub

Design criteria

There must not be any weld lines on the Tub. The fill pattern must be balanced, with the pressure to fill less than 100 MPa. Valve gates will be used to prevent weld line formation between the drops. The three gate locations are needed to keep the flow length shorter and uniform in length, in order to create a balanced fill and a pressure drop below 100 MPa.



This part is a Fusion model, but the procedures for modeling and running an analysis with valve gates is the same for all 3 mesh types.

Setup

To open a project

1. Click the file open icon or **File** ➡ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Valve_Gates**.
2. Double click the project file **Valve_Gates.mpi**.
3. Click **File** ➡ **Preferences** and ensure that **System Units** are set to **Metric**.

To review the model

1. Open the model **Tub**.
2. Investigate the model geometry using the **model manipulation** tools.

Creating the runner system

The Runner System Wizard will be used to create the runner system. The properties of the drops and gates will need to be modified manually also in order to the correct properties. The drawing in Figure 124 shows the dimensions for the runners, and Figure 125 shows the size of the gate, both figures are used in the Runner System Wizard. You will create a layer just for the gates to make gate manipulation easy.

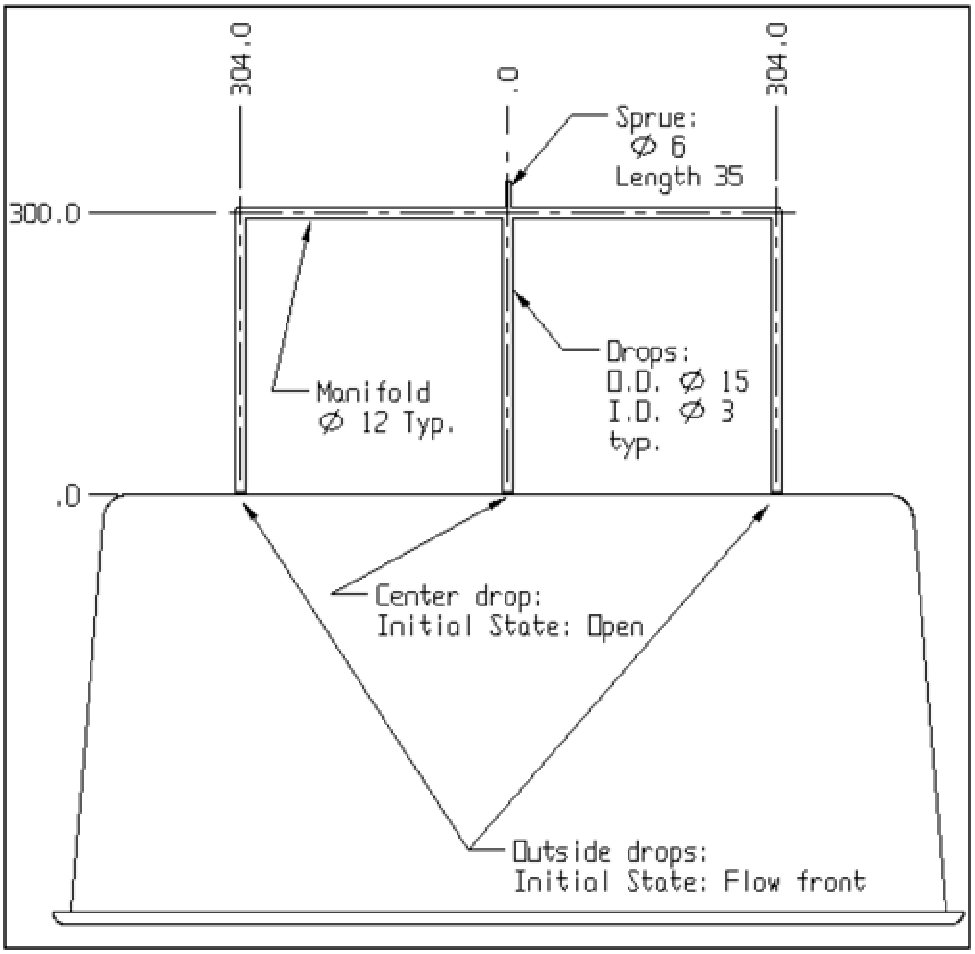


Figure 124: Runner Layout

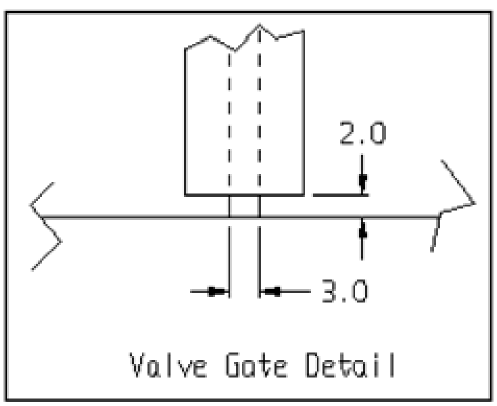


Figure 125: Gate Detail

To create the runner system

1. Click **Modeling ➡ Runner System Wizard**.
2. On the first page set:
   - **2.1.** Ensure the **X** and **Y** sprue positions are **0**.
   - **2.2. Check** the box **I would like to use a hot runner system**.
   - **2.3.** Enter **300** in the Top runner plane Z: field.
   - **2.4.** Click **Next**.
3. On the second page set:
   - **3.1.** Enter **6** in the Sprue Orifice diameter field.
   - **3.2.** Ensure **0** is the value for the Sprue Included angle.
   - **3.3.** Enter **35** into the Sprue Length field.
   - **3.4.** Enter **12** into the Runners Diameter field.
   - **3.5.** Enter **15** into the Drops Bottom diameter field.
   - **3.6.** Ensure **0** is the value for the Drops Included angle.
   - **3.7.** Click **Next**.
4. On the third page set:
   - **4.1.** Enter **3** into the Top gates Start and End diameter fields.
   - **4.2.** Enter **2** into the Top gates Length field.
   - **4.3.** Click **Finish**.
   - The runner system is created on a new layer called **Runner System**.

To set the hot drop properties

1. Click the Bottom View Icon  on the Viewpoint toolbar.
2. Band-select all the elements in all 3 of the vertical hot drops.
   - If any additional entities are selected reselect the drops.
3. Right-click to select **Properties**  or click **Edit ➡ Properties**.
   - **3.1.** Select **Annular** in the Cross-section drop-down list.
   - **3.2.** Select **Non-tapered** in the Shape drop-down list.
   - **3.3.** Click the **Edit dimensions** button.
     - ➤ Set the Outer diameter to **15**.
     - ➤ Set the Inner diameter to **3**.
     - ➤ Click **OK**.
   - **3.4.** Enter **Annular drop 15 mm x 3 mm** in the Name field.
   - **3.5.** Click **OK**.

The gates will be moved to the gates layer, the nodes will be moved to the nodes layer and the injection location will be moved to the Runner System layer.

To create a gate layer

1. Click in the Layers pane.
2. Enter **Gates** for the new layer's name.

To move entities to different layers

1. Move the gates to the new layer.
   1.1. Select **Edit ➨ Select by ➨ Properties**, or use **Ctrl+B**.
   1.2. Select **Beam element** and **Curve** in the **By entity types** field.
   1.3. Select **Hot gate** in **the By properties** field.
   1.4. Click **OK**.
   1.5. Highlight the **Gates** layer and click (Assign).
2. Move the nodes to a different layer.
   2.1. Click on the model and click **Ctrl+B**.
   2.2. Click **Node** as the entity type.
   2.3. Click **OK**
   2.4. Highlight the **New Nodes** layer and click .
3. Move the injection location to a different layer.
   3.1. Select the injection location cone.
   3.2. Highlight the **Runner System layer** and click .

Assigning the gate properties

All elements at each gate location will be assigned a new property with a different valve gate controller. At the rotation –35 89 40, the gate to the lower left a negative X value, the gate in the middle is at 0 X and the gate in the upper right has a positive X value. The gate properties will be called:

- Gate –x.
- Gate center.
- Gate +x.
- All the gates will be set with an initial state of open, which will simulate a normal hot runner system. Then a second analysis will be run with the +x and –x gate set to an initial state of flow front activated.

To prepare for setting the valve gate properties

1. Right-click on the **Gates** layer and select **Hide all other layers** from the context menu.
   - Now, only the 3 gates should be visible on screen.
2. Rotate the model to –**35 89 40** using the **Enter Rotation Angles** field 

, in the **Viewpoint** toolbar.
3. Zoom up on the gates to fill the screen similar to Figure 126.

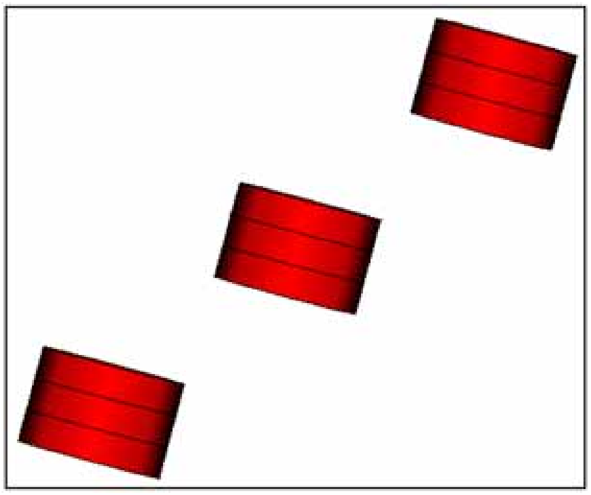


Figure 126: Zoomed in on the 3 gates for the tub

To set the valve gate properties for the -X gate

1. Select the bottom element of the lower left gate.
2. Click  or **Edit** ➨ **Assign Property**.
3. Select **New** ➨ **Hot Gate**.
4. On the Gate Properties tab set:

   4.1. The Cross-section to **Circular**.

   4.2. The Shape to **Non-tapered**.

   4.3. The Diameter is **3**.

5. Click the **Valve Control** tab.

   5.1. Click the **Select** button in the Valve gate controller frame.

   5.2. Click **Select** in the Select Valve gate controller dialog.

   - By clicking select, you are creating a valve gate controller property that can be edited.

   5.3. Click **Edit** in the Valve gate controller group.

   5.4. Ensure **Time** is the Valve gate control type.

   5.5. Click the **Edit Timing** button.

   - Select the Initial state of **Open.**
   - Ensure the timer settings are **0** and **30**.
   - Click **OK**.

   5.6. Enter **Gate –X** in the name field.

   5.7. Click **OK** in the Valve gate controller dialog.

6. Enter **Gate –X** in the name field of the Hot gate dialog.

7. Click **OK** in the Hot gate dialog.

8. Click **OK** in the Assign property dialog.

To assign the valve gate properties to the other gates.

1. Select the bottom element of the **center** gate.

   1.1. Repeat steps 2-8 of the previous task, for the center gate.

   1.2. The name should be **Gate Center**.

   1.3. The initial state should be **Open**.

2. Select the bottom element of the **upper right** gate.

   2.1. Repeat steps 2-8 of the previous task, for the **Gate +X**.

   2.2. The name should be **Gate +X**.

   2.3. The initial state should be **Open**.

To ensure controllers are assigned correctly

1. Click **Edit ➧ Select by ➧ Properties**, or use **Ctrl+B**.

   1.1. Select the **Beam Element** entity type.

   1.2. Select the property **Hot gate (default) #1**, as shown in Figure 127.

   1.3. Click **OK**.

   - Six elements should be selected, the top two elements on each gate.

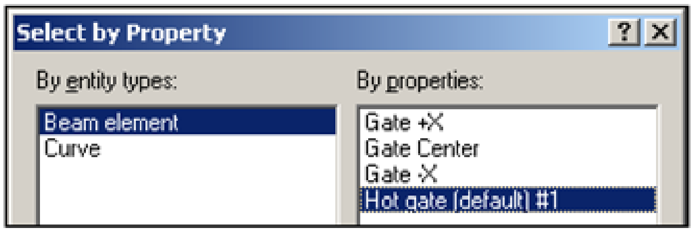


Figure 127: Selecting the hot gate property

2. Right-click to select **Properties**  or click **Edit ➡ Properties**.

2.1. Click the **Valve Control** tab.

2.2. Ensure there is a blank field for the valve gate controller.

- This will ensure there is no valve gate controllers assigned.
- Only one element is needed at every gate with a valve gate controller.

2.3. Un-assign a controller if necessary and click **OK**.

Figure 128: Valve gate controller not assigned.

Having multiple valve gates controllers in a gate will not be a problem unless the control type is Flow front. If more than one element is a gate has flow front control, the gates will not open.

3. Click **Edit ➡ Select by ➡ Properties**, or use **Ctrl+B**.

3.1. Select the **Beam Element** entity type.

3.2. Select the property **Gate -X.**

3.3. Click **OK**.

➤ One element should be selected.

4. Click  or **Edit ➡ Assign Property**.

- The **Gate -X** property should be highlighted.

4.1. Click the **Edit** button.

4.2. Click the **Valve Control** tab.

4.3. Ensure the **Gate -X** valve gate controller is the defined controller.

➤ Assign the controller as necessary.

4.4. Click **Cancel** to exit the dialog.

5. Repeat steps 3 and 4 for the **Center** and **+X gates**.

Run and review the first analysis

To run the filling analysis

1. Turn on the layer:
   - New Triangles.
   - Runner System.
   - Gates.
   - All other layers should be off.
2. Double-click and pick the material:
   - **Prime Polymer Co Ltd**.
   - **Hipol CJ700**.
3. Click or **Analysis** ➡ **Process Settings Wizard** and specify the values shown in Table 33. All other settings remain at their default values.

Table 33: Inputs for the tub fill analysis

Parameter	Value
Mold temperature:	40ºC.
Melt temperature:	230ºC.
Injection time:	3.75 sec.

4. Click , or **File** ➡ **Save Study**.
5. Click , or **File** ➡ **Save Study as** and enter the name **Tub no valve gates**.
6. Double-click , or **Analysis** ➡ **Analyze Now!**

To review the weld line results

1. Click on the **Weld lines** result.
   - The weld lines are plotted relative to the meeting angle.
   - Click the **Plot properties** icon on the results tool bar.

   1.1. Click the Highlight tab.

   1.2. Click the icon.

   1.3. Highlight **Temperature at flow front**.

   1.4. Click **OK**.
   - The weld line is now plotted with flow front temperature. The closer the flow front temperature is to the melt temperature the higher the weld line quality will be. In this case however, no weld lines are acceptable.

To review the fill time and weld line results

1. Display the fill time result.
   - **1.1.** Highlight **Fill time**.
   - **1.2.** Right-click and select **Overlay**.
2. Set the result to contour lines.
   - **2.1.** Highlight **Fill time**.
   - **2.2.** Right-click and select **Properties**.
   - **2.3.** Select the **Methods** tab.
   - **2.4.** Select **Contour**.
   - **2.5.** Click **OK**.
   - The fill time and weld line results are now displayed over each other. However, the weld line is difficult to see as some of the colors used to display the weld line blend into the contours for fill time. The weld lines can be made to be a solid color.
3. Highlight **Weld lines**.
   - **3.1.** Right-click and select **Properties**.
   - **3.2.** Select the **Optional Settings** tab.
   - **3.3.** Select **Single color** in the Color combo box.
   - **3.4.** Click the Select button.
   - **3.5.** Select the color **Black**.
   - **3.6.** Click **OK** on the color dialog.
   - **3.7.** Click **OK** on the plot properties dialog.

Running a second analysis with valve gate timings

To adjust the valve gate timing

1. Click , or **File ➡ Save Study**.
2. Click , or **File ➡ Save Study as** and enter the name **Tub with valve gates**.
3. Click **Edit ➡ Select by ➡ Properties**, or use **Ctrl+B**.
   - **3.1.** Select **Beam element**, **Gate +X** and click **OK**.

4. Click or **Edit** ➨ **Assign Property**.
   - The **Gate -X** property should be highlighted.

   4.1. Click the **Edit** button.

   4.2. Click the **Valve Control** tab.

   4.3. Ensure the **Gate -X** valve gate controller is the defined controller.

   4.4. Click the **Edit** button.

   4.5. Set **Flow front** as the valve gate control.

   4.6. Click **OK** three times to make the changes.
5. Repeat the steps 3 - 4 for the **Beam element**, **Gate +x.**
6. Click **Analysis** ➨ **Valve gate timings**.

   6.1. Click on **Flow front** as the control.

   6.2. Set the **Trigger Location** to **Gate Node** for both the **Gate -X** and **Gate +X** groups.

   6.3. Set the **Delay Time** to **0.25** sec. for both groups.

   6.4. Click **OK**.
   - Refer to Figure 129.
7. Click , or **File** ➨ **Save Study**.
8. Double-click , or **Analysis** ➨ **Analyze Now!**

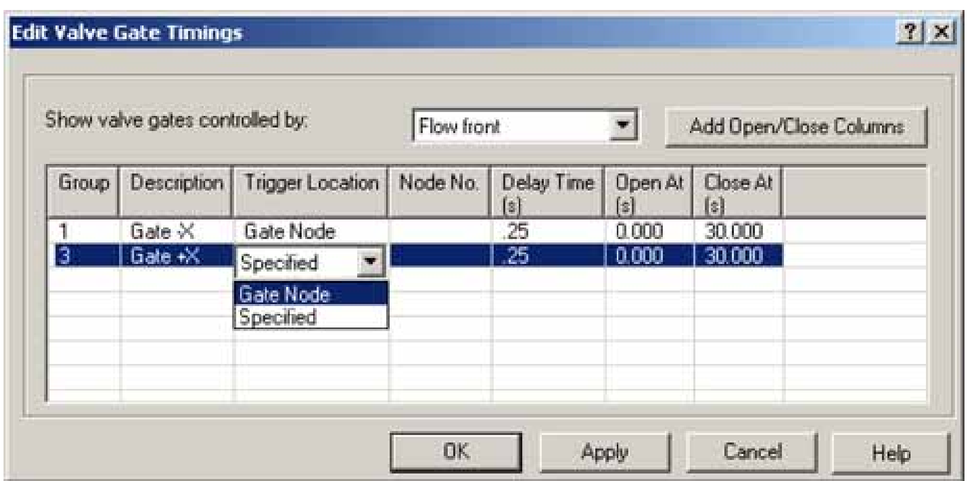


Figure 129: Valve gate timing settings

Comparing results

To prepare for reviewing results

1. Ensure that only the following studies are open.
   - **Tub no valve gates.**
   - **Tub with valve gates.**
2. Click **Windows ➡ Tile Horizontally**.
3. Click **View ➡ Lock ➡ All Views**, or  (not on default toolbars).
4. Click **View ➡ Lock ➡ All Animations**, or  (not on default toolbars).
5. Click **View ➡ Lock ➡ All Plots**, or  (not on default toolbars).

To review the fill time and weld line results

1. Click in the window containing the study **Tub no valve gates** to make it active.
2. Rotate and magnify the models so you can see both the top and sides.
   - A suggested rotation is -50 15 10.
3. Click on the **Fill time** result.
   - Because the windows are locked for views animations and plots, the fill time result is displayed with contour lines for both studies.
4. Right-click on the **Weld lines** result in the **Tub no valve gates** study and select **Overlay**.
   - Notice that the weld lines are gone for the **Tub with valve gates** study and the balance is very similar between the studies, as shown in Figure 130.

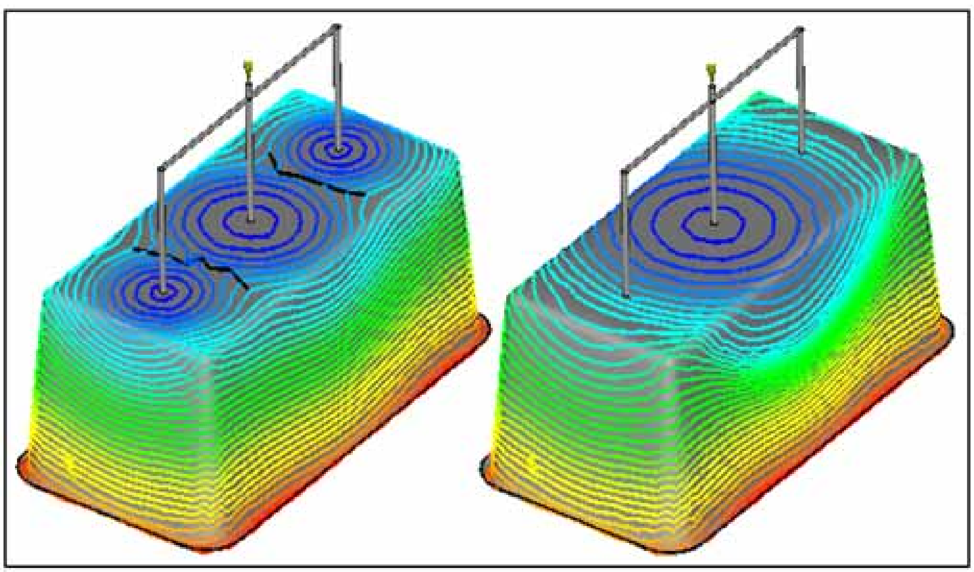
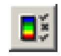


Figure 130: Filling pattern for the tub with and without valve gates

To review the following results for both studies

1. Plot each result:

- Scale them to the same range.
- Animate the results.
- Notice the differences.

1.1. To scale the result, open **Plot Properties**, .

1.2. Click on the **Scaling** tab.

1.3. Click **Specified**.

1.4. Enter the minimum and maximum value between both studies.

1.5. Click **OK**.

2. Plot the following results for each model:

2.1. Pressure at V/P Switchover.

2.2. Bulk temperature.

2.3. Frozen layer fraction.

2.4. Average velocity.

➤ Scale the maximum velocity to **75 cm/s** and keep the **extended color** on.

3. Check for the time that the valve gates opened in the Screen output for the **tub with valve gates** study. The output should look similar to Figure 131.

```
|-------------------------------------------------------------|
|   Time   | Volume|   Pressure    | Clamp force|Flow rate|Status |
|   (s)    |  (%)  |    (MPa)      |  (tonne)   |(cm^3/s) |        |
|-------------------------------------------------------------|
|    0.00 |   0.01 | valve gate =   2 (Elem =    7843) opened.|
|    0.20 |   1.81 |         20.31 |      10.07 |  575.53 |    V    |
|    0.38 |   5.64 |         27.36 |      44.19 |  722.64 |    V    |
|    0.57 |   9.92 |         32.48 |      97.13 |  766.26 |    V    |
|    0.76 |  14.32 |         36.63 |     171.01 |  768.13 |    V    |
|    0.95 |  18.76 |         40.28 |     244.76 |  781.75 |    V    |
|    1.13 |  22.97 |         43.59 |     321.18 |  788.89 |    V    |
|    1.18 |  24.21 | valve gate =   1 (Elem =    7840) opened.|
|    1.20 |  24.64 | valve gate =   3 (Elem =    7846) opened.|
|    1.32 |  28.55 |         32.90 |     305.76 |  928.01 |    V    |
```

Figure 131: Screen output showing valve gate timings

Competency Check - Using Valve Gates

1. What properties need to be changed or set to convert a gate to a valve gate?
2. How many elements at each gate need to be defined as a valve gate?

Evaluation Sheet - Using Valve Gates

1.	**What properties need to be changed or set to convert a gate to a valve gate?**
•	Select a valve gate controller.
•	Select the control method.
•	Set the parameters for the control method.
2.	**How many elements at each gate need to be defined as a valve gate?**
	At least one element must be used with a property of hot gate. If others elements are used in the gate, they must all have the same valve gate controller. If the control type “flow front” is used, only one element in the gate is allowed.

CHAPTER 20

Flow Leaders and Deflectors

Aim

The aim of this chapter is to use a flow deflector to move the location of a weld line.

Why do it

Flow leaders and deflectors are small changes in wall thickness to accomplish a balance or to move a weld line.

Flow leaders and flow deflectors can be useful tools to solve balance problems within parts or to move weld lines.

Design constraints sometimes prevent us from moving gates. For example, a particular part may have a shape that doesn't allow for a gate location that creates a balance in the part with a nominal wall thickness. In cases like this, flow leaders and deflectors can be used to move weld lines.

Overview

In this chapter, you will move the weld line around a square hole from the center of the hole's edge to a corner. You will achieve this by creating changing wall thickness to form a flow deflector in the relevant section of the model.

Moving the weld line makes the weld line less visible to the user because of its location and the angle at which it is formed.

In this practice, you will perform the following:

- Run a filling analysis.
- Review filling results.
- Review the weld line location.
- Create the flow deflector by decreasing the wall thickness.
- Re-run and review results.

Practice - Flow Leaders and Deflectors

Design criteria

The gate location on the window cover model is fixed. The weld line formed with the nominal wall of the part and the gate location is on the far side of the window from the gate. At this location, the weld line formed is quite noticeable and weaker. The weld line needs to be moved to the upper right corner of the window. This will be accomplished by adding a flow deflector.

Two different thicknesses for the flow deflector will be evaluated.

Figure 132: Window cover model

Setup

To open a project

1. Click the file open icon  or **File ➡ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Flow_Leaders**.
2. Double click the project file **Flow_Leaders.mpi**.
3. Click **File ➡ Preferences** and ensure that **System Units** are set to **Metric**.

To review the model

1. Open the model **Window Cover**.
2. Investigate the model geometry using the **model manipulation** tools.
3. Turn on and off the layers.
   - Notice the flow deflector has its own layer to facilitate changing the thickness.

The first flow analysis

The first flow analysis is run on a part with a uniform wall thickness of 2.54mm.

To run a filling analysis

1. Select the material.

1.1. **Lanxess** as the Manufacturer.

1.2. **Lustran 450C** as the Trade name.

2. Double-click the **Process Settings Wizard**.

3. Set the process settings to the values listed in the table below, leaving all others at default values.

Parameter	Value
Mold surface temperature	65º
Melt temperature	250º
Filling Control	Fill time
Fill time	2.5 Seconds

4. Click **Analyze Now**.

- The analysis should take 10 minutes or less to complete.

5. Uncheck the **logs** box when the analysis is complete.

Review the results

To display the fill contours

1. Click on the plot the **Fill Time** result.

2. Click **Results** ➔ **Plot Properties**.

2.1. Click the **Methods** tab.

2.2. Click the **Contour** radio button.

2.3. Click the **Mesh Display** tab.

2.4. Click the **Element lines** option in the **Edge Display** area.

2.5. Click **OK**.

3. Animate the result.

To overlay the weld lines

1. Right-click the **Weld Lines** result.

2. Select **Overlay**.

- Notice the weld line appears is displayed over the fill time plot.

3. Zoom up on the location of the weld line.

4. Open the **Plot Properties** for the weld line.

   4.1. Click the **Highlight** tab.

   4.2. Click the icon .

   4.3. Select **Temperature at flow front**.

   4.4. Click **OK** twice, to exit plot properties.

   - The color scale for the weld line represents the temperature at which the weld line is formed. It should be very near the melt temperature. If the temperature drops too much, the weld line may be too weak.

To plot frozen layer fraction at end of fill

1. Click on the **Frozen layer fraction at end of fill** plot to display the result.
2. Query the Frozen layer fraction at the vicinity of the weld line.
   - The frozen layer fraction should be low and uniform on both sides of the weld line.

Results discussion

The fill time and weld line results show the location of the weld line as seen in Figure 133. The weld line is not perfectly straight or directly opposite the gate because the location of the weld line is predicted by node locations, and the node locations are not symmetrical with the gate location and the hole.

The temperature at the flow front indicates what the midstream temperature when the weld line is formed. It is quite good, as it is just over 250ºC.

The frozen layer fraction shows no problems either. The maximum value is around 0.06, which is very low.

The only problem is the location of the weld line.

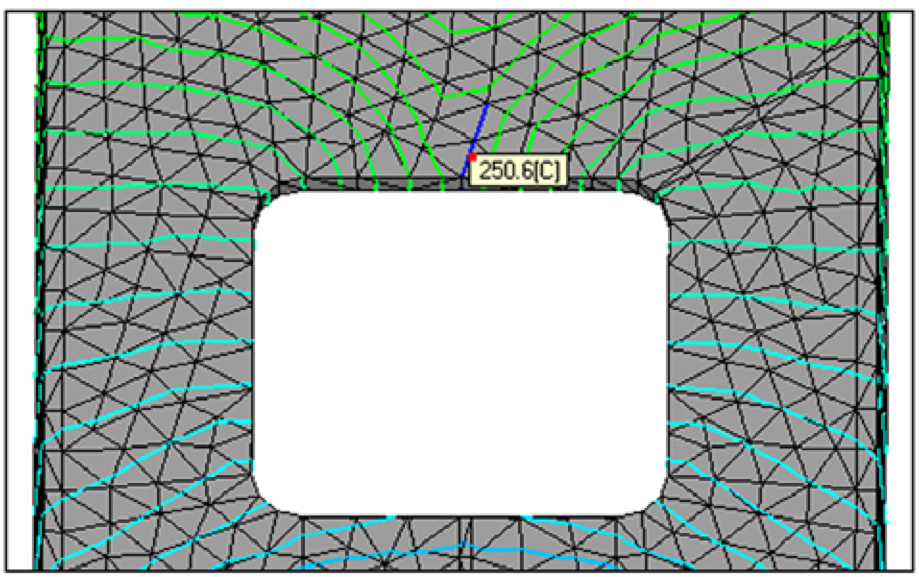


Figure 133: Original weld line location with nominal wall

Run a second flow analysis

The second flow analysis is run on a part with a 1.90 mm wall thickness for the flow deflector and a 2.5 mm nominal wall.

To change the thickness of the flow leader

1. Turn off all layers except the flow deflector.
2. Select one of the elements in the flow deflector.
3. Right-click and select **Properties**.
4. Set the thickness to **1.9** mm in the Part surface (fusion) dialog.
5. Ensure the **Apply to all entities that share this property** box is selected.
6. Enter **Flow Deflector 1.9mm** in the Name field**.**
7. Click **OK**.
8. Click the **Create Copy** button to keep the original model and save a new model with the thickness change.

To run a filling analysis

1. Save the new model with the name **Window Cover FD190**.
2. Use the same material and processing conditions as in the previous analysis:
3. Double-click **Analyze Now** to start the filling analysis.
    - The analysis should take 10 minutes or less to complete
4. Uncheck the **logs** box when the analysis is complete.

Review the results

To display the Fill Time contours

1. Display the **Body** layer again.
2. Click the **Fill Time** result.
3. Use **Plot properties** to set the results to contours.

To overlay the Weld Lines result

1. Overlay the **Weld Lines** result.
2. Open the **Plot Properties** for the weld line.

    2.1. Click the **Highlight** tab.

    2.2. Click the icon .

    2.3. Select **Temperature at flow front**.

    2.4. Click **OK** twice, to exit plot properties.

To plot Temperature at Flow Front result

1. Click on the **Temperature at Flow Front** plot to display the result.
   - Note the difference in temperature at the weld line.

To plot frozen layer fraction at end of fill

1. Click on the **Frozen layer fraction at end of fill** plot to display the result.
2. Query the Frozen layer fraction at the vicinity of the weld line.
   - The frozen layer fraction should be low and uniform on both sides of the weld line.

Results discussion

The fill time and weld line results show the location of the weld line as seen in Figure 134. The weld line has moved closer to the corner but not enough.

The temperature at the flow front indicates what the midstream temperature when the weld line is formed. It is still good, as it is over 250ºC. There is a slight difference in temperature between each side of the weld line due to the flow rates of the flow fronts that formed the weld line.

The flow deflector has a noticeable effect on the frozen layer fraction. The flow deflector has the thickest frozen layer fraction in the part, but the maximum value is under 0.1. This value is reasonably low.

The main problem is the location of the weld line has not moved enough.

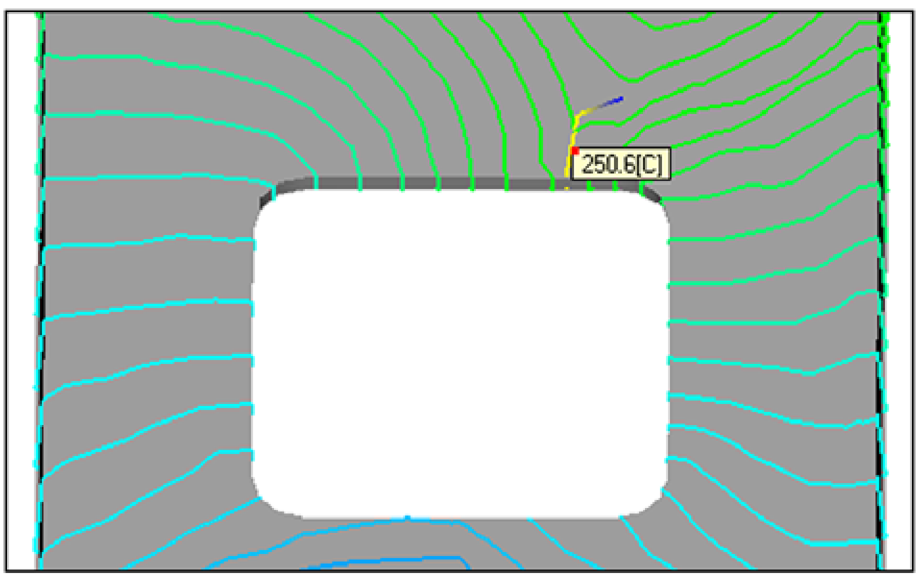


Figure 134: Weld line location with 1.9 mm deflector

Run a third flow analysis

The third flow analysis is run on a part with a 1.27 mm wall thickness for the flow deflector and a 2.5 mm nominal wall.

To change the thickness of the flow leader

1. Turn off all layers except the flow deflector.
2. Select one of the elements in the flow deflector.
3. Right-click and select **Properties**.
4. Set the thickness to **1.27** mm in the Part surface (fusion) dialog.
5. Ensure the **Apply to all entities that share this property** box is selected.
6. Enter **Flow Deflector 1.27mm** in the Name field.
7. Click **OK**.
8. Click the **Create Copy** button to keep the original model and save a new model with the thickness change.

To run a filling analysis

1. Save the new model with the name **Window Cover FD127**.
2. Use the same material and processing conditions as in the previous analysis:
3. Double-click **Analyze Now** to start the filling analysis.
    - The analysis should take 10 minutes or less to complete
4. Uncheck the **logs** box when the analysis is complete.

Review the results

To display the Fill Time contours

1. Display the **Body** layer again.
2. Click the **Fill Time** result.
3. Use **Plot properties** to set the results to contours.

To overlay the Weld Lines result

1. Overlay the **Weld Lines** result.
2. Open the **Plot Properties** for the weld line.

    2.1. Click the **Highlight** tab.

    2.2. Click the icon .

    2.3. Select **Temperature at flow front**.

    2.4. Click **OK** twice, to exit plot properties.

To plot Temperature at Flow Front result

1. Click on the **Temperature at Flow Front** plot to display the result.
   - Note the difference in temperature at the weld line.

To plot frozen layer fraction at end of fill

1. Click on the **Frozen layer fraction at end of fill** plot to display the result.
2. Query the Frozen layer fraction at the vicinity of the weld line.
   - The frozen layer fraction should be low and uniform on both sides of the weld line.

Results discussion

The fill time and weld line results show the weld line has moved to the corner as seen in Figure 135. The weld line then turns and heads down the part because of the sidewalls have not changed in thickness. The temperature at the flow front indicates what the midstream temperature when the weld line is formed. It is good because the temperature is above 249ºC, but the effect of the deflector is becoming apparent.

The flow deflector has a significant effect on the frozen layer fraction. The flow deflector has the thickest frozen layer fraction in the part, with the maximum value of 0.15, which is starting to become high.

The weld line has been moved to the desired location. Now you need to decide if the wall thickness change is too significant. A fill analysis by itself can't answer that question. You can see that the frozen layer fraction and temperatures are affected, but you should also consider the shrinkage and warpage of the part. Also, find out if there any structural or impact requirements the thinner wall may have a problem with. These questions need to be answered in consultation with the part design engineer among others.

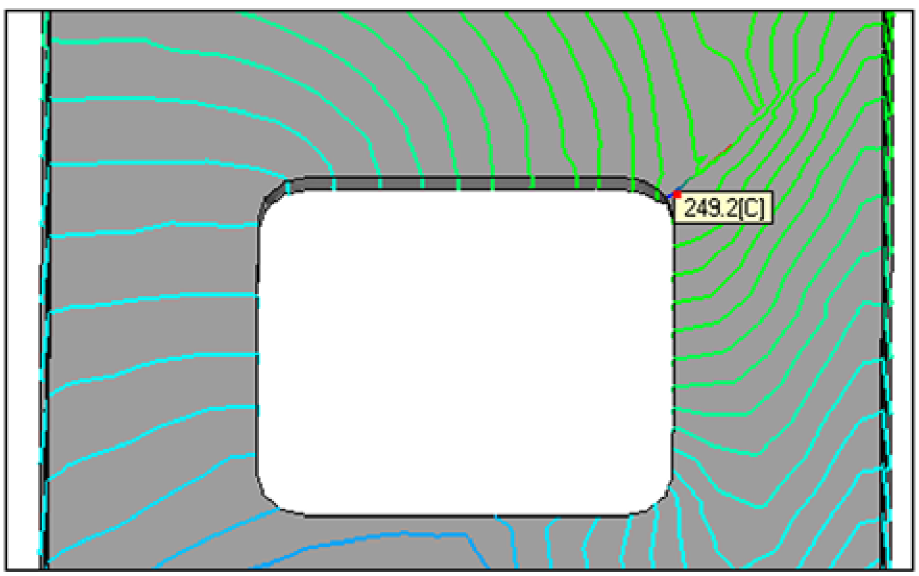


Figure 135: Weld line location with a 1.27 mm deflector

Competency Check - Flow Leaders and Deflectors

What is the procedure for changing/creating a flow leader?
1.
2.
3.
4.
5.

Evaluation Sheet - Flow Leaders and Deflectors

What is the procedure for changing / creating a flow leader?
1. Change the thickness and run the analysis again as necessary
2. Assign the elements to a new layer.
3. Create a new element property to define the thickness
4. Run a flow analysis to see the effect of the thickness change.
5. Change the thickness and run the analysis again as necessary

CHAPTER 21

Flow Analysis Process Settings

There is no practice for this subject.

CHAPTER 22

Creating Reports

Aim

The aim of this chapter is to learn about the features of the Report Generation Wizard to quickly make HTML, Power Point or Word Document reports.

Why do it

Using the Report Generation wizard is a very quick and easy way to create reports that can then be sent to anyone with an Internet browser or Microsoft office (Word or Powerpoint). There are a few options that can make the report easier to generate and more customizable which you will learn after completing this chapter.

Overview

The basic process for creating a report include the following steps:

1. Start the Report Generation Wizard.

2. Pick studies to include.

3. Pick the plots to include.

4. Select the format for the report.

5. Generate the report.

This chapter will take these simple steps and expand them.

Practice - Create Reports

Design Criteria

Two analyses were done on the lid with different gating locations.

1. Create a report to describe the advantages and disadvantages of each gate location.
2. Include your recommendations for continued analysis.

The goal is to decide on the gate location so a tool design can be developed for the part. The main quality criterion is flatness.

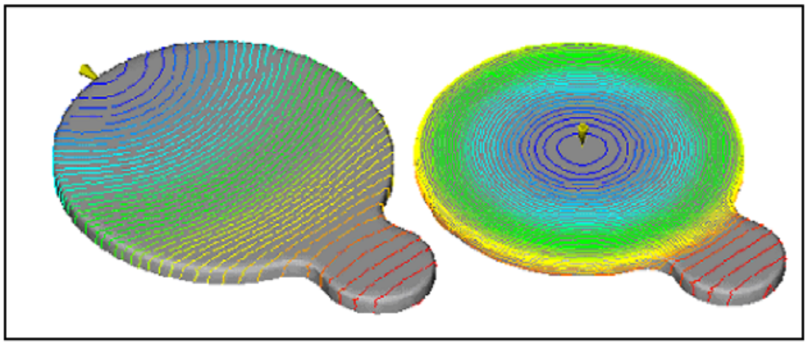


Figure 136: Two gate locations for a lid

Setup

To open a project

1. Click the file open icon or **File ➡ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Create_Reports**.
2. Double click the project file **Create_Reports.mpi**.
3. Click **File ➡ Preferences** and ensure that **System Units** are set to **Metric**.

To review the models

1. Open both models.
2. Tile the results vertically or horizontally using the appropriate tool in the **Windows** menu.
3. Click **View ➡ Lock** and lock the windows for:
   - Views.
   - Animations.
   - Plots.
4. Investigate the model geometry using the **model manipulation** tools.

Creating a report

To view the results

1. In the Study Tasks pane, plot the following results to interpret the results for both parts, using the property modification shown in Table 34.
   - Fill time.
   - Average velocity.
   - Pressure at injection location: XY Plot.
   - Frozen layer fraction (end of filling).
   - Shear stress at wall.
   - Volumetric shrinkage (at ejection).
   - Screen output.
   - Any other plots as necessary.
2. After viewing each result, decide which gate location is the best.
3. Right-click on each result in each study listed above and select **Plot notes**.

   3.1. Write a brief description about the definition of the result and it's significance.

Table 34: Customized images required

Result	Properties
Fill time.	Method of contour.
Average velocity.	Scaled with a maximum value of 100 cm/s and with Extended Color checked. Save the file at about 1.05s.
Pressure at injection location: XY Plot.	Default properties.
Frozen layer fraction (end of filling).	Default properties.
Shear stress at wall.	Scaled with a minimum value of 0.1 MPa, (shear stress limit), Extended Color deselected, and Nodal Averaging deselected. Save the file at a time where the shear stress is highest.
Volumetric shrinkage (at ejection).	Default properties.
Any other plots as necessary.	As needed.

To create the report

1. Click on the **Grabit_Center_Gate** study to make it the active study.
2. Click the **Fill time** result to activate it.
3. Rotate, pan, and zoom as necessary, to get the part at the size and rotation you want for the report.
4. Right-click the **Fill time** result in the Study tasks list and select **Add report image**.

   4.1. Accept the default name.

   4.2. Accept the default screen shot properties.

   4.3. Ensure an image icon has been added to the study tasks list under Fill time.

5. Add an image for each plot for both studies.

To edit the report

1. Double-click the **Report** icon in the Project view.
2. Click **Next** twice to get to the third page.
3. Check **Cover Page** and edit the properties as necessary.
4. Click and drag the items in the list of outputs to get the order of the results the way you want.
   - As an example, you may want to have the results in the order shown in Table 34, with the center gated part first and the end gate second.
5. Click Descriptive text for each plot so the plot notes will be added.
   - Click the Edit button to review your notes as necessary.
6. Click **Generate**.
   - The report will open up in synergy.
   - Review the report and make changes as necessary by editing the report again.

Competency check - Create Reports

1. What kind of formats can you create inside MPI?
2. What are the basic steps to follow to create a report?
3. When can you make changes to the report?

Evaluation Sheet - Create Reports

1. What kind of formats can you create inside MPI? • HTML. • Microsoft Word. • Microsoft PowerPoint.
2. What are the basic steps to follow to create a report? **1.** Start the Report Generation Wizard. **2.** Pick studies to include. **3.** Pick the plots to include. **4.** Select the format for the report. **5.** Generate the report.
3. When can you make changes to the report? At any time, the report updates automatically.

CHAPTER 23

Moldflow Communicator

Aim

To learn about the Moldflow Communicator functionality.

Why do it

By understanding the extended functionality that Moldflow Communicator offers, MPI users will be able to decide if using Moldflow Communicator will be more appropriate than an MPI report, for distributing information about a project.

Overview

In this chapter you will learn about the different features of the Moldflow Communicator and you will practice creating and using a Moldflow Communicator results' file. The sections covered in this chapter are:

- Moldflow Communicator introduction.
- Moldflow Communicator interface.
- Using Moldflow Communicator.
- Preparing the Moldflow results file and criteria file in Synergy.

Practice - Moldflow Communicator

Design criteria

There are two separate practices for Moldflow Communicator. The first will create a Moldflow Results file and Criteria file in Synergy. The second practice with review these results inside Moldflow Communicator.

For further practice on Moldflow Communicator usage you may complete the tutorial from the **Help ➡ Tutorials.**

Practice 1 - Creating results files in MPI for Moldflow Communicator

To open a project

1. Click the file open icon or **File ➡ Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Create_Reports**.
2. Double click the project file **Create_Reports.mpi**.
3. Click **File ➡ Preferences** and ensure that **System Units** are set to **Metric**.

> Any project that has results will work. You don't have to use the Create Reports folder.

To review the models

1. Open the study or studies from which you would like to create the results file.
2. Investigate the model or models' geometry using the **model manipulation** tools.

Selecting results for export

Moldflow results files are designed to be as small as possible by limiting the information that is in the file. One way this is done is to limit the results that are written to the Moldflow results file. Every result to be exported to the Moldflow results file must be marked for export in MPI.

To mark results for export to the Moldflow results file

1. Plot the result you would like to export.
2. Ensure the plot properties are correctly set.
3. Right click on the result of your interest from the first study. (For example **fill time** result).
4. Select **Mark for export** as shown in Figure 137.
   - An asterisk "*" is appended to the result name to indicate the result has been marked for export.

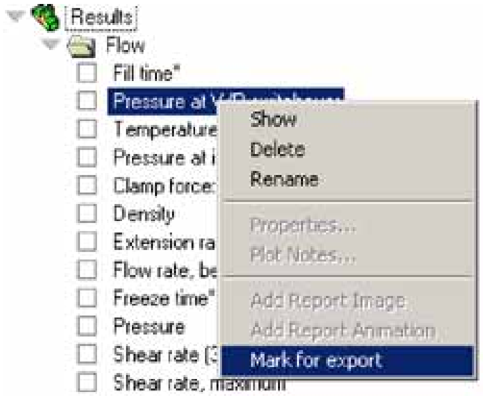


Figure 137: Result - Mark for export

5. **Repeat** steps 1 and 4 to mark for export the other results (you may mark up to 8 per study).
6. **Repeat** steps 1 thorough 4 for the other study if you have a second study in the project that you wish to include in the results file.

Creating plot notes

Plot notes must be created before the results file is created.

To create plot notes

1. Plot a result you would like notes on.
2. Click **Results ➡ Plot notes**.
3. Type the notes.
4. Click **OK**.

Entering project property information

Project property information includes data on who preformed the analysis. The information includes:

- Creator.
- Company.
- Contact information.

To enter project property information

1. Click **File ➡ Project Properties**.
2. Fill in the following fields:
    - Creator.
    - Company.
    - Contact Info.

Creating the Moldflow Results file

Once all the results are marked for export, and all the appropriate layers are turned on, the studies must be saved. Once saved, the command **File ➡ Export** is used to create the Moldflow results file.

To create a Moldflow results file

1. Ensure the layers displayed for the exported are only the ones you would like to export.
2. Select **both** studies to be exported in the Project view on the tasks pane in MPI.
3. Click **File ➡ Export**.
4. Navigate to the folder where the files are to be written.
5. Enter a name for the result file.

   5.1. Ensure the file type is “**Moldflow Results File**”.
6. Click **Save**.
7. Click **OK** to the message window on the screen.

Creating an Analysis Criteria file

A criteria file describes the values and ranges that attributes must have for the part to be considered of high quality. Values outside the desired range are highlighted in yellow when you compare a study with a criteria file.

To create a criteria file

1. Click **Tools ➡ Criteria editor...**
   - The criteria editor dialog opens (see Figure 138).
2. Activate the check boxes besides the names of the quality criteria you would like included in the criteria file.
3. Enter a value for the attribute, or choose a value from the drop-down list. A range can be specified by using a colon (:).
4. Click **OK** to save the criteria file.
   - A dialog appears asking for a name for the new criteria file.

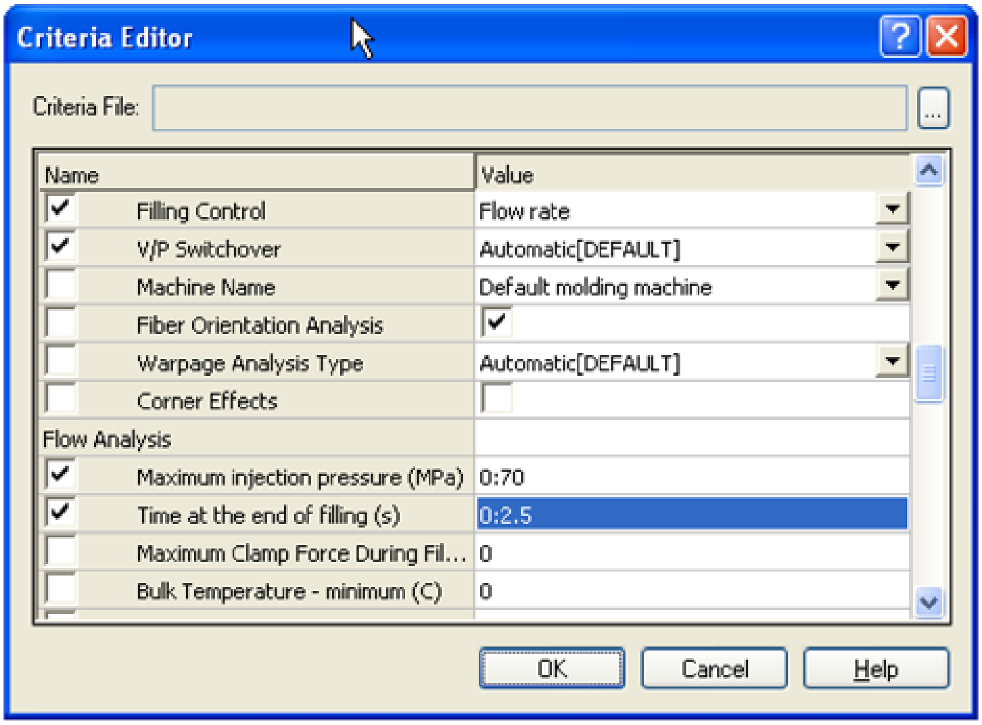


Figure 138: Criteria editor dialog

Practice 2 - Using Moldflow Communicator

Starting Moldflow Communicator

To open the Moldflow Communicator

1. Go to **Start** ➡ **Programs** ➡ **Moldflow Communicator 1.0** ➡ **Communicator 1.0.**
    - As soon as you open Moldflow Communicator the Launch wizard dialog appears, as shown in Figure 139.

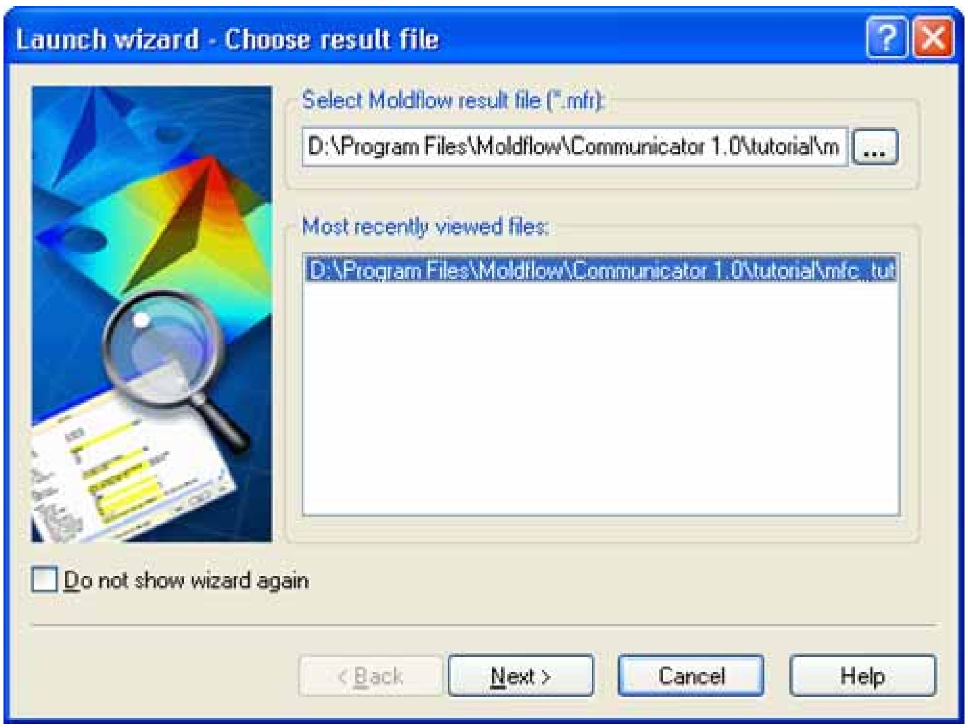


Figure 139: Launch wizard - Choose result file

Opening a Moldflow results file

To open the results file

1. Click the icon ... and Browse to the file you created in the first practice (*.mfr file).

If the Launch wizard dialog doesn't open, you can open browse for the result file by clicking on the icon Open or select **File** ➡ **Open Project.**

2. Highlight the file.
3. Click **Open**.
4. Click **Next** on the Launch Wizard.

5. Select **Compare results** from the Launch wizard - Select action page.

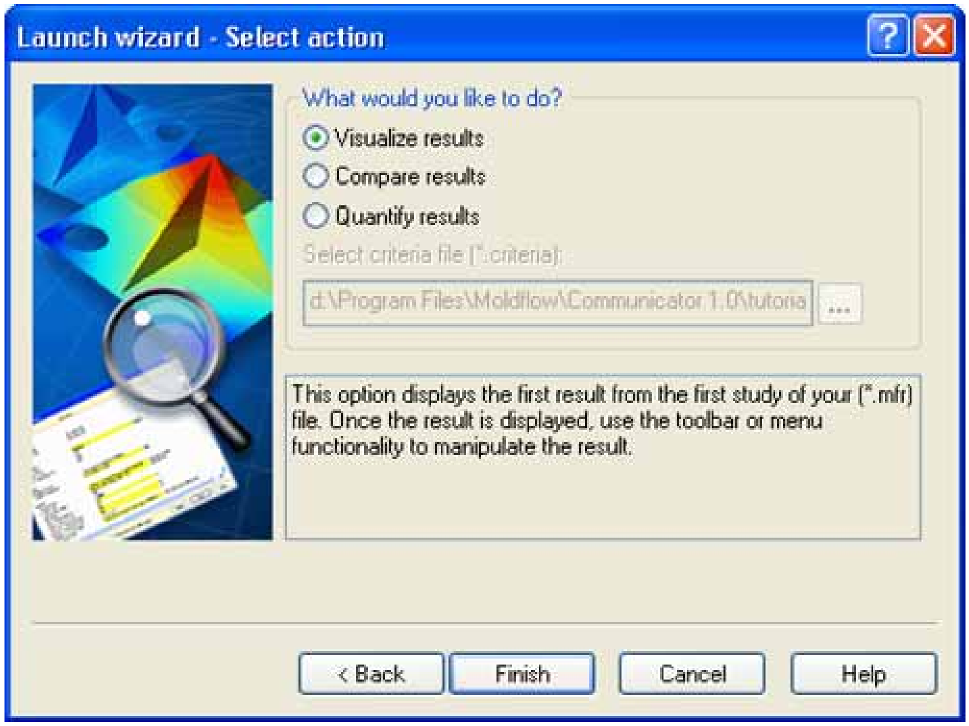


Figure 140: Launch wizard - Selection action

6. Click **Finish**.

- The program will open, tile, lock views and plots for both studies.

Comparing results

To view results

1. Ensure the Dynamic help is displayed on the right side of the screen.

1.1. If not, click **View** ➡ **Dynamic help**.

2. Click on a result.

3. Click the Animate icon 

.

4. Rotate, pan, zoom in on the model and use other animation tools as necessary to understand the results.

5. Click on all the results and animate them.

To display a clipping plane

1. Click on a result you want to use with the clipping plane.

2. Click the Clip icon 

.

3. Click the **Plane ZX** box.
4. Click **Flip**.

To move a clipping plane

1. Click **Make Active** on the Cutting Planes dialog.
2. Drag the cursor up and down.
3. Move the cutting plane towards one corner.
4. Inspect the hot core and cooler cavity.

Quantifying results

To quantify the results

1. Click the Quantify icon 

.
   - An Open dialog appears.
2. Navigate to the folder that has the criteria file you created in it.
3. Highlight the file the criteria file.
4. Click **Open**.
   - A dialog similar to Figure 141 will open.
5. Review the similarities and differences between the two studies and the criteria.

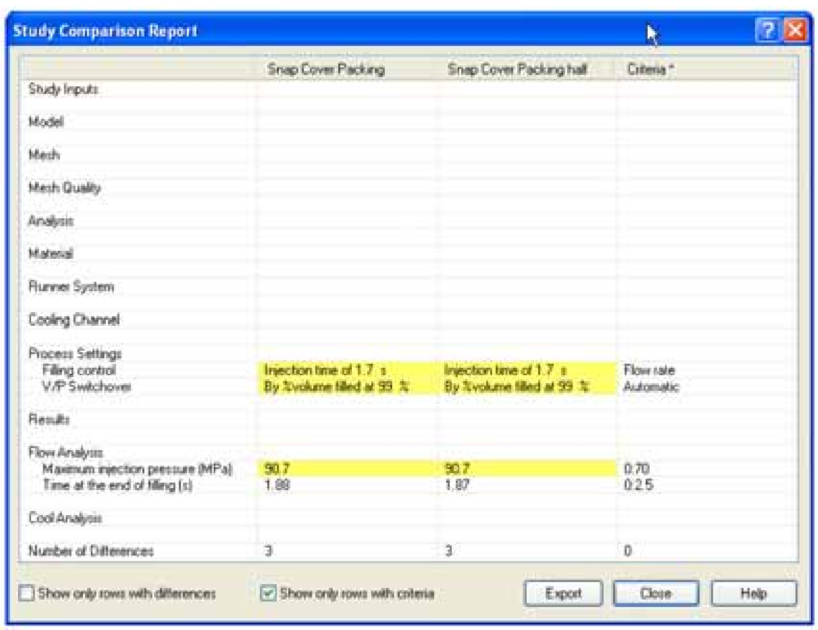


Figure 141: Study comparison report.

To view project properties

1. Click **File ➡ Project Properties**.
   - This opens a dialog shown in Figure 142.
2. Review the information.
3. Click **OK** when done.

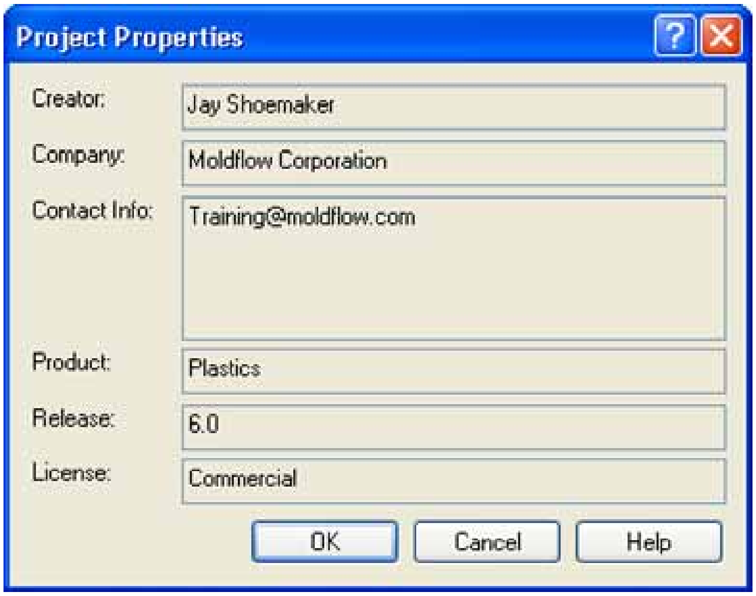


Figure 142: Project Properties

CHAPTER 24

Job Manager

There is no practice for this subject.

CHAPTER 25

Guided Project

Aim

The aim of this chapter is to work through an entire project; from receiving a CAD model and running different analysis, to finding and solving the problems associated with the part.

Why do it

By doing this in-depth practice of the entire analysis process, you will be much more confident in how the entire analysis process works, and will be able to apply the analysis procedure to your own part designs. This practice includes using the mesh tools to fix many kinds of model problems, using various analyses to identify filling-related problems, interpreting results to understand the problems, and test corrective actions to find a resolution. During the course of the chapter, you will employ Moldflow design principles to understand and solve the analysis problems.

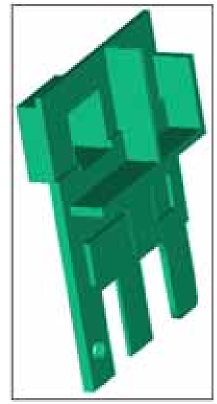
Overview

This is a quick look at the procedures involved in this project which include:

1. Understand the design criteria.
2. Import the model.
3. Check the model for mesh errors.
4. Fix the mesh errors.
5. Run a gate location analysis.
6. Run a molding window analysis.
7. Run a fill analysis.
8. Interpret the results to identify problems.
9. Determine possible solutions.
10. Change the model or process settings to fix the problem.
11. Run an analysis to verify the problem is fixed.
12. Create a runner system for the part.
13. Analyze the initial runner system design.
14. Use flow analysis to optimize the runner design.

Practice - Guided Project

Design criteria



For the base, shown on the right, the study file provided will be meshed, but will need to be cleaned up. The base will intentionally have errors in it. The typical cleanup process will involve using the mesh repair wizard. However for this part only manual cleanup methods will be used so there is extra practice with using mesh cleanup tools.

Once the base is cleaned, the base will be analyzed to determine the gate location, eliminate filling problems, determine optimum molding conditions, model and size the runners. Several flow analyses will be done to accomplish these tasks.

Project setup

To open a project

1. Click the file open icon or **File** ➡ **Open Project**, and navigate to the folder **My MPI 6.0 Projects\MPI_Fundamentals\Guided_Project**.
2. Double click the project file **Guided_Project.mpi**.
3. Click **File** ➡ **Preferences** and ensure that **System Units** are set to **Metric**.

Preparing the base for analysis

For most commands used in Synergy, there are two or three ways to access them. For the commands related to mesh diagnostics and cleanup for the base, the commands will be assessed via the **Toolbox** on the **Tools** tab, shown in Figure 143. These commands can also be accessed in the **Mesh** menu, and on various standard and customized toolbars. Use the method of access you prefer.

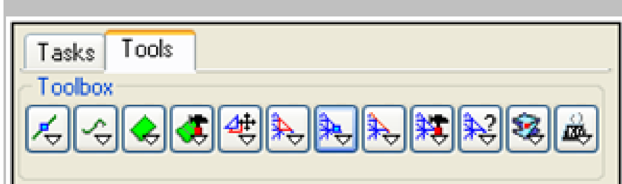


Figure 143: Toolbox

To open the base_mesh model

1. Double-click on the file **base_mesh** in the Project View pane.
2. Rotate the model around to review the geometry.
3. Turn on and off layers to see what they contain.

Check the model for errors

To check the mesh quality

1. Click **Mesh** ➡ **Mesh Statistics** or the icon .
2. Record the values in Column **Original Mesh** in Table 39 on page 493.
3. Fix the mesh errors

Fixing connectivity problems

To find connectivity problems

1. Click the Mesh Diagnostics icon in the toolbox and select **Connectivity Diagnostic**.
2. Select one element on the model.
   - Notice that the triangle element number is updated into the **Start connectivity search from entity** field.
3. Click **Show**.
4. Review the result.
   - The Mesh Connectivity diagnostic shows the elements that are connected to the single element you selected in blue. All elements that are not connected to the element selected are displayed in red.
   - There is more than one connectivity region due to an error in the original CAD model or the translation file cut in the CAD model. A second solid region was created and not merged into the first.
   - The elements that you need to delete are inside the fusion model. To correct the model, use the cutting plane feature.
5. Rotate the part to **125 5 0**, by using the **Enter Rotation Angle** field on the viewpoint toolbar.
6. Click the **Edit Cutting Plane** icon .

   6.1. Check the **Plane ZX** box.

   6.2. Click the **Make Active** button.

   6.3. Uncheck **Show active plane**.

   6.4. Click and drag the cutting plane so most of the unconnected finger is not visible as shown in Figure 144.

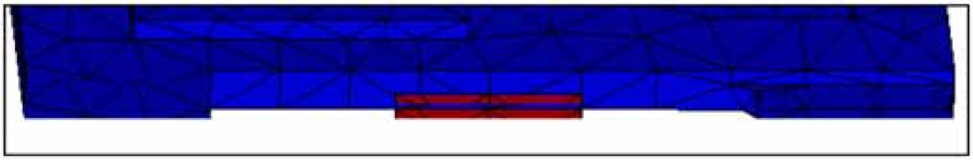


Figure 144: Cutting plane location

To delete the necessary elements

1. Rotate the part to about **100 0 0**.
2. Click the Global Mesh Tools icon  and select **Delete elements.**
3. Click the **Select Facing Items Only** icon  on the selection toolbar.
4. Band select inside the finger as shown in Figure 145 to select the elements to delete.
5. Click **Apply**.

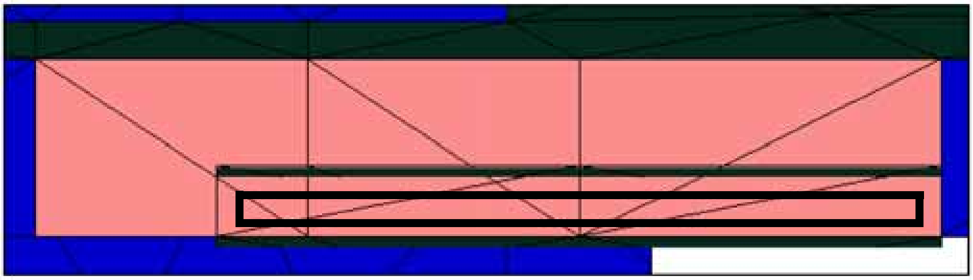


Figure 145: Selection in the finger

6. Rotate the model to about **-131 -170 2**.
7. Select the two elements that intersect the unconnected elements of the finger as shown in Figure 146.

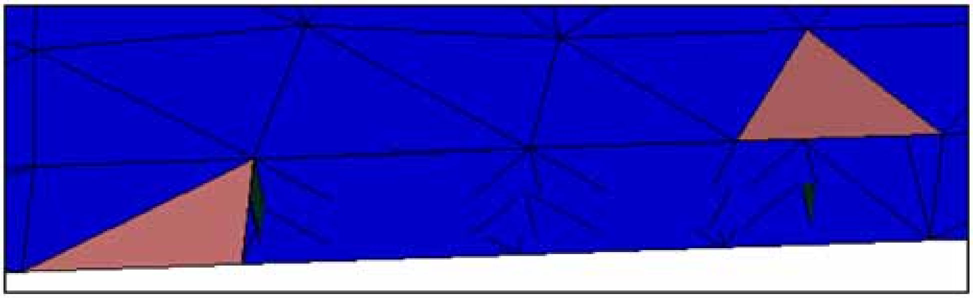


Figure 146: Two elements selected around the finger

8. Click Apply to delete them.

To fill the holes created

1. Click the Global Mesh Tools icon  and select **Global merge**.

2. Uncheck **Preserve Fusion**.

3. Click **Apply**.

- This will connect the middle finger with the rest of the model. The finger will turn blue to indicate it is now connected.

4. Click **Mesh ➡ Show Diagnostics**.

- This will toggle off the diagnostic display.

5. Click the Edge Mesh Tools icon  and select **Fill hole.**

5.1. Select the 4 nodes shown in Figure 147, forming a triangle on the top of the part.

5.2. Click **Apply**.

5.3. To fix the other hole, click on one of the nodes that is on the free edge (edge of hole.).

5.4. Click the **Search** key.

5.5. Review the highlight line to determine if the nodes selected form a proper edge.

5.6. Click **Apply**.

5.7. Select **Close** to close the **Tools** dialog.

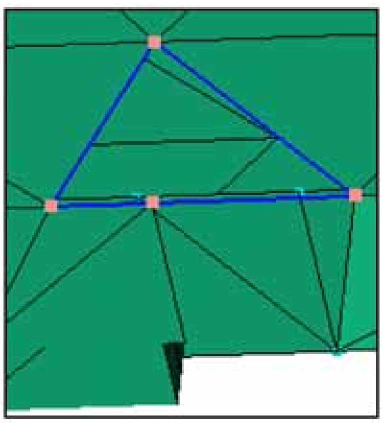


Figure 147: Fill hole node selection 1

6. Click the **Edit Cutting Plane** icon 

7. Uncheck **Plane ZX**.

7.1. Click **Close.**

- This removes the cutting plane from the display.

8. Click the icon  to save the model.

To display overlapping and intersecting elements

1. Click the Mesh Diagnostics icon  in the toolbox and select **Overlapping Elements Diagnostic.**

   1.1. Check the **Place results in diagnostics layer** box.

   1.2. Click **Show**.

2. Expand the diagnostics results layer.

   2.1. Select the **Diagnostics Results** layer.

   2.2. Click the Expand Layer icon .

   2.3. Click **OK** to accept the default value of one level.

3. Make the triangles transparent.

   3.1. Select the **New Triangles** layer.

   3.2. Click the Layer Display icon .

   3.3. Select **Transparent** in the Show as drop-down list.

   3.4. Click **Close**.

   - The overlapping elements are shown in Figure 148. Two areas are identified indicated with numbers 1 and 2.

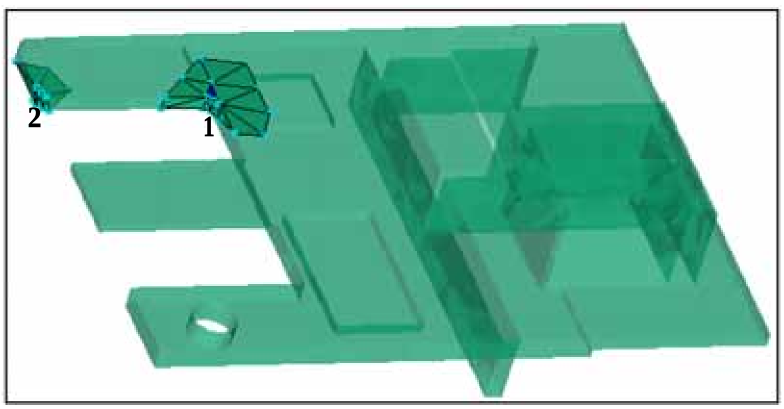


Figure 148: Overlapping elements

To correct overlapping element area 1

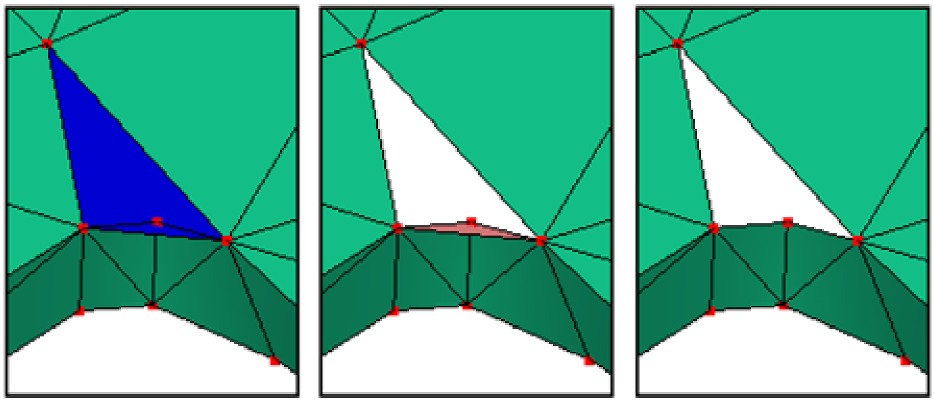
1. Refer to Figure 148 to identify the location of overlapping element area 1.
2. Uncheck the **New Triangles** layer.
3. Rotate the model to **-155 155 -45**.
4. Zoom in on problem area 1.

5. Click the Global Mesh Tools icon  and select **Delete elements.**

5.1. Select the blue element indicated in Figure 149A, and click **Apply**.

5.2. Select the second small element shown in Figure 149B, and click **Apply**.

- The hole should look like Figure 149C.



A
Original

B
Delete

C
Result

Figure 149: Problem area 1, first 3 steps to delete

6. Click the Nodal Mesh Tools icon  and select **Insert nodes**.

6.1. Select the two nodes indicated in Figure 150A, and click **Apply**.

➤ The inserted node is shown in Figure 150B.

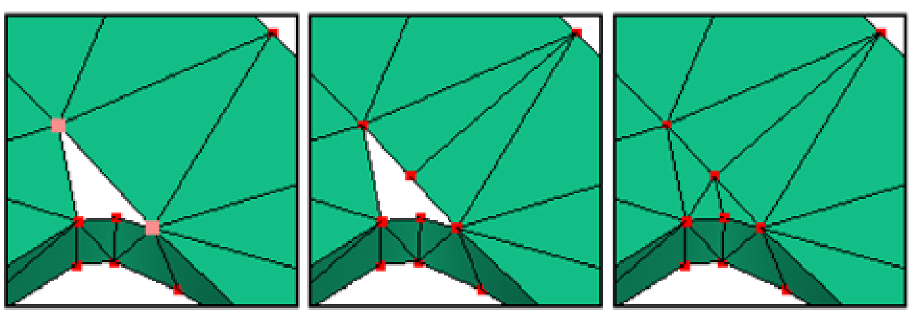
6.2. Click the Edge Mesh Tools icon  and select **Fill hole**.

6.3. Select one node on the edge of the hole.

6.4. Click **Search.**

6.5. Click **Apply**.

➤ The filled hole should look like Figure 150C.



A
Insert

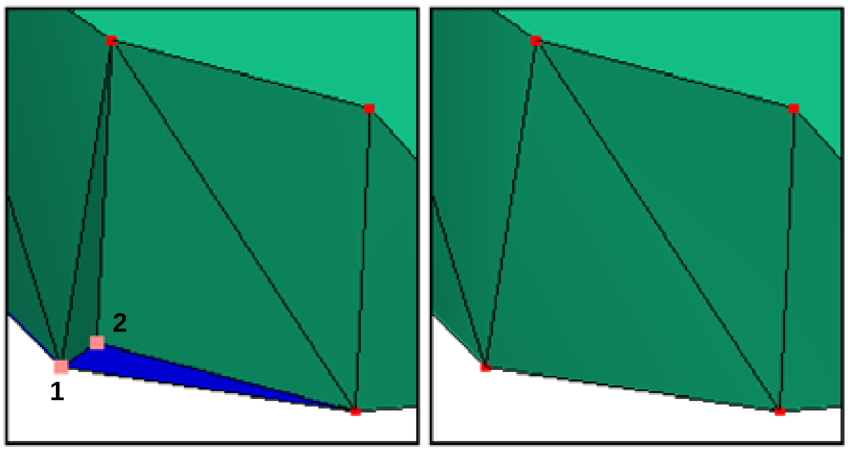
B
Insert Node Result

C
Fill Hole Result

Figure 150: Problem area 1, last 3 steps to delete

To correct overlapping element area 2

1. Refer to Figure 148 to identify the location of overlapping element area 2.
2. Click the **Go to next diagnostics** icon .
3. Click the Nodal Mesh Tools icon and select **Merge Nodes**.
   - **3.1.** Select the left node, labeled 1, then select the right one labeled 2, as indicated in Figure 151A.
   - **3.2.** Click **Apply.**
     - ➤ Merging the nodes fixes the overlap as shown in Figure 151B.
   - **3.3.** Click **Close**.
4. Save the model.



A Selected nodes	B Fixed model

Figure 151: Problem 2, Merge elements

Fixing free edges

To fix the free edges

1. Click the Mesh Diagnostics icon in the toolbox and select **Free edges Diagnostic**.
2. Check the **Place results in diagnostics layer** box.
3. Click **Show**.

4. Expand the diagnostics results layer.
   4.1. Select the **Diagnostics Results** layer.
   4.2. Click the Expand Layer icon .
   4.3. Enter **2** into the Expand Layer dialog.
   4.4. Click **OK**.
5. Right-click on the **Diagnostics results** layer and select **Hide All Other Layers**.
6. Click the Fit to Window icon , then rotate the model to see the problem area.
7. Click the Edge Mesh Tools icon and select **Fill hole**.
   7.1. Select one node on the edge of the hole.
   7.2. Click **Search.**
   7.3. Click **Apply**.

To check the mesh quality

1. Click **Mesh** ➡ **Mesh Statistics** or the icon .
2. Record the values in Column **In process** in Table 39 on page 493.

> This is an intermediate mesh check to confirm you have fixed all the intersections, overlaps and holes. The new problem, **Elements not oriented** will be fixed at the end of the mesh editing.

Fixing the Aspect ratio problems

To find high aspect ratio elements

1. Click the Mesh Diagnostics icon in the toolbox and select **Aspect Ratio Diagnostic**.
   1.1. Ensure 6 is in the Minimum field.
   1.2. Check **Place results in diagnostics layer**.
   1.3. Click **Show**.
2. Expand the diagnostics results layer.
   2.1. Select the **Diagnostics Results** layer.
   2.2. Click the Expand Layer icon .
   2.3. Click **OK** to accept the default value of one level.

3. Make the triangles solid with edges.

   3.1. Select the **New Triangles** layer.

   3.2. Click the Layer Display icon .

   3.3. Select **Solid + Element Edges** in the Show as drop-down list.

   3.4. Click **Close**.

4. Check the **New Nodes** layer to turn it on.

Many of the high aspect ratio elements are at the end of a rib. Several steps will be done to fix the aspect ratio problems in this area.

To add a node

1. Click the **Go to first diagnostics** icon .
2. Rotate the part to **30 175 -10**.
3. Zoom in or out so your image looks about like Figure 152.
4. Click the Measure icon and click on positions 1 then 2 as shown in Figure 152.
5. Click the Create Node Tools icon and select **Node by Offset**.

   5.1. Click on the node at position 1 in Figure 152 for the **Base** coordinate.

   5.2. Click on the **Offset** field.

   5.3. Add the offset dx dy and dz values **0 0 -4.5**.

   5.4. Click **Apply**.

   ➤ This creates a new node at the X and Y location of node 1 and the Z of node 2.

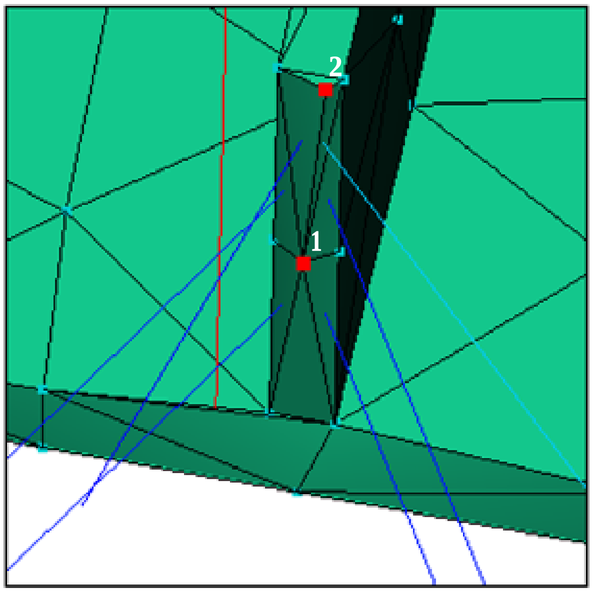


Figure 152: Node positions to measure

To merge nodes

1. Click the Nodal Mesh Tools icon and select **Merge Nodes**.
   - **1.1.** Select the new node just created.
   - **1.2.** Select the node labeled 2 in Figure 152.
   - **1.3.** Uncheck preserve fusion.
   - **1.4.** Click **Apply.**
     - ➤ The fixed corner looks like Figure 153.

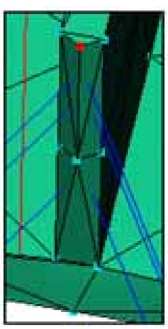
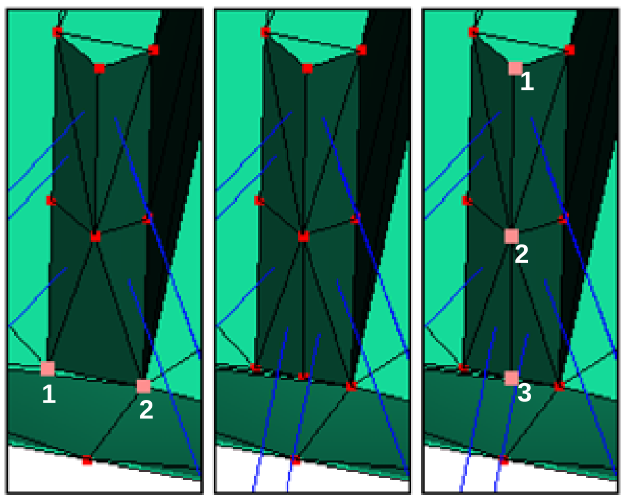


Figure 153: Merged node moves corner

To create a node at the base of the rib

1. Click the Nodal Mesh Tools icon  and select **Insert Nodes**.
   1.1. Select the node labeled 1 in Figure 154A.
   1.2. Select the node labeled 2 in Figure 154A.
   1.3. Click **Apply.**

2. Click the Nodal Mesh Tools icon  and select **Align Nodes**.
   2.1. Select the node labeled 1 in Figure 154C.
   2.2. Select the node labeled 2 in Figure 154C.
   2.3. Select the node labeled 3 in Figure 154C.
   2.4. Click **Apply.**
   - The aligned node should make the rib look like Figure 155A.

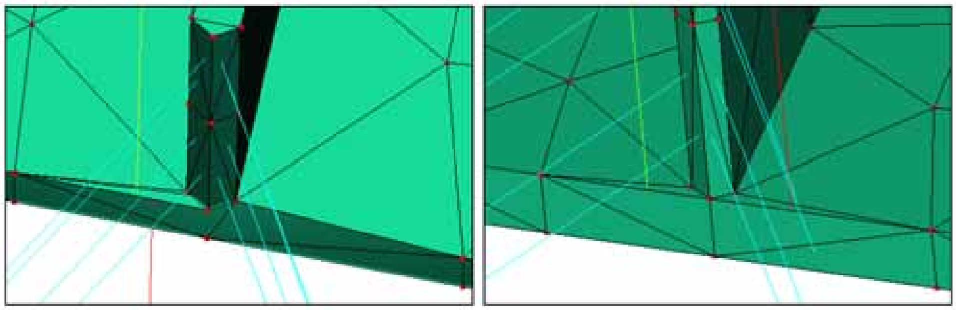


A Insert nodes | **B** Node created | **C** Align nodes

Figure 154: Inserting nodes on the rib

To properly connect the rib to the plate

1. Use mesh tools as necessary to fix the mesh so it looks like Figure 155B.
   - You can delete and create elements to make the changes or, insert, merge, and swap nodes.
   - Use any combination of commands you would like to use.
2. Save the model.



A
Node inserted

B
Rib to plate connection fixed

Figure 155: Rib connection to the plate

To insert nodes

1. Click the Nodal Mesh Tools icon and select **Insert Nodes**.

   1.1. Select the node labeled 1 in Figure 156.

   1.2. Select the node labeled 2 in Figure 156.

   1.3. Click **Apply.**

   - This forms the node labeled 4 in Figure 156.

2. Repeat the insert creation to create node 5 in Figure 156.
3. Insert two nodes on the other side of the rib, so both sides look the same.

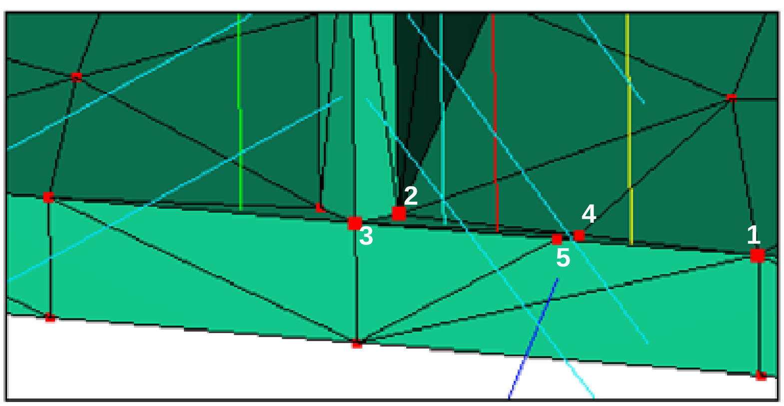


Figure 156: Nodes inserted on the plate edge

To merge nodes

1. Click the Nodal Mesh Tools icon and select **Merge Nodes**.

   1.1. Select the node labeled 5 in Figure 156.

   1.2. Select the node labeled 4 in Figure 156.

   1.3. Click **Apply.**

2. Repeat the node merging on the other side of the rib. The part should look like Figure 157.

To move nodes

1. Ensure the Aspect ratio diagnostic is on. If not, do the following:

   1.1. Click the Mesh Diagnostics icon  in the toolbox and select **Aspect Ratio Diagnostic**.

   1.2. Ensure 6 is in the Minimum field.

   1.3. Click **Show**.

2. Click the Nodal Mesh Tools icon  and select **Move Nodes**.
3. Select the node labeled 1 in Figure 157.
4. Drag the node closer to the rib.
5. Click **Apply**.
6. Ensure the element on the top of the part no longer has an aspect ratio line. If it does, move the node again.
7. Repeat for node 2 shown in Figure 157.
8. Save the model.

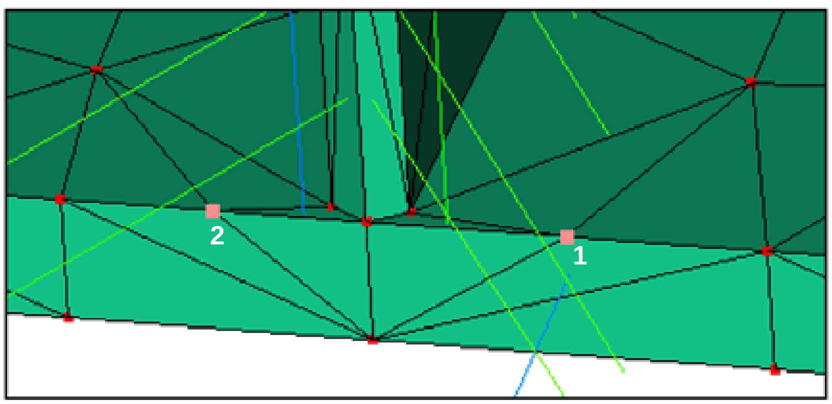
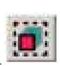


Figure 157: Nodes to be moved

To Remesh the end of the rib

1. Click the Global Mesh Tools icon  and select **Remesh Area.**
2. Select the elements at the front of the rib. The elements can be easily selected two ways:
   - Band select the elements on the edge of the rib with the **Facing Items Only** icon  on.
   - Band select the elements by banding around the selecting the high aspect ratio lines at a rotation nothing else will be selected, as shown in Figure 158.

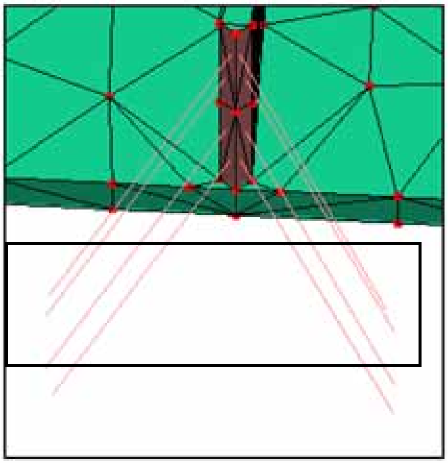


Figure 158: Band selecting using high aspect ratio lines

3. Enter **2.2** (mm) for Target edge length.
4. Click **Apply**.
   - The remeshed rib should look like Figure 159.

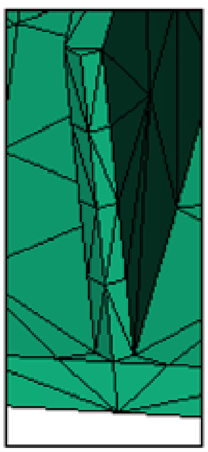


Figure 159: Rib edge remeshed

To remove other high aspect ratio elements

1. Use the Diagnostic navigator to move to other problem areas.
2. Use the necessary tools to fix the aspect ratio problems. Common tools to use include:
   - Merge nodes.
   - Swap edge.
   - Insert node.

3. Continue to fix elements until the aspect ratio is below 6:1.

- The scale on the screen will disappear when the aspect ratio is below 6:1.

If the diagnostic navigator buttons are grayed out but the scale has not disappeared, display the Aspect Ratio diagnostic again. There should be only one element to fix.

Fixing miscellaneous issues

To smooth nodes

1. Ensure all layers with triangles or nodes are on. This would include all layers except the Stl Representation layer. Turn on the layers by:

- Check all the layers individually, or
- Right-click on a layer and select **Show All Layers**.

2. Make the triangles transparent.

2.1. Select the **New Triangles** layer.

2.2. Click the Layer Display icon  .

2.3. Select **Transparent + edges** in the Show as drop-down list.

2.4. Click **Close**.

3. View the correct area on the part.

3.1. Click the Back view icon .

3.2. Pan to the lower left corner of the model.

4. Click the Global Mesh Tools icon  and select **Smooth Nodes.**

4.1. Select the 2 nodes labeled 1 and 2 in Figure 160.

4.2. Click **Apply**.

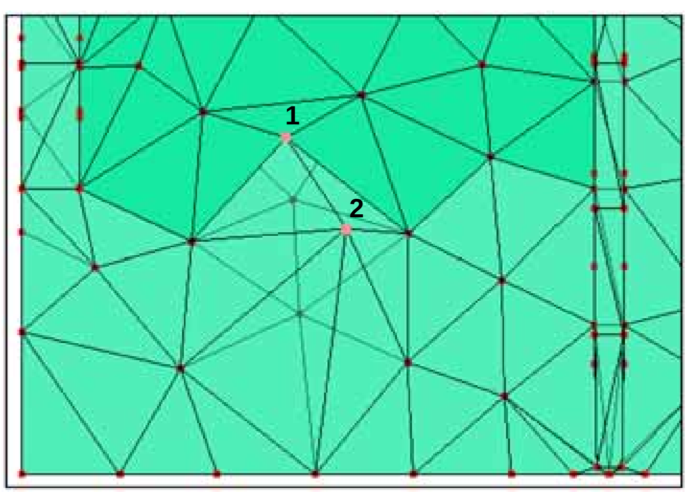


Figure 160: Nodes to be smoothed

To delete disconnected nodes

1. Click the Nodal Mesh Tools icon and select **Purge Nodes**.
2. Click **Apply**.
   - There may not be any disconnected nodes.
3. Click **Close**.

To reorganize the layers

1. Right-click on any layer and select **Show All Layers**.
2. Select by properties by one of the following methods:
   - Click the icon on the selection toolbar.
   - Click or **Edit** ➡ **Select By** ➡ **Properties**.
   - Enter **Ctrl + B**.

   2.1. Select **Nodes** in the By entity types window.

   2.2. Click **OK**.

   2.3. Select the layer **New Nodes.**

   2.4. Click the **Assign Layer** icon .
3. Click the icon on the selection toolbar, or any other method.

   3.1. Select **Triangle elements** in the By entity types window.

   3.2. Click **OK**.

   3.3. Select the layer **New Triangles.**

   3.4. Click the **Assign Layer** icon .
4. Highlight the **New Triangles** layer.

   4.1. Click the Layer Display icon .

   4.2. Select **Default** in the Show as drop-down list.

   4.3. Click **Close**.
5. Click the icon Clean layers .
   - This will delete all layers with no entities on it.

To fix the mesh orientation

1. Click the Global Mesh Tools icon and select **Orient all.**

To check the element mesh match ratio

1. Click **Mesh** ➡ **Mesh Statistics** or the icon .
2. Record the values in Column **Fixed mesh** in Table 39 on page 493.
   - The mesh match ratio should be above 90% for warpage.
   - All problems should be fixed now and the model should be error free!

To save the model

1. Click the icon or click **File** ➡ **Save Study**.
2. Click **File** ➡ **Close**.
   - A study is provided for the analysis section to ensure the model is consistent with the practice.

Part analysis

Now that the base mesh has been cleaned up, the analysis can be done. To ensure your model is consistent with the practice, a new model will be used. The analysis will include the following steps:

- Finding the gate location.
- Finding and solving fill problems.
- Optimizing the molding conditions.
- Modeling the runners.
- Running a Flow analysis with runners.

Finding the gate location

Introduction

In this section, you will perform a gate location analysis, and four filling analyses to determine the optimum gate location. All five analyses will be used together to decide on the gate location for the part.

To open the model

1. Right-click on the **Base_fixed** model in the Project View pane and select **Duplicate**.
2. Name the new study **Base_gate**.
3. Double-click on the file **Base_gate** to open the study.

To launch the gate Location analysis

1. Double-click the **Analysis Sequence** icon in the Study Tasks pane.

1.1. Select **Gate location.**

1.2. Click **OK**.

2. Double-click the **Select Material** icon in the Study Tasks pane and select the material:

2.1. Manufacturer: **Shell**.

2.2. Trade Name: **KMT 6100**.

2.3. Click the **Details button** and record the information requested in Table 40 on page 493.

2.4. Click **OK** twice to close the Details window and select the material.

3. Double-click the **Process Settings Wizard** icon .

- Notice that the recommended melt and mold surface temperatures are already specified.
- These temperatures will be used for the analysis.
- Record the values in Table 42 on page 494.

4. Click **Select** in the Injection Molding Machine frame.

4.1. Click on **Battenfeld BA 1300/400 BK** to highlight it.

4.2. Click **Details**.

4.3. Calculate the Maximum injection pressure by multiplying the Maximum machine hydraulic pressure and the intensification ratio.

4.4. Record the information requested in Table 41 on page 493.

4.5. Click **OK.**

4.6. Click **Select.**

4.7. Click **OK.**

- ➤ This closes the Process Settings wizard.
- ➤ A specific molding machine is not required for the gate location analysis, but this study will be duplicated for other analysis work. Once the injection molding machine is defined, it stays with the study. Using the exact molding machine is useful when doing a fill and flow analysis because you have the machine's actual capacity for pressure, clamp force, etc. plus the injection unit is defined which is needed for using ram speed profiles.

5. Double-click the **Analyze now!** icon.

   5.1. De-select **Run a full analysis**.

   5.2. Select **Run the analysis only**.

   5.3. Click **OK**.

   - When the gate location analysis is done, the recommended gate location is shown in the screen output.

To view the Screen Output

1. Record the **Near node** value (node number) in Table 42 on page 494.

To find the recommended gate location on the model by Query

1. Ensure that the New Triangles and New Nodes layers are visible.

2. Click the query entities icon or **Modeling ➡ Query Entities**.

   2.1. Enter **N** followed by the “near node” number determined in step 1.

   ➤ For example N4711.

   2.2. Click **Show**.

   - You may need to select and de-select the **New Triangles** layer to view the location of the queried node.
   - You may need to rotate the model to locate the node.

To find the recommended gate location on the model by graphic result

1. Click **Best gate location** in the study tasks list.

   - This will give you a general idea of the best gate location.

2. Right-click on the **Best gate location** result and select **Properties**.

3. Click the **Scaling** tab.

   3.1. Click the **Specified** button.

   3.2. Set the **Min** field to **0.95**.

   3.3. Uncheck **Extended color**.

4. Click the **Mesh Display** tab.

   4.1. Click the **Element lines** button.

   4.2. Click **OK**.

   - This will pinpoint the best gate location graphically.
   - Click the icon or click **File ➡ Save Study**.

Running filling analyses to investigate gate locations

To further investigate the best position for the injection location, you will run 4 fill analyses with different injection locations.

To create the four studies with different gate locations

1. Duplicate the study **Base_gate**.
2. Rename the study using the name **Base_gate_loc**.
   - **2.1.** Open the study.
   - **2.2.** Double-click the **Analysis Sequence** icon in the Study Tasks pane.
   - **2.3.** Select **Fill**.
   - **2.4.** Click **OK**.
   - This study will use the gate location determined by the best gate location analysis.
3. Duplicate the study **Base_ gate_loc**.
   - **3.1.** Rename the study using the name **Base_box_mid**.
   - This study will use a gate location in the center of the box like projection.
4. Duplicate the study **Base_ gate_loc**.
   - **4.1.** Rename the study using the name **Base_ box_edge**.
   - This study will use a gate location in the on the rim of the box.
5. Duplicate the study **Base_ gate_loc**.
   - **5.1.** Rename the study using the name **Base_hot_drop**.
   - This study will use a gate location on the parting line as close as possible to the best gate location result so a hot drop can be used.

To run the base_gate_loc fill analysis

1. Ensure the **Base_Gate_Loc** analysis is active.
2. Click the injection location icon in the study tasks list or click **Analysis ➨ Set Injection Locations.**
3. Set the injection location on the node that was determined as the best gate location.
   - Open the Base_Gate study if necessary to determine the correct node location.
4. Right-click the mouse and select **Finish Injection Locations** when you are done.
5. Double-click the **Analyze now!** icon .
   - **5.1.** De-select **Run a full analysis**.
   - **5.2.** Select **Run the analysis only**.
   - **5.3.** Click **OK**.

To run the base_box_mid fill analysis

1. Double-click the **Base_box_mid** study to open it.
2. Click the injection location icon in the study tasks list or click **Analysis ➡ Set Injection Locations.**
3. Set the injection location in the center of the box structure as shown in Figure 161.
4. Right-click the mouse and select **Finish Injection Locations** when you are done.
5. Double-click the **Analyze now!** icon .

   5.1. De-select **Run a full analysis**.

   5.2. Select **Run the analysis only**.

   5.3. Click **OK**.

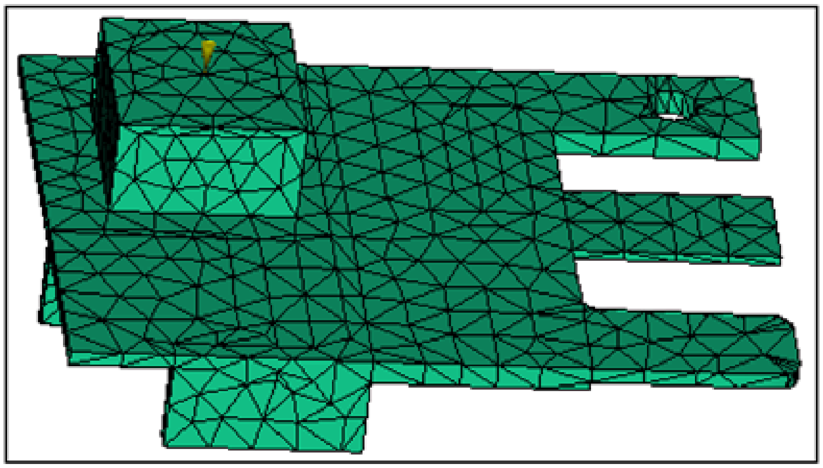


Figure 161: Gate location in the middle of the box

To run the base_box_edge fill analysis

1. Double-click the **Base_box_edge** study to open it.
2. Rotate the part to -175 -125 80.
3. Zoom up on the lower right corner of the box.
4. Click the injection location icon in the study tasks list or click **Analysis ➡ Set Injection Locations.**
5. Set the injection location on the edge of the box structure as shown in Figure 162.
6. Right-click the mouse and select **Finish Injection Locations** when you are done.
7. Double-click the **Analyze now!** icon .

   7.1. De-select **Run a full analysis**.

   7.2. Select **Run the analysis only**.

   7.3. Click **OK**.

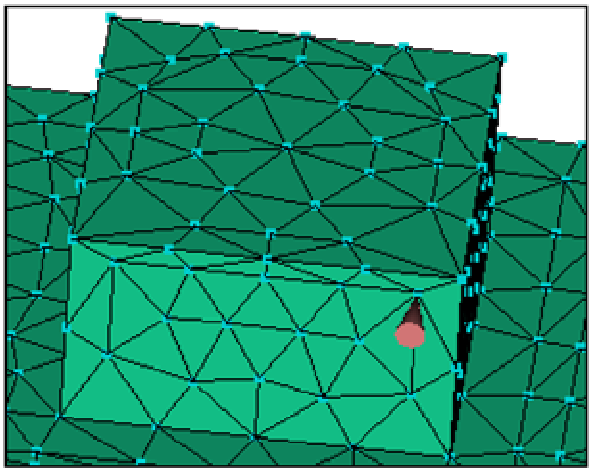


Figure 162: Gate location on the edge of the box

For the fourth model, you will create a new injection location at a realistic area considering the use of a hot drop. If you review the gate location results, you will see that the blue area is near the box. To add a runner, you need space for the hot drop, and you also cannot inject opposite a rib. For this gate location, a new node will be created.

To create the new node location

1. Double-click the **Base_hot_drop** study to open it.
2. Rotate the part to **5 -5 0**.
3. Zoom in on the center of the part.
4. Click the Nodal Mesh Tools icon  and select **Insert Nodes**.
   - **4.1.** Select the node labeled 1 in Figure 163.
   - **4.2.** Select the node labeled 2 in Figure 163.
   - **4.3.** Click **Apply.**
     - ➤ The inserted node is shown in Figure 164.

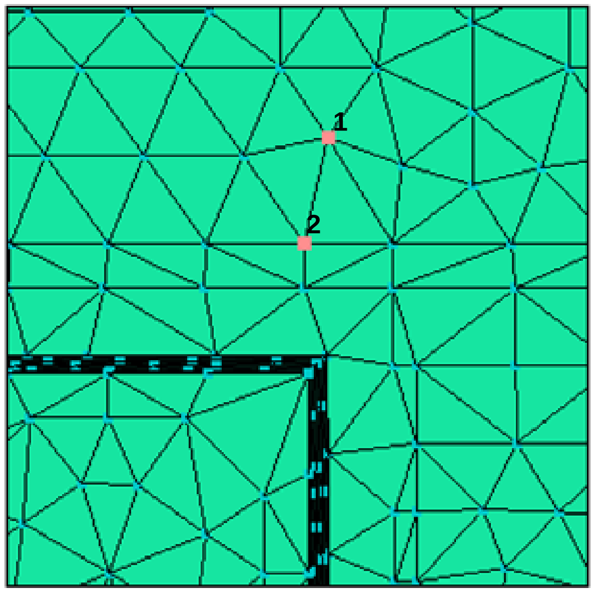


Figure 163: Selection to insert a node

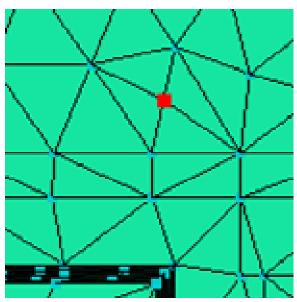
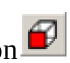


Figure 164: Inserted node

To match a node on the other side of the mesh from the new gate node

1. Click the front view icon  to rotate the part to 0 0 0.
2. Zoom up to the newly created gate node.
3. Make the triangles transparent.

   3.1. Select the **New Triangles** layer.

   3.2. Click the Layer Display icon  .

   3.3. Select **Transparent + Element edges** in the Show as drop-down list.

   3.4. Click **Close**.

4. Click the Nodal Mesh Tools icon and select **Match Nodes**.

   4.1. Select the new node created for the gate, shown in Figure 165.

   4.2. Click the Back view icon to rotate the part to 0 180 0.

   4.3. Use the Ctrl key and select the two elements on the opposite side of the selected node for the **Triangles for nodes to project to** field, as shown in Figure 165.

   4.4. Click **Apply**.

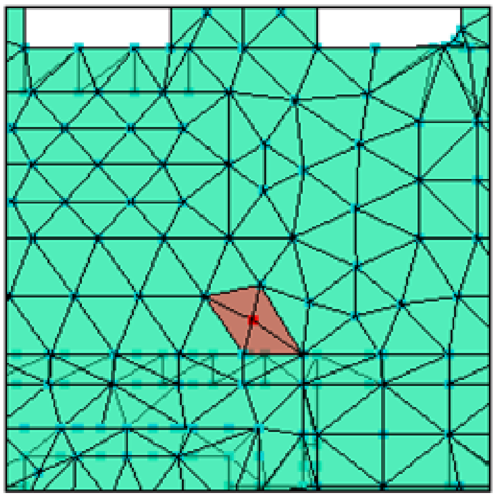


Figure 165: Selection for the mesh matching

5. Make the triangles solid.

   5.1. Select the **New Triangles** layer.

   5.2. Click the Layer Display icon .

   5.3. Select **Solid + Element edges** in the Show as drop-down list.

   5.4. Click **Close**.

To run the base_box_hot_drop fill analysis

1. Click the injection location icon in the study tasks list or click **Analysis ➡ Set Injection Locations.**
2. Set the injection location at the new node location shown in Figure 166.
3. Right-click the mouse and select **Finish Injection Locations** when you are done.
4. Double-click the **Analyze now!** icon .

   4.1. De-select **Run a full analysis**.

   4.2. Select **Run the analysis only**.

   4.3. Click **OK**.

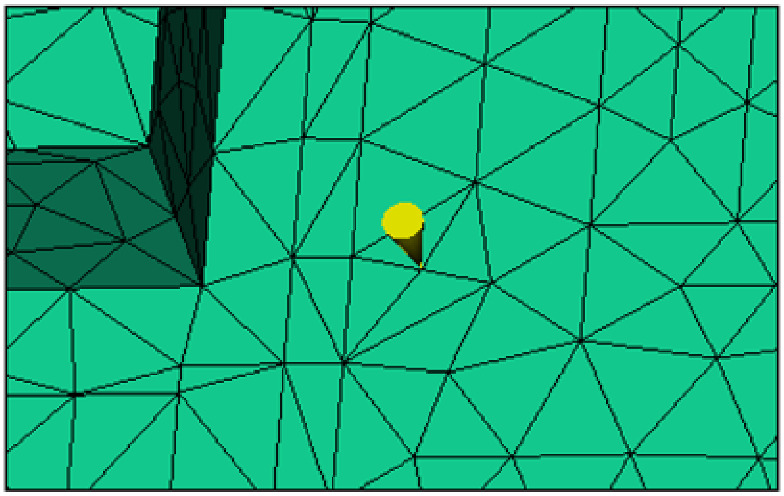


Figure 166: Injection location for the hot_drop

Review results from the four filling analyses

After all analyses have been completed, view the results. Look for the filling pattern, pressure variations, any weld lines and air traps, in order to find the gate location(s) that is the most desirable for this part. Later, we will optimize the process parameters for the chosen injection location.

To view the results summary file

1. Ensure all four filling analyses are open.
2. Click on the **Base_Gate_Loc** tab on the bottom of the Synergy window to activate that study.
3. Check the **Logs** button at the bottom of the Synergy window, as shown in Figure 167.
4. Click the Fill tab.
5. Scroll through the results summary file and find the listing for Maximum injection pressure, and record it in Table 43 on page 494.
6. Repeat this for the remaining analyses.

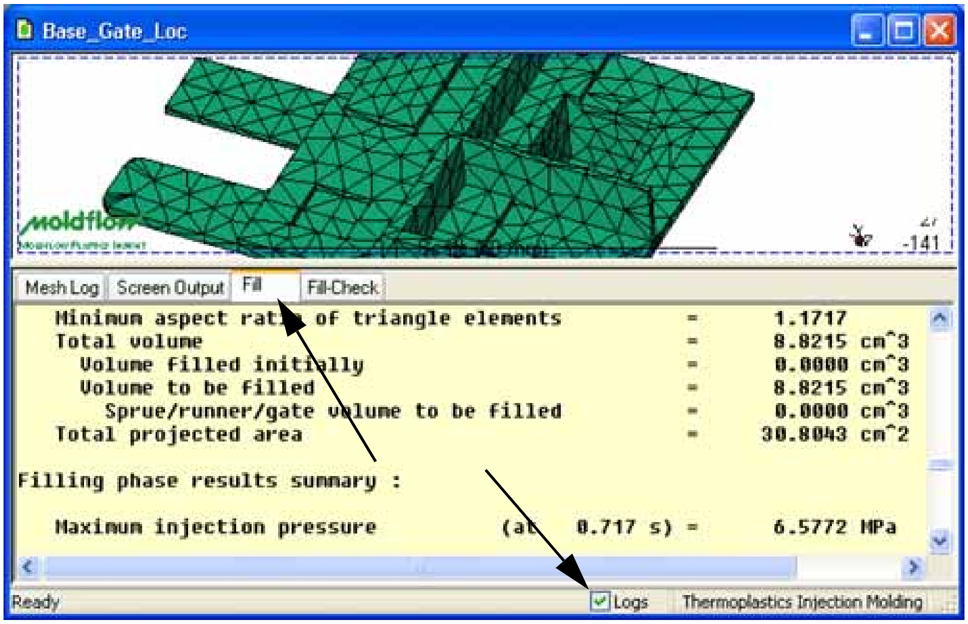


Figure 167: Results summary

To view the results graphically

1. Select **Windows ➨ Tile Horizontally** to view all four studies at once.
1. Uncheck the **logs** box for all the studies.
2. Click **View ➨ Lock ➨ All Views**.
   - This will lock the views so if one study is rotated, panned or zoomed, all will be manipulated at the same time and amount.
3. Click **View ➨ Lock ➨ All Animations**.
   - This will animate all the results at the same time and lock the time steps. This will only work for results that have the same number of animations defined. Sometimes intermediate results do not have the same number of recorded time steps even though the number of steps in the process settings are the same.
4. Click **View ➨ Lock ➨ All Plots**.
   - This locks the result and result properties for all the plots. If you select fill time for one, Synergy will plot fill time for them all. If a result property is changed in one plot, all locked plots are changed.
5. Click on the results listed in Table 35 and record your observations in Table 43 on page 494.
   - An example of plotting fill time is shown in Figure 168.
   - An example of an air trap is shown in Figure 169.

Table 35: Results to plot to evaluate gate location

Result	What to look for	Plot Property
Fill time	Is the flow pattern balanced or not, Y/N?	• Methods ➡ Contour, (Optional)
Air Trap	Are there air traps that can't be vented, Y/N?	
Weld Lines	Any objectionable weld lines caused by the gate position, Y/N?	
Shear stress at wall	Is the shear stress above the material limit, Y/N?	• Remove Lock all plots before adjusting scale. Only work on studies with stress higher than 0.25 MPa. • Scaling ➡ Min 0.25, • Scaling ➡ Extended color, Unchecked • Mesh Display ➡ Transparent • Optional Settings ➡ Nodal averaged, Unchecked

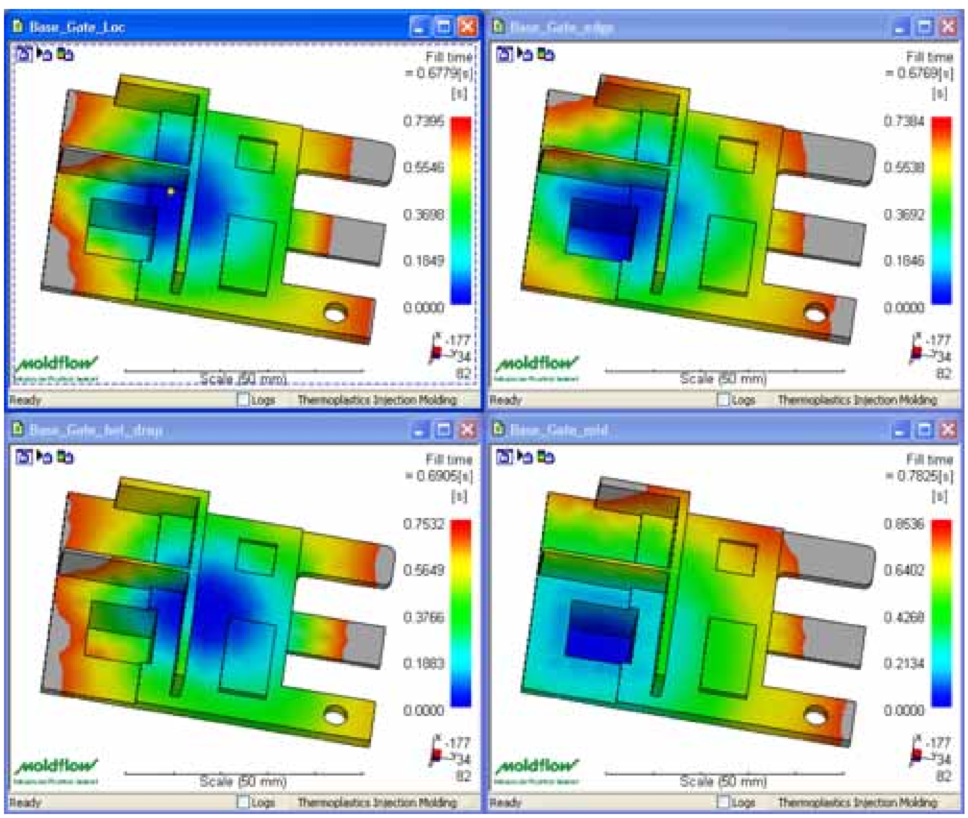


Figure 168: Example of fill time plots

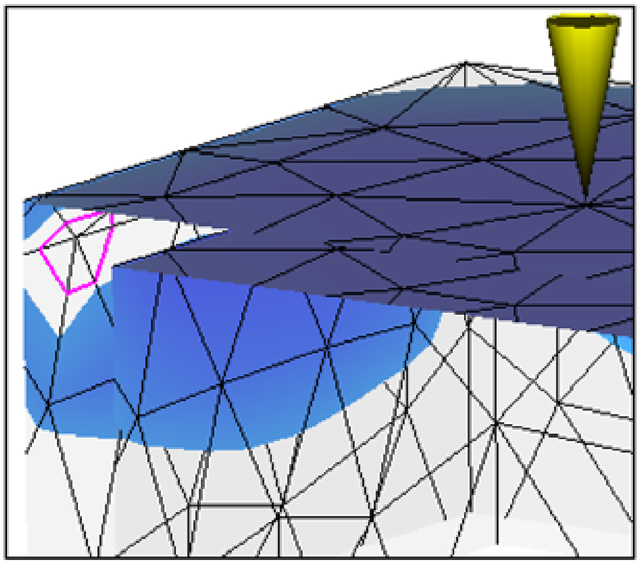


Figure 169: Air trap display, with fill time overlay and a cutting plane

Air trap explanation

For the study **base_box_mid**, air traps are predicted on the side of the box. There will not be an air trap on the actual part. This is caused by the Fusion mesh's unmatched elements in the corner. When viewing air traps on Fusion meshes, carefully look at the air trap to decide if the air trap is real or not.

To create Figure 169, the air trap result was overlaid the fill time result, with the model set to transparent, and the element lines on. In addition, cutting planes were turned on in two planes.

Discussion of results

The most important factor to consider is that the part should have a balanced filling pattern.

Weld lines influence the cosmetic and the stiffness of the part. If they cannot be removed, view the temperature at the position of the weld line with the result **Temperature at flow front**. Weld lines can be acceptable if the temperature is hot when it is formed. Also review the **pressure** result and look if the pressure is not too low after the formation of the weld line.

If air traps are present, then a flow leader can possibly be used to remove them. If this does not remove the air trap, you need to add a vent location.

It is also important to take into account that the pressure distribution does not cause a flash. The maximum pressure value during the filling analysis will not always indicate what machine size needs to be used, as it depends on the influence of clamp tonnage. Most presses, no matter what their clamp tonnage capacity, have approximately the same pressure capacity. If the clamp force drops enough, you can make a statement about the required injection molding machine. Qualify the statement by indicating it also has to do with how the part is packed out.

For the studies Base_gate_loc, Base_box_edge and Base_hot_drop, the weld line result does not show a weld line in the box region. However, the filling pattern shows a weld line will occur. The course mesh contributes to this issue.

To get a part without air traps and weld lines on the box, the **Base_box_mid** study can be used. However, this model displays an unbalanced filling pattern, which is a big disadvantage. Also the shear stress values are very high.

Base_gate_loc has the best balanced filling but the space for an runner system is to small. So we will have to look for a different gate location. **Base_box_edge** or **Base_hot_drop** are the last possible solutions. The fill pattern for **Base_box_edge** is more balanced compared to the **Base_hot_drop**.

Flow leaders or flow deflectors could possibly assist in removing the air trap and weld line at the box. If this is possible, it means that the study **Base_box_edge** is the best compromise of all four studies and will be used for the next analysis.

Identify and fix problems step by step

Introduction

After the injection location is selected, you should continue and remove any filling related problems associated with the part. In this section, you will attempt to remove the air traps and weld lines at the box. You will also identify if there are any other filling problems and try to remove them.

To open the model

1. Double-click on the study **Base_box_edge** to open the model if it is not already open.

To view the following results graphically

1. View the following results:
    - Fill time.
    - Weld lines.
    - Air traps.
    - Time to freeze.
    - Bulk temperature.
2. Identify the problems with the analysis.
3. Refer to the Results Interpretation training unit to help you to identify problems.
4. Compare your findings with the list in Table 36 on page 469.
5.

Table 36: Problems to be addressed with iterations

No	Result	Problem	Location	Propose	Solve in
1	Air traps	Air trap present	Box	• Create flow leaders and / or deflectors.	Iteration 1

Table 36: Problems to be addressed with iterations

No	Result	Problem	Location	Propose	Solve in
2	Time to freeze	Cooling time not uniform; very high value	Thick area (3.8mm)	• Decrease thickness	Iteration 2
3	Bulk temperature	Temperature decrease too much	Rib, finger with hole	• Increase wall thickness • Profile ram speed • Make the mold hotter at the middle finger	-
4	Weld lines	Weld lines present	At hole in finger	• Cannot be removed	-

Further work

The gate location on **Base_box_edge** is not possible to get to with a tunnel gate. You will move the gate location to up the wall of the box so a sub gate and cold runner can be used. The temperature will be influenced greatly by the injection time. After the air trap and cooling time problems are solved, you will run a molding window analysis to optimize the injection time which will minimize flow front temperature variation. A runner system will be added. The injection time will be modified to ensure the fill time of the part is still optimum.

Iteration 1 – Change box thickness

To open the model

1. Duplicate the model **base_box_edge.**
2. Change the name to **base_it1**.
3. Double-click on the file **base_it1** to open the model.

To create a new layer

1. Click the **New Layer** icon in the Layers pane.
2. Change the layer name to **Box**.

To assign element to the box layer

1. Click the Front View Icon on the Viewpoint toolbar to rotate the part to 0 0 0.
2. Click the Select Enclosed Items Only Icon on the Selection toolbar.
3. Zoom as necessary to see the Box.
4. If you are not sure where the box is, rotate the part a little then click the Front View Icon again.
5. Band-Select around the box as shown in Figure 170A.

6. Ensure the **Box** layer is highlighted and click the Assign Layer icon.
   - Some elements were moved to the box layer that should be on the New Triangles layer. These elements need to be moved back.
7. Move elements that don't belong on the **Box** layer.

   7.1. Rotate the model to **-90 -90 0**.

   7.2. Zoom up on the box.

   7.3. Band-select the two areas as shown in Figure 170B, by holding down the Ctrl key.

   7.4. Rotate the part to ensure that only elements not part of the box are selected.

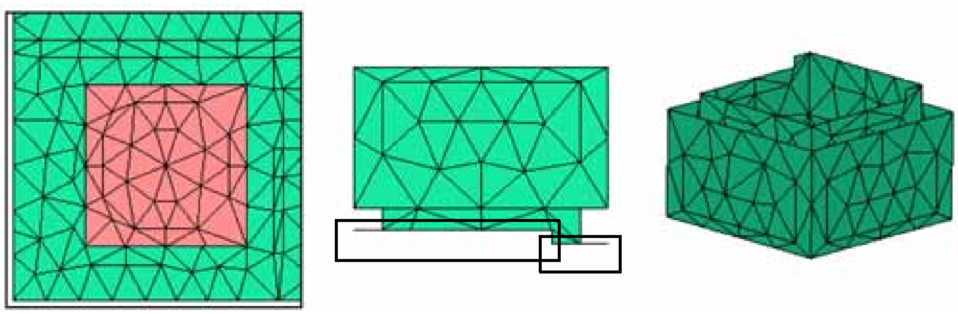
8. Highlight the **New Triangles** layer and click the Assign Layer icon.
   - The elements on the box layer are shown in Figure 170C.

To change the color of the elements on the box layer

1. Highlight the **Box** layer.
2. Click the **Layer Display** icon.
3. Click the **Specify** radio button.
4. Click the **Select** button.
5. Pick a color of your choosing.
6. Click **OK**.
7. Click **Close** on the Display dialog.

To change the thickness of the box

1. De-select all layers except the **Box** layer.
2. Click the Select All icon, **Edit ➨ Select All**, or **Ctrl+A**.
3. Click the Edit properties icon or **Edit ➨ Properties**.
4. Select all listed properties and click **OK**.
5. Select **Specified** in the Thickness drop-down list.
6. Enter **1.8** (mm) into the Thickness field.
7. Uncheck the **Apply to all entities that share this property** box.
8. Click **OK**.
9. Click the icon or click **File ➨ Save Study**.



A
Original selection
of box

B
Band selection
of edges

C
Elements on the
box layer

Figure 170: Box moving to a different layer

To run an analysis and interpret results

1. Double-click the **Analyze now!** icon .
   - **1.1.** De-select **Run a full analysis**.
   - **1.2.** Select **Run the analysis only**.
   - **1.3.** Click **OK**.
2. Review the following results and record if the problem is solved in Table 44 on page 495.
   - Air traps.
   - Weld lines.
   - Fill time.

The **air trap** result shows an air trap at the box region but from the filling pattern you can see that no air trap will be available here. So the problem of weld lines and air traps in the box region is fixed.

The next main problem is the different time to freeze through the part. That should be fixed at the next iteration step 2. Try to reduce that to 15sec maximum.

Iteration 2 – Decrease plate thickness

To open the model

1. Duplicate the model **base_it1** and change the name to **base_it2**.
2. Double-click on the file **base_it2** to open the model.

To create a new layer

1. Click the **New Layer** icon in the Layers pane.
2. Change the layer name to **Thick plate**.

To assign elements to the thick plate layer

1. Click the icon or in the toolbox click and select **Thickness Diagnostic.**
2. Click **Show**.
3. Click **Close**.
4. Click the front view icon to rotate the part to 0 0 0.
5. Ensure the Select Enclosed Items Only Icon on the Selection toolbar is selected.
6. Band-select all elements of the thick area (3.8mm) Be sure you get the edges as well.
7. Highlight the **Thick Plate** layer and click **Assign**.

To set the plate thickness

1. De-select all layers except the **Thick plate** layer.
2. Click the Select All icon , **Edit ➨ Select All**, or **Ctrl+A**.
3. Click the Edit properties icon or **Edit ➨ Properties**.
4. Select all listed properties and click **OK**.
5. Select **Specified** in the Thickness drop-down list.
   5.1. Enter a value (of your own choosing) into the **Thickness** field that will reduce the time to freeze below 15 sec.
   - Refer to the **Base_it1** results as a guide.
6. Click **OK**.
7. Click the icon or click **File ➨ Save Study**.

To run an analysis and interpret results

1. Double-click the **Analyze now!** icon .
   1.1. De-select **Run a full analysis**.
   1.2. Select **Run the analysis only**.
   1.3. Click **OK**.
2. Review the Time to freeze result and record if the problem is solved in Table 44 on page 495.

Optimize the part for the chosen gate location

Optimizing the part involves determining the optimum molding conditions, then running a full flow analysis to ensure the molding conditions are acceptable and all other issues are good.

Determine the Optimum Molding Conditions

In this section, the molding window analysis will be run to determine the optimum process settings for the study.

To run the molding window analysis

1. Duplicate **base_it2** and rename the study **base_moldwin**.
2. Double-click on the file **base_moldwin** to open the model.
3. Change the Analysis Sequence to **Molding Window**.
4. Double click the Process Settings icon .

   4.1. Click the Advanced Options button.

   4.2. Set the **Flow front temp. drop limit; Maximum drop** to **20ºC**.

   4.3. Set the **Flow front temp. rise limit; Maximum rise** to **2ºC**.

   - These values set the size of a good molding window. Normally temperature range limits the window size.

   4.4. Click **OK** twice to exit the wizard.
5. Run the analysis.
6. The analysis should only take a few moments.

Review the molding window results

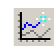
To view molding window analysis screen output

1. Click the Logs box in the **Study Tasks** pane, then click the **Screen output** tab.
2. View and record the recommended Mold temperature, Melt temperature and Injection time values in Table 45 on page 496.

To view the 2D zone plot

1. Click the **Zone (molding window):2D Slice Plot** in the Study Tasks pane.
2. Right-Click the **Zone (molding window):2D Slice Plot** and select **Properties**.
3. Set the Cut Axis to **Mold temperature**.
4. Set the Cut position to the **recommended mold temperature** from the Screen Output file.

5. Click **OK**.

- The green area is the preferred molding window. The larger the window the better.
- The molding window is defined by the Advanced options set. The vertical boundaries are defined primarily by the temperature range. The "optimum" injection time for any melt temperature has a minimum flow front temperature equal to the melt temperature.
- The preferred zone is defined by a 2ºC rise and a 20ºC drop in temperature. Therefore, the optimum injection time is near the bottom of the preferred window, for a given melt temperature.
- Use **Query results** tool  to determine the melt temperature and injection time at any location on the graph.
- The Cut axis can be animated with the add XY Curve tool .

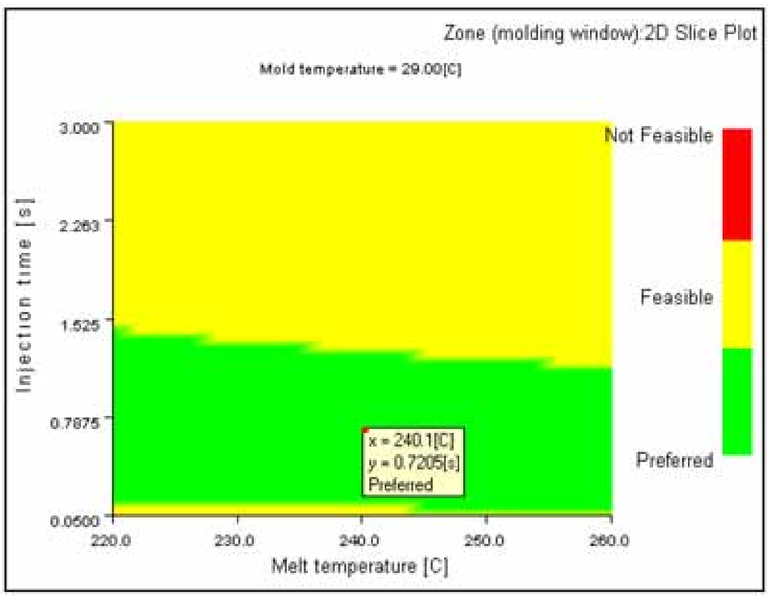


Figure 171: Zone plot

To view the quality plot

1. Click the **Quality (MW): XY** plot in the Study Tasks pane.

2. Right-click the **Quality (MW): XY** plot and select **Properties**.

3. Set the X-axis to **Injection time** by clicking the check box.

3.1. Scroll the mold and melt temperature sliders to their recommended values.

- Notice how the maximum quality changes as the sliders are moved.

4. Record the data point with the highest quality value at the recommended mold and melt temperature in Table 45 on page 496.

- Use **Query results** tool  to determine the value.

5. Check to see if the recommended mold and melt temperatures are at the minimum or maximum value of the range to be analyzed.
   - Find this range in the **Screen Output** file or Material details.
   - For example, if the recommended mold temperature is 60ºC. and the maximum range of mold temperature is also 60ºC, then the optimum molding conditions are at the edge of the molding window.
   - A better solution is to use the mid-range mold and melt temperatures, when possible. This will provide greater flexibility during production.
6. Move the mold and melt temperature sliders to mid-range values and see how the quality is affected. Record up to three alternative molding conditions and the quality values in Table 45 on page 496.

To find the injection time

1. Adjust the Mold and Melt temperature sliders to one of the molding conditions you choose.

1. Select the data point at the highest quality using the **Query results** tool and note the X value (Injection time).
   - This represents the new optimum injection time.
   - This time will produce a minimum flow front temperature that is very close to the melt temperature.
2. Repeat this for all the molding conditions you picked.

To review results

1. Review the following results at the new recommended process settings.
   - Injection pressure.
   - Minimum flow front temperature.
   - Maximum Shear stress.
2. Ensure the results are good.

> Refer to the molding window training unit for more on Molding Window Analysis Interpretation.

3. The following the process setting will be used in the following analysis:

Table 37: Optimum Processing conditions

Parameter	Value
Mold surface temperature:	40° C.
Melt temperature:	250° C.
Injection time:	0.2 Sec.

4. The mold and melt temperatures were rounded slightly from the values derived from the molding window analysis. The mold surface and melt temperatures are near the recommended values, and the quality does not decrease much.

Running the fill analysis

To run a filling analysis

1. Duplicate **base_moldwin** and rename the new study **base_ fill**.
2. Open the **base_fill** study.
3. Change the Analysis Sequence to Fill.
4. Open the process settings wizard and use the following settings. All other settings are at their default values:

Parameter	Value
Mold surface temperature:	40° C.
Melt temperature:	250° C.
Injection time:	0.2 Sec.

5. Start the analysis.

Review results

Review all necessary results and compare them with the following list of filling issues:

- Flow balancing, constant pressure gradient
    - The flow path is not balanced, as shown by the fill time plot. Balance is easiest to see with contour lines. The flow front does not reach all corners of the model at the same time. To move the injection location slightly will change the fill pattern extremely due to the box. Also, if the part is filled at the parting line, other problems such as air traps at the box will arise. Therefore, this injection location is the best for this model. If we could change the thickness in some areas, this could also help to balance the filling pattern.
    - The pressure gradient as shown with contour lines on the Pressure at end of fill result, shows the same issues as the fill time. It highlights the thin middle finger.
- Maximum shear stress
    - If shear stresses during filling are larger that the critical level, this can be a problem. However, it depends on the application if the stresses need to be reduced. For parts used in demanding applications it should be reduced. In the thin middle finger high values occur, which could lead to warpage.
- Uniform cooling time
    - The time to freeze is not uniform and the range of values is very high (1-13.8ec). Making the part wall thickness more uniform will help make the cooling time uniform. However, a uniform wall is not practical for this part. Flow leaders were added to the part to remove gas traps. Cooling can be made more uniform

with the cooling line design. More cooling around thick features and less around thin ones.

- Avoid hesitation
  - ➢ Hesitation still occurs in the middle finger of the part. It is not very practical that the middle finger is the thinnest of the 3 fingers; however, a change of the design of the part may need to be discussed to remove this problem.
- Avoid underflow
  - ➢ A minimal amount of underflow occurs due to the unbalanced filling. This can be seen by overlying fill time plotted with contour lines and average velocity. The velocity vectors should be perpendicular to the fill time contours.
- Low temperature in the thin rib
  - ➢ The variation in bulk temperature should be under 5°C but is more than 20°C with the automatic injection time used in previous analyses. The rib has the lowest bulk temperature compared with the whole part. The shorter injection time has significantly reduced the temperature variation in the part.

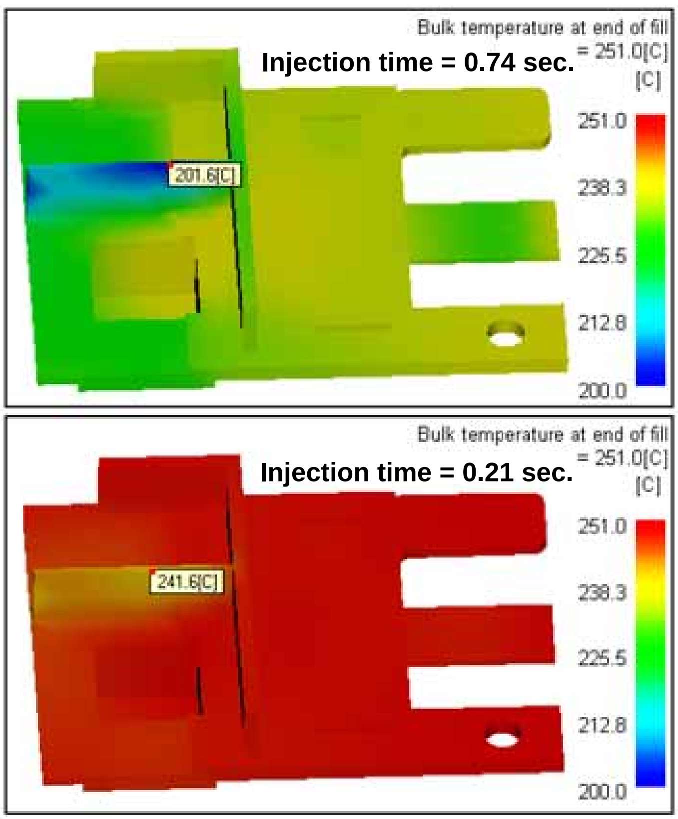


Figure 172: Bulk temperature differences due to injection time

- Possible design changes
  - ➢ Reduce thickness of the thick plate area further to reduce the time to freeze.
  - ➢ Increase the thickness of the middle finger to help reduce the hesitation effect.

- To balance the flow pattern, deflectors and leaders can be used.

> The suggestions above are possible design changes that will not be done here in this chapter.

Modeling the feed system

Overview

In this section, you will create a runner system manually. This will allow you to analyze the feed system so it can be optimized and to determine the effect on the part.

There are two ways that a runner system can be created. The Runner System Wizard, or by manually create the runners. The Runner System Wizard does not allow you control over the layout of the runner(s). The Wizard is primarily used to build geometrically balanced runner systems.

When manually creating runners, two methods can be used. Beam elements can be defined directly, or curves can be created and then meshed. If beam elements are to be created directly, they must be non-tapered. If they are tapered, you must use curves. In both cases, there is significant control of the mesh density.

Most of the time, runners are placed on a different layer than the part, and often the feed system is broken into several layers. For instance, the gates, runner, and sprue can be placed on different layers. There may be cases where different potions of the runners are even on different layers.

In this section you will add a runner system to the model and run an analysis.

Feed system creation

To open the model

1. Duplicate the study **base_fill** and rename the new study **base_ runner**.
2. Open the **base_runner** study.
3. Select the yellow injection cone, right-click and select **Delete**.
   - You may need to turn on some layers to find it.
   - Note the node location that the injection location was set on.

To create a layer

1. Click the **New Layer** icon in the Layers pane.
2. Change the layer name to **Runner**.

For most commands used in Synergy, there are two or three ways to access them. For the commands related to creation of curves used for the feed system, the commands will be assessed via the **Toolbox** on the **Tools** tab, shown in Figure 173. These commands can also be accessed in the **Modeling** menu, and on various standard and customized toolbars.The mesh generator is available in the **Mesh** menu, and from a mesh icon in the Study task list such as the Fusion mesh icon, . Use the method of access you prefer.

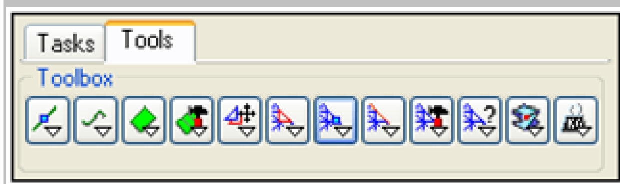


Figure 173: Toolbox

To create the gate

1. Ensure the **Nodes** layer is on.
2. Ensure the **Runners** layer is active (**Bold**).
3. Click the Create Curves icon  and select **Create Line**.

   3.1. Set the **Filter** to **Node**.

   3.2. Select the node where the injection location was set.

   3.3. Click the **Relative** radio button in the Coordinate 2 area.

   3.4. Enter **2.48 0 2.48** in the **Second** field.

   3.5. Click icon with the 3 dots  in the Selection option frame.

   3.6. Click **New** and select **Cold gate**.

   3.7. Select **Tapered (by end Dimensions)** in the **Shape is** drop-down list.

   3.8. Click **Edit dimensions.**

   - Enter **0.5** (mm) into the Start diameter field.
   - Enter **4** (mm) into the End diameter field.
   - Click **OK**.

   3.9. Enter **Cold gate 0.5 mm x 4 mm** in the name field.

   3.10. Click **OK** twice to exit the properties dialog.

   3.11. Click **Apply** to create the gate curve.

   3.12. Click **Close** exit the tool.

4. Click **Mesh ➨ Generate Mesh**.

   4.1. Enter **1** (mm) into the **Global edge length** field.

   4.2. Click **Preview**.

   4.3. Ensure there are 4 preview nodes displayed on the cold gate curve.

   4.4. Check the **Place mesh in active layer** box.

   4.5. Click **Mesh Now**.

To create a modeling plane

1. Click **File ➨ Preferences**.

1. Enter **4** (mm) into the **Grid size** field.

1. Ensure the **Snap to grid** box is checked.

   1.1. Click **OK**.

2. Select **Modeling ➨ Local Coordinate System/ Modeling plane ➨ Define**.

   2.1. Select **Node** in the Filter field.

   2.2. Select the node at the end of the cold gate.

   2.3. Click **Apply**.

   2.4. Click **Close**.

3. Select the **LCS** icon on the cold gate.

4. Right-click and select **Activate as modeling plane**.

   - The modeling grid should appear.

To create the runners

1. Click the Create Curves icon and select **Create Line**.

2. Click icon with the 3 dots in the Selection option frame.

3. Click **Select** and select **Cold runner**.

   3.1. Highlight **Cold runner 4mm**.

   3.2. Click **Select**.

   3.3. Click **OK**.

4. Click the front view icon to rotate the part to 0 0 0.

5. Set the Filter to **Modeling Plane**.

6. Create the runner system as shown in Figure 174.

    6.1. Click on the end of the gate to fill in the **First** field.

    6.2. Ensure the coordinate value is **0 0 0**.

    6.3. Click in the **Second** field.

    6.4. Click on the grid location 2 lines to the right of the gate.

    6.5. Click **Apply**.

    6.6. Notice how the coordinate that was in the **Second** field was moved to the **First** field.

    6.7. Continue to click on the grid locations as necessary and hit **Apply**.

7. Click **Close** when you have finished.

8. Right-click over the model and de-select **Activate as modeling plane**.

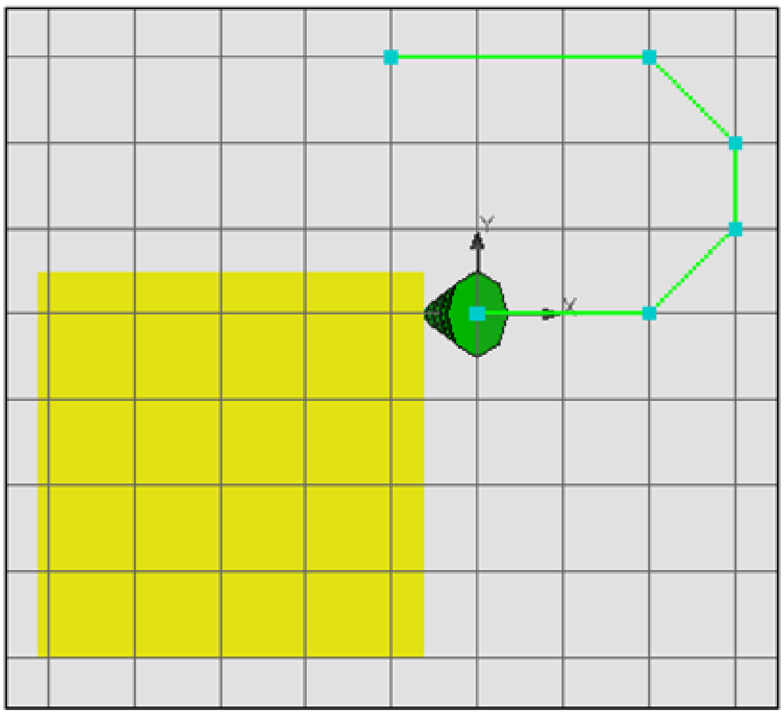


Figure 174: Runner system layout

To create the sprue

1. Click the Create Nodes Tools icon and select **Node By Offset**.
   1.1. Set the Filter to **Node**.
   1.2. Select the last node created for the runner system.
   1.3. Enter an Offset of **0 0 20**.
   1.4. Click **Apply**.
   1.5. Click **Close**.
   - This defines the end of the sprue.

2. Click the Create Curves icon and select **Create Line**.
   2.1. Set the filter to **Node**.
   2.2. Select the node just created.
   2.3. Select the last node on the runner system.
   2.4. Click icon with the 3 dots in the Selection option frame.
   2.5. Click **Select** and select **Cold sprue**.
   - ➤ Highlight **Cold sprue (3.5mm 1.5deg)**.
   - ➤ Click **Select**.
   - ➤ Click **OK**.

   2.6. Click **Apply**.
   2.7. Click **Close**.

To generate the mesh for the runner and sprue

1. Click **Mesh ➜ Generate Mesh**.
   1.1. Enter **3** (mm) into the **Global edge length** field.
   1.2. Click **Preview**.
   1.3. Check the **Place mesh in active layer** box.
   1.4. Click **Mesh Now**.

2. Click the injection location icon in the study tasks list or click **Analysis ➜ Set Injection Locations.**
3. Select the node at the top of the sprue.
4. Right-click and select **Finish Injection Locations**.
5. Select the **LCS** and click the **Delete** key to Delete the LCS icon.
6. Save the Study.

Feed system analysis

Previous analysis work has optimized the fill time based on just the part. Now that a feed system has been added, the injection time should be longer to account for the time to fill the runner. Instead of calculating a new injection time, a flow rate is calculated based on the part volume and injection time for the part. By using a flow rate, no matter what the runner volume is, the part will fill in the correct time. The overall fill time changes with the runner volume.

To determine the flow rate used for analysis

1. Open the **Base_Fill** analysis.
2. Click the **Logs** box.
3. Click the **Fill** tab.
4. Find the Fill time in the Process parameters section.
5. Find the part volume in the model details section.
6. Record the information below and calculate the flow rate.

Total volume of the part =		cm^3
Injection time=		s
Flow rate = Total volume / injection time =		cm^3/s

Running the analysis

To run a fill analysis with the runners

1. Open the **Base_Runner** study if necessary.
2. Ensure that the Analysis Sequence is set to **Fill**.
3. Double click the Process Settings Wizard .
4. Set the Filling control to **Flow rate**.
5. Enter the calculated flow rate value (from above) into the flow rate field.
6. Click **OK**.
7. Double-click the **Analyze now!** icon to start the analysis.

Review the analysis results

Runner design check list

When reviewing results of the runner system, there are several things that should be checked related to the gate and runner sizes.

Is the shear rate below the design guidelines?

The shear rate is a critical factor when sizing the runner system. In most cases, the only location where the shear rate may be high is in the gate. If the shear rate is too high, the polymer may be degraded. Before the polymer degrades, additives such as glass fibers, stabilizers, colorants, bulk fillers etc. will be adversely affected.

A good general rule to follow is to keep the shear rate as low as practically possible. A good target is 20,000 1/sec. This is NOT a practical value for many gate geometries, however, get the shear rate as low as practical for the gate geometry.

Are the runner freeze times practical?

The runners should not be so large that they control the cycle time of the mold. Generally, the maximum cooling time of the runner should be no more than 200% to 300% of the part cooling time. This is often difficult to achieve.

The runners should also not be too small. The smaller the runner, the greater the possibility it will limit the ability to pack the part out. The minimum freeze time of the runner should be about 80% of the part's freeze time. If the part has critical dimensions or sink criteria, the minimum freeze time should be about 100% of the part. The best check for this is the volumetric shrinkage result from a Flow analysis.

Is the pressure drop in the mold acceptable?

The total pressure drop within the tool should be no more that 65% to 75% of the machine limit. If the pressure to fill the tool is too high, the molding process may not be stable. Previous analysis work determined the pressure to fill the part. Now that the runners are included, the total pressure to fill the tool is determined.

If the pressure to fill the runner is high compared to the part, molding problems may occur, particularly in multi-cavity tools. For example, if the tool takes 100 MPa to fill and the parts only take 10 MPa to fill, small variations in the pressure drop in the runners will be very noticeable in the part. Some times high pressure drops are unavoidable but if possible, the runner pressure drop should not be excessive compared to the part. There may be a compromise between runner pressure drop, and runners controlling the cycle time of the mold.

Are the runners a practical size?

Practical size can mean a couple of things, including; are the runners too small to be cut? This situation may occur when using the runner balance analysis. It may calculate a runner diameter that produces a balanced fill, however, is really too small to cut and run efficiently.

Most likely, the practical size refers to the runner volume compared to the part volume. The smaller the runner volume the better. Preferably the runner volume should be under 20% of the part volume.

Review results

Results will be plotted to determine if the runners meet the guidelines stated above.

To plot the Shear rate, bulk result

1. Select the **Shear rate, bulk** result in the Study Tasks list.
1. Right-click the result name in the Study Tasks list and select **Properties.**
2. Click the **Scaling** tab.
   2.1. Click the **Specified** button.
   2.2. Set the Min field to **40,000**.
   2.3. Uncheck **Extended color**.
3. Click the **Mesh Display** tab.
   3.1. Set the Element surface display to **Transparent**.
4. Click the **Optional Settings** tab.
   4.1. Uncheck **Nodal averaged**.
5. Click **OK**.
6. Move the animation slider to view the result over time and review the gate region.
   - Setting the plot properties for this result was done so the high shear rate was very easy to find.
7. Record the shear rate in the gate in Table 46 on page 496.

To plot shear rate in an XY plot

1. Click **Results ➡ New Plot**.
2. Select **Shear rate, bulk** as the result.
3. Select **XY plot** as the plot type.
4. Click **OK**.
5. Select the smallest element in the gate.
   - Rotate and zoom up on the gate as necessary.
6. Click the select icon to finish picking elements to plot.

Discussion of the Shear rate, bulk result

Both results show a very high shear rate for a long period of time, which will destroy the material. Therefore, the shear rate at the gate needs to be lowered.

The recommended maximum shear rate for the material is 100,000 1/s. Therefore, the result value of 40,000 1/s. in the feed system is well within the recommended limits.

The value of 20,000 1/s is a preferred target, however, for this part and material, and because of the short injection time, 40,000 1/s is used. The freeze time of the gate is much better.

How to lower shear rate values?

Several things can be done to reduce the shear rate including:

- Lower the flow rate.
- Add a second injection location.
- Increase the gate diameter.

All options are fixed apart from the last solution: Increase the gate diameter. This is the only possibility that can be used to lower the shear rate at the gate.

To plot the time to freeze

1. Click on the time to freeze results.
2. Click the Query result icon .
3. Hold the Ctrl key down and select on several locations including:
    - The small end of the gate.
    - The box near the gate.
    - The runner.
    - The base of the sprue.
    - The thickest area of the part.

Discussion of the time to freeze result

The cooling time of the runner is not a controlling factor for this part, mainly because of the very thick section that still exists on the part. The smallest runner cooling time is about equal to the middle section of the part. The runner and sprue diameter could be made smaller.

The cooling time in the gate is rather small compared to the part, however the gate size will be increasing to reduce the amount of shear rate in the gate. If the gate freeze time is near or above the cooling time of the part, the gate may be too large.

To plot the pressure

1. Click **Results** ➡ **New Plot**.
2. Select **Pressure at end of fill** as the result.
3. Select **Path plot** as the plot type.
4. Click **OK**.
5. Rotate and zoom as necessary to select nodes at the following locations:
    - Top of the sprue.
    - Base of the sprue.
    - Runner/gate intersection.
    - Gate/part intersection.
    - End of the middle finger.
6. Click the select icon to finish picking elements to plot.
7. Record the maximum pressure in Table 47 on page 496.

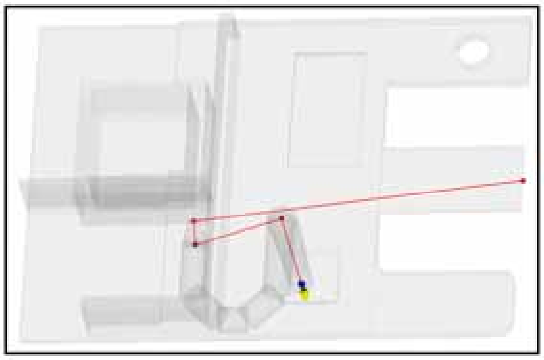


Figure 175: Path plot locations

Discussion of the pressure result

By plotting pressure using a path plot, you can see that the feed system takes just under half of the total pressure. The plot also shows how the runner has the largest pressure drop in the feed system.

To compare the runner volume to part volume

1. Click the **Logs** box.
2. Click the **Fill** tab.
3. Find the part volume and runner volume in the mesh details section, similar to Figure 176.

Discussion of the time to freeze result

The runner volume is under 10% of the part volume. If the runner volume was more than 20% of the part volume, the runners should be reduced if possible.

```
Part volume to be filled            =   8.6841 cm^3
Sprue/runner/gate volume to be filled     =   0.7161 cm^3
```

Figure 176: Part and runner volume from the Fill tab.

Modify the gate size and re-run the analysis

The primary problem with the existing feed system is the gate size, creating a high shear rate. Duplicate the base_runner study and open the gate size. Keep in mind the wall thickness where the gate goes into the part is 1.8 mm. Preferably, the shear rate should be about 20,000 1/sec. but 40,000 1/sec. is acceptable.

To modify the gate size

1. Duplicate **base_runner** and rename the new study **base_ runner2**.
2. Select one element in the gate, right-click and select **Properties**.
3. Click **Edit Dimensions** and enter a larger start diameter of your choosing.

4. Click **OK** twice to save the new gate size.
   - Notice that all the elements of the gates change.

To re-run the analysis

1. Double-click the **Analyze now!** icon to start the analysis.

To plot shear rate

1. Select the **Shear rate, bulk** result in the Study Tasks list.
1. Right-click the result name in the Study Tasks list and select **Properties.**
2. Click the **Mesh Display** tab.
   2.1. Set the Element surface display to **Transparent**.
3. Click the **Optional Settings** tab.
   3.1. Uncheck **Nodal averaged**.
4. Click **OK**.
5. Move the animation slider to view the result over time and review the gate region.
6. Duplicate the study, modify the gate size and re-run analysis if the shear rate is not low enough.

Discussion of results

To get a shear rate around 40,000 1/s, the gate needs to be made very large. This can make it difficult to de-gate the part at ejection. Also, a thick injection location is not visually appealing.

Packing analysis

Once the runners and gates are sized, a packing analysis is run. The primary output from a packing analysis is the volumetric shrinkage. The packing analysis is also used to define the packing time by determining when the gate freezes.

To run a packing analysis

1. Duplicate the study with the finalized gate and runner sizes.
2. Name the study **Base_Pack1**.
3. Open the study.
4. Double-click the analysis sequence icon, .
   4.1. Select Flow as the analysis sequence.
   4.2. Click **OK**.
5. Double-click the process settings wizard icon .

6. Ensure the following is set.

Table 38: Packing analysis inputs

Parameter	Value
Velocity/pressure switch-over	Automatic
Pack/holding control	%Filling pressure vs time
Pack/holding profile	0 80 10 80
Cooling time	Automatic

7. Double-click **Analyze now**.

To view the volumetric shrinkage at ejection results

1. Ensure only layers with triangles are on, and layers with the feed system are off.
2. Click the result, **Volumetric shrinkage at ejection**.
3. Review the results.
    - The volumetric shrinkage should be as uniform as possible. There should be no spikes in shrinkage, or no very low areas. Preferably, the range of volumetric shrinkage should be under 3%. The maximum allowable shrinkage will depend on the material and the part.
    - In this case, the volumetric shrinkage should have a maximum of 6%.
    - For this part with the wide range of thicknesses, keeping the shrinkage variation down is impossible to do. However, there is a spike in shrinkage near the gate that can be addressed.

To plot frozen layer fraction results.

1. Turn on the **Runner** layer.
2. Click the Logs box.
    - 2.1. Click the **Screen Output** tab and scroll through the packing phase table and determine when the pressure was released.
    - 2.2. Uncheck the Logs box.
3. Click the result, **Frozen layer fraction**.
4. Click **File ➡ Plot Properties**.
    - 4.1. Click the **Optional Settings** tab.
    - 4.2. Uncheck **Nodal averaged**.
    - 4.3. Click **OK**.
5. Zoom up around the gate so you can see the profile of the gate.
6. Animate the result one frame at a time to the time pressure was released.
    - 6.1. Determine if the gate is frozen, (Gate has a value of one).
        - ➤ If the gate has not frozen, the packing time is not long enough.

To plot the time to freeze results

1. Turn off the **Runner** layer.
2. Click the result, **Time to freeze**.
3. Note the maximum time to freeze on the part.
   - This will become the new packing time. This should be long enough to ensure the gate has frozen. The time to freeze result is based on the ejection temperature and the frozen layer fraction result is based on the transition temperature. Since the ejection temperature is lower, the time to freeze should be longer. The time to freeze plot also assumes no shear so the freeze time is an estimate.

To run a packing analysis with new packing time.

1. Save the **Base_Pack1** study.
2. Duplicate the **Base_Pack1** study and name the new study **Base_Pack2**.
3. Open the new study.
4. Double-click the process settings wizard icon .

   4.1. Click the **Edit Profile** button.

   4.2. Change the packing **Duration** to the time to freeze found for the part, rounded to the nearest second.

   4.3. Click **OK** twice to exit the wizard.
5. Run the analysis.

To review the results

1. Plot the following results:
   - Volumetric shrinkage at ejection.
   - Frozen layer fraction.
2. Answer the questions:
   - Did the volumetric shrinkage improve? If so, how?
   - Did the gate freeze before pressure was released? If so, what can the packing time be reduced to?
   - Is the volumetric shrinkage acceptable? If not, what should be improved.
3. If you are not satisified with the results, modifiy the packing profile and run the analysis again.

Final review of results

During this exercise you found the best possible injection location and optimum process parameters. Also, a possible location where air could be vented was identified, and ways that you can modify the design of the part that reduce costs by:

- The thick area of the part makes the time to freeze too long. This area of the part design could be changed so that the cycle time will be reduced dramatically.
- The rib and middle fingers also cause some problems. They are thin and need higher pressures to fill. The temperature rises in this area during filling, and freeze off occurs quickly compared to the rest of the model. If these sections of the part were made thicker, the part would be easier to mold with fewer problems.
- If the thickness of the rib and middle finger was increased, the injection location with the hot drop could be used. The part would have a better balance, and the tool design would be simpler.

What's wrong with the design?

Realistically, the gate location and runner layout are not possible. As a 3-plate tool design, the floating plate would be too thin, and as a 2-plate design, the slide required would be large and would be a maintanance problem. This gate location highlights a potential problem with simulation, **communication** or the lack of time. If the person running the analysis does not have a strong tooling background they may not know that this gate location and runner system has significant problems. Using reports, or Moldflow Communicator during the process to kept everyone involved is a critical part of the total analysis process. The groups that should be kept involved include:

- Materials.
- Product design.
- Tooling, (design and build).
- Production.

Generally, the Moldflow analyst, belongs to one of those groups, or has expertiese in one or two areas. Rarely does the analyst have expertise in all of these areas.

For this part, just moving the gate to the top of the box rather than on the side would have solved the tooling issues without any significant change in the filling.

> The gate location was chosen in part to provide additional practice with runner modeling.

Worksheets

Please fill out the tables below.

Table 39: Result statistics list

	Original Mesh	In Process	Fixed Mesh
Surface triangles			
Beams			
Connectivity regions			
Free edges			
Non-manifold edges			
Elements not oriented			
Element intersections			
Fully overlapping elements			
Maximum aspect ratio			
Average aspect ratio			
Match ratio			

Table 40: Material

Manufacturer: Shell, Grade: KMT 6100	
Family Abbreviation:	
Material Structure:	
Recommended melt temperature: °C	
Recommended mold surface temperature: °C	
Mold temperatures: °C	
Minimum:	
Maximum:	
Melt temperatures: °C	
Minimum:	
Maximum:	
Ejection temperature: °C	
Max. Shear stress: MPa	
Max shear rate: 1/s	

Table 41: Injection Molding Machine

Parameter	Value
Manufacturer	
Trade Name	

Table 41: Injection Molding Machine

Parameter	Value
Max. hydraulic pressure MPa	
Intensification ratio	
Max. injection pressure MPa	
Clamp force tonne	

Table 42: Gate Location Analysis summary

Parameter	Value
Analysis name	Base_gate
Analysis sequence	Gate location
Mold temperature °C	
Melt temperature °C	
Near Node	

Table 43: Result summary of the four injection locations

	Analysis Name Base_+			
Parameter	Gate_Loc	Box_Mid	Box_edge	Hot_drop
Analysis Sequence:	Fill	Fill	Fill	Fill
Mold Temperature: ºC	40	40	40	40
Melt Temperature: ºC	240	240	240	240
Injection time: Sec.	Auto	Auto	Auto	Auto
Switchover:	Auto	Auto	Auto	Auto
Maximum Injection Pressure: MPa				
Fill Time: Sec. Rank the balance, 1 best, 4 worst				
Air Traps? Are there any, can they be vented? y/n				
Weld lines in box y/n				
Shear Stress: MPa, Over limit?				

Additional remarks:

Table 44: Result Summary iterations to solve problems

Analysis name	base_it1	base_it2
Analysis sequence	Fill	Fill
Problem	Air trap and weld line position box	Time to freeze not uniform; very high value at thick area (3.8mm)
Change	Thickness change 1.8mm for hole box	Reduce to 15sec. maximum! New thickness______
Solved, y/n?		

Additional remarks:

Table 45: Process settings

	Mold Temp °C	Melt Temp °C	Inj. time s	Quality	Indicate conditions used
Recommended					
Own values					
Own values					
Own values					

Table 46: Shear rate comparison

Analysis Name	Start diameter (mm)	Shear rate, bulk at gate (1/s) inside gate	Indicate analysis used
1. Base_runner	0.5		
2.			
3.			
4.			

Table 47: Pressure comparison

Analysis name	Gate diameter mm	Maximum injection pressure MPa
1. Base_runner	0.5	
2.		
3.		
4.		

Worksheet Solutions.

Table 48: Result statistics list

	Original Mesh	In Process	Fixed Mesh
Surface triangles	1798	1806	1824 (depends on cleanup method)
Beams	0	0	0
Connectivity regions	2	1	1
Free edges	16	0	0
Non-manifold edges	0	0	0
Elements not oriented	0	8	0
Element intersections	28	0	0
Fully overlapping elements	3	0	0
Maximum aspect ratio	66.9	66.9	6.0
Average aspect ratio	2.2	2.2	2.2
Match ratio	95.2	95.2	95.2

Table 49: Material

Manufacturer: Shell, Grade: KMT 6100	
Family Abbreviation:	pp
Material Structure:	Crystalline
Recommended melt temperature: °C	40
Recommended mold surface temperature: °C	240
Mold temperatures: °C	
Minimum:	20
Maximum:	60
Melt temperatures: °C	
Minimum:	220
Maximum:	260
Ejection temperature: °C	101
Max. Shear stress: MPa	0.25
Max shear rate: 1/s	100,000

Table 50: Injection Molding Machine

Parameter	Value
Manufacturer	Battenfeld
Trade Name	BA 1300/400 BK
Max. hydraulic pressure MPa	17.5
Intensification ratio	12.47
Max. injection pressure MPa	218.2
Clamp force tonne	132.5

Table 51: Gate Location Analysis summary

Parameter	Value
Analysis name	Base_gate
Analysis sequence	Gate location
Mold temperature °C	40
Melt temperature °C	240
Near Node	72

Table 52: Result summary of the four injection locations

	Analysis Name Base_+			
Parameter	Gate_Loc	Box_Mid	Box_edge	hot_drop
Analysis Sequence:	Fill	Fill	Fill	Fill
Mold Temperature: ºC	40	40	40	40
Melt Temperature: ºC	240	240	240	240
Injection time: Sec.	Auto	Auto	Auto	Auto
Switchover:	Auto	Auto	Auto	Auto
Maximum Injection Pressure: MPa	6.6	10.3	7.8	5.6
Fill Time: Sec. Rank the balance, 1 best, 4 worst	1	4	2	3
Air Traps? Are there any, can they be vented? y/n	Y	Y[1]	Y	Y
Weld lines in box y/n	Y	N	Y[2]	Y
Shear Stress: MPa, Over limit?	No	No	No	No

Additional remarks:

Base_gate_loc has the best balanced filling pattern. Then **Base_box_edge**.

[1]Air traps are available but not valid.

[2]The result Weld line do not show a weld line but from the filling pattern you can see that weld line will occur.

Table 53: Result Summary iterations to solve problems

Analysis name	base_it1	base_it2
Analysis sequence	Fill	Fill
Problem	Air trap and weld line position box	Time to freeze not uniform; very high value at thick area (3.8mm)
Change	Thickness change 1.8mm for hole box	Reduce to 15sec. maximum! New thickness______
Solved, y/n?	Yes	Yes

Table 54: Process settings

	Mold Temp °C	Melt Temp °C	Inj. time s	Quality	Indicate conditions used
Recommended	28.89	260	0.17	0.905	
Own values	51.11	232.6	0.23	0.902	
Own values	42.22	249.5	0.23	0.886	X
Own values					

Table 55: Shear rate comparison

Analysis Name	Start diameter (mm)	Shear rate, bulk at gate (1/s) inside gate	Indicate analysis used
1. Base_runner	0.5	~535,000	
2. Base_runner2	1.8	~49,000	X
3. Base_runner3	2.4	~25,000	
4.			

Table 56: Pressure comparison

Analysis name	Gate diameter mm	Maximum injection pressure MPa
1. Base_runner	0.5	21.15
2. Base_runner2	1.8	18.94
3. Base_runner3	2.4	18.73
4.		

Competency check - Guided Project

1. What can you do to reduce shear rate values in gates?

Evaluation Sheet - Guided Project

1. What can you do to reduce shear rate values in gates?

3 possible solutions:

1. Lower the flow rate.

2. Add a second injection location.

3. Increase the gate diameter.

No. 3 has the biggest influence.

Index

Numerics

A

B

C

D

E

N

O

P

Q

R