佐证材料 8-1-2：开发注塑模具模流分析（CAE）资源

开发注塑模具模注分析的国际化资源，同步将 CAE 技术融入专业课程体系，改革模具设计与制造专业的专业核心课程。

材料：资源展示

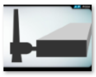


Diagram of injection molding process-01

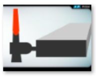


Diagram of injection molding process-02

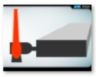


Diagram of injection molding process-03

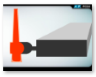


Diagram of injection molding process-04

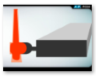


Diagram of injection molding process-05

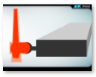


Diagram of injection molding process-06

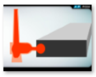


Diagram of injection molding process-07

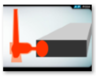


Diagram of injection molding process-08

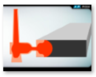


Diagram of injection molding process-09

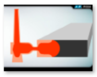


Diagram of injection molding process-10

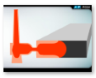


Diagram of injection molding process-11

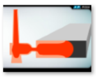


Diagram of injection molding process-12

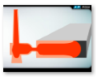


Diagram of injection molding process-13

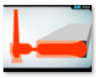


Diagram of injection molding process-14

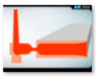


Diagram of injection molding process-15

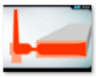


Diagram of injection molding process-16

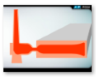


Diagram of injection molding process-17

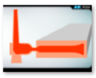


Diagram of injection molding process-18

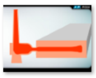


Diagram of injection molding process-19

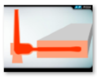


Diagram of injection molding process-20

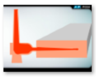


Diagram of injection molding process-21

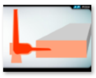


Diagram of injection molding process-22

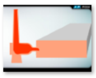


Diagram of injection molding process-23

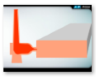


Diagram of injection molding process-24

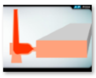


Diagram of injection molding process-25

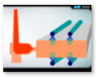


Diagram of injection molding process-26

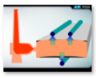


Diagram of injection molding process-27

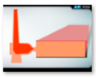


Diagram of injection molding process-28

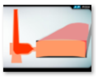


Diagram of injection molding process-29

2023/10/17

Getting started (tutorial)

SHARE

This tutorial will familiarize you with the user interface and teach you how to import a model.

Tutorial duration: Approximately 50 minutes

This product helps you determine the ideal combination of part geometry, material, mold design, and processing conditions that will produce quality finished parts.

The main tasks that you will perform are:

1. Importing a model.
2. Creating a mesh to represent the model geometry (also known as meshing the model).
3. Repairing the mesh if it contains errors.
4. Modeling the mold (cavity layout, feed system, and cooling).
5. Analyzing the study.
6. Viewing and interpreting the results of the analyses.
7. Adjusting the molding parameters based on the analysis results.

Importing a model

SHARE

A model of the part needs to be imported for analysis. The model is saved into a **Study** in a **Project** .

A project is used to store **studies** . Each study is a version of the original imported model and can be analyzed with different combinations of features, such as injection locations, processing parameters, runner/cooling configurations, material, analysis sequences, etc. Analysis results for a study are stored in separate files from the study and are linked to the study in the project. You can generate a new study at any time by using File > Save Study As.

For file management reasons, most users create a new project for each mold analyzed.

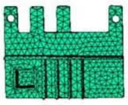
You will now create a new project and import a study into it.

Creating a new Project

1. Click (Start & Learn tab > Launch panel > New Project). The Create New Project dialog appears.
   The default file location for a project is defined during installation, but can be changed at any time from the Options dialog associated with the Application menu. Select the Directories tab of the dialog. If you want to create the project in a different location than the one shown in the Project directory textbox, click Browse and navigate to the required folder.
2. Enter `Getting Started` in the Project name text box.
   Note that the Create in text box outlines the pathway to where the project will be stored, and that the new Project name you have entered is included in this description. A subfolder called "Getting_Started" has been

3. Click OK.
   A new Getting Started project is created and opened.
4. Click (Home tab > Import panel > Import).
5. Select the Files of type drop-down list. The list of file types directly supported is shown. Select **Study files (*.sdy)** .
6. Navigate to the **Tutorial** folder, typically C:\Users\Public\Public Documents\Autodesk\Moldflow Synergy 20xx\tutorial).
7. Click on the file tutorial_model.sdy and click Open.
   The model is imported, and a new study is automatically created in the "Getting Started" project. This is illustrated in the Project View pane.
8. Click (View tab > Navigate panel > Zoom). Click anywhere in the model display pane and drag the cursor vertically up or down the screen to resize the model.

The imported study should appear as follows:



Note: You **import** a model from outside this product and store it within a project as a study. Once it is stored within a project, you **Open** the study from within the project.

Click the **Next topic** link below to move on to the next task of the tutorial.

Model manipulation

SHARE

In this task, will look at the tools that are available to rotate and zoom in and out of sections of the model you opened in the previous task.

Ensure the model from the previous task is open.

1. Click (View tab > Navigate panel > Orbit).
2. Click anywhere on the model and, with the left mouse button held down, move the mouse. The model will pivot around a central point. This is useful when investigating the geometry of the part.

3. Using the ViewCube, click Front view to return the model to its original orientation.
   Other views of the model can be quickly obtained by clicking on the five other sides of the ViewCube.
4. This view can be adjusted by clicking (View tab > Navigate panel > Pan). Click and drag on the model until it is in the desired position.

5. Click (View tab > Navigate panel > Zoom Window). Holding the left mouse button down, select the area around the top left hole in the model. The area you selected is now expanded to fill the Model pane.

6. Click (View tab > Navigate panel > Zoom). Click and drag down in the Model Display panel, shrinking the image until the whole of the model is visible.
   The model is now showing in the lor right hand section of the panel.

7. Rotate the model (see Step 1). The model moves around the center of the panel.
8. You can select the pivot point by clicking (View tab > Navigate panel > Center). Click on the upper right of the model and rotate the part again.

   This feature is used to help investigate local conditions.
9. To return the model to its original condition, select Front View using the ViewCube and then (View tab > Navigate panel > Zoom All).

User interface-model pane

SHARE

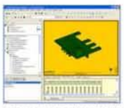
In this section, you will investigate the different sections of the graphical user interface.

The graphical user interface consists of five main sections:

1. Model pane
2. Toolbar panel
3. Project panel
4. Layers panel
5. Logs pane

Model pane

The Model pane is the largest section of the user interface.



This pane displays a 3D representation of the active study. Animations of the molding process and visual representations of the mold features (feed and cooling systems) can also be presented in this pane.

There are tabs across the bottom of the Model pane. Each tab represents a different study that is open in this session of the project. The active study has its tab to the front.

There is also a default tab that links to the Discussion Group (Application Menu > Options > Internet tab > Automatically launch the Discussion Forum). The Discussion Group is a web page designed for you to share your knowledge, learn tips and new features, and network with other users. The Discussion Group can also be accessed from the Autodesk website (www.autodesk.com) support pages. To be able to use the Discussion Group, you need to register.

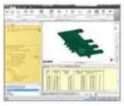
User interface-project panel

SHARE

The Project panel displays information about the model that is being analyzed.

Ensure the study used in the previous task is open. Alternatively, open **tutorial_model** from the **Getting Started** project you created in the **Importing a model** tutorial.

The Project panel is located in the upper left of the user interface.



If this panel is not displayed, it can be accessed by clicking (View tab > Windows panel > User Interface) and checking Project and Study Tasks.

At the top of this panel are two tabs, the Tasks tab and the Tools tab.

The Tasks tab

There are two sections to the Tasks tab

1. **Project View pane**

   The Project View pane is the upper section of the Tasks tab. This is where the studies in the current project are listed. The icons to the right of the study name are analysis icons and these will be discussed in a later tutorial.

The pane you are displaying shows that the current project is **Getting Started** and the only study currently available is **tutorial_model** (the study you originally imported).

2. **Study Tasks pane**

The Study Tasks pane is the lower section of the Tasks tab and this is where details about your active study are displayed. The status of the tasks required before analysis and the results generated are also shown.

The Tools tab

The Tools tab is the second tab of the Project panel.

Models usually require some manipulation of geometry or definition of features prior to analysis. The modeling dialogs open in the Tools tab.

1. From the Results section of the Study Tasks pane, tick **Fill Time** for a visual representation of the time required to fill the mold.
2. Rotate the model to investigate this result.
3. Experiment with the other analysis results that are available.

4. Click (Mesh tab > Mesh Diagnostics > Aspect Ratio). The Aspect Ratio Diagnostic dialog opens in the Tools tab. This feature is explained in the meshing tutorial.
5. Ensure the **Minimum Input** parameter is the default of 6.
6. Tick the Place results in diagnostics layer option.
7. Click Show.
8. The lines that appear in the Model pane indicate the location of elements with an aspect ratio greater than 6. The color of the line indicates the magnitude of the issue.
9. Rotate and zoom into sections of the model to investigate the elements with a high aspect ratio. will revisit these results in the Layer panel task.
10. Right-click on the model and select Mesh Diagnostics to remove the lines.
11. Return the model to its original orientation as outlined in the Model manipulation task. Using the

ViewCube, select Front and then click (View tab > Navigate panel > Zoom All).

User interface-layers panel

SHARE

Layers are an organizational tool that allow you to isolate components of your model. They assist in visualizing, manipulating, and editing your model more efficiently.

Layers Panel

The Layers Panel allows you to add, activate, delete, and modify the layers associated with your active model. Each layer can be displayed either individually or with other layers.

The Layers Panel is located in the bottom left of the graphics user interface.

If this panel is not showing, click (View tab > Windows panel > User Interface) and select Layers from the drop-down menu.

Ensure the working model from the previous task is open. If this model is not open, click File > Import. Navigate to the **Tutorial** folder found typically at C:\Users\Public\Public Documents\Autodesk\Moldflow Synergy 20xx\tutorial\tutorial, click **tutorial_model** and click Open.

Although they are not displayed, there are two cooling channels and an injection point defined in the model. Each of these features, along with the visible triangles and nodes (to be explained in the Meshing tutorial), are represented by an entry in the Layers Panel.

1. Experiment by ticking different check boxes in the Layers Panel and observing the effect in the Model pane. You may want to rotate the model to view the two cooling channels.
2. Tick **Triangles** in the Layers pane to return the model to its original form.

User interface-logs pane

SHARE

After an analysis has been run, the Logs pane is displayed at the bottom of the Model pane.

Logs Pane

The results of an analysis can be represented in several ways. You have already vied the graphical representation of **Fill time** and high **Aspect Ratios** . The numerical values of the available results are also of interest. An analysis overview is presented in the Logs pane.

The Logs pane can be hidden or displayed at any time by ticking the Logs box in the Study Tasks pane.

Ensure the model from the previous task is open. Alternatively, select **tutorial_model** from the **Getting Started** project.

1. Select the Tasks tab from the top of the Project panel.
2. Select the Logs box in the Study Tasks pane.
3. Select the Analysis Log tab from the top of the Logspane.
4. Scroll up and down the screen to view the output results.

Finally, you can generate customized reports that outline specific combinations of features required. This will be addressed in the **Post-Processing** tutorial.

Meshing (tutorial)

SHARE

This tutorial will teach you how to select and generate the appropriate mesh type, and make you aware of problems with the mesh that may arise.

Tutorial duration: Approximately 55 minutes

Analyzing a model is a complex operation. Modeling the way molten plastic (a non-Newtonian fluid) flows, especially through complex model geometry, is difficult.

To overcome these issues, a part is divided into smaller elements that are joined together. The result is a mesh. Temperature, pressure, and velocity of flow are then calculated for each element within the mesh. The cumulative effect is a representation of the whole model.

Three different types of mesh (analysis technologies) are available:

- Midplane
- Dual Domain
- 3D

Each of these will be discussed in this tutorial along with:

- How to select and generate the appropriate mesh type
- Problems with a mesh

Midplane mesh

SHARE

Historically, a part was meshed by determining the thickness of the part and then assigning a surface plane through the middle of the thickness.

This surface was then meshed with triangular elements that are joined along their edges and corners (called nodes). The thickness of the part is stored in each element. The analysis then incorporates this measurement by dividing the thickness of the part into thin layers called laminae. This provides a defined part volume on which calculations can be performed.

If you want to use a Midplane analysis, ensure the CAD package you use can directly generate a Midplane model. Converting a model into Midplane is a very time-consuming task.

In this task, you will:

- Import a Midplane model
- Investigate how the mesh is structured

1. Click (Start & Learn tab > Launch panel > New Project).
2. Enter `Mesh tutorial` for the **Project Name** .
3. Click OK.
4. Click (Home tab > Import panel > Import).
5. Select the Files of type drop-down list. The list of file types directly supported is shown. Select **Study files (*.sdy)** .
6. Navigate to the **Tutorial** folder, typically C:\Users\Public\Public Documents\Autodesk\Moldflow Synergy 20xx\tutorial.
7. Select dustpan_midplane.sdy then click Open.
8. Rotate and zoom in on the model to investigate the part geometry and the way the model has been represented as a Midplane model.
9. Note that the Midplane Mesh type and number of elements in the model are displayed in the Study Tasks pane.

Dual Domain mesh

SHARE

A Dual Domain mesh represents a solid CAD model by covering the surface of the model with triangular elements. The corners of the triangular elements are called nodes.

The model could be visualized as a hollow body covered with a surface shell.

When analyzing a part, the volume of the model is represented by layers through the thickness of the part (typically 10 or more layers through the thickness). Elements on the opposite faces of the part must be matching so the layers can be built. These matched elements ensure the alignment of the layers through the thickness. These layers enable an accurate representation of thin, cross-sectioned parts, where there is a rapidly changing characteristic profile, for example, temperature and flow-front velocity.

In this task, you will:

- Import a Dual Domain model
- Investigate how the mesh is structured

1. Ensure the **Mesh tutorial** project you used in the previous task is active. If it is not, click (Start & Learn tab > Launch panel > Open Project) and select Mesh tutorial.
2. Click (Home tab > Import panel > Import).
3. From the Files of type drop-down list, select **Study files (*.sdy)** .
4. Navigate to the **Tutorial** folder, typically C:\Users\Public\Public Documents\Autodesk\Moldflow Synergy 20xx\tutorial.
5. Select dustpan_dual_domain.sdy then click Open.
6. Click View tab > Navigate panel > Select).
7. Click on an element (triangle) in the model and press Delete on your keyboard.
8. Click Center and click on an element adjacent to the deleted element. This will centralize the area under investigation.
9. Rotate and zoom in on the model to understand the way the model has been represented.
10. Note that the Dual Domain Mesh type and number of elements in the model are displayed in the Study Tasks pane.

2023/10/17

3D mesh

SHARE

A 3D mesh represents the CAD model by filling the volume of the model with four-node, tetrahedral elements (tetras).

3D meshes work well for parts that are thick or solid because tetras give a true 3D representation of the model. A 3D analysis does not make the assumptions that are made for Midplane or Dual Domain analyses. Therefore, 3D analyses often require additional computational time to complete. This makes a 3D mesh more appropriate for thick models with complicated shapes, while Midplane and Dual Domain meshes are more applicable for thin-walled, shell-like parts.

In this task, you will:

- Import a 3D model
- Investigate how the mesh is structured

1. Ensure the **Mesh tutorial** project you used in the previous task is active. Alternatively click (Start & Learn tab > Launch panel > Open Project) and select **Mesh tutorial** .
2. Click (Home tab > Import panel > Import).
3. From the Files of type drop-down list, select **Study files (*.sdy)** .
4. Navigate to the **Tutorial** folder where the software is installed, typically C:\Users\Public\Public Documents\Autodesk\Moldflow Synergy 20xx\tutorial.
5. Select dustpan_3d.sdy then click Open.
6. Click (View tab > Navigate panelSelect).
7. Click on an element (tetra) in the model and press Delete on your keyboard.
8. Rotate and zoom in on the model to investigate the way the model has been represented.
9. Note that the 3D Mesh type and number of elements in the model are displayed in the Study Tasks pane.
10. When you have finished comparing the three model types generated in the last tasks, close the studies.

Meshing a model

SHARE

The quality of the mesh used in your analysis will determine the accuracy of your results.

The models you have worked with so far have already been meshed, but this is not how a model will be imported from a CAD program.

In this task, you will:

- Import a CAD model
- Generate a mesh on the model
- Compare the effect of different mesh densities

A Dual Domain mesh is the imported mesh type that is usually selected, and the one you will select. The reasons for this are:

- Most part geometry is shell-like in appearance, which is suitable for Dual Domain analysis.
- Midplane meshes usually require more cleanup than Dual Domain meshes.

  When an analysis that can only be performed on a midplane mesh is required, the mesh is usually generated in an external CAD package and imported as a midplane mesh.
- A 3D mesh requires a good Dual Domain mesh of the model as its starting point. This Dual Domain mesh is then converted into a 3D mesh. A solid part that obviously requires a 3D mesh analysis would still be imported as a Dual Domain mesh.

1. Ensure the **Mesh tutorial** project you used in the previous task is active. If it is not already open, click (Start & Learn tab > Launch panel > Open Project) and select Mesh tutorial.
2. Click (Home tab > Import panel > Import).
3. In the Files of type drop down list, select **IGES(*.igs,iges).**
4. Navigate to Tutorial folder, typically C:\Users\Public\Public Documents\Autodesk\Moldflow Synergy 20xx\tutorial.
5. Select Dustpan.IGS and click Open.
6. In the Import dialog, ensure **Dual Domain** is the mesh type selected.

   Note: If you have Autodesk Moldflow Design Link installed, you can check the Process using Autodesk Moldflow Design Link option to improve the quality of the imported mesh.
7. Click OK.

   Later in this task, you are going to compare two different mesh densities. You need to duplicate this model so that you can compare results.
8. Select **Dustpan_study** in the Study Tasks pane.
9. Right-click on **Dustpan_study** and select Duplicate from the drop-down menu.
10. Right-click on **Dustpan_study (copy)** , and select Rename from the drop-down menu.
11. Enter the new study name Dustpan_mesh and click Enter.
12. You will now generate a Dual Domain mesh for this part. Double-click **Dustpan_study** in the Study Tasks pane.
13. In the Study Tasks pane, double-click Create Mesh.

    The Generate Mesh pane of the Tools tab appears.
14. Ensure the General tab is selected.
15. Note the suggested **Global edge length on surface** of about 11.2 mm.
16. Click Preview. A representation of the major nodes that you will generate is shown.
17. Change the **Global edge length on surface** to 5 mm.
18. Click Preview to view the finer mesh your change would represent.
19. Click Mesh Now. The Mesh Generation message dialog appears. Click Close to dismiss the dialog and begin the meshing operation. The generation of the mesh will take a short time. The Mesh Complete message dialog appears. Click OK to dismiss the dialog.
20. Rotate and zoom in on the part to investigate the mesh, especially at rounded corners and thin edges.
21. In the Study Tasks pane, note that the third line down now identifies the mesh type as **Dual Domain Mesh** . Note the number of elements in the mesh.

    Although a finer mesh gives a better result, there is a point beyond which additional mesh density fails to increase the value of the analysis results and only increases computational time.
22. Double-click **Dustpan_mesh** in the Study Tasks pane.
23. In the Study task pane, double-click Create Mesh.
24. Ensure the General tab is selected.
25. Leave the **Global edge length on surface** at the default (about 11.2 mm).
26. Click Mesh Now. Close the message dialogs that appear before and after the meshing operation.
27. When the mesh has been created, rotate the part to investigate the mesh.
28. In the Study Tasks pane, note the number of elements in the mesh. This mesh has fewer elements than the first model.

    The mesh density should be optimized as far as possible. It is advisable to accept the default **Global edge length on surface** as an initial estimate. This length is often too coarse and may have to be refined to one-third to one-half of the initial value. Alternatively, a Global edge length on surface should be about 2 times the nominal wall thickness for a 3D mesh, and 2 to 5 times the nominal wall thickness for a Dual Domain mesh. You will now alter the mesh density.
29. Make sure the **Dustpan_mesh** study is the active study.
30. In the Study Tasks pane, right-click Dual Domain Mesh.
31. Select Generate Mesh.
32. From the General tab, alter the Global edge length on surface to 8.5 mm.
33. Check the Remesh already meshed parts of the model checkbox that is located above the Mesh Now button.
34. In the Layers panel, if **IGES Surface** and **IGES Curves** are listed, ensure they are not ticked.
35. Click Mesh Now. Close the message dialogs that appear before and after the meshing operation.
36. The mesh density of the model has been altered. In the Study Tasks pane, note the updated number of elements in the mesh.

2023/10/17

Checking the mesh for errors

SHARE

CAD programs represent complex models by combining simpler surfaces together into the finished design.

When importing or meshing a part, the interface between these separate surfaces can be misinterpreted. This is the cause of many errors, especially in areas of fine detail. The inclusion of surface texture, small radii, or fillets, while important to the finished product, leads to unnecessary complexity in mesh generation.

The Mesh Statistics report is divided into a number of different sections that provide a quick overview of various characteristics of the mesh. We will look at each of these sections.

- In the Entity Counts section, it is important to check the Connectivity regions value. There should only be one region. Any more than this will indicate disconnected sections of the part or runner system.
- In the Edge Details section, the integrity of the surface edges are determined.

  Free edges are not connected to another surface. This value should be zero in Dual Domain and 3D meshes.

  A manifold edge is a mesh edge that has two elements attached to it. This is the only edge type that is allowed for a Dual Domain mesh.

  A non-manifold edge has more than two elements attached to it. For a Dual Domain mesh, this value should be zero.
- In the Orientation Details section, the Elements not oriented value should be zero. Orientation is not a meshing error but a data handling requirement of the program. This will be explained in the Mesh editing tutorial.

- In the Intersection Details section, shared surfaces are reported. All values should be zero.
- The Surface Triangle Aspect Ratio section refers to the geometry of the mesh elements. The aspect ratio of an element is the ratio of the longest side to the height perpendicular to that side (a / b in the following figure).

  The aspect ratio value should ideally be less than:

For:	Max. Aspect Ratio
Midplane/Dual Domain	6:1
Midplane/Dual Domain-noncritical areas	20:1
Tetra elements	50:1
Dual Domain mesh before conversion to 3D	20:1
Cool and Warp analysis	6:1

  Very high aspect ratio triangles should be avoided, especially when the longest side is in the direction of flow as this can affect localized results. The report indicates that there is at least one element with a high aspect ratio. You will investigate this later.
- In the Match Percentage section, the mesh match values should ideally be 85% or higher. Mesh matching is only applicable for Dual Domain meshes. This is a measure of how elements on one surface correspond with elements on the opposite surface. This measure is very important for correct part thickness determination and fiber orientation prediction.

In this task, you will:

- Create a Mesh Statistics report to assess the quality of the mesh you generated in the last task
- Run the Mesh Repair Wizard to check for and repair a number of common mesh defects

1. Open the Mesh tutorial project you created in the previous task. Click (Start & Learn tab > Launch panel > Open Project) and navigate to the project directory where your projects are stored.
2. From the Mesh_tutorial folder, select **Mesh tutorial.mpi** .
3. Click Open.
4. Click (Home tab > Import panel > Import). From the Tutorial folder, typically found at C:\Users\Public\Public Documents\Autodesk\Moldflow Synergy 20xx\tutorial, select mesh_tutorial5.sdy.
5. Click (Mesh tab > Mesh Diagnostics panel > Mesh Statistics).
6. Click Close when you have finished reviewing the information in the Mesh Statistics dialog.

7. You will now run the Mesh Repair Wizard to check for further mesh defects, and to try to correct the high aspect ratio elements automatically. Click (Mesh tab > Mesh Repair panel > Mesh Repair Wizard). The Mesh Repair Wizard opens at the Stitch Free Edges page.

   The wizard has automatically run a diagnostic to check for edges that are not connected to another edge. This is illustrated on the left of the dialog. There are "0 free edges found".

   On each page of the wizard:
   - If a diagnostic indicates a problem, click Fix to try to correct the problem. Fix can be clicked repeatedly to correct problems.
   - Click Skip 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 not see if the problem was resolved.
8. Click Next to proceed to the Fill Hole page. The result indicates that there are no holes in the model.
9. Click Next to proceed to the Overhang page. The result indicates that there are no elements that are not part of the model surface. This is a Dual Domain related issue only.
10. Click Next to proceed to the Degenerate Elements page. A degenerate element has two nodes close to each other which can be resolved by merging nodes. (You would expect these elements to also have a high aspect ratio). For this model, no degenerate elements were found.

11. Click Next to proceed to the Flip Normal page. The result indicates that all elements in the model are oriented consistently.
12. Click Next to proceed to the Fix Overlap page. The result indicates that there are no intersections or overlapped surfaces in the model.
13. Click Next to proceed to the Collapsed Faces page. Where boundaries on opposite surfaces unexpectedly touch, you have a collapsed surface. From an analysis point of view, a collapsed face represents an obstacle to the flow and so will impact on results in the local area. The result indicates that there is no collapse on the model boundary.
14. Click Next to proceed to the Aspect Ratio page. As in the Mesh Statistics report, the minimum, maximum, and average aspect ratio values are reported.
15. Select the Show diagnostics box to display the aspect ratio diagnostic plot in the Model display pane. An element with a high aspect ratio can be very thin and elongated, making it difficult to locate in the Model display pane. The Aspect Ratio Diagnostic plot therefore uses a special technique to help you locate high aspect ratio elements. The aspect ratio of each element is represented by a spike projecting from the center of the element. The longer the spike, the higher the aspect ratio. The color of the spike also indicates the magnitude of the aspect ratio. The color grid on the right of the pane is a guide to the aspect ratio value.
16. Click Fix on the Aspect Ratio page to run the automatic repair function. The new results are reported on the dialog, and the plot is updated automatically. There are still some high aspect ratio elements that could not be fixed automatically.

17. Examine the model after using the Mesh Repair Wizard. The wizard can fix most problems automatically. However, in some cases not all errors can be repaired, and new errors may be introduced. In the next task you will learn how to correct mesh errors manually.
18. Click Next to proceed to the Summary page. The summary identifies the number of elements fixed and by what method.
19. Click Close to exit the wizard.
20. Save and close the study.