Chapter 24. Creating Surfaces for Displaying and Reporting Data

FLUENT allows you to select portions of the domain to be used for visualizing the flow field. The domain portions are called surfaces, and there are a variety of ways to create them. Surfaces are required for graphical analysis of 3D problems, since you cannot display vectors, contours, etc. or create an XY plot for the entire domain at once. In 2D you can usually visualize the flow field on the entire domain, but to create an XY plot of a variable in a portion of the interior of the domain, you must generate a surface. In addition, in both 2D and 3D, you will need one or more surfaces if you want to generate a surface-integral report. Note that FLUENT will automatically create a surface for each boundary zone in the domain. Surface information is stored in the case file.

The following sections explain how to create surfaces, rename, group, and delete them, and determine their sizes.

- Section 24.1: Using Surfaces
- Section 24.2: Zone Surfaces
- Section 24.3: Partition Surfaces
- Section 24.4: Point Surfaces
- Section 24.5: Line and Rake Surfaces
- Section 24.6: Plane Surfaces
- Section 24.7: Quadric Surfaces
- Section 24.8: Isosurfaces
- Section 24.9: Clipping Surfaces
Creating Surfaces for Displaying and Reporting Data

- Section 24.10: Transforming Surfaces
- Section 24.11: Grouping, Renaming, and Deleting Surfaces

24.1 Using Surfaces

In order to visualize the internal flow of a 3D problem or create XY plots of solution variables for 3D results, you must select portions of the domain (surfaces) on which the data are to be displayed. Surfaces can also be used for visualizing or plotting data for 2D problems, and for generating surface-integral reports.

FLUENT provides methods for creating several kinds of surfaces, and stores all surfaces in the case file. These surfaces and their uses are described briefly below:

Zone Surfaces: If you want to create a surface that will contain the same cells/faces as an existing cell/face zone, you can generate a zone surface. This kind of surface is useful for displaying results on boundaries.

Partition Surfaces: When you are using the parallel version of FLUENT, you may find it useful to create surfaces that are defined by the boundaries between grid partitions (see Chapter 28 for more information about running the parallel solver). You can then display data on each side of a partition boundary.

Point Surfaces: To monitor the value of some variable or function at a particular location in the domain, you can create a surface consisting of a single point.

Line and Rake Surfaces: To generate and display pathlines, you must specify a surface from which the particles are released. Line and rake surfaces are well-suited for this purpose and for obtaining data for comparison with wind tunnel data. A rake surface consists of a specified number of points equally spaced between two specified endpoints. A line surface is simply a line that includes the specified endpoints and extends through the domain; data points will be at the centers of the cells through which the line passes, and consequently will not be equally spaced.
24.2 Zone Surfaces

**Plane Surfaces:** If you want to display flow-field data on a specific plane in the domain, you can create a plane surface. A plane surface is simply a plane that passes through three specified points.

**Quadric Surfaces:** To display data on a line (2D), plane (3D), circle (2D), sphere (3D), or quadric surface you can specify the surface by entering the coefficients of the quadric function that defines it. This feature provides you with an explicit method for defining surfaces.

**Isosurfaces:** You can use an isosurface to display results on cells that have a constant value for a specified variable. Generating an isosurface based on \( x \), \( y \), or \( z \) coordinate, for example, will give you an \( x \), \( y \), or \( z \) cross-section of your domain. Generating an isosurface based on pressure will allow you to display data for another variable on a surface of constant pressure.

24.2 Zone Surfaces

Zone surfaces are useful for displaying results on boundaries. For example, you may want to plot contours of velocity magnitude at the inlet and outlet of the problem domain, or temperature contours on the domain’s walls. To do so, you need to have a surface that contains the same faces (or cells) as an existing face (or cell) zone. Zone surfaces are created automatically for all boundary face zones in the domain, so you will generally not need to create any zone surfaces unless you accidentally delete one.

To create a zone surface, you will use the **Zone Surface** panel (Figure 24.2.1).

To create a zone surface, you will use the **Zone Surface** panel (Figure 24.2.1).

The steps for creating the zone surface are as follows:

1. In the **Zone** list, select the zone for which you want to create a surface.

2. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is
Creating Surfaces for Displaying and Reporting Data

the concatenation of the surface type and an integer which is the new surface ID (e.g., \texttt{zone-surface-6}). (If the \texttt{New Surface Name} you enter is the same as the name of a surface that already exists, FLUENT will automatically assign the default name to the new surface when it is created.)

3. Click on the Create button. The new surface name will be added to the \texttt{Surfaces} list in the panel.

If you want to delete or otherwise manipulate any surfaces, click on the Manage... button to open the \texttt{Surfaces} panel. See Section 24.11 for details.

Figure 24.2.1: The Zone Surface Panel
24.3 Partition Surfaces

If you are using the parallel version of FLUENT (see Chapter 28), you may find it useful to create data surfaces defined by the boundaries of grid partitions. As described in Section 28.4, partitioning the grid divides it into groups of cells that can be solved on separate processors when you use a parallel solver. A partition surface will contain faces or cells on the boundary of two grid partitions. For example, you can plot solution values on the partition surface to determine how the solution is changing across a partition interface. Figure 24.3.1 shows cell-partition contours on a partition surface which is overlaid on the grid.

Figure 24.3.1: Contours of Cell Partitions on Partition Surface Overlaid on Grid

To create a partition surface, you will use the Partition Surface panel (Figure 24.3.2).

The steps for creating the partition surface are as follows:
1. Specify the partition boundary you are interested in by indicating the two bordering partitions under the Partitions heading. The boundary that defines the partition surface is the boundary between the “interior partition” and the “exterior partition”. \textit{Int Part} indicates the ID number of the interior partition (i.e., the partition under consideration), and \textit{Ext Part} indicates the ID number of the bordering (exterior) partition. The \textit{Min} and \textit{Max} fields will indicate the minimum and maximum ID numbers of the grid partitions. (The minimum is always zero, and the maximum is one less than the number of processors.) If there are more than two grid partitions, each interior partition will share boundaries with several exterior partitions. By setting the appropriate values for \textit{Int Part} and \textit{Ext Part}, you can create surfaces for any of these boundaries.

2. Choose interior or exterior faces or cells to be contained in the partition surface by turning \textit{Cells} and \textit{Interior} on or off under \textit{Options}. To obtain a surface consisting of cells that are on the “interior” side of the partition boundary, turn on both \textit{Cells} and \textit{Interior}. To
create one consisting of cells that are on the “exterior” side, turn on Cells and turn off Interior. If you want the surface to contain the faces on the boundary instead of the cells, turn off the Cells option. To have the faces reflect data values for the interior cells, turn the Interior check button on, and to have them reflect values for the exterior cells, turn it off.

3. If you do not want to use the default name assigned to the surface, enter a new name under New Surface Name. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., partition-surface-6). (If the New Surface Name you enter is the same as the name of a surface that already exists, FLUENT will automatically assign the default name to the new surface when it is created.)

4. Click on the Create button. The new surface name will be added to the Surfaces list in the panel.

If you want to delete or otherwise manipulate any surfaces, click on the Manage... button to open the Surfaces panel. See Section 24.11 for details.

24.4 Point Surfaces

You may often be interested in displaying results at a single point in the domain. For example, you may want to monitor the value of some variable or function at a particular location. To do this, you must first create a “point” surface, consisting of a single point. When you display node-value data on a point surface, the value displayed will be a linear average of the neighboring node values. If you display cell-value data, the value at the cell in which the point lies will be displayed.

To create a point surface, you will use the Point Surface panel (Figure 24.4.1).

Surface —> Point...

The steps for creating the point surface are as follows:
Creating Surfaces for Displaying and Reporting Data

1. Specify the location of the point. There are three different ways to do this:
   - Enter the coordinates \((x_0, y_0, z_0)\) under \textit{Coordinates}.
   - Click on the \textit{Select Point With Mouse} button and then select the point by clicking on a location in the active graphics window with the mouse-probe button. (See Section 25.3 for information about setting mouse button functions.)
   - Use the \textit{Point Tool} option to interactively position a point in the graphics window. You can set the initial location of this point using one of the two methods described above for specifying the point’s position (or you can start from the position defined by the default \textit{Coordinates}). See Section 24.4.1 for information about using the point tool.

2. If you do not want to use the default name assigned to the surface, enter a new name under \textit{New Surface Name}. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., \textit{point-5}). (If the \textit{New Surface Name} you enter...
24.4 Point Surfaces

is the same as the name of a surface that already exists, FLUENT will automatically assign the default name to the new surface when it is created.)

3. Click on the Create button to create the new surface.

If you want to check that your new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click on the Manage... button to open the Surfaces panel. See Section 24.11 for details.

24.4.1 Using the Point Tool

The point tool allows you to interactively fine-tune the definition of a point using graphics. Starting from an initial point, you can translate the point until its position is as desired. For example, if you need to position a point surface at the center of a duct, just past the inlet, you can start with the point tool near the desired location (e.g., on the inlet), and translate it until it is in the proper place. (You may find it helpful to display grid faces to ensure that the point tool is correctly positioned inside the domain.)

Initializing the Point Tool

Before turning on the Point Tool option, set the Coordinates to suitable starting values. You can enter values manually, or use the Select Point With Mouse button. Often it is convenient to display the grid for an inlet or isosurface on or near which the point is to be located, and then select a point on that grid to specify the initial position of the point tool. Once you have specified the appropriate Coordinates, activate the tool by turning on the Point Tool option. The point tool, an eight-sided polygon, will appear in the graphics window, as shown in Figure 24.4.2.

You can then translate the point tool as described below. The point surface you create will be located at the center of the point tool.
Translating the Point Tool

To translate the point tool in the direction along the red axis, click the mouse-probe button (the right button by default—see Section 25.3 for information about changing the mouse functions) anywhere on the gray part of the point tool and drag the mouse until the tool reaches the desired location. Green arrows will show the direction of motion.

To translate the tool in the transverse directions (i.e., along either of the other axes), press the <Shift> key, click the mouse-probe button anywhere on the gray part of the point tool, and drag the mouse until the tool reaches the desired location. Two sets of green arrows will show the possible directions of motion. (In 2D, there will be only one set of green arrows, since there is only one other direction for translation.) If you find the perspective distracting when performing this type of translation, you can turn it off in the Camera Parameters panel (opened from the Views panel), as described in Section 25.4.2.
24.5 Line and Rake Surfaces

Resetting the Point Tool

If you “lose” the point tool, or want to reset it for any other reason, you can either click on the Reset button to return the point tool to the default position and start from there, or turn the tool off and reinitialize it as described above. In the default position, the point tool will lie at the center of the domain.

24.5 Line and Rake Surfaces

You can create lines and rakes in the domain for releasing particles, obtaining data for comparison with tunnel data, etc. A rake consists of a specified number of points equally spaced between two specified endpoints. A line is simply a line that includes the specified endpoints and extends through the domain; data points will be located where the line intersects the faces of the cell, and consequently may not be equally spaced.

To create a line or rake surface, you will use the Line/Rake Surface panel (Figure 24.5.1).

Surface — Line/Rake...

The steps for creating the line or rake surface are as follows:

1. Indicate whether you are creating a Line surface or a Rake surface by selecting the appropriate item in the Type drop-down list.

2. If you are creating a rake surface, specify the Number of Points to be equally spaced between the two endpoints.

3. Specify the location of the line or rake surface. There are three different ways to define the location:
   - Enter the coordinates of the first point \((x_0,y_0,z_0)\) and the last point \((x_1,y_1,z_1)\) under End Points.
   - Click on the Select Points With Mouse button and then select the endpoints by clicking on locations in the active graphics window with the mouse-probe button. (See Section 25.3 for information about setting mouse button functions.)
Use the Line Tool option to interactively position a line in the graphics window. You can set the initial location of this line using one of the two methods described above for specifying endpoints (or you can start from the position defined by the default End Points). See Section 24.5.1 for information about using the line tool.

Note that the coordinates of the End Points will be updated automatically when you use the second or third method described above.

4. If you do not want to use the default name assigned to the surface, enter a new name under New Surface Name. The default name is the concatenation of the surface type and an integer which is...
the new surface ID (e.g., line-5 or rake-6). (If the New Surface Name you enter is the same as the name of a surface that already exists, FLUENT will automatically assign the default name to the new surface when it is created.)

5. Click on the Create button to create the new surface.

If you want to check that your new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click on the Manage... button to open the Surfaces panel. See Section 24.11 for details.

24.5.1 Using the Line Tool

The line tool allows you to interactively fine-tune the definition of a line or rake using graphics. Starting from an initial line, you can translate, rotate, and resize the line until its position, orientation, and length are as desired. For example, if you need to position a rake surface just inside the inlet to a duct, you can start with the line tool near the desired location (e.g., on the inlet), and translate, rotate, and resize it until you are satisfied. (You may find it helpful to display grid faces to ensure that the line tool is correctly positioned inside the domain.)

Initializing the Line Tool

Before turning on the Line Tool option, set the End Points to suitable starting values. You can enter values manually, or use the Select Points With Mouse button. Often it is convenient to display the grid for an inlet or isosurface on or near which you wish to place the line or rake surface and then select two points on that grid to specify the initial position of the line tool. Once you have specified the appropriate End Points, activate the tool by turning on the Line Tool option. The line tool will appear in the graphics window, as shown in Figure 24.5.2.

You can then translate, rotate, and/or resize the line tool as described below.
**Translating the Line Tool**

To translate the line tool in the direction along the red axis, click the mouse-probe button (the right button by default—see Section 25.3 for information about changing the mouse functions) anywhere on the “line” part of the tool (see note below) and drag the mouse until the tool reaches the desired location. Green arrows will show the direction of motion.

Do not click on the axes of the line tool that have arrows on the ends. These axes control rotation of the tool. Click only on the portion of the tool that represents the prospective line surface. This portion is designated by the rectangles attached to each end.

To translate the tool in the transverse directions (i.e., along either of the axes within the plane perpendicular to the red axis), press the <Shift> key, click the mouse-probe button anywhere on the “line” part of the tool (see note above), and drag the mouse until the tool reaches the desired location. Two sets of green arrows will show the possible directions of motion. (In 2D, there will be only one set of green arrows, since there
is only one other direction for translation.) If you find the perspective distracting when performing this type of translation, you can turn it off in the Camera Parameters panel (opened from the Views panel), as described in Section 25.4.2.

**Rotating the Line Tool**

To rotate the line tool, you will click the mouse-probe button on one of the white axes with arrows. When you click on one of these axes, a green ribbon will encircle the other arrowed axis, designating it as the axis of rotation. As you drag the mouse along the circle to rotate the tool, the green circle will become yellow.

! Do not click on the red axis to rotate the line tool.

**Resizing the Line Tool**

If you plan to generate a rake surface, you can resize the line tool to define the length of the rake. Click the mouse-probe button in one of the white rectangles at the ends of the “line” part of the tool (shown in black in Figure 24.5.2) and drag the mouse to lengthen or shorten the tool. Green arrows will show the direction of stretching/shrinking.

**Resetting the Line Tool**

If you “lose” the line tool, or want to reset it for any other reason, you can either click on the Reset button to return the line tool to the default position and start from there, or turn the tool off and reinitialize it as described above. In the default position, the line tool will lie midway along the $x$ and $y$ lengths of the domain, spanning the $z$ domain extent.
24.6 Plane Surfaces

To display flow-field data on a specific plane in the domain, you will use a plane surface. You can create surfaces that cut through the solution domain along arbitrary planes only in 3D; this feature is not available in 2D.

There are six types of plane surfaces that you can create:

- Intersection of the domain with the infinite plane: This is the default plane surface created. The extents of the plane will be determined by the extents of the domain. Since the plane is slicing through the domain, the data points will, by default, be located where the plane intersects the faces of a cell, and consequently may not be equally spaced.

- Bounded plane: This plane will be a bounded parallelepiped, for which 3 of the 4 corners are the 3 points that define the plane equation (or the 4 corners are the corners of the “plane tool”). Like the default plane surface described above, this type of surface will also have unequally spaced data points.

- Bounded plane with equally spaced data points: This plane is the same as the bounded plane described above, except you will specify the density of points along the 2 directions of the parallelepiped, creating a uniform distribution of data points.

- Plane having a certain normal vector and passing through a specified point: To create this type of plane, you will define a normal vector and a point. A plane with the specified normal and passing through the specified point will be created.

- Plane aligned with an existing surface: To create this type of plane, you will define a single point and a surface. A plane parallel to the selected surface and passing through the specified point will be created.

- Plane aligned with the view in the graphics window: To create this type of plane, you will define a single point. A plane parallel to
the current view in the active graphics window and passing through the specified point will be created.

To create a plane surface, you will use the Plane Surface panel (Figure 24.6.1).

![Plane Surface Panel](image)

Figure 24.6.1: The Plane Surface Panel

The steps for creating the plane surface are as follows:

1. Decide which of the six types of planes described above you want to
Creating Surfaces for Displaying and Reporting Data

create. If you are creating the default plane type (the intersection of the infinite plane with the domain), go directly to step 2. To create a bounded plane, turn on Bounded under Options. To create a bounded plane with equally spaced data points, turn on both Bounded and Sample Points, and then set the number of data points under Sample Density. You will specify the point density in each direction by entering the appropriate values for Edge 1 and Edge 2. (Edge 1 extends from point 0 to point 1, and edge 2 extends from point 1 to point 2. The points are specified in step 2, below.)

To define a plane aligned with an existing surface, select Aligned With Surface, and then choose the surface in the Surfaces list and specify a single point using one of the first two methods described below in step 2.

To define a plane aligned with the view plane, select Aligned With View Plane, and then choose a single point using one of the first two methods described below in step 2.

To define a plane having a certain normal vector and passing through a specified point, select Point And Normal, and then specify the normal vector by entering values in the ix, iy, and iz fields under Normal, and a single point using one of the first two methods described below in step 2.

2. Specify the location of the plane surface. There are three different ways to define the location:

- Enter the coordinates of the three Points defining the planar surface: (x0,y0,z0), (x1,y1,z1), and (x2,y2,z2).
- Click on the Select Points button and then select the three points by clicking on locations in the active graphics window with the mouse-probe button. (See Section 25.3 for information about setting mouse button functions.)
- Use the Plane Tool option to interactively position a plane in the graphics window. You can set the initial location of this plane using one of the two methods described above for specifying the defining points (or you can start from the position defined by the default Points). See Section 24.6.1 for information about using the plane tool.
24.6 Plane Surfaces

Note that the coordinates of the Points will be updated automatically when you use the second or third method described above.

3. If you do not want to use the default name assigned to the surface, enter a new name under New Surface Name. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., plane-7). (If the New Surface Name you enter is the same as the name of a surface that already exists, FLUENT will automatically assign the default name to the new surface when it is created.)

4. Click on the Create button to create the new surface.

If you want to check that your new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click on the Manage... button to open the Surfaces panel. See Section 24.11 for details.

24.6.1 Using the Plane Tool

The plane tool allows you to interactively fine-tune the definition of a plane using graphics. Starting from an initial plane, you can translate, rotate, and resize the plane until its position, orientation, and size are as desired. For example, if you need to position a plane surface at a cross-section of an irregularly-shaped, curved duct, you can start with the plane tool near the desired location, resize it, translate it until it is within the duct walls, and rotate it to the proper orientation. (You may find it helpful to display grid faces to ensure that the plane tool is correctly positioned inside the domain.)

Initializing the Plane Tool

Before turning on the Plane Tool option, set the Points to suitable starting values. You can enter values manually, or use the Select Points button. Often it is convenient to display the grid for an inlet or isosurface that is similar to the desired plane surface, and then select three points on that grid to position the initial plane. Once you have specified the appropriate
Creating Surfaces for Displaying and Reporting Data

Figure 24.6.2: The Plane Tool

Points, activate the tool by turning on the Plane Tool option. The plane tool will appear in the graphics window, as shown in Figure 24.6.2.

You can then translate, rotate, and/or resize the plane tool as described below.

**Translating the Plane Tool**

To translate the plane tool in the direction normal to the plane, click the mouse-probe button (the right button by default—see Section 25.3 for information about changing the mouse functions) anywhere on the gray part of the plane tool and drag the mouse until the tool reaches the desired location. Green arrows will show the direction of motion.

To translate the tool in the transverse directions (i.e., along either of the axes that lie within the plane), press the <Shift> key, click the mouse-probe button anywhere on the gray part of the plane tool, and drag the mouse until the tool reaches the desired location. Two sets of green arrows will show the possible directions of motion. If you find
the perspective distracting when performing this type of translation, you can turn it off in the Camera Parameters panel (opened from the Views panel), as described in Section 25.4.2.

**Rotating the Plane Tool**

To rotate the plane tool, you will click the mouse-probe button on one of the white arrows at the tips of the plane’s axes. Clicking on any arrow allows you to rotate the tool about either of the other two axes: when you click on the arrow, two green ribbons will encircle the plane tool, forming circles about each of the two possible axes of rotation. Drag the mouse along the desired circle to rotate the tool. As you do so, the circle along which the tool is rotating will become yellow.

The following notes may help you when you are rotating the plane tool:

- Once you move your mouse along one circle, you cannot change the direction of rotation unless you release the mouse-probe button and try again. Be careful to start moving your mouse very steadily so that you can choose the correct direction.

- Do not click on the red arrow to rotate.

- Do not try to rotate by clicking on an arrow that is pointing away from you. It will be very difficult for you to judge which direction of rotation is correct from this point of view. Since there are two arrows on each axis, there will always be an appropriate arrow available.

- Do not rotate the plane tool more than 90° or so at once. If you rotate the tool by a large angle, the arrow on which you are clicking will begin to point away from you, and you will have trouble controlling the rotation (as discussed in the item above).

**Resizing the Plane Tool**

If you plan to generate a bounded plane, you can resize the plane tool to define the plane’s boundaries. Click the mouse-probe button in one of the white squares at the plane tool’s corners (shown in black in Figure 24.6.2).
Creating Surfaces for Displaying and Reporting Data

and drag the mouse to stretch or shrink the tool. Green arrows will show the direction of the plane’s diagonal.

! Be careful not to drag your mouse across any of the axes while resizing the tool. This will flip the tool over and corrupt it. If you accidentally do this, reset the plane tool and start again.

Resetting the Plane Tool

If you “lose” the plane tool, or want to reset it for any other reason, you can either click on the Reset Points button to return the plane tool to the default position and start from there, or turn the tool off and reinitialize it as described above. In the default position, the plane tool will lie midway along the $x$ length of the domain, spanning the $y$ and $z$ domain extents.

24.7 Quadric Surfaces

If you want to display data on a line (2D), plane (3D), circle (2D), sphere (3D), or general quadric surface, you can specify the surface by entering the coefficients of the quadric function that defines it. This feature provides you with an explicit method for defining surfaces. See Sections 24.5 and 24.6 for additional methods for creating line and plane surfaces.

To create a quadric surface, you will use the Quadric Surface panel (Figure 24.7.1).

The steps for creating the quadric surface are as follows:

1. Decide which type of quadric surface you want to create. In 3D, choose Plane, Sphere, or (general) Quadric in the Type drop-down list. In 2D, choose Line, Circle, or Quadric.

2. Specify the defining equation for the surface in SI units.
   - Line or plane surface: If you have selected Line (in 2D) or Plane (in 3D) as the surface type, the surface will consist of
24.7 Quadric Surfaces

Figure 24.7.1: The Quadric Surface Panel

all points on the domain that satisfy the equation \( ix \cdot x + iy \cdot y + iz \cdot z = \text{distance} \). You will input \( ix \) (the coefficient of \( x \)), \( iy \) (the coefficient of \( y \)), \( iz \) (the coefficient of \( z \)), and \( \text{distance} \) (the distance of the line or plane from the origin) in the fields to the right of the Type drop-down list. When you click on the Update button under the Quadric Function heading, the display of the quadric function coefficients will change to reflect your inputs.

- Circle or sphere surface: If you have selected Circle (in 2D) or Sphere (in 3D) as the surface type, the surface will consist of all points on the domain that satisfy the equation \((x - x0)^2 + (y - y0)^2 + (z - z0)^2 = r^2\). You will input \(x0, y0, z0\) (the \(x\), \(y\), and \(z\) coordinates of the sphere or circle’s center) and \(r^2\) (the square of the radius) in the fields to the right of the Type drop-down list. When you click on the Update button
under the **Quadric Function** heading, the display of the quadric function coefficients will change to reflect your inputs.

- **Quadric surface**: If you have selected **Quadric** as the surface type, the surface will consist of all points in the domain that satisfy the general quadric function $Q = \text{value}$. You will input the coefficients of the quadric function $Q$ (the coefficients of the terms $x^2, y^2, z^2, xy, yz, zx, x, y, z$ and the constant term) directly in the **Quadric Function** box, and you will set value to the right of the **Type** drop-down list. Note that the **Update** button will be disabled when you choose this type of surface.

3. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., sphere-slice-7 or quadric-slice-10). (If the **New Surface Name** you enter is the same as the name of a surface that already exists, FLUENT will automatically assign the default name to the new surface when it is created.)

4. Click on the **Create** button to create the new surface.

If you want to check that your new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click on the **Manage...** button to open the **Surfaces** panel. See Section 24.11 for details.

### 24.8 Isosurfaces

If you want to display results on cells that have a constant value for a specified variable, you will need to create an isosurface of that variable. Generating an isosurface based on $x$, $y$, or $z$ coordinate, for example, will give you an $x$, $y$, or $z$ cross-section of your domain; generating an isosurface based on pressure will allow you to display data for another variable on a surface of constant pressure. You can create an isosurface from an existing surface or from the entire domain.

Note that you cannot create an isosurface until you have initialized the solution, performed calculations, or read a data file.
To create an isosurface, you will use the Iso-Surface panel (Figure 24.8.1).

The steps for creating the isosurface are as follows:

1. Choose the scalar variable to be used for isosurfacing in the **Surface of Constant** drop-down list. First, select the desired category in the upper list. You can then select from related quantities from the lower list. (See Chapter 27 for an explanation of the variables in the list.)

2. If you wish to create an isosurface from an existing surface (i.e., generate a new surface of constant \( x \), \( y \), temperature, pressure, etc. that is a subset of another surface), choose that surface in the **From Surface** list. If you do not select a surface from the list, the isosurfacing will be performed on the entire domain.

![Figure 24.8.1: The Iso-Surface Panel](image)
Creating Surfaces for Displaying and Reporting Data

3. Click on the Compute button to calculate the minimum and maximum values of the selected scalar field in the domain or on the selected surface (in the From Surface list). The minimum and maximum values will be displayed in the Min and Max fields.

4. Set the isovalue using one of the following methods. (Note that the second method will allow you to define multiple isovales in a single isosurface.)
   - You can set an isovalue interactively by moving the slider with the left mouse button. The value in the Iso-Values field will be updated automatically. This method will also create a temporary isosurface in the graphics window. Using the slider allows you to preview an isosurface before creating it.
   - Even though the isosurface is displayed, it is only a temporary surface. To create an isosurface, use the Create button after deciding on a particular isovalue.
   - You can type in isovales in the Iso-Values field directly, separating multiple values by white space. Multiple isovales will be contained in a single isosurface; i.e., you cannot select subsurfaces within the resulting isosurface.

5. If you do not want to use the default name assigned to the surface, enter a new name under New Surface Name. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., z-coordinate-6). (If the New Surface Name you enter is the same as the name of a surface that already exists, FLUENT will automatically assign the default name to the new surface when it is created.)

6. Click on the Create button. The new surface name will be added to the From Surface list in the panel.

If you want to delete or otherwise manipulate any surfaces, click on the Manage... button to open the Surfaces panel. See Section 24.11 for details.
24.9 Clipping Surfaces

If you have created a surface, but you do not want to use the whole surface to display data, you can clip the surface between two isovalue limits to create a new surface that spans a specified subrange of a specified scalar quantity. The clipped surface consists of those points on the selected surface where the scalar field values are within the specified range. For example, in Figure 24.9.1 the external wall has been clipped to values of $x$ coordinate less than 0 to show only the back half of the wall, allowing you to see the valve inside the intake port.

![Figure 24.9.1: External Wall Surface Isoclipped to Values of $x$ Coordinate](image)

To clip an existing surface, you will use the Iso-Clip panel (Figure 24.9.2).

Surface — Iso-Clip...

The steps for clipping a surface are as follows:

1. Choose the scalar variable on which the clipping will be based in the Clip To Values Of drop-down list. First, select the desired category...
Creating Surfaces for Displaying and Reporting Data

Figure 24.9.2: The Iso-Clip Panel

in the upper list. You can then select from related quantities from the lower list. (See Chapter 27 for an explanation of the variables in the list.)

2. Select the surface to be clipped in the Clip Surface list.

3. Click on the Compute button to calculate the minimum and maximum values of the selected scalar field on the selected surface. The minimum and maximum values will be displayed in the Min and Max fields.

4. Define the clipping range using one of the following methods.
   - You can set the upper and lower limits of the clipping range interactively by moving the indicator in each dial (i.e., the dial above the Min or Max field) with the left mouse button. The value in the corresponding Min or Max field will be updated automatically. This method will also create a temporary sur-
face in the graphics window. Using the dials allows you to preview a clipped surface before creating it.

Even though the clipped surface is displayed, it is only a temporary surface. To create the new surface, use the Clip button after deciding on the clipping range.

- You can type the minimum and maximum values in the clipping range directly in the Min and Max fields.

5. If you do not want to use the default name assigned to the surface, enter a new name under New Surface Name. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., clip-density-8). (If the New Surface Name you enter is the same as the name of a surface that already exists, FLUENT will automatically assign the default name to the new surface when it is created.)

6. Click on the Clip button. The new surface name will be added to the Clip Surface list in the panel. (The original surface will remain unchanged.)

If you want to delete or otherwise manipulate any surfaces, click on the Manage... button to open the Surfaces panel. See Section 24.11 for details.

24.10 Transforming Surfaces

You can create a new data surface from an existing surface by rotating and/or translating the original surface. For example, you can rotate the surface of a complicated turbomachinery blade to plot data in the region between blades. You can also create a new surface at a constant normal distance from the original surface.

To transform an existing surface to create a new one, you will use the Transform Surface panel (Figure 24.10.1).

The steps for transforming a surface are as follows:
1. Select the surface to be transformed in the Transform Surface list.

2. Set the appropriate transformation parameters, as described below. You can perform any combination of translation, rotation, and “isodistancing” on the surface.

   - Rotation: To rotate a surface, you will specify the origin about which the rotation is performed, and the angle by which the surface is rotated.

   In the About box under Rotate, you will specify a point, and the origin of the coordinate system for the rotation will be set to that point. (The $x$, $y$, and $z$ directions will be the same as for the global coordinate system.) For example, if
you specified the point (1,5,3) in 3D, rotation would be about the x, y, and z axes anchored at (1,5,3). You can either enter the point’s coordinates in the x,y,z fields or click on the Mouse Select button and select a point in the graphics window using the mouse-probe button. (See Section 25.3 for information about mouse button functions.)

In the Angles box under Rotate, you will specify the angles about the x, y, and z axes (i.e., the axes of the coordinate system with the origin defined under About) by which the surface is rotated. For 2D problems, you can specify rotation about the z axis only.

- Translation: To translate a surface, you will simply define the distance by which the surface is translated in each direction. Set the x, y, and z translation distances under Translate.
- Isodistancing: To create a surface positioned at a constant normal distance from the original surface, you need to set only that normal distance between the original surface and the transformed surface. Set the value for d under Iso-Distance.

3. If you do not want to use the default name assigned to the surface, enter a new name under New Surface Name. The default name is the concatenation of the surface type and an integer which is the new surface ID (e.g., transform-9). (If the New Surface Name you enter is the same as the name of a surface that already exists, FLUENT will automatically assign the default name to the new surface when it is created.)

4. Click on the Create button. The new surface name will be added to the Transform Surface list in the panel. (The original surface will remain unchanged.)

If you want to delete or otherwise manipulate any surfaces, click on the Manage... button to open the Surfaces panel. See Section 24.11 for details.
24.11 Grouping, Renaming, and Deleting Surfaces

Once you have created a number of surfaces, you can interactively rename, delete, and group surfaces and obtain information about their components. Grouping surfaces is useful if you want to perform post-processing on a number of surfaces at a time. For example, you may want to group several wall surfaces together to generate a contour plot of temperature on all walls. To postprocess results on each wall surface individually, you will simply “ungroup” the surfaces.

Manipulation of existing surfaces is performed with the Surfaces panel (Figure 24.11.1).

You can also open this panel by clicking on the Manage... button in one of the surface creation panels described in the previous sections.
24.11 Grouping, Renaming, and Deleting Surfaces

Grouping Surfaces

As mentioned above, you may want to group several surfaces together in order to perform postprocessing on all of them at once. To create a surface group, select the surfaces to be grouped in the Surfaces list. You can define a new name for the group in the Name field, or you can use the default name, which is the name of the first surface you selected in the Surfaces list. Then click on the Group button. The selected surfaces will disappear from the Surfaces list, and the name of the surface group will be added to the list.

Note that the Group button will not appear until you have selected at least two surfaces. As soon as you choose a second surface in the Surfaces list, the Rename button will change to the Group button.

To ungroup the surfaces, simply select the surface group in the Surfaces list and click on the UnGroup button. The group name will disappear from the list and the names of the original surfaces in the group will reappear in the list.

Renaming Surfaces

To change the name of an existing surface, select the surface in the Surfaces list, enter a new name in the Name field, and then click on the Rename button. The new name will replace the old name in the Surfaces list and the surface will be otherwise unchanged.

Note that the Rename button will not appear in the panel if you have selected more than one surface. When more than one surface is selected, the Rename button is replaced by the Group button.

Deleting Surfaces

If you find that a surface is no longer useful, you may want to delete it to prevent the list of surfaces from becoming too cluttered. Select the surface or surfaces to be deleted in the Surfaces list, and then click on the Delete button. The delete operation is not reversible, so if you want to get a deleted surface back again you will need to recreate it using one of the surface-creation panels described in the previous sections.
Creating Surfaces for Displaying and Reporting Data

**Surface Statistics**

You can also use the **Surfaces** panel to retrieve topological information about surfaces. **Points** is the total number of nodes in a surface. **0D Facets** is the number of isolated nodes in a surface (i.e., nodes that have no connectivity, such as point surfaces or nodes in a rake). **1D Facets** is the number of linear faces (consisting of two connected nodes) in a surface in a 2D problem, and **2D Facets** is the number of 2D faces (triangular or quadrilateral) in a surface in a 3D problem. Note that an interior zone surface in a 3D problem consists of 2D facets, and similarly an interior zone surface in a 2D problem consists of 1D facets.

These statistics are listed for the surface(s) selected in the **Surfaces** list. If more than one surface is selected, the sum over all selected surfaces is displayed for each quantity.

Note that if you want to check these statistics for a surface that was read from a case file, you will need to first display it.