Chapter 6. Boundary Conditions

This chapter describes the boundary condition options available in FLUENT. Details regarding the boundary condition inputs you must supply and the internal treatment at boundaries are provided.

The information in this chapter is divided into the following sections:

- Section 6.1: Overview of Defining Boundary Conditions
- Section 6.2: Flow Inlets and Exits
- Section 6.3: Pressure Inlet Boundary Conditions
- Section 6.4: Velocity Inlet Boundary Conditions
- Section 6.5: Mass Flow Inlet Boundary Conditions
- Section 6.6: Inlet Vent Boundary Conditions
- Section 6.7: Intake Fan Boundary Conditions
- Section 6.8: Pressure Outlet Boundary Conditions
- Section 6.9: Pressure Far-Field Boundary Conditions
- Section 6.10: Outflow Boundary Conditions
- Section 6.11: Outlet Vent Boundary Conditions
- Section 6.12: Exhaust Fan Boundary Conditions
- Section 6.13: Wall Boundary Conditions
- Section 6.14: Symmetry Boundary Conditions
- Section 6.15: Periodic Boundary Conditions
Boundary Conditions

- Section 6.16: Axis Boundary Conditions
- Section 6.17: Fluid Conditions
- Section 6.18: Solid Conditions
- Section 6.19: Porous Media Conditions
- Section 6.20: Fan Boundary Conditions
- Section 6.21: Radiator Boundary Conditions
- Section 6.22: Porous Jump Boundary Conditions
- Section 6.23: User-Defined Fan Model
- Section 6.24: Heat Exchanger Model
- Section 6.25: Boundary Profiles
- Section 6.26: Fixing the Values of Variables
- Section 6.27: Defining Mass, Momentum, Heat, and Other Sources
- Section 6.28: Coupling Boundary Conditions with GT-Power

6.1 Overview of Defining Boundary Conditions

Boundary conditions specify the flow and thermal variables on the boundaries of your physical model. They are, therefore, a critical component of your FLUENT simulations and it is important that they are specified appropriately.

6.1.1 Available Boundary Types

The boundary types available in FLUENT are classified as follows:

- Flow inlet and exit boundaries: pressure inlet, velocity inlet, mass flow inlet, inlet vent, intake fan, pressure outlet, pressure far-field, outflow, outlet vent, exhaust fan
6.1 Overview of Defining Boundary Conditions

- Wall, repeating, and pole boundaries: wall, symmetry, periodic, axis
- Internal cell zones: fluid, solid (porous is a type of fluid zone)
- Internal face boundaries: fan, radiator, porous jump, wall, interior

(The internal face boundary conditions are defined on cell faces, which means that they do not have a finite thickness and they provide a means of introducing a step change in flow properties. These boundary conditions are used to implement physical models representing fans, thin porous membranes, and radiators. The “interior” type of internal face zone does not require any input from you.)

In this chapter, the boundary conditions listed above will be described, and an explanation of how to set them and when they are most appropriately used will be provided. Note that while periodic boundaries are described in Section 6.15, additional information about modeling fully-developed periodic flows is provided in Section 8.3.

6.1.2 The Boundary Conditions Panel

The Boundary Conditions panel (Figure 6.1.1) allows you to change the boundary zone type for a given zone and open other panels to set the boundary condition parameters for each zone.

Define → Boundary Conditions...

Sections 6.1.3–6.1.6 explain how to perform these operations with the Boundary Conditions panel, and how to use the mouse and the graphics display in conjunction with the panel.

6.1.3 Changing Boundary Zone Types

Before you set any boundary conditions, you should check the zone types of all boundary zones and change any if necessary. For example, if your grid includes a pressure inlet, but you want to use a velocity inlet instead, you will need to change the pressure-inlet zone to a velocity-inlet zone.

The steps for changing a zone type are as follows:
Figure 6.1.1: The Boundary Conditions Panel
6.1 Overview of Defining Boundary Conditions

1. In the **Boundary Conditions** panel, select the zone to be changed in the **Zone** list.

2. Choose the correct zone type in the **Type** list.

3. Confirm the change when prompted by the **Question** dialog box.

   ![Question Dialog Box]

Once you have confirmed the change, the zone type will be changed, the name will change automatically (if the original name was the default name for that zone—see Section 6.1.7), and the panel for setting conditions for the zone will open automatically.

! Note that you cannot use this method to change zone types to or from the periodic type, since additional restrictions exist for this boundary type. Section 5.7.5 explains how to create and uncouple periodic zones.

! If you are using one of the general multiphase models (VOF, mixture, or Eulerian), the procedure for changing types is slightly different. See Section 20.6.14 for details.

**Categories of Zone Types**

You should be aware that you can only change boundary types within each category listed in Table 6.1.1. (Note that a double-sided face zone is a zone that separates two different cell zones or regions.)
Table 6.1.1: Zone Types Listed by Category

<table>
<thead>
<tr>
<th>Category</th>
<th>Zone Types</th>
</tr>
</thead>
<tbody>
<tr>
<td>Faces</td>
<td>axis, outflow, mass flow inlet, pressure far-field, pressure inlet, pressure outlet, symmetry, velocity inlet, wall, inlet vent, intake fan, outlet vent, exhaust fan</td>
</tr>
<tr>
<td>Double-Sided Faces</td>
<td>fan, interior, porous jump, radiator, wall</td>
</tr>
<tr>
<td>Periodic</td>
<td>periodic</td>
</tr>
<tr>
<td>Cells</td>
<td>fluid, solid (porous is a type of fluid cell)</td>
</tr>
</tbody>
</table>

6.1.4 Setting Boundary Conditions

In FLUENT, boundary conditions are associated with zones, not with individual faces or cells. If you want to combine two or more zones that will have the same boundary conditions, see Section 5.7.3 for information about merging zones.

To set boundary conditions for a particular zone, perform one of the following sequences:

1. Choose the zone in the Boundary Conditions panel’s Zone list.

2. Click on the Set... button.

or

1. Choose the zone in the Zone list.

2. Click on the selected zone type in the Type list.

or

1. Double-click on the zone in the Zone list.
6.1 Overview of Defining Boundary Conditions

The panel for the selected boundary zone will open, and you can specify the appropriate boundary conditions.

If you are using one of the general multiphase models (VOF, mixture, or Eulerian), the procedure for setting boundary conditions is slightly different from that described above. See Section 20.6.14 for details.

6.1.5 Copying Boundary Conditions

You can copy boundary conditions from one zone to other zones of the same type. If, for example, you have several wall zones in your model and they all have the same boundary conditions, you can set the conditions for one wall, and then simply copy them to the others.

The procedure for copying boundary conditions is as follows:

1. In the Boundary Conditions panel, click the Copy... button. This will open the Copy BCs panel (Figure 6.1.2).

![Copy BCs Panel](image)

Figure 6.1.2: The Copy BCs Panel

2. In the From Zone list, select the zone that has the conditions you want to copy.
3. In the To Zones list, select the zone or zones to which you want to copy the conditions.

4. Click **Copy**. FLUENT will set *all* of the boundary conditions for the zones selected in the To Zones list to be the same as the conditions for the zone selected in the From Zone list. (You cannot copy a subset of the conditions, such as only the thermal conditions.)

Note that you cannot copy conditions from external walls to internal (i.e., two-sided) walls, or vice versa, if the energy equation is being solved, since the thermal conditions for external and internal walls are different.

! If you are using one of the general multiphase models (VOF, mixture, or Eulerian), the procedure for copying boundary conditions is slightly different. See Section 20.6.14 for details.

### 6.1.6 Selecting Boundary Zones in the Graphics Display

Whenever you need to select a zone in the **Boundary Conditions** panel, you can use the mouse in the graphics window to choose the appropriate zone. This feature is particularly useful if you are setting up a problem for the first time, or if you have two or more zones of the same type and you want to determine the zone IDs (i.e., figure out which zone is which). To use this feature, do the following:

1. Display the grid using the **Grid Display** panel.

2. Use the mouse probe button (the right button, by default—see Section 25.3 to modify the mouse button functions) to click on a boundary zone in the graphics window.

The zone you select in the graphics display will automatically be selected in the Zone list in the **Boundary Conditions** panel, and its name and ID will be printed in the console window.
6.1 Overview of Defining Boundary Conditions

6.1.7 Changing Boundary Zone Names

The default name for a zone is its type plus an ID number (e.g., pressure-inlet-7). In some cases, you may want to assign more descriptive names to the boundary zones. If you have two pressure-inlet zones, for example, you might want to rename them small-inlet and large-inlet. (Changing the name of a zone will not change its type. Instructions for changing a zone’s type are provided in Section 6.1.3.)

To rename a zone, follow these steps:

1. Select the zone to be renamed in the Zones list in the Boundary Conditions panel.
2. Click Set... to open the panel for the selected zone.
3. Enter a new name under Zone Name.
4. Click the OK button.

Note that if you specify a new name for a zone and then change its type, the name you specified will be retained; the automatic name change that accompanies a change in type occurs only if the name of the zone is its type plus its ID.

6.1.8 Defining Non-Uniform Boundary Conditions

Most conditions at each type of boundary zone can be defined as profile functions instead of constant values. You can use a profile contained in an externally generated boundary profile file, or a function that you create using a user-defined function (UDF). Boundary condition profiles are described in Section 6.25, and user-defined functions are described in the separate UDF Manual.
6.1.9 Defining Transient Boundary Conditions

There are two ways you can specify transient boundary conditions:

- transient profile with a format similar to the standard boundary profiles described in Section 6.25
- transient profile in a tabular format

For both methods, the boundary condition will vary only in time; it must be spatially uniform.

Standard Transient Profiles

The format of the standard transient profile file (based on the profiles described in Section 6.25) is

```
((profile-name transient n periodic?)
 (field_name-1 a1 a2 a3 .... an)
 (field_name-2 b1 b2 b3 .... bn)
 .
 .
 .
 (field_name-r r1 r2 r3 .... rn))
```

One of the field names should be used for the time field, and the time field section must be in ascending order. The periodic? entry indicates whether or not the profile is time-periodic. Set it to 1 for a time-periodic profile, or 0 if the profile is not time-periodic.

An example is shown below:

```
((sampleprofile transient 3 1)
 (time
  1
  2
```

© Fluent Inc. November 28, 2001
6.1 Overview of Defining Boundary Conditions

All quantities, including coordinate values, must be specified in SI units because FLUENT does not perform unit conversion when reading profile files. Also, boundary profile names must have all lowercase letters (e.g., \texttt{name}). Uppercase letters in boundary profile names are not acceptable.

You can read this file into FLUENT using the Boundary Profiles panel or the File/Read/Profile... menu item.

\[
\text{Define} \rightarrow \text{Profiles...}
\]
\[
\text{File} \rightarrow \text{Read} \rightarrow \text{Profile...}
\]

See Section 6.25.3 for details.

**Tabular Transient Profiles**

The format of the tabular transient profile file is

\[
\text{profile-name n\_field n\_data periodic?}
\text{field-name-1 field-name-2 field-name-3 \ldots field-name-n\_field}
\text{v-1-1 v-2-1 \ldots \ldots \ldots v-n\_field-1}
\text{v-1-2 v-2-2 \ldots \ldots \ldots v-n\_field-2}
\ldots
\ldots
\ldots
\ldots
\text{v-1-n\_data v-2-n\_data \ldots \ldots \ldots v-n\_field-n\_data}
\]
Boundary Conditions

One of the field-names should be used for the time field, and the time field section must be in ascending order. The periodic? entry indicates whether or not the profile is time-periodic. Set it to 1 for a time-periodic profile, or 0 if the profile is not time-periodic.

An example is shown below:

sampletabprofile 2 3 1
  time u
  1 10
  2 20
  3 30

This file defines the same transient profile as the standard profile example above.

All quantities, including coordinate values, must be specified in SI units because FLUENT does not perform unit conversion when reading profile files. Also, boundary profile names must have all lowercase letters (e.g., name). Uppercase letters in boundary profile names are not acceptable.

You can read this file into FLUENT using the read-transient-table text command.

6.1.10 Saving and Reusing Boundary Conditions

You can save boundary conditions to a file so that you can use them to specify the same conditions for a different case, as described in Section 3.10.
6.2 Flow Inlets and Exits

FLUENT has a wide range of boundary conditions that permit flow to enter and exit the solution domain. To help you select the most appropriate boundary condition for your application, this section includes descriptions of how each type of condition is used, and what information is needed for each one. Recommendations for determining inlet values of the turbulence parameters are also provided.

6.2.1 Using Flow Boundary Conditions

This section provides an overview of flow boundaries in FLUENT and how to use them.

FLUENT provides 10 types of boundary zone types for the specification of flow inlets and exits: velocity inlet, pressure inlet, mass flow inlet, pressure outlet, pressure far-field, outflow, inlet vent, intake fan, outlet vent, and exhaust fan.

The inlet and exit boundary condition options in FLUENT are as follows:

- Velocity inlet boundary conditions are used to define the velocity and scalar properties of the flow at inlet boundaries.
- Pressure inlet boundary conditions are used to define the total pressure and other scalar quantities at flow inlets.
- Mass flow inlet boundary conditions are used in compressible flows to prescribe a mass flow rate at an inlet. It is not necessary to use mass flow inlets in incompressible flows because when density is constant, velocity inlet boundary conditions will fix the mass flow.
- Pressure outlet boundary conditions are used to define the static pressure at flow outlets (and also other scalar variables, in case of backflow). The use of a pressure outlet boundary condition instead of an outflow condition often results in a better rate of convergence when backflow occurs during iteration.
- Pressure far-field boundary conditions are used to model a free-stream compressible flow at infinity, with free-stream Mach number
**Boundary Conditions**

and static conditions specified. This boundary type is available only for compressible flows.

- Outflow boundary conditions are used to model flow exits where the details of the flow velocity and pressure are not known prior to solution of the flow problem. They are appropriate where the exit flow is close to a fully developed condition, as the outflow boundary condition assumes a zero normal gradient for all flow variables except pressure. They are not appropriate for compressible flow calculations.

- Inlet vent boundary conditions are used to model an inlet vent with a specified loss coefficient, flow direction, and ambient (inlet) total pressure and temperature.

- Intake fan boundary conditions are used to model an external intake fan with a specified pressure jump, flow direction, and ambient (intake) total pressure and temperature.

- Outlet vent boundary conditions are used to model an outlet vent with a specified loss coefficient and ambient (discharge) static pressure and temperature.

- Exhaust fan boundary conditions are used to model an external exhaust fan with a specified pressure jump and ambient (discharge) static pressure.

### 6.2.2 Determining Turbulence Parameters

When the flow enters the domain at an inlet, outlet, or far-field boundary, FLUENT requires specification of transported turbulence quantities. This section describes which quantities are needed for specific turbulence models and how they must be specified. It also provides guidelines for the most appropriate way of determining the inflow boundary values.

**Specification of Turbulence Quantities Using Profiles**

If it is important to accurately represent a boundary layer or fully-developed turbulent flow at the inlet, you should ideally set the turbulence quantities by creating a boundary profile file (see Section 6.25)
from experimental data or empirical formulas. If you have an analytical
description of the profile, rather than data points, you can either use
this analytical description to create a boundary profile file, or create a
user-defined function to provide the inlet boundary information. (See
the separate UDF Manual for information on user-defined functions.)

Once you have created the profile function, you can use it as described
below:

- Spalart-Allmaras model: Choose Turbulent Viscosity or Turbulent
  Viscosity Ratio in the Turbulence Specification Method drop-down
  list and select the appropriate profile name in the drop-down list
  next to Turbulent Viscosity or Turbulent Viscosity Ratio. FLUENT
  computes the boundary value for the modified turbulent viscosity,
  \( \tilde{\nu} \), by combining \( \mu_t/\mu \) with the appropriate values of density and
  molecular viscosity.

- \( k-\epsilon \) models: Choose K and Epsilon in the Turbulence Specification
  Method drop-down list and select the appropriate profile names in
  the drop-down lists next to Turb. Kinetic Energy and Turb. Dissi-
  pation Rate.

- \( k-\omega \) models: Choose K and Omega in the Turbulence Specification
  Method drop-down list and select the appropriate profile names in
  the drop-down lists next to Turb. Kinetic Energy and Spec. Dissi-
  pation Rate.

- Reynolds stress model: Choose K and Epsilon in the Turbulence
  Specification Method drop-down list and select the appropriate pro-
  file names in the drop-down lists next to Turb. Kinetic Energy and Turb.
  Dissipation Rate. Choose Reynolds-Stress Components in the
  Reynolds-Stress Specification Method drop-down list and select the
  appropriate profile name in the drop-down list next to each of the
  individual Reynolds-stress components.

**Uniform Specification of Turbulence Quantities**

In some situations, it is appropriate to specify a uniform value of the
turbulence quantity at the boundary where inflow occurs. Examples are
Boundary Conditions

fluid entering a duct, far-field boundaries, or even fully-developed duct flows where accurate profiles of turbulence quantities are unknown.

In most turbulent flows, higher levels of turbulence are generated within shear layers than enter the domain at flow boundaries, making the result of the calculation relatively insensitive to the inflow boundary values. Nevertheless, caution must be used to ensure that boundary values are not so unphysical as to contaminate your solution or impede convergence. This is particularly true of external flows where unphysically large values of effective viscosity in the free stream can “swamp” the boundary layers.

You can use the turbulence specification methods described above to enter uniform constant values instead of profiles. Alternatively, you can specify the turbulence quantities in terms of more convenient quantities such as turbulence intensity, turbulent viscosity ratio, hydraulic diameter, and turbulence length scale. These quantities are discussed further in the following sections.

Turbulence Intensity

The turbulence intensity, $I$, is defined as the ratio of the root-mean-square of the velocity fluctuations, $u'$, to the mean flow velocity, $u_{avg}$.

A turbulence intensity of 1% or less is generally considered low and turbulence intensities greater than 10% are considered high. Ideally, you will have a good estimate of the turbulence intensity at the inlet boundary from external, measured data. For example, if you are simulating a wind-tunnel experiment, the turbulence intensity in the free stream is usually available from the tunnel characteristics. In modern low-turbulence wind tunnels, the free-stream turbulence intensity may be as low as 0.05%.

For internal flows, the turbulence intensity at the inlets is totally dependent on the upstream history of the flow. If the flow upstream is under-developed and undisturbed, you can use a low turbulence intensity. If the flow is fully developed, the turbulence intensity may be as high as a few percent. The turbulence intensity at the core of a fully-developed duct flow can be estimated from the following formula derived from an empirical correlation for pipe flows:
6.2 Flow Inlets and Exits

\[ I \equiv \frac{u'}{u_{avg}} = 0.16(Re_{D_H})^{-1/8} \]  \hspace{1cm} (6.2-1)

At a Reynolds number of 50,000, for example, the turbulence intensity will be 4%, according to this formula.

Turbulence Length Scale and Hydraulic Diameter

The turbulence length scale, \( \ell \), is a physical quantity related to the size of the large eddies that contain the energy in turbulent flows.

In fully-developed duct flows, \( \ell \) is restricted by the size of the duct, since the turbulent eddies cannot be larger than the duct. An approximate relationship between \( \ell \) and the physical size of the duct is

\[ \ell = 0.07L \]  \hspace{1cm} (6.2-2)

where \( L \) is the relevant dimension of the duct. The factor of 0.07 is based on the maximum value of the mixing length in fully-developed turbulent pipe flow, where \( L \) is the diameter of the pipe. In a channel of noncircular cross-section, you can base \( L \) on the hydraulic diameter.

If the turbulence derives its characteristic length from an obstacle in the flow, such as a perforated plate, it is more appropriate to base the turbulence length scale on the characteristic length of the obstacle rather than on the duct size.

It should be noted that the relationship of Equation 6.2-2, which relates a physical dimension (\( L \)) to the turbulence length scale (\( \ell \)), is not necessarily applicable to all situations. For most cases, however, it is a suitable approximation.

Guidelines for choosing the characteristic length \( L \) or the turbulence length scale \( \ell \) for selected flow types are listed below:

- For fully-developed internal flows, choose the Intensity and Hydraulic Diameter specification method and specify the hydraulic diameter \( L = D_H \) in the Hydraulic Diameter field.
Boundary Conditions

- For flows downstream of turning vanes, perforated plates, etc., choose the Intensity and Hydraulic Diameter method and specify the characteristic length of the flow opening for $L$ in the Hydraulic Diameter field.

- For wall-bounded flows in which the inlets involve a turbulent boundary layer, choose the Intensity and Length Scale method and use the boundary-layer thickness, $\delta_{99}$, to compute the turbulence length scale, $\ell$, from $\ell = 0.4\delta_{99}$. Enter this value for $\ell$ in the Turbulence Length Scale field.

Turbulent Viscosity Ratio

The turbulent viscosity ratio, $\mu_t/\mu$, is directly proportional to the turbulent Reynolds number ($Re_t \equiv k^2/(\nu)$. $Re_t$ is large (on the order of 100 to 1000) in high-Reynolds-number boundary layers, shear layers, and fully-developed duct flows. However, at the free-stream boundaries of most external flows, $\mu_t/\mu$ is fairly small. Typically, the turbulence parameters are set so that $1 < \mu_t/\mu < 10$.

To specify quantities in terms of the turbulent viscosity ratio, you can choose Turbulent Viscosity Ratio (for the Spalart-Allmaras model) or Intensity and Viscosity Ratio (for the $k-\epsilon$ models, the $k-\omega$ models, or the RSM).

Relationships for Deriving Turbulence Quantities

To obtain the values of transported turbulence quantities from more convenient quantities such as $I$, $L$, or $\mu_t/\mu$, you must typically resort to an empirical relation. Several useful relations, most of which are used within FLUENT, are presented below.

Estimating Modified Turbulent Viscosity from Turbulence Intensity and Length Scale

To obtain the modified turbulent viscosity, $\bar{\nu}$, for the Spalart-Allmaras model from the turbulence intensity, $I$, and length scale, $\ell$, the following equation can be used:
6.2 Flow Inlets and Exits

\[ \bar{v} = \sqrt{\frac{3}{2}} u_{\text{avg}} I \ell \]  

(6.2-3)

This formula is used in FLUENT if you select the Intensity and Hydraulic Diameter specification method with the Spalart-Allmaras model. \( \ell \) is obtained from Equation 6.2-2.

**Estimating Turbulent Kinetic Energy from Turbulence Intensity**

The relationship between the turbulent kinetic energy, \( k \), and turbulence intensity, \( I \), is

\[ k = \frac{3}{2} (u_{\text{avg}} I)^2 \]  

(6.2-4)

where \( u_{\text{avg}} \) is the mean flow velocity.

This relationship is used in FLUENT whenever the Intensity and Hydraulic Diameter, Intensity and Length Scale, or Intensity and Viscosity Ratio method is used instead of specifying explicit values for \( k \) and \( \epsilon \).

**Estimating Turbulent Dissipation Rate from a Length Scale**

If you know the turbulence length scale, \( \ell \), you can determine \( \epsilon \) from the relationship

\[ \epsilon = C_\mu^{3/4} \frac{k^{3/2}}{\ell} \]  

(6.2-5)

where \( C_\mu \) is an empirical constant specified in the turbulence model (approximately 0.09). The determination of \( \ell \) was discussed previously.

This relationship is used in FLUENT whenever the Intensity and Hydraulic Diameter or Intensity and Length Scale method is used instead of specifying explicit values for \( k \) and \( \epsilon \).
**Boundary Conditions**

*Estimating Turbulent Dissipation Rate from Turbulent Viscosity Ratio*

The value of $\epsilon$ can be obtained from the turbulent viscosity ratio $\mu_t/\mu$ and $k$ using the following relationship:

$$\epsilon = \rho C_\mu \frac{k^2}{\mu} \left( \frac{\mu_t}{\mu} \right)^{-1}$$  \hspace{1cm} (6.2-6)

where $C_\mu$ is an empirical constant specified in the turbulence model (approximately 0.09).

This relationship is used in FLUENT whenever the Intensity and Viscosity Ratio method is used instead of specifying explicit values for $k$ and $\epsilon$.

*Estimating Turbulent Dissipation Rate for Decaying Turbulence*

If you are simulating a wind-tunnel situation in which the model is mounted in the test section downstream of a grid and/or wire mesh screens, you can choose a value of $\epsilon$ such that

$$\epsilon \approx \frac{\Delta k U_\infty}{L_\infty}$$ \hspace{1cm} (6.2-7)

where $\Delta k$ is the approximate decay of $k$ you wish to have across the flow domain (say, 10% of the inlet value of $k$), $U_\infty$ is the free-stream velocity, and $L_\infty$ is the streamwise length of the flow domain. Equation 6.2-7 is a linear approximation to the power-law decay observed in high-Reynolds-number isotropic turbulence. Its basis is the exact equation for $k$ in decaying turbulence, $U \partial k/\partial x = -\epsilon$.

If you use this method to estimate $\epsilon$, you should also check the resulting turbulent viscosity ratio $\mu_t/\mu$ to make sure that it is not too large, using Equation 6.2-6.

Although this method is not used internally by FLUENT, you can use it to derive a constant free-stream value of $\epsilon$ that you can then specify directly by choosing $K$ and $\text{Epsilon}$ in the Turbulence Specification Method drop-down list. In this situation, you will typically determine $k$ from $I$ using Equation 6.2-4.
6.2 Flow Inlets and Exits

Estimating Specific Dissipation Rate from a Length Scale

If you know the turbulence length scale, \( \ell \), you can determine \( \omega \) from the relationship

\[
\omega = \frac{k^{1/2}}{C_\mu^{1/4} \ell}
\]

(6.2-8)

where \( C_\mu \) is an empirical constant specified in the turbulence model (approximately 0.09). The determination of \( \ell \) was discussed previously.

This relationship is used in FLUENT whenever the Intensity and Hydraulic Diameter or Intensity and Length Scale method is used instead of specifying explicit values for \( k \) and \( \omega \).

Estimating Specific Dissipation Rate from Turbulent Viscosity Ratio

The value of \( \omega \) can be obtained from the turbulent viscosity ratio \( \mu_\ell/\mu \) and \( k \) using the following relationship:

\[
\omega = \rho \frac{k}{\mu} \left( \frac{\mu_\ell}{\mu} \right)^{-1}
\]

(6.2-9)

This relationship is used in FLUENT whenever the Intensity and Viscosity Ratio method is used instead of specifying explicit values for \( k \) and \( \omega \).

Estimating Reynolds Stress Components from Turbulent Kinetic Energy

When the RSM is used, if you do not specify the values of the Reynolds stresses explicitly at the inlet using the Reynolds-Stress Components option in the Reynolds-Stress Specification Method drop-down list, they are approximately determined from the specified values of \( k \). The turbulence is assumed to be isotropic such that

\[
\overline{u'_i u'_j} = 0
\]

(6.2-10)

and
Boundary Conditions

\[
\bar{u}_\alpha \bar{u}_\alpha = \frac{2}{3} k
\]  
(6.2-11)

(no summation over the index \(\alpha\)).

FLUENT will use this method if you select K or Turbulence Intensity in the Reynolds-Stress Specification Method drop-down list.

Specifying Inlet Turbulence for LES

The turbulence intensity value specified at a velocity inlet for LES, as described in Section 10.10.2, is used to randomly perturb the instantaneous velocity field at the inlet. It does not specify a modeled turbulence quantity. Instead, the stochastic components of the flow at the inlet boundary are accounted for by superposing random perturbations on individual velocity components as described in Section 10.7.3.
6.3 Pressure Inlet Boundary Conditions

Pressure inlet boundary conditions are used to define the fluid pressure at flow inlets, along with all other scalar properties of the flow. They are suitable for both incompressible and compressible flow calculations. Pressure inlet boundary conditions can be used when the inlet pressure is known but the flow rate and/or velocity is not known. This situation may arise in many practical situations, including buoyancy-driven flows. Pressure inlet boundary conditions can also be used to define a “free” boundary in an external or unconfined flow.

You can find the following information about pressure inlet boundary conditions in this section:

- Section 6.3.1: Inputs at Pressure Inlet Boundaries
- Section 6.3.2: Default Settings at Pressure Inlet Boundaries
- Section 6.3.3: Calculation Procedure at Pressure Inlet Boundaries

For an overview of flow boundaries, see Section 6.2.

6.3.1 Inputs at Pressure Inlet Boundaries

Summary

You will enter the following information for a pressure inlet boundary:

- Total (stagnation) pressure
- Total (stagnation) temperature
- Flow direction
- Static pressure
- Turbulence parameters (for turbulent calculations)
- Radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
Boundary Conditions

- Chemical species mass fractions (for species calculations)
- Mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- Progress variable (for premixed or partially premixed combustion calculations)
- Discrete phase boundary conditions (for discrete phase calculations)
- Multiphase boundary conditions (for general multiphase calculations)

All values are entered in the Pressure Inlet panel (Figure 6.3.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).

Pressure Inputs and Hydrostatic Head

The pressure field \( p'_s \) and your pressure inputs \( p'_s \) or \( p'_0 \) include the hydrostatic head, \( \rho_0gx \). That is, the pressure in FLUENT is defined as

\[
p'_s = p_s - \rho_0gx \tag{6.3-1}
\]

or

\[
\frac{\partial p'_s}{\partial x} = \frac{\partial p_s}{\partial x} - \rho_0g \tag{6.3-2}
\]

This definition allows the hydrostatic head to be taken into the body force term, \( (\rho - \rho_0)g \), and excluded from the pressure calculation when the density is uniform. Thus your inputs of pressure should not include hydrostatic pressure differences, and reports of pressure \( p'_s \) will not show any influence of the hydrostatic pressure. See Section 11.5 for information about buoyancy-driven (natural-convection) flows.
6.3 Pressure Inlet Boundary Conditions

Figure 6.3.1: The Pressure Inlet Panel
Defining Total Pressure and Temperature

Enter the value for total pressure in the Gauge Total Pressure field in the Pressure Inlet panel. Total temperature is set in the Total Temperature field.

Remember that the total pressure value is the gauge pressure with respect to the operating pressure defined in the Operating Conditions panel. Total pressure for an incompressible fluid is defined as

\[ p_0 = p_s + \frac{1}{2} \rho \| \vec{v} \|^2 \]  

(6.3-3)

and for a compressible fluid as

\[ p_0 = p_s \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{\gamma / (\gamma - 1)} \]  

(6.3-4)

where

- \( p_0 \) = total pressure
- \( p_s \) = static pressure
- \( M \) = Mach number
- \( \gamma \) = ratio of specific heats (\( c_p/c_v \))

If you are modeling axisymmetric swirl, \( \vec{v} \) in Equation 6.3-3 will include the swirl component.

If the adjacent cell zone is moving (i.e., if you are using a rotating reference frame, multiple reference frames, a mixing plane, or sliding meshes) and you are using the segregated solver, the velocity in Equation 6.3-3 (or the Mach number in Equation 6.3-4) will be absolute or relative to the grid velocity, depending on whether or not the Absolute velocity formulation is enabled in the Solver panel. For the coupled solvers, the velocity in Equation 6.3-3 (or the Mach number in Equation 6.3-4) is always in the absolute frame.

Defining the Flow Direction

You can define the flow direction at a pressure inlet explicitly, or you can define the flow to be normal to the boundary. If you choose to specify
6.3 Pressure Inlet Boundary Conditions

the direction vector, you can set either the (Cartesian) $x$, $y$, and $z$ components, or the (cylindrical) radial, tangential, and axial components.

For moving zone problems calculated using the segregated solver, the flow direction will be absolute or relative to the grid velocity, depending on whether or not the Absolute velocity formulation is enabled in the Solver panel. For the coupled solvers, the flow direction will always be in the absolute frame.

The procedure for defining the flow direction is as follows, referring to Figure 6.3.1:

1. Choose which method you will use to specify the flow direction by selecting Direction Vector or Normal to Boundary in the Direction Specification Method drop-down list.

2. If you selected Normal to Boundary in step 1 and you are modeling axisymmetric swirl, enter the appropriate value for the Tangential-Component of Flow Direction. If you chose Normal to Boundary and your geometry is 3D or 2D without axisymmetric swirl, there are no additional inputs for flow direction.

3. If you chose in step 1 to specify the direction vector, and your geometry is 3D, you will next choose the coordinate system in which you will define the vector components. Choose Cartesian ($X, Y, Z$), Cylindrical (Radial, Tangential, Axial), or Local Cylindrical (Radial, Tangential, Axial) in the Coordinate System drop-down list.

   - The Cartesian coordinate system is based on the Cartesian coordinate system used by the geometry.

   - The Cylindrical coordinate system uses the axial, radial, and tangential components based on the following coordinate systems:

     - For problems involving a single cell zone, the coordinate system is defined by the rotation axis and origin specified in the Fluid panel.

     - For problems involving multiple zones (e.g., multiple reference frames or sliding meshes), the coordinate system...
is defined by the rotation axis specified in the Fluid (or Solid) panel for the fluid (or solid) zone that is adjacent to the inlet.

For all of the above definitions of the cylindrical coordinate system, positive radial velocities point radially out from the rotation axis, positive axial velocities are in the direction of the rotation axis vector, and positive tangential velocities are based on the right-hand rule using the positive rotation axis (see Figure 6.3.2).

Figure 6.3.2: Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains

- The Local Cylindrical coordinate system allows you to define a coordinate system specifically for the inlet. When you use the local cylindrical option, you will define the coordinate system right here in the Pressure Inlet panel. The local cylindrical coordinate system is useful if you have several inlets with different rotation axes.

4. If you chose in step 1 to specify the direction vector, define the vector components as follows:

- If your geometry is 2D non-axisymmetric, or you chose in step
6.3 Pressure Inlet Boundary Conditions

3 to input Cartesian vector components, enter the appropriate values for X, Y, and (in 3D) Z-Component of Flow Direction.

- If your geometry is 2D axisymmetric, or you chose in step 3 to input Cylindrical components, enter the appropriate values for Axial, Radial, and (if you are modeling axisymmetric swirl or using cylindrical coordinates) Tangential-Component of Flow Direction.

- If you are using Local Cylindrical coordinates, enter the appropriate values for Axial, Radial, and Tangential-Component of Flow Direction, and then specify the X, Y, and Z-Component of Axis Direction and the X, Y, and Z-Coordinate of Axis Origin.

Figure 6.3.2 shows the vector components for these different coordinate systems.

Defining Static Pressure

The static pressure (termed the Supersonic/Initial Gauge Pressure) must be specified if the inlet flow is supersonic or if you plan to initialize the solution based on the pressure inlet boundary conditions. Solution initialization is discussed in Section 22.13.

Remember that the static pressure value you enter is relative to the operating pressure set in the Operating Conditions panel. Note the comments in Section 6.3.1 regarding hydrostatic pressure.

The Supersonic/Initial Gauge Pressure is ignored by FLUENT whenever the flow is subsonic, in which case it is calculated from the specified stagnation quantities. If you choose to initialize the solution based on the pressure-inlet conditions, the Supersonic/Initial Gauge Pressure will be used in conjunction with the specified stagnation pressure to compute initial values according to the isentropic relations (for compressible flow) or Bernoulli’s equation (for incompressible flow). Therefore, for a subsonic inlet it should generally be set based on a reasonable estimate of the inlet Mach number (for compressible flow) or inlet velocity (for incompressible flow).
Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in Section 6.2.2. Turbulence modeling in general is described in Chapter 10.

Defining Radiation Parameters

If you are using the P-1 radiation model, the DTRM, the DO model, or the surface-to-surface model, you will set the Internal Emissivity and (optionally) Black Body Temperature. See Section 11.3.16 for details. (The Rosseland radiation model does not require any boundary condition inputs.)

Defining Species Mass Fractions

If you are modeling species transport, you will set the species mass fractions under Species Mass Fractions. For details, see Section 13.1.5.

Defining Non-Premixed Combustion Parameters

If you are using the non-premixed or partially premixed combustion model, you will set the Mean Mixture Fraction and Mixture Fraction Variance (and the Secondary Mean Mixture Fraction and Secondary Mixture Fraction Variance, if you are using two mixture fractions), as described in Section 14.3.3.

Defining Premixed Combustion Boundary Conditions

If you are using the premixed or partially premixed combustion model, you will set the Progress Variable, as described in Section 15.3.5.

Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the pressure inlet. See Section 19.10 for details.
6.3 Pressure Inlet Boundary Conditions

Defining Multiphase Boundary Conditions

If you are using the VOF, mixture, or Eulerian model for multiphase flow, you will need to specify volume fractions for secondary phases and (for some models) additional parameters. See Section 20.6.14 for details.

6.3.2 Default Settings at Pressure Inlet Boundaries

Default settings (in SI) for pressure inlet boundary conditions are as follows:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gauge Total Pressure</td>
<td>0</td>
</tr>
<tr>
<td>Supersonic/Initial Gauge Pressure</td>
<td>0</td>
</tr>
<tr>
<td>Total Temperature</td>
<td>300</td>
</tr>
<tr>
<td>X-Component of Flow Direction</td>
<td>1</td>
</tr>
<tr>
<td>Y-Component of Flow Direction</td>
<td>0</td>
</tr>
<tr>
<td>Z-Component of Flow Direction</td>
<td>0</td>
</tr>
<tr>
<td>Turb. Kinetic Energy</td>
<td>1</td>
</tr>
<tr>
<td>Turb. Dissipation Rate</td>
<td>1</td>
</tr>
</tbody>
</table>

6.3.3 Calculation Procedure at Pressure Inlet Boundaries

The treatment of pressure inlet boundary conditions by FLUENT can be described as a loss-free transition from stagnation conditions to the inlet conditions. For incompressible flows, this is accomplished by application of the Bernoulli equation at the inlet boundary. In compressible flows, the equivalent isentropic flow relations for an ideal gas are used.

Incompressible Flow Calculations at Pressure Inlet Boundaries

When flow enters through a pressure inlet boundary, FLUENT uses the boundary condition pressure you input as the total pressure of the fluid at the inlet plane, $p_0$. In incompressible flow, the inlet total pressure and the static pressure, $p_s$, are related to the inlet velocity via Bernoulli’s equation:

$$p_0 = p_s + \frac{1}{2} \rho v^2 \quad (6.3-5)$$
Boundary Conditions

With the resulting velocity magnitude and the flow direction vector you assigned at the inlet, the velocity components can be computed. The inlet mass flow rate and fluxes of momentum, energy, and species can then be computed as outlined in Section 6.4.3.

For incompressible flows, density at the inlet plane is either constant or calculated as a function of temperature and/or species mass fractions, where the mass fractions are the values you entered as an inlet condition.

If flow exits through a pressure inlet, the total pressure specified is used as the static pressure. For incompressible flows, total temperature is equal to static temperature.

Compressible Flow Calculations at Pressure Inlet Boundaries

In compressible flows, isentropic relations for an ideal gas are applied to relate total pressure, static pressure, and velocity at a pressure inlet boundary. Your input of total pressure, \( p'_0 \), at the inlet and the static pressure, \( p'_s \), in the adjacent fluid cell are thus related as

\[
\frac{p'_0 + p_{op}}{p'_s + p_{op}} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\gamma/(\gamma-1)}
\]

(6.3-6)

where

\[
M \equiv \frac{v}{c} = \frac{v}{\sqrt{\gamma R T_s}}
\]

(6.3-7)

\( c = \) the speed of sound, and \( \gamma = c_p/c_v \). Note that the operating pressure, \( p_{op} \), appears in Equation 6.3-6 because your boundary condition inputs are in terms of pressure relative to the operating pressure. Given \( p'_0 \) and \( p'_s \), Equations 6.3-6 and 6.3-7 are used to compute the velocity magnitude of the fluid at the inlet plane. Individual velocity components at the inlet are then derived using the direction vector components.

For compressible flow, the density at the inlet plane is defined by the ideal gas law in the form
6.4 Velocity Inlet Boundary Conditions

\[ \rho = \frac{p_s' + p_{op}}{RT_s} \]  

(6.3-8)

The specific gas constant, \( R \), is computed from the species mass fractions, \( Y_i \), that you defined as boundary conditions at the pressure inlet boundary. The static temperature at the inlet, \( T_s \), is computed from your input of total temperature, \( T_0 \), as

\[ \frac{T_0}{T_s} = 1 + \frac{\gamma - 1}{2} M^2 \]  

(6.3-9)

6.4 Velocity Inlet Boundary Conditions

Velocity inlet boundary conditions are used to define the flow velocity, along with all relevant scalar properties of the flow, at flow inlets. The total (or stagnation) properties of the flow are not fixed, so they will rise to whatever value is necessary to provide the prescribed velocity distribution.

This boundary condition is intended for incompressible flows, and its use in compressible flows will lead to a nonphysical result because it allows stagnation conditions to float to any level. You should also be careful not to place a velocity inlet too close to a solid obstruction, since this could cause the inflow stagnation properties to become highly non-uniform.

In special instances, a velocity inlet may be used in FLUENT to define the flow velocity at flow exits. (The scalar inputs are not used in such cases.) In such cases you must ensure that overall continuity is maintained in the domain.

You can find the following information about velocity inlet boundary conditions in this section:

- Section 6.4.1: Inputs at Velocity Inlet Boundaries
- Section 6.4.2: Default Settings at Velocity Inlet Boundaries
- Section 6.4.3: Calculation Procedure at Velocity Inlet Boundaries

For an overview of flow boundaries, see Section 6.2.
6.4.1 Inputs at Velocity Inlet Boundaries

Summary

You will enter the following information for a velocity inlet boundary:

- Velocity magnitude and direction or velocity components
- Swirl velocity (for 2D axisymmetric problems with swirl)
- Temperature (for energy calculations)
- Outflow gauge pressure (for calculations with the coupled solvers)
- Turbulence parameters (for turbulent calculations)
- Radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- Chemical species mass fractions (for species calculations)
- Mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- Progress variable (for premixed or partially premixed combustion calculations)
- Discrete phase boundary conditions (for discrete phase calculations)
- Multiphase boundary conditions (for general multiphase calculations)

All values are entered in the Velocity Inlet panel (Figure 6.4.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).
6.4 Velocity Inlet Boundary Conditions

Figure 6.4.1: The Velocity Inlet Panel
**Boundary Conditions**

**Defining the Velocity**

You can define the inflow velocity by specifying the velocity magnitude and direction, the velocity components, or the velocity magnitude normal to the boundary. If the cell zone adjacent to the velocity inlet is moving (i.e., if you are using a rotating reference frame, multiple reference frames, or sliding meshes), you can specify either relative or absolute velocities. For axisymmetric problems with swirl in FLUENT, you will also specify the swirl velocity.

The procedure for defining the inflow velocity is as follows:

1. Choose which method you will use to specify the flow direction by selecting *Magnitude and Direction*, *Components*, or *Magnitude, Normal to Boundary* in the *Velocity Specification Method* drop-down list.

2. If the cell zone adjacent to the velocity inlet is moving, you can choose to specify relative or absolute velocities by selecting *Relative to Adjacent Cell Zone* or *Absolute* in the *Reference Frame* drop-down list. If the adjacent cell zone is not moving, *Absolute* and *Relative to Adjacent Cell Zone* will be equivalent, so you need not visit the list.

3. If you are going to set the velocity magnitude and direction or the velocity components, and your geometry is 3D, you will next choose the coordinate system in which you will define the vector or velocity components. Choose *Cartesian (X, Y, Z)*, *Cylindrical (Radial, Tangential, Axial)*, or *Local Cylindrical (Radial, Tangential, Axial)* in the *Coordinate System* drop-down list. See Section 6.3.1 for information about Cartesian, cylindrical, and local cylindrical coordinate systems.

4. Set the appropriate velocity parameters, as described below for each specification method.

**Setting the Velocity Magnitude and Direction**

If you selected *Magnitude and Direction* as the *Velocity Specification Method* in step 1 above, you will enter the magnitude of the velocity vector at
6.4 Velocity Inlet Boundary Conditions

the inflow boundary (the Velocity Magnitude) and the direction of the vector:

- If your geometry is 2D non-axisymmetric, or you chose in step 3 to use the Cartesian coordinate system, you will define the $X$, $Y$, and (in 3D) $Z$-Component of Flow Direction.

- If your geometry is 2D axisymmetric, or you chose in step 3 to use a Cylindrical coordinate system, enter the appropriate values of Radial, Axial, and (if you are modeling axisymmetric swirl or using cylindrical coordinates) Tangential-Component of Flow Direction.

- If you chose in step 3 to use a Local Cylindrical coordinate system, enter the appropriate values for Axial, Radial, and Tangential-Component of Flow Direction, and then specify the $X$, $Y$, and $Z$-Component of Axis Direction and the $X$, $Y$, and $Z$-Coordinate of Axis Origin.

Figure 6.3.2 shows the vector components for these different coordinate systems.

Setting the Velocity Magnitude Normal to the Boundary

If you selected Magnitude, Normal to Boundary as the Velocity Specification Method in step 1 above, you will enter the magnitude of the velocity vector at the inflow boundary (the Velocity Magnitude). If you are modeling 2D axisymmetric swirl, you will also enter the Tangential-Component of Flow Direction.

Setting the Velocity Components

If you selected Components as the Velocity Specification Method in step 1 above, you will enter the components of the velocity vector at the inflow boundary as follows:

- If your geometry is 2D non-axisymmetric, or you chose in step 3 to use the Cartesian coordinate system, you will define the $X$, $Y$, and (in 3D) $Z$-Velocity.
Boundary Conditions

- If your geometry is 2D axisymmetric without swirl, you will set the radial and axial-velocity.
- If your model is 2D axisymmetric with swirl, you will set the axial, radial, and swirl-velocity, and (optionally) the swirl angular velocity, as described below.
- If you chose in step 3 to use a cylindrical coordinate system, you will set the radial, tangential, and axial-velocity, and (optionally) the angular velocity, as described below.
- If you chose in step 3 to use a local cylindrical coordinate system, you will set the radial, tangential, and axial-velocity, and (optionally) the angular velocity, as described below, and then specify the X, Y, and Z-component of axis direction and the X, Y, and Z-coordinate of axis origin.

! Remember that positive values for x, y, and z velocities indicate flow in the positive x, y, and z directions. If flow enters the domain in the negative x direction, for example, you will need to specify a negative value for the x velocity. The same holds true for the radial, tangential, and axial velocities. Positive radial velocities point radially out from the axis, positive axial velocities are in the direction of the axis vector, and positive tangential velocities are based on the right-hand rule using the positive axis.

Setting the Angular Velocity

If you chose Components as the Velocity Specification Method in step 1 above, and you are modeling axisymmetric swirl, you can specify the inlet swirl angular velocity $\Omega$ in addition to the swirl-velocity. Similarly, if you chose Components as the Velocity Specification Method and you chose in step 3 to use a cylindrical or local cylindrical coordinate system, you can specify the inlet angular velocity $\Omega$ in addition to the tangential-velocity.

If you specify $\Omega$, $v_\theta$ is computed for each cell as $\Omega r$, where $r$ is the radial coordinate in the coordinate system defined by the rotation axis and origin. If you specify both the swirl-velocity and the swirl angular
Velocity, or the Tangential Velocity and the Angular Velocity, FLUENT will add \( v_\theta \) and \( \Omega r \) to get the swirl or tangential velocity at each cell.

**Defining the Temperature**

For calculations in which the energy equation is being solved, you will set the static temperature of the flow at the velocity inlet boundary in the Temperature field.

**Defining Outflow Gauge Pressure**

If you are using one of the coupled solvers, you can specify an Outflow Gauge Pressure for a velocity inlet boundary. If the flow exits the domain at any face on the boundary, that face will be treated as a pressure outlet with the pressure prescribed in the Outflow Gauge Pressure field.

**Defining Turbulence Parameters**

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in Section 6.2.2. Turbulence modeling in general is described in Chapter 10.

**Defining Radiation Parameters**

If you are using the P-1 radiation model, the DTRM, the DO model, or the surface-to-surface model, you will set the Internal Emissivity and (optionally) Black Body Temperature. See Section 11.3.16 for details. (The Rosseland radiation model does not require any boundary condition inputs.)

**Defining Species Mass Fractions**

If you are modeling species transport, you will set the species mass fractions under Species Mass Fractions. For details, see Section 13.1.5.
Boundary Conditions

Defining Non-Premixed Combustion Parameters

If you are using the non-premixed or partially premixed combustion model, you will set the Mean Mixture Fraction and Mixture Fraction Variance (and the Secondary Mean Mixture Fraction and Secondary Mixture Fraction Variance, if you are using two mixture fractions), as described in Section 14.3.3.

Defining Premixed Combustion Boundary Conditions

If you are using the premixed or partially premixed combustion model, you will set the Progress Variable, as described in Section 15.3.5.

Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the velocity inlet. See Section 19.10 for details.

Defining Multiphase Boundary Conditions

If you are using the VOF, mixture, or Eulerian model for multiphase flow, you will need to specify volume fractions for secondary phases and (for some models) additional parameters. See Section 20.6.14 for details.

6.4.2 Default Settings at Velocity Inlet Boundaries

Default settings (in SI) for velocity inlet boundary conditions are as follows:
6.4 Velocity Inlet Boundary Conditions

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature</td>
<td>300</td>
</tr>
<tr>
<td>Velocity Magnitude</td>
<td>0</td>
</tr>
<tr>
<td>X-Component of Flow Direction</td>
<td>1</td>
</tr>
<tr>
<td>Y-Component of Flow Direction</td>
<td>0</td>
</tr>
<tr>
<td>Z-Component of Flow Direction</td>
<td>0</td>
</tr>
<tr>
<td>X-Velocity</td>
<td>0</td>
</tr>
<tr>
<td>Y-Velocity</td>
<td>0</td>
</tr>
<tr>
<td>Z-Velocity</td>
<td>0</td>
</tr>
<tr>
<td>Turb. Kinetic Energy</td>
<td>1</td>
</tr>
<tr>
<td>Turb. Dissipation Rate</td>
<td>1</td>
</tr>
<tr>
<td>Outflow Gauge Pressure</td>
<td>0</td>
</tr>
</tbody>
</table>

6.4.3 Calculation Procedure at Velocity Inlet Boundaries

FLUENT uses your boundary condition inputs at velocity inlets to compute the mass flow into the domain through the inlet and to compute the fluxes of momentum, energy, and species through the inlet. This section describes these calculations for the case of flow entering the domain through the velocity inlet boundary and for the less common case of flow exiting the domain through the velocity inlet boundary.

Treatment of Velocity Inlet Conditions at Flow Inlets

When your velocity inlet boundary condition defines flow entering the physical domain of the model, FLUENT uses both the velocity components and the scalar quantities that you defined as boundary conditions to compute the inlet mass flow rate, momentum fluxes, and fluxes of energy and chemical species.

The mass flow rate entering a fluid cell adjacent to a velocity inlet boundary is computed as

\[ \dot{m} = \int \rho \vec{v} \cdot d\vec{A} \]  \hspace{1cm} (6.4-1)

Note that only the velocity component normal to the control volume face contributes to the inlet mass flow rate.
**Boundary Conditions**

**Treatment of Velocity Inlet Conditions at Flow Exits**

Sometimes a velocity inlet boundary is used where flow exits the physical domain. This approach might be used, for example, when the flow rate through one exit of the domain is known or is to be imposed on the model.

In such cases you must ensure that overall continuity is maintained in the domain.

In the segregated solver, when flow exits the domain through a velocity inlet boundary FLUENT uses the boundary condition value for the velocity component normal to the exit flow area. It does not use any other boundary conditions that you have input. Instead, all flow conditions except the normal velocity component are assumed to be those of the upstream cell.

In the coupled solvers, if the flow exits the domain at any face on the boundary, that face will be treated as a pressure outlet with the pressure prescribed in the Outflow Gauge Pressure field.

**Density Calculation**

Density at the inlet plane is either constant or calculated as a function of temperature, pressure, and/or species mass fractions, where the mass fractions are the values you entered as an inlet condition.
6.5 Mass Flow Inlet Boundary Conditions

Mass flow boundary conditions can be used in FLUENT to provide a prescribed mass flow rate or mass flux distribution at an inlet. Physically, specifying the mass flux permits the total pressure to vary in response to the interior solution. This is in contrast to the pressure inlet boundary condition (see Section 6.3), where the total pressure is fixed while the mass flux varies.

A mass flow inlet is often used when it is more important to match a prescribed mass flow rate than to match the total pressure of the inflow stream. An example is the case of a small cooling jet that is bled into the main flow at a fixed mass flow rate, while the velocity of the main flow is governed primarily by a (different) pressure inlet/outlet boundary condition pair.

The adjustment of inlet total pressure might result in a slower convergence, so if both the pressure inlet boundary condition and the mass flow inlet boundary condition are acceptable choices, you should choose the former.

It is not necessary to use mass flow inlets in incompressible flows because when density is constant, velocity inlet boundary conditions will fix the mass flow.

You can find the following information about mass flow inlet boundary conditions in this section:

- Section 6.5.1: Inputs at Mass Flow Inlet Boundaries
- Section 6.5.2: Default Settings at Mass Flow Inlet Boundaries
- Section 6.5.3: Calculation Procedure at Mass Flow Inlet Boundaries

For an overview of flow boundaries, see Section 6.2.
6.5.1 Inputs at Mass Flow Inlet Boundaries

Summary

You will enter the following information for a mass flow inlet boundary:

- Mass flow rate, mass flux, or (primarily for the mixing plane model) mass flux with average mass flux
- Total (stagnation) temperature
- Static pressure
- Flow direction
- Turbulence parameters (for turbulent calculations)
- Radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- Chemical species mass fractions (for species calculations)
- Mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- Progress variable (for premixed or partially premixed combustion calculations)
- Discrete phase boundary conditions (for discrete phase calculations)
- Multiphase boundary conditions (for general multiphase calculations)

All values are entered in the Mass-Flow Inlet panel (Figure 6.5.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).
6.5 Mass Flow Inlet Boundary Conditions

Figure 6.5.1: The Mass-Flow Inlet Panel

<table>
<thead>
<tr>
<th>Zone Name</th>
<th>mass-flow-inlet-6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass Flow Specification Method</td>
<td>Mass Flow Rate</td>
</tr>
<tr>
<td>Mass Flow Rate (kg/s)</td>
<td>1</td>
</tr>
<tr>
<td>Total Temperature (K)</td>
<td>300</td>
</tr>
<tr>
<td>Supersonic/Initial Gauge Pressure (pascal)</td>
<td>0 constant</td>
</tr>
<tr>
<td>Direction Specification Method</td>
<td>Direction Vector</td>
</tr>
<tr>
<td>Coordinate System</td>
<td>Cartesian (X, Y, Z)</td>
</tr>
<tr>
<td>X-Component of Flow Direction</td>
<td>constant</td>
</tr>
<tr>
<td>Y-Component of Flow Direction</td>
<td>constant</td>
</tr>
<tr>
<td>Z-Component of Flow Direction</td>
<td>constant</td>
</tr>
<tr>
<td>Turbulence Specification Method</td>
<td>K and Epsilon</td>
</tr>
<tr>
<td>Turb. Kinetic Energy (m^2/s^2)</td>
<td>constant</td>
</tr>
<tr>
<td>Turb. Dissipation Rate (m^2/s^3)</td>
<td>constant</td>
</tr>
</tbody>
</table>

© Fluent Inc. November 28, 2001
**Boundary Conditions**

**Defining the Mass Flow Rate or Mass Flux**

You can specify the mass flow rate through the inlet zone and have FLUENT convert this value to mass flux, or specify the mass flux directly. For cases where the mass flux varies across the boundary, you can also specify an average mass flux; see below for more information about this specification method.

If you set the mass flow rate, it will be converted internally to a uniform mass flux over the zone by dividing the flow rate by the area of the zone. You can define the mass flux (but not the mass flow rate) using a boundary profile or a user-defined function.

The inputs for mass flow rate or flux are as follows:

1. Choose the method you will use to specify the mass flow by selecting **Mass Flow Rate**, **Mass Flux**, or **Mass Flux with Average Mass Flux** in the **Mass Flow Specification Method** drop-down list.

2. If you selected **Mass Flow Rate** (the default), set the prescribed mass flow rate in the **Mass Flow-Rate** field.

! Note that for axisymmetric problems, this mass flow rate is the flow rate through the entire (2π-radian) domain, not through a 1-radian slice.

If you selected **Mass Flux**, set the prescribed mass flux in the **Mass Flux** field.

! Note that for axisymmetric problems, this mass flux is the flux through a 1-radian slice of the domain.

If you selected **Mass Flux with Average Mass Flux**, set the prescribed mass flux and average mass flux in the **Mass Flux** and **Average Mass Flux** fields.

! Note that for axisymmetric problems, this mass flux is the flux through a 1-radian slice of the domain.

**More About Mass Flux and Average Mass Flux**

As noted above, you can specify an average mass flux with the mass flux. If, for example, you specify a mass flux profile such that the average mass
6.5 Mass Flow Inlet Boundary Conditions

flux integrated over the zone area is 4.7, but you actually want to have a total mass flux of 5, you can keep the profile unchanged, and specify an average mass flux of 5. FLUENT will maintain the profile shape but adjust the values so that the resulting mass flux across the boundary is 5.

The mass flux with average mass flux specification method is also used by the mixing plane model described in Section 9.4. If the mass flow inlet boundary is going to represent one of the mixing planes, then you do not need to specify the mass flux or flow rate; you can keep the default Mass Flow-Rate of 1. When you create the mixing plane later on in the problem setup, FLUENT will automatically select the Mass Flux with Average Mass Flux method in the Mass-Flow Inlet panel and set the Average Mass Flux to the value obtained by integrating the mass flux profile for the upstream zone. This will ensure that mass is conserved between the upstream zone and the downstream (mass flow inlet) zone.

Define the Total Temperature

Enter the value for the total (stagnation) temperature of the inflow stream in the Total Temperature field in the Mass-Flow Inlet panel.

Define Static Pressure

The static pressure (termed the Supersonic/Initial Gauge Pressure) must be specified if the inlet flow is supersonic or if you plan to initialize the solution based on the pressure inlet boundary conditions. Solution initialization is discussed in Section 22.13.

The Supersonic/Initial Gauge Pressure is ignored by FLUENT whenever the flow is subsonic. If you choose to initialize the flow based on the mass flow inlet conditions, the Supersonic/Initial Gauge Pressure will be used in conjunction with the specified stagnation quantities to compute initial values according to isentropic relations.

Remember that the static pressure value you enter is relative to the operating pressure set in the Operating Conditions panel. Note the comments in Section 6.3.1 regarding hydrostatic pressure.
Boundary Conditions

Defining the Flow Direction

You can define the flow direction at a mass flow inlet explicitly, or you can define the flow to be normal to the boundary.

For moving zone problems calculated using the segregated solver, the flow direction will be absolute or relative to the grid velocity, depending on whether or not the Absolute velocity formulation is enabled in the Solver panel. For the coupled solvers, the flow direction will always be in the absolute frame.

The procedure for defining the flow direction is as follows, referring to Figure 6.5.1:

1. Choose which method you will use to specify the flow direction by selecting Direction Vector or Normal to Boundary in the Direction Specification Method drop-down list.

2. If you selected Normal to Boundary, there are no additional inputs for flow direction.

   Note that if you are modeling axisymmetric swirl, the flow direction will be normal to the boundary; i.e., there will be no swirl component at the boundary for axisymmetric swirl.

3. If you selected Direction Vector and your geometry is 2D, go to the next step. If your geometry is 3D, you will next choose the coordinate system in which you will define the flow direction components. Choose Cartesian (X, Y, Z), Cylindrical (Radial, Tangential, Axial), or Local Cylindrical (Radial, Tangential, Axial) in the Coordinate System drop-down list. See Section 6.3.1 for information about Cartesian, cylindrical, and local cylindrical coordinate systems.

4. If you selected Direction Vector, set the vector components as follows:

   - If your geometry is 2D non-axisymmetric, or you chose to use a 3D Cartesian coordinate system, enter the appropriate values for X, Y, and (in 3D) Z-Component of Flow Direction.
6.5 Mass Flow Inlet Boundary Conditions

- If your geometry is 2D axisymmetric, or you chose to use a 3D Cylindrical coordinate system, enter the appropriate values for Axial, Radial, and (if you are modeling swirl or using cylindrical coordinates) Tangential-Component of Flow Direction.

- If you chose to use a 3D Local Cylindrical coordinate system, enter the appropriate values for Axial, Radial, and Tangential-Component of Flow Direction, and then specify the X, Y, and Z-Component of Axis Direction and the X, Y, and Z-Coordinate of Axis Origin.

Figure 6.3.2 shows the vector components for these different coordinate systems.

**Defining Turbulence Parameters**

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in Section 6.2.2. Turbulence modeling is described in Chapter 10.

**Defining Radiation Parameters**

If you are using the P-1 radiation model, the DTRM, the DO model, or the surface-to-surface model, you will set the Internal Emissivity and (optionally) Black Body Temperature. See Section 11.3.16 for details. (The Rosseland radiation model does not require any boundary condition inputs.)

**Defining Species Mass Fractions**

If you are modeling species transport, you will set the species mass fractions under Species Mass Fractions. For details, see Section 13.1.5.

**Defining Non-Premixed Combustion Parameters**

If you are using the non-premixed or partially premixed combustion model, you will set the Mean Mixture Fraction and Mixture Fraction Variance (and the Secondary Mean Mixture Fraction and Secondary Mixture
Boundary Conditions

Fraction Variance, if you are using two mixture fractions), as described in Section 14.3.3.

Defining Premixed Combustion Boundary Conditions

If you are using the premixed or partially premixed combustion model, you will set the Progress Variable, as described in Section 15.3.5.

Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the mass flow inlet. See Section 19.10 for details.

Defining Multiphase Boundary Conditions

If you are using the VOF, mixture, or Eulerian model for multiphase flow, you will need to specify volume fractions for secondary phases and (for some models) additional parameters. See Section 20.6.14 for details.

6.5.2 Default Settings at Mass Flow Inlet Boundaries

Default settings (in SI) for mass flow inlet boundary conditions are as follows:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass Flow-Rate</td>
<td>1</td>
</tr>
<tr>
<td>Total Temperature</td>
<td>300</td>
</tr>
<tr>
<td>Supersonic/Initial Gauge Pressure</td>
<td>0</td>
</tr>
<tr>
<td>X-Component of Flow Direction</td>
<td>1</td>
</tr>
<tr>
<td>Y-Component of Flow Direction</td>
<td>0</td>
</tr>
<tr>
<td>Z-Component of Flow Direction</td>
<td>0</td>
</tr>
<tr>
<td>Turb. Kinetic Energy</td>
<td>1</td>
</tr>
<tr>
<td>Turb. Dissipation Rate</td>
<td>1</td>
</tr>
</tbody>
</table>

6.5.3 Calculation Procedure at Mass Flow Inlet Boundaries

When mass flow boundary conditions are used for an inlet zone, a velocity is computed for each face in that zone, and this velocity is used to compute the fluxes of all relevant solution variables into the domain.
With each iteration, the computed velocity is adjusted so that the correct mass flow value is maintained.

To compute this velocity, your inputs for mass flow rate, flow direction, static pressure, and total temperature are used.

There are two ways to specify the mass flow rate. The first is to specify the total mass flow rate, $\dot{m}$, for the inlet. The second is to specify the mass flux, $\rho v_n$ (mass flow rate per unit area). If a total mass flow rate is specified, FLUENT converts it internally to a uniform mass flux by dividing the mass flow rate by the total inlet area:

$$\rho v_n = \frac{\dot{m}}{A} \quad (6.5-1)$$

If the direct mass flux specification option is used, the mass flux can be varied over the boundary by using profile files or user-defined functions. If the average mass flux is also specified (either explicitly by you or automatically by FLUENT), it is used to correct the specified mass flux profile, as described earlier in this section.

Once the value of $\rho v_n$ at a given face has been determined, the density, $\rho$, at the face must be determined in order to find the normal velocity, $v_n$. The manner in which the density is obtained depends upon whether the fluid is modeled as an ideal gas or not. Each of these cases is examined below.

**Flow Calculations at Mass Flow Boundaries for Ideal Gases**

If the fluid is an ideal gas, the static temperature and static pressure are required to compute the density:

$$p = \rho RT \quad (6.5-2)$$

If the inlet is supersonic, the static pressure used is the value that has been set as a boundary condition. If the inlet is subsonic, the static pressure is extrapolated from the cells inside the inlet face.
Boundary Conditions

The static temperature at the inlet is computed from the total enthalpy, which is determined from the total temperature that has been set as a boundary condition. The total enthalpy is given by

\[ h_0(T_0) = h(T) + \frac{1}{2}v^2 \] (6.5-3)

where the velocity is related to the mass flow rate given by Equation 6.5-1. Using Equation 6.5-2 to relate density to the (known) static pressure and (unknown) temperature, Equation 6.5-3 can be solved to obtain the static temperature.

Flow Calculations at Mass Flow Boundaries for Incompressible Flows

When you are modeling incompressible flows, the static temperature is nearly the same as the total temperature. The density at the inlet is either constant or readily computed as a function of the temperature and (optionally) the species mass fractions. The velocity is then computed using Equation 6.5-1.

Flux Calculations at Mass Flow Boundaries

To compute the fluxes of all variables at the inlet, the flux velocity, \( v_n \), is used along with the inlet value of the variable in question. For example, the flux of mass is \( \rho v_n \), and the flux of turbulence kinetic energy is \( \rho k v_n \). These fluxes are used as boundary conditions for the corresponding conservation equations during the course of the solution.
6.6 Inlet Vent Boundary Conditions

Inlet vent boundary conditions are used to model an inlet vent with a specified loss coefficient, flow direction, and ambient (inlet) pressure and temperature.

6.6.1 Inputs at Inlet Vent Boundaries

You will enter the following information for an inlet vent boundary:

- Total (stagnation) pressure
- Total (stagnation) temperature
- Flow direction
- Static pressure
- Turbulence parameters (for turbulent calculations)
- Radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- Chemical species mass fractions (for species calculations)
- Mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- Progress variable (for premixed or partially premixed combustion calculations)
- Discrete phase boundary conditions (for discrete phase calculations)
- Multiphase boundary conditions (for general multiphase calculations)
- Loss coefficient
Boundary Conditions

All values are entered in the Inlet Vent panel (Figure 6.6.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4). The first 11 items listed above are specified in the same way that they are specified at pressure inlet boundaries. See Section 6.3.1 for details. Specification of the loss coefficient is described here.

Figure 6.6.1: The Inlet Vent Panel
6.7 Intake Fan Boundary Conditions

**Specifying the Loss Coefficient**

An inlet vent is considered to be infinitely thin, and the pressure drop through the vent is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient that you supply. That is, the pressure drop, \( \Delta p \), varies with the normal component of velocity through the vent, \( v \), as follows:

\[
\Delta p = k_L \frac{1}{2} \rho v^2 \tag{6.6-1}
\]

where \( \rho \) is the fluid density, and \( k_L \) is the nondimensional loss coefficient.

\( \Delta p \) is the pressure drop in the direction of the flow; therefore the vent will appear as a resistance even in the case of backflow.

You can define the Loss Coefficient across the vent as a constant, polynomial, piecewise-linear, or piecewise-polynomial function of the normal velocity. The panels for defining these functions are the same as those used for defining temperature-dependent properties. See Section 7.1.3 for details.

6.7 Intake Fan Boundary Conditions

Intake fan boundary conditions are used to model an external intake fan with a specified pressure jump, flow direction, and ambient (intake) pressure and temperature.

6.7.1 Inputs at Intake Fan Boundaries

You will enter the following information for an intake fan boundary:

- Total (stagnation) pressure
- Total (stagnation) temperature
- Flow direction
- Static pressure
Boundary Conditions

- Turbulence parameters (for turbulent calculations)
- Radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- Chemical species mass fractions (for species calculations)
- Mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- Progress variable (for premixed or partially premixed combustion calculations)
- Discrete phase boundary conditions (for discrete phase calculations)
- Multiphase boundary conditions (for general multiphase calculations)
- Pressure jump

All values are entered in the Intake Fan panel (shown in Figure 6.7.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).

The first 11 items listed above are specified in the same way that they are specified at pressure inlet boundaries. See Section 6.3.1 for details. Specification of the pressure jump is described here.

Specifying the Pressure Jump

An intake fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the velocity through the fan. In the case of reversed flow, the fan is treated like an outlet vent with a loss coefficient of unity.

You can define the Pressure-Jump across the fan as a constant, polynomial, piecewise-linear, or piecewise-polynomial function of the normal velocity. The panels for defining these functions are the same as those used for defining temperature-dependent properties. See Section 7.1.3 for details.
6.7 Intake Fan Boundary Conditions

![Intake Fan Panel](image)

Figure 6.7.1: The Intake Fan Panel
6.8 Pressure Outlet Boundary Conditions

Pressure outlet boundary conditions require the specification of a static (gauge) pressure at the outlet boundary. The value of static pressure specified is used only while the flow is subsonic. Should the flow become locally supersonic, the specified pressure is no longer used; pressure will be extrapolated from the flow in the interior. All other flow quantities are extrapolated from the interior.

A set of "backflow" conditions is also specified to be used if the flow reverses direction at the pressure outlet boundary during the solution process. Convergence difficulties will be minimized if you specify realistic values for the backflow quantities.

FLUENT also provides an option to use a radial equilibrium outlet boundary condition. See Section 6.8.1 for details.

You can find the following information about pressure outlet boundary conditions in this section:

- Section 6.8.1: Inputs at Pressure Outlet Boundaries
- Section 6.8.2: Default Settings at Pressure Outlet Boundaries
- Section 6.8.3: Calculation Procedure at Pressure Outlet Boundaries

For an overview of flow boundaries, see Section 6.2.

6.8.1 Inputs at Pressure Outlet Boundaries

Summary

You will enter the following information for a pressure outlet boundary:

- Static pressure
- Backflow conditions
  - Total (stagnation) temperature (for energy calculations)
6.8 Pressure Outlet Boundary Conditions

- Turbulence parameters (for turbulent calculations)
- Chemical species mass fractions (for species calculations)
- Mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- Progress variable (for premixed or partially premixed combustion calculations)
- Multiphase boundary conditions (for general multiphase calculations)

- Radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)

- Discrete phase boundary conditions (for discrete phase calculations)

All values are entered in the Pressure Outlet panel (Figure 6.8.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).

Figure 6.8.1: The Pressure Outlet Panel
**Boundary Conditions**

### Defining Static Pressure

To set the static pressure at the pressure outlet boundary, enter the appropriate value for **Gauge Pressure** in the **Pressure Outlet** panel. This value will be used for subsonic flow only. Should the flow become locally supersonic, the pressure will be extrapolated from the upstream conditions.

Remember that the static pressure value you enter is relative to the operating pressure set in the **Operating Conditions** panel. Note the comments in Section 6.3.1 regarding hydrostatic pressure.

**FLUENT** also provides an option to use a radial equilibrium outlet boundary condition. To enable this option, turn on **Radial Equilibrium Pressure Distribution**. When this feature is active, the specified gauge pressure applies only to the position of minimum radius (relative to the axis of rotation) at the boundary. The static pressure on the rest of the zone is calculated from the assumption that radial velocity is negligible, so that the pressure gradient is given by

\[ \frac{\partial p}{\partial r} = \frac{\rho v_y^2}{r} \]  

(6.8-1)

where \( r \) is the distance from the axis of rotation and \( v_y \) is the tangential velocity. Note that this boundary condition can be used even if the rotational velocity is zero. For example, it could be applied to the calculation of the flow through an annulus containing guide vanes.

Note that the radial equilibrium outlet condition is available only for 3D and axisymmetric swirl calculations.

### Defining Backflow Conditions

Backflow properties consistent with the models you are using will appear in the **Pressure Outlet** panel. The specified values will be used only if flow is pulled in through the outlet.

- The **Backflow Total Temperature** should be set for problems involving energy calculation.
For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in Section 6.2.2. Turbulence modeling in general is described in Chapter 10.

If you are modeling species transport, you will set the backflow species mass fractions under Species Mass Fractions. For details, see Section 13.1.5.

If you are modeling combustion using the non-premixed or partially premixed combustion model, you will set the backflow mixture fraction and variance values. See Section 14.3.3 for details.

If you are modeling combustion using the premixed or partially premixed combustion model, you will set the backflow Progress Variable value. See Section 15.3.5 for details.

If you are using the VOF, mixture, or Eulerian model for multiphase flow, you will need to specify volume fractions for secondary phases and (for some models) additional parameters. See Section 20.6.14 for details.

If backflow occurs, the pressure you specified as the Gauge Pressure will be used as total pressure, so you need not specify a backflow pressure value explicitly. The flow direction in this case will be normal to the boundary.

If the cell zone adjacent to the pressure outlet is moving (i.e., if you are using a rotating reference frame, multiple reference frames, mixing planes, or sliding meshes) and you are using the segregated solver, the velocity in the dynamic contribution to total pressure (see Equation 6.3-3) will be absolute or relative to the motion of the cell zone, depending on whether or not the Absolute velocity formulation is enabled in the Solver panel. For the coupled solvers, the velocity in Equation 6.3-3 (or the Mach number in Equation 6.3-4) is always in the absolute frame.

Even if no backflow is expected in the converged solution, you should al-
ways set realistic values to minimize convergence difficulties in the event that backflow does occur during the calculation.

**Defining Radiation Parameters**

If you are using the P-1 radiation model, the DTRM, the DO model, or the surface-to-surface model, you will set the *Internal Emissivity* and (optionally) *Black Body Temperature*. See Section 11.3.16 for details. (The Rosseland radiation model does not require any boundary condition inputs.)

**Defining Discrete Phase Boundary Conditions**

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the pressure outlet. See Section 19.10 for details.

### 6.8.2 Default Settings at Pressure Outlet Boundaries

Default settings (in SI) for pressure outlet boundary conditions are as follows:

<table>
<thead>
<tr>
<th>Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gauge Pressure</td>
<td>0</td>
</tr>
<tr>
<td>Backflow Total Temperature</td>
<td>300</td>
</tr>
<tr>
<td>Backflow Turb. Kinetic Energy</td>
<td>1</td>
</tr>
<tr>
<td>Backflow Turb. Dissipation Rate</td>
<td>1</td>
</tr>
</tbody>
</table>

### 6.8.3 Calculation Procedure at Pressure Outlet Boundaries

At pressure outlets, FLUENT uses the boundary condition pressure you input as the static pressure of the fluid at the outlet plane, $p_s$, and extrapolates all other conditions from the interior of the domain.
6.9 Pressure Far-Field Boundary Conditions

Pressure far-field conditions are used in FLUENT to model a free-stream condition at infinity, with free-stream Mach number and static conditions being specified. The pressure far-field boundary condition is often called a characteristic boundary condition, since it uses characteristic information (Riemann invariants) to determine the flow variables at the boundaries.

This boundary condition is applicable only when the density is calculated using the ideal-gas law (see Section 7.2). Using it for other flows is not permitted. To effectively approximate true infinite-extent conditions, you must place the far-field boundary far enough from the object of interest. For example, in lifting airfoil calculations, it is not uncommon for the far-field boundary to be a circle with a radius of 20 chord lengths.

You can find the following information about pressure far-field boundary conditions in this section:

- Section 6.9.1: Inputs at Pressure Far-Field Boundaries
- Section 6.9.2: Default Settings at Pressure Far-Field Boundaries
- Section 6.9.3: Calculation Procedure at Pressure Far-Field Boundaries

For an overview of flow boundaries, see Section 6.2.

6.9.1 Inputs at Pressure Far-Field Boundaries

Summary

You will enter the following information for a pressure far-field boundary:

- Static pressure
- Mach number
- Temperature
Boundary Conditions

- Flow direction
- Turbulence parameters (for turbulent calculations)
- Radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- Chemical species mass fractions (for species calculations)
- Discrete phase boundary conditions (for discrete phase calculations)

All values are entered in the Pressure Far-Field panel (Figure 6.9.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).

Defining Static Pressure, Mach Number, and Static Temperature

To set the static pressure and temperature at the far-field boundary, enter the appropriate values for Gauge Pressure and Temperature in the Pressure Far-Field panel. You will also set the Mach Number there. The Mach number can be subsonic, sonic, or supersonic.

Defining the Flow Direction

You can define the flow direction at a pressure far-field boundary by setting the components of the direction vector. If your geometry is 2D non-axisymmetric or 3D, enter the appropriate values for X, Y, and (in 3D) Z-Component of Flow Direction in the Pressure Far-Field panel. If your geometry is 2D axisymmetric, enter the appropriate values for Axial, Radial, and (if you are modeling axisymmetric swirl) Tangential-Component of Flow Direction.

Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in Section 6.2.2. Turbulence modeling is described in Chapter 10.
Figure 6.9.1: The Pressure Far-Field Panel
**Boundary Conditions**

**Defining Radiation Parameters**

If you are using the P-1 radiation model, the DTRM, the DO model, or the surface-to-surface model, you will set the *Internal Emissivity* and (optionally) *Black Body Temperature*. See Section 11.3.16 for details.

**Defining Species Transport Parameters**

If you are modeling species transport, you will set the species mass fractions under *Species Mass Fractions*. See Section 13.1.5 for details.

**Defining Discrete Phase Boundary Conditions**

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the pressure far-field boundary. See Section 19.10 for details.

**6.9.2 Default Settings at Pressure Far-Field Boundaries**

Default settings (in SI) for pressure far-field boundary conditions are as follows:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gauge Pressure</td>
<td>0</td>
</tr>
<tr>
<td>Mach Number</td>
<td>0.6</td>
</tr>
<tr>
<td>Temperature</td>
<td>300</td>
</tr>
<tr>
<td>X-Component of Flow Direction</td>
<td>1</td>
</tr>
<tr>
<td>Y-Component of Flow Direction</td>
<td>0</td>
</tr>
<tr>
<td>Z-Component of Flow Direction</td>
<td>0</td>
</tr>
<tr>
<td>Turb. Kinetic Energy</td>
<td>1</td>
</tr>
<tr>
<td>Turb. Dissipation Rate</td>
<td>1</td>
</tr>
</tbody>
</table>

**6.9.3 Calculation Procedure at Pressure Far-Field Boundaries**

The pressure far-field boundary condition is a non-reflecting boundary condition based on the introduction of Riemann invariants (i.e., characteristic variables) for a one-dimensional flow normal to the boundary. For flow that is subsonic there are two Riemann invariants, corresponding to incoming and outgoing waves:
6.9 Pressure Far-Field Boundary Conditions

\[ R_\infty = v_{n\infty} - \frac{2c_{\infty}}{\gamma - 1} \]  \hspace{1cm} (6.9-1)

\[ R_i = v_{n_i} + \frac{2c_i}{\gamma - 1} \]  \hspace{1cm} (6.9-2)

where \( v_n \) is the velocity magnitude normal to the boundary, \( c \) is the local speed of sound and \( \gamma \) is the ratio of specific heats (ideal gas). The subscript \( \infty \) refers to conditions being applied at infinity (the boundary conditions), and the subscript \( i \) refers to conditions in the interior of the domain (i.e., in the cell adjacent to the boundary face). These two invariants can be added and subtracted to give the following two equations:

\[ v_n = \frac{1}{2}(R_i + R_\infty) \]  \hspace{1cm} (6.9-3)

\[ c = \frac{\gamma - 1}{4}(R_i - R_\infty) \]  \hspace{1cm} (6.9-4)

where \( v_n \) and \( c \) become the values of normal velocity and sound speed applied on the boundary. At a face through which flow exits, the tangential velocity components and entropy are extrapolated from the interior; at an inflow face, these are specified as having free-stream values. Using the values for \( v_n \), \( c \), tangential velocity components, and entropy the values of density, velocity, temperature, and pressure at the boundary face can be calculated.
6.10 Outflow Boundary Conditions

Outflow boundary conditions in FLUENT are used to model flow exits where the details of the flow velocity and pressure are not known prior to solution of the flow problem. You do not define any conditions at outflow boundaries (unless you are modeling radiative heat transfer, a discrete phase of particles, or split mass flow): FLUENT extrapolates the required information from the interior. It is important, however, to understand the limitations of this boundary type.

Note that outflow boundaries cannot be used in the following cases:

- if a problem includes pressure inlet boundaries; use pressure outlet boundary conditions (see Section 6.8) instead
- if you are modeling compressible flow
- if you are modeling unsteady flows with varying density, even if the flow is incompressible

You can find the following information about outflow boundary conditions in this section:

- Section 6.10.1: FLUENT’s Treatment at Outflow Boundaries
- Section 6.10.2: Using Outflow Boundaries
- Section 6.10.3: Mass Flow Split Boundary Conditions
- Section 6.10.4: Other Inputs at Outflow Boundaries

For an overview of flow boundaries, see Section 6.2.

6.10.1 FLUENT’s Treatment at Outflow Boundaries

The boundary conditions used by FLUENT at outflow boundaries are as follows:

- A zero diffusion flux for all flow variables
• An overall mass balance correction

The zero diffusion flux condition applied at outflow cells means that the conditions of the outflow plane are extrapolated from within the domain and have no impact on the upstream flow. The extrapolation procedure used by FLUENT updates the outflow velocity and pressure in a manner that is consistent with a fully-developed flow assumption, as noted below, when there is no area change at the outflow boundary.

The zero diffusion flux condition applied by FLUENT at outflow boundaries is approached physically in fully-developed flows. Fully-developed flows are flows in which the flow velocity profile (and/or profiles of other properties such as temperature) is unchanging in the flow direction.

It is important to note that gradients in the cross-stream direction may exist at an outflow boundary. Only the diffusion fluxes in the direction normal to the exit plane are assumed to be zero.

6.10.2 Using Outflow Boundaries

As noted in Section 6.10.1, the outflow boundary condition is obeyed in fully-developed flows where the diffusion flux for all flow variables in the exit direction are zero. However, you may also define outflow boundaries at physical boundaries where the flow is not fully developed—and you can do so with confidence if the assumption of a zero diffusion flux at the exit is expected to have a small impact on your flow solution. The appropriate placement of an outflow boundary is described by example below.

• Outflow boundaries where normal gradients are negligible: Figure 6.10.1 shows a simple two-dimensional flow problem and several possible outflow boundary location choices. Location C shows the outflow boundary located upstream of the plenum exit but in a region of the duct where the flow is fully-developed. At this location, the outflow boundary condition is exactly obeyed.

• Ill-posed outflow boundaries: Location B in Figure 6.10.1 shows the outflow boundary near the reattachment point of the recirculation in the wake of the backward-facing step. This choice of outflow
boundary condition is ill-posed as the gradients normal to the exit plane are quite large at this point and can be expected to have a significant impact on the flow field upstream. Because the outflow boundary condition ignores these axial gradients in the flow, location B is a poor choice for an outflow boundary. The exit location should be moved downstream from the reattachment point.

Figure 6.10.1 shows a second ill-posed outflow boundary at location A. Here, the outflow is located where flow is pulled into the FLUENT domain through the outflow boundary. In situations like this the FLUENT calculation typically does not converge and the results of the calculation have no validity. This is because when flow is pulled into the domain through an outflow, the mass flow rate through the domain is “floating” or undefined. In addition, when flow enters the domain through an outflow boundary, the scalar properties of the flow are not defined. For example, the temperature of the flow pulled in through the outflow is not defined. (FLUENT chooses the temperature using the temperature of the fluid adjacent to the outflow, inside the domain.) Thus you
should view all calculations that involve flow entering the domain through an outflow boundary with skepticism. For such calculations, pressure outlet boundary conditions (see Section 6.8) are recommended.

Note that convergence may be affected if there is recirculation through the outflow boundary at any point during the calculation, even if the final solution is not expected to have any flow reentering the domain. This is particularly true of turbulent flow simulations.

### 6.10.3 Mass Flow Split Boundary Conditions

In FLUENT, it is possible to use multiple outflow boundaries and specify the fractional flow rate through each boundary. In the Outflow panel, set the Flow Rate Weighting to indicate what portion of the outflow is through the boundary.

![Figure 6.10.2: The Outflow Panel](image)

The Flow Rate Weighting is a weighting factor:

$$\frac{\text{percentage flow through boundary}}{= \frac{\text{Flow Rate Weighting specified on boundary}}{\text{sum of all Flow Rate Weightings}}} \quad (6.10-1)$$

By default, the Flow Rate Weighting for all outflow boundaries is set to 1. If the flow is divided equally among all of your outflow boundaries (or
Boundary Conditions

if you have just one outflow boundary, you need not change the settings from the default; FLUENT will scale the flow rate fractions to obtain equal fractions through all outflow boundaries. Thus, if you have two outflow boundaries and you want half of the flow to exit through each one, no inputs are required from you. If, however, you want 75% of the flow to exit through one, and 25% through the other, you will need to explicitly specify both Flow Rate Weightings, i.e., 0.75 for one boundary and 0.25 for the other.

If you specify a Flow Rate Weighting of 0.75 at the first exit and leave the default Flow Rate Weighting (1.0) at the second exit, then the flow through each boundary will be

\[
\text{Boundary 1} = \frac{0.75}{0.75 + 1.0} = 0.429 \text{ or } 42.9\%
\]

\[
\text{Boundary 2} = \frac{1.0}{0.75 + 1.0} = 0.571 \text{ or } 57.1\%
\]

6.10.4 Other Inputs at Outflow Boundaries

Radiation Inputs at Outflow Boundaries

In general, there are no boundary conditions for you to set at an outflow boundary. If, however, you are using the P-1 radiation model, the DTRM, the DO model, or the surface-to-surface model, you will set the Internal Emissivity and (optionally) Black Body Temperature in the Outflow panel. These parameters are described in Section 11.3.16. The default value for Internal Emissivity is 1 and the default value for Black Body Temperature is 300.

Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the outflow boundary. See Section 19.10 for details.
6.11 Outlet Vent Boundary Conditions

Outlet vent boundary conditions are used to model an outlet vent with a specified loss coefficient and ambient (discharge) pressure and temperature.

6.11.1 Inputs at Outlet Vent Boundaries

You will enter the following information for an outlet vent boundary:

- Static pressure
- Backflow conditions
  - Total (stagnation) temperature (for energy calculations)
  - Turbulence parameters (for turbulent calculations)
  - Chemical species mass fractions (for species calculations)
  - Mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
  - Progress variable (for premixed or partially premixed combustion calculations)
  - Multiphase boundary conditions (for general multiphase calculations)
- Radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- Discrete phase boundary conditions (for discrete phase calculations)
- Loss coefficient

All values are entered in the Outlet Vent panel (Figure 6.11.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4). The first 4 items listed above are specified in the same way that they are specified at pressure outlet boundaries. See Section 6.8.1 for details. Specification of the loss coefficient is described here.
Boundary Conditions

Figure 6.11.1: The Outlet Vent Panel
6.12 Exhaust Fan Boundary Conditions

**Specifying the Loss Coefficient**

An outlet vent is considered to be infinitely thin, and the pressure drop through the vent is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient which you supply. That is, the pressure drop, $\Delta p$, varies with the normal component of velocity through the vent, $v$, as follows:

$$\Delta p = k_L \frac{1}{2} \rho v^2$$  \hspace{1cm} (6.11-1)

where $\rho$ is the fluid density, and $k_L$ is the nondimensional loss coefficient.

$\Delta p$ is the pressure drop in the direction of the flow; therefore the vent will appear as a resistance even in the case of backflow.

You can define a constant, polynomial, piecewise-linear, or piecewise-polynomial function for the Loss Coefficient across the vent. The panels for defining these functions are the same as those used for defining temperature-dependent properties. See Section 7.1.3 for details.

6.12 Exhaust Fan Boundary Conditions

Exhaust fan boundary conditions are used to model an external exhaust fan with a specified pressure jump and ambient (discharge) pressure.

6.12.1 Inputs at Exhaust Fan Boundaries

You will enter the following information for an exhaust fan boundary:

- Static pressure
- Backflow conditions
  - Total (stagnation) temperature (for energy calculations)
  - Turbulence parameters (for turbulent calculations)
  - Chemical species mass fractions (for species calculations)
  - Mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
Boundary Conditions

- Progress variable (for premixed or partially premixed combustion calculations)
- Multiphase boundary conditions (for general multiphase calculations)

- Radiation parameters (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- Discrete phase boundary conditions (for discrete phase calculations)
- Pressure jump

All values are entered in the Exhaust Fan panel (Figure 6.12.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).

The first 4 items listed above are specified in the same way that they are specified at pressure outlet boundaries. See Section 6.8.1 for details. Specification of the pressure jump is described here.

Specifying the Pressure Jump

An exhaust fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the local fluid velocity normal to the fan. You can define a constant, polynomial, piecewise-linear, or piecewise-polynomial function for the Pressure-Jump across the fan. The panels for defining these functions are the same as those used for defining temperature-dependent properties. See Section 7.1.3 for details.

You must be careful to model the exhaust fan so that a pressure rise occurs for forward flow through the fan. In the case of reversed flow, the fan is treated like an inlet vent with a loss coefficient of unity.
6.12 Exhaust Fan Boundary Conditions

![Exhaust Fan Panel](Image)

**Figure 6.12.1: The Exhaust Fan Panel**

<table>
<thead>
<tr>
<th>Zone Name</th>
<th>exhaust-fan-1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gauge Pressure (pascal)</td>
<td>0</td>
</tr>
<tr>
<td>Radial Equilibrium Pressure Distribution</td>
<td></td>
</tr>
<tr>
<td>Turbulence Specification Method</td>
<td>K and Epsilon</td>
</tr>
<tr>
<td>Backflow Turb. Kinetic Energy (m^2/s^2)</td>
<td>6e-05</td>
</tr>
<tr>
<td>Backflow Turb. Dissipation Rate (m^2/s^3)</td>
<td>3.29e-08</td>
</tr>
<tr>
<td>Pressure Jump (pascal)</td>
<td>polynomial</td>
</tr>
</tbody>
</table>

© Fluent Inc. November 28, 2001 6-77
**6.13 Wall Boundary Conditions**

Wall boundary conditions are used to bound fluid and solid regions. In viscous flows, the no-slip boundary condition is enforced at walls by default, but you can specify a tangential velocity component in terms of the translational or rotational motion of the wall boundary, or model a “slip” wall by specifying shear. (You can also model a slip wall with zero shear using the symmetry boundary type, but using a symmetry boundary will apply symmetry conditions for all equations. See Section 6.14 for details.)

The shear stress and heat transfer between the fluid and wall are computed based on the flow details in the local flow field.

You can find the following information about wall boundary conditions in this section:

- Section 6.13.1: Inputs at Wall Boundaries
- Section 6.13.2: Default Settings at Wall Boundaries
- Section 6.13.3: Shear-Stress Calculation Procedure at Wall Boundaries
- Section 6.13.4: Heat Transfer Calculation Procedure at Wall Boundaries

**6.13.1 Inputs at Wall Boundaries**

**Summary**

You will enter the following information for a wall boundary:

- Thermal boundary conditions (for heat transfer calculations)
- Wall motion conditions (for moving or rotating walls)
- Shear conditions (for slip walls, optional)
- Wall roughness (for turbulent flows, optional)
6.13 Wall Boundary Conditions

- Species boundary conditions (for species calculations)
- Chemical reaction boundary conditions (for surface reactions)
- Radiation boundary conditions (for calculations using the P-1 model, the DTRM, the DO model, or the surface-to-surface model)
- Discrete phase boundary conditions (for discrete phase calculations)
- Multiphase boundary conditions (for VOF calculations, optional)

Defining Thermal Boundary Conditions at Walls

When you are solving the energy equation, you need to define thermal boundary conditions at wall boundaries. Five types of thermal conditions are available:

- Fixed heat flux
- Fixed temperature
- Convective heat transfer
- External radiation heat transfer
- Combined external radiation and convection heat transfer

If the wall zone is a “two-sided wall” (a wall that forms the interface between two regions, such as the fluid/solid interface for a conjugate heat transfer problem) a subset of these thermal conditions will be available, but you will also be able to choose whether or not the two sides of the wall are “coupled”. See below for details.

The inputs for each type of thermal condition are described below. If the wall has a non-zero thickness, you should also set parameters for calculating thin-wall thermal resistance and heat generation in the wall, as described below.

You can model conduction within boundary walls and internal (i.e., two-sided) walls of your model. This type of conduction, called shell
**Boundary Conditions**

Conduction, allows you to more conveniently model heat conduction on walls where the wall thickness is small with respect to the overall geometry (e.g., finned heat exchangers or sheet metal in automobile under-hoods). Meshing these walls with solid cells would lead to high-aspect-ratio meshes and a significant increase in the total number of cells. See below for details about shell conduction.

Thermal conditions are entered in the Thermal section of the Wall panel (Figure 6.13.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4). To view the thermal conditions, click the Thermal tab.

![Figure 6.13.1: The Wall Panel (Thermal Section)](image)

© Fluent Inc. November 28, 2001
6.13 Wall Boundary Conditions

Heat Flux Boundary Conditions

For a fixed heat flux condition, choose the Heat Flux option under Thermal Conditions. You will then need to set the appropriate value for the heat flux at the wall surface in the Heat Flux field. You can define an adiabatic wall by setting a zero heat flux condition. This is the default condition for all walls.

Temperature Boundary Conditions

To select the fixed temperature condition, choose the Temperature option under Thermal Conditions in the Wall panel. You will need to specify the temperature at the wall surface (Temperature). The heat transfer to the wall is computed using Equation 6.13-7 or Equation 6.13-8.

Convective Heat Transfer Boundary Conditions

For a convective heat transfer wall boundary, select Convection under Thermal Conditions. Your inputs of Heat Transfer Coefficient and Free Stream Temperature will allow FLUENT to compute the heat transfer to the wall using Equation 6.13-11.

External Radiation Boundary Conditions

If radiation heat transfer from the exterior of your model is of interest, you can enable the Radiation option in the Wall panel and set the External Emissivity and External Radiation Temperature.

Combined Convection and External Radiation Boundary Conditions

You can choose a thermal condition that combines the convection and radiation boundary conditions by selecting the Mixed option. With this thermal condition, you will need to set the Heat Transfer Coefficient, Free Stream Temperature, External Emissivity, and External Radiation Temperature.
**Boundary Conditions**

**Thin-Wall Thermal Resistance Parameters**

By default, a wall will have a thickness of zero. You can, however, in conjunction with any of the thermal conditions, model a thin layer of material between two zones. For example, you can model the effect of a piece of sheet metal between two fluid zones, a coating on a solid zone, or contact resistance between two solid regions. FLUENT will solve a 1D conduction equation to compute the thermal resistance offered by the wall and the heat generation in the wall.

To include these effects in the heat transfer calculation you will need to specify the type of material, the thickness of the wall, and the heat generation rate in the wall. Select the material type in the Material Name drop-down list, and specify the thickness in the Wall Thickness field. If you want to check or modify the properties of the selected material, you can click Edit... to open the Material panel; this panel contains just the properties of the selected material, not the full contents of the standard Materials panel.

The thermal resistance of the wall is \( \frac{\Delta x}{k} \), where \( k \) is the conductivity of the wall material and \( \Delta x \) is the wall thickness. The thermal boundary condition you set will be specified on the outside of the thin wall, as shown in Figure 6.13.2, where \( T_b \) is the fixed Temperature specified at the wall.

Specify the heat generation rate inside the wall in the Heat Generation Rate field. This option is useful if, for example, you are modeling printed circuit boards where you know the electrical power dissipated in the circuits.

**Thermal Conditions for Two-Sided Walls**

If the wall zone has a fluid or solid region on each side, it is called a “two-sided wall”. When you read a grid with this type of wall zone into FLUENT, a “shadow” zone will automatically be created so that each side of the wall is a distinct wall zone. In the Wall panel, the shadow zone’s name will be shown in the Shadow Face Zone field. You can choose to specify different thermal conditions on each zone, or to couple the two zones:
6.13 Wall Boundary Conditions

![Diagram of a thin wall with fluid or solid cells, outer surface, inner surface, Ax, Tb, and thin wall labeled.]

Figure 6.13.2: Thermal Conditions are Specified on the Outside of a Thin Wall

- To couple the two sides of the wall, select the Coupled option under Thermal Conditions. (This option will appear in the Wall panel only when the wall is a two-sided wall.) No additional thermal boundary conditions are required, because the solver will calculate heat transfer directly from the solution in the adjacent cells. You can, however, specify the material type, wall thickness, and heat generation rate for thin-wall thermal resistance calculations, as described above. Note that the resistance parameters you set for one side of the wall will automatically be assigned to its shadow wall zone. Specifying the heat generation rate inside the wall is useful if, for example, you are modeling printed circuit boards where you know the electrical power dissipated in the circuits but not the heat flux or wall temperature.

- To uncouple the two sides of the wall and specify different thermal conditions on each one, choose Temperature or Heat Flux as the thermal condition type. (Convection and Radiation are not applicable for two-sided walls.) The relationship between the wall and
Boundary Conditions

its shadow will be retained, so that you can couple them again at a later time, if desired. You will need to set the relevant parameters for the selected thermal condition, as described above. The two uncoupled walls can have different thicknesses, and are effectively insulated from one another. If you specify a non-zero wall thickness for the uncoupled walls, the thermal boundary conditions you set will be specified on the outer sides of the two thin walls, as shown in Figure 6.13.3, where $T_{b1}$ is the Temperature (or $q_{b1}$ is the Heat Flux) specified on one wall and $T_{b2}$ is the Temperature (or $q_{b2}$ is the Heat Flux) specified on the other wall. $k_{w1}$ and $k_{w2}$ are the thermal conductivities of the uncoupled thin walls. Note that the gap between the walls in Figure 6.13.3 is not part of the model; it is included in the figure only to show where the thermal boundary condition for each uncoupled wall is applied.

![Figure 6.13.3: Thermal Conditions are Specified on the Outer Sides of the Uncoupled Thin Walls](image)

Figure 6.13.3: Thermal Conditions are Specified on the Outer Sides of the Uncoupled Thin Walls
Shell Conduction in Walls

To enable shell conduction for a wall, turn on the Shell Conduction option in the Wall boundary condition panel. When this option is enabled, FLUENT will compute heat conduction within the wall, in addition to conduction across the wall (which is always computed when the energy equation is solved). The Shell Conduction option will appear in the Wall panel for all walls when solution of the energy equation is active. For a boundary wall, the thermal conditions are applied as described above for thin walls.

! You must specify a non-zero Wall Thickness in the Wall panel, because the shell conduction model is relevant only for walls with non-zero thickness.

! Note that the shell conduction model has several limitations:

- It is available only in 3D.
- It is available only when the segregated solver is used.
- It cannot be used with the non-premixed or partially premixed combustion model.
- It cannot be used with the multiphase mixture, VOF, or Eulerian model.
- When used in conjunction with the discrete ordinates (DO) radiation model, shell conducting walls cannot be semi-transparent.
- Shell conducting walls cannot be split or merged. If you need to split or merge a shell conducting wall, disable the Shell Conduction option for the wall, perform the split or merge operation, and then enable Shell Conduction for the new wall zones.
- The shell conduction model cannot be used on a wall zone that has been adapted. If you want to perform adaption elsewhere in the computational domain, be sure to use the mask register described in Section 23.9.1 to ensure that no adaption is performed on the shell conducting wall.
Boundary Conditions

- Fluxes at the ends of a shell conducting wall are not included in the heat balance reports. These fluxes are accounted for correctly in the FLUENT solution, but not in the flux report itself.

Defining Wall Motion

Wall boundaries can be either stationary or moving. The stationary boundary condition specifies a fixed wall, whereas the moving boundary condition can be used to specify the translational or rotational velocity of the wall, or the velocity components.

Wall motion conditions are entered in the Momentum section of the Wall panel (Figure 6.13.4), which is opened from the Boundary Conditions panel (as described in Section 6.1.4). To view the wall motion conditions, click the Momentum tab.

Defining a Stationary Wall

For a stationary wall, choose the Stationary Wall option under Wall Motion.

Defining Velocity Conditions for Moving Walls

If you wish to include tangential motion of the wall in your calculation, you need to define the translational or rotational velocity, or the velocity components. Select the Moving Wall option under Wall Motion. The Wall panel will expand, as shown in Figure 6.13.4, to show the wall velocity conditions.

Note that you cannot use the moving wall condition to model problems where the wall has a motion normal to itself. FLUENT will neglect any normal component of wall motion that you specify using the methods below.

Specifying Relative or Absolute Velocity

If the cell zone adjacent to the wall is moving (e.g., if you are using a moving reference frame or a sliding mesh), you can choose to specify velocities relative to the zone motion by enabling the Relative to Adjacent
Figure 6.13.4: The Wall Panel for a Moving Wall
**Boundary Conditions**

**Cell Zone** option. If you choose to specify relative velocities, a velocity of zero means that the wall is stationary in the relative frame, and therefore moving at the speed of the adjacent cell zone in the absolute frame. If you choose to specify absolute velocities (by enabling the **Absolute** option), a velocity of zero means that the wall is stationary in the absolute frame, and therefore moving at the speed of the adjacent cell zone—but in the opposite direction—in the relative reference frame.

If you are using one or more moving reference frames, sliding meshes, or mixing planes, and you want the wall to be fixed in the moving frame, it is recommended that you specify relative velocities (the default) rather than absolute velocities. Then, if you modify the speed of the adjacent cell zone, you will not need to make any changes to the wall velocities, as you would if you specified absolute velocities.

Note that if the adjacent cell zone is not moving, the absolute and relative options are equivalent.

**Translational Wall Motion**

For problems that include linear translational motion of the wall boundary (e.g., a rectangular duct with a moving belt as one wall) you can enable the **Translational** option and specify the wall’s **Speed** and **Direction** \((X,Y,Z)\) vector). By default, wall motion is “disabled” by the specification of **Translational** velocity with a **Speed** of zero.

If you need to define non-linear translational motion, you will need to use the **Components** option, described below.

**Rotational Wall Motion**

For problems that include rotational wall motion you can enable the **Rotational** option and define the rotational **Speed** about a specified axis. To define the axis, set the **Rotation-Axis Direction** and **Rotation-Axis Origin**. This axis is independent of the axis of rotation used by the adjacent cell zone, and independent of any other wall rotation axis. For 3D problems, the axis of rotation is the vector passing through the specified **Rotation-Axis Origin** and parallel to the vector from \((0,0,0)\) to the \((X,Y,Z)\) point.
specified under **Rotation-Axis Direction**. For 2D problems, you will specify only the **Rotation-Axis Origin**; the axis of rotation is the \(z\)-direction vector passing through the specified point. For 2D axisymmetric problems, you will not define the axis: the rotation will always be about the \(x\) axis, with the origin at \((0,0)\).

Note that the modeling of tangential rotational motion will be correct only if the wall bounds a surface of revolution about the prescribed axis of rotation (e.g., a circle or cylinder). Note also that rotational motion can be specified for a wall in a stationary reference frame.

**Wall Motion Based on Velocity Components**

For problems that include linear or non-linear translational motion of the wall boundary you can enable the **Components** option and specify the **X-Velocity**,** Y-Velocity**, and **Z-Velocity** of the wall. You can define non-linear translational motion using a boundary profile or a user-defined function for the **X-Velocity**,** Y-Velocity**, and/or **Z-Velocity** of the wall.

**Wall Motion for Two-Sided Walls**

As discussed earlier in this section, when you read a grid with a two-sided wall zone (which forms the interface between fluid/solid regions) into **FLUENT**, a “shadow” zone will automatically be created so that each side of the wall is a distinct wall zone. For two-sided walls, it is possible to specify different motions for the wall and shadow zones, whether or not they are coupled. Note, however, that you cannot specify motion for a wall (or shadow) that is adjacent to a solid zone.

**Defining Shear Conditions at Walls**

Three types of shear conditions are available:

- no-slip
- specified shear
- Marangoni stress
**Boundary Conditions**

Note that all moving walls are no-slip; the other shear conditions are relevant only for stationary walls.

The no-slip condition is the default, and it indicates that the fluid sticks to the wall and moves with the same velocity as the wall, if it is moving. The specified shear and Marangoni stress boundary conditions are useful in modeling situations in which the shear stress (rather than the motion of the fluid) is known. Examples of such situations are applied shear stress, slip wall (zero shear stress), and free surface conditions (zero shear stress or shear stress dependent on surface tension gradient). The specified shear boundary condition allows you to specify the $x$, $y$, and $z$ components of the shear stress as constant values or boundary profiles. The Marangoni stress boundary condition allows you to specify the gradient of the surface tension with respect to the temperature at this surface. The shear stress is calculated based on the surface gradient of the temperature and the specified surface tension gradient. The Marangoni stress option is available only for calculations in which the energy equation is being solved.

Shear conditions are entered in the **Momentum** section of the **Wall** panel, which is opened from the **Boundary Conditions** panel (as described in Section 6.1.4). To view the shear conditions, click the **Momentum** tab.

**Modeling No-Slip Walls**

You can model a no-slip wall by selecting the **No Slip** option under **Shear Condition**. This is the default for all walls in viscous flows.

**Specified Shear**

In addition to the no-slip wall that is the default for viscous flows, you can model a slip wall by specifying zero or non-zero shear. To specify the shear, select the **Specified Shear** option under **Shear Condition** (see Figure 6.13.5). You can then enter $x$, $y$, and $z$ components of shear under **Shear Stress**. Wall functions for turbulence are not used with the **Specified Shear** option.
6.13 Wall Boundary Conditions

Figure 6.13.5: The Wall Panel for Specified Shear
Marangoni Stress

**FLUENT** can also model shear stresses caused by the variation of surface tension due to temperature. The shear stress applied at the wall is given by

\[ \tau = \frac{d\sigma}{dT} \nabla_s T \]  

(6.13-1)

where \( d\sigma/dT \) is the surface tension gradient with respect to temperature, and \( \nabla_s T \) is the surface gradient. This shear stress is then applied to the momentum equation.

To model Marangoni stress for the wall, select the **Marangoni Stress** option under **Shear Condition** (see Figure 6.13.6). This option is available only for calculations in which the energy equation is being solved. You can then enter the surface tension gradient (\( d\sigma/dT \) in Equation 6.13-1) in the **Surface Tension Gradient** field. Wall functions for turbulence are not used with the **Marangoni Stress** option.

**Modeling Wall Roughness Effects in Turbulent Wall-Bounded Flows**

Fluid flows over rough surfaces are encountered in diverse situations. Examples are, among many others, flows over the surfaces of airplanes, ships, turbomachinery, heat exchangers, and piping systems, and atmospheric boundary layers over terrain of varying roughness. Wall roughness affects drag (resistance) and heat and mass transfer on the walls.

If you are modeling a turbulent wall-bounded flow in which the wall roughness effects are considered to be significant, you can include the wall roughness effects through the law-of-the-wall modified for roughness.

**Law-of-the-Wall Modified for Roughness**

Experiments in roughened pipes and channels indicate that the mean velocity distribution near rough walls, when plotted in the usual semi-logarithmic scale, has the same slope (1/\( \kappa \)) but a different intercept...
6.13 Wall Boundary Conditions

Figure 6.13.6: The Wall Panel for Marangoni Stress
(additive constant $B$ in the log-law). Thus, the law-of-the-wall for mean velocity modified for roughness has the form

$$\frac{u_p u^*}{\tau_w / \rho} = \frac{1}{\kappa} \ln\left(\frac{E u^* y_p}{\mu}\right) - \Delta B$$

(6.13-2)

where $u^* = C_1^{1/4} k^{1/2}$ and $\Delta B$ is a roughness function that quantifies the shift of the intercept due to roughness effects.

$\Delta B$ depends, in general, on the type (uniform sand, rivets, threads, ribs, mesh-wire, etc.) and size of the roughness. There is no universal roughness function valid for all types of roughness. For a sand-grain roughness and similar types of uniform roughness elements, however, $\Delta B$ has been found to be well-correlated with the nondimensional roughness height, $K_s^+ = \rho K_s u^*/\mu$, where $K_s$ is the physical roughness height and $u^* = C_1^{1/4} k^{1/2}$. Analyses of experimental data show that the roughness function, $\Delta B$, is not a single function of $K_s^+$, but takes different forms depending on the $K_s^+$ value. It has been observed that there are three distinct regimes:

- Hydrodynamically smooth ($K_s^+ < 3 \sim 5$)
- Transitional ($3 \sim 5 < K_s^+ < 70 \sim 90$)
- Fully rough ($K_s^+ > 70 \sim 90$)

According to the data, roughness effects are negligible in the hydrodynamically smooth regime, but become increasingly important in the transitional regime, and take full effect in the fully rough regime.

In FLUENT, the whole roughness regime is subdivided into the three regimes, and the formulas proposed by Cebeci and Bradshaw based on Nikuradse’s data [31] are adopted to compute the roughness function, $\Delta B$, for each regime.

For the hydrodynamically smooth regime ($K_s^+ < 2.25$):

$$\Delta B = 0$$

(6.13-3)
For the transitional regime \((2.25 < K_s^+ < 90)\):

\[
\Delta B = \frac{1}{\kappa} \ln \left[ \frac{K_s^+ - 2.25}{87.75} + C_{K_s} K_s^+ \right] \times \sin \{0.4258(\ln K_s^+ - 0.811)\}
\]

where \(C_{K_s}\) is a roughness constant, and depends on the type of the roughness.

In the fully rough regime \((K_s^+ > 90)\):

\[
\Delta B = \frac{1}{\kappa} \ln(1 + C_{K_s} K_s^+)
\]

In the solver, given the roughness parameters, the roughness function \(\Delta B(K_s^+)\) is evaluated using the corresponding formula (Equation 6.13-3, 6.13-4, or 6.13-5). The modified law-of-the-wall in Equation 6.13-2 is then used to evaluate the shear stress at the wall and other wall functions for the mean temperature and turbulent quantities.

**Setting the Roughness Parameters**

The roughness parameters are in the **Momentum** section of the Wall panel (see Figure 6.13.6), which is opened from the **Boundary Conditions** panel (as described in Section 6.1.4). To view the wall roughness parameters, click the **Momentum** tab.

To model the wall roughness effects, you must specify two roughness parameters: the **Roughness Height**, \(K_s\), and the **Roughness Constant**, \(C_{K_s}\). The default roughness height \((K_s)\) is zero, which corresponds to smooth walls. For the roughness to take effect, you must specify a non-zero value for \(K_s\). For a uniform sand-grain roughness, the height of the sand-grain can simply be taken for \(K_s\). For a non-uniform sand-grain, however, the mean diameter \((D_{50})\) would be a more meaningful roughness height. For other types of roughness, an “equivalent” sand-grain roughness height should be used for \(K_s\).

Choosing a proper roughness constant \((C_{K_s})\) is dictated mainly by the type of the given roughness. The default roughness constant \((C_{K_s} = 0.5)\)
was determined so that, when used with $k$-$\varepsilon$ turbulence models, it reproduces Nikuradse’s resistance data for pipes roughened with tightly-packed, uniform sand-grain roughness. You may need to adjust the roughness constant when the roughness you want to model departs much from uniform sand-grain. For instance, there is some experimental evidence that, for non-uniform sand-grains, ribs, and wire-mesh roughness, a higher value ($C_{K_s} = 0.5 \sim 1.0$) is more appropriate. Unfortunately, a clear guideline for choosing $C_{K_s}$ for arbitrary types of roughness is not available.

Note that it is not physically meaningful to have a mesh size such that the wall-adjacent cell is smaller than the roughness height. For best results, make sure that the distance from the wall to the centroid of the wall-adjacent cell is greater than $K_s$.

**Defining Species Boundary Conditions for Walls**

By default, a zero-gradient condition for all species is assumed at walls (except for species that participate in surface reactions), but it is also possible to specify species mass fractions at walls. That is, Dirichlet boundary conditions such as those that are specified at inlets can be used at walls as well.

If you wish to retain the default zero-gradient condition for a species, no inputs are required. If you want to specify the mass fraction for a species at the wall, the steps are as follows:

1. Click the Species tab in the Wall panel to view the species boundary conditions for the wall (see Figure 6.13.7).

2. Under Species Boundary Condition, select Specified Mass Fraction (rather than Zero Diffusive Flux) in the drop-down list to the right of the species name. The panel will expand to include space for Species Mass Fractions.

3. Under Species Mass Fractions, specify the mass fraction for the species.
The boundary condition type for each species is specified separately, so you can choose to use different methods for different species.

Note that if you use Dirichlet conditions for species in a turbulent flow, FLUENT will not use wall functions for calculating the species diffusion flux at the walls.

**Defining Reaction Boundary Conditions for Walls**

If you have enabled the modeling of wall surface reactions in the **Species Model** panel, you can indicate whether or not surface reactions should
**Boundary Conditions**

be activated for the wall. In the Species section of the Wall panel (Figure 6.13.7), turn the Surface Reactions option on or off.

Note that a zero-gradient condition is assumed at the wall for species that do not participate in any surface reactions.

**Defining Radiation Boundary Conditions for Walls**

If you are using the P-1 radiation model, the DTRM, the DO model, or the surface-to-surface model, you will need to set the emissivity of the wall (Internal Emissivity) in the Radiation section of the Wall panel. If you are using the Rosseland model you do not need to set the emissivity, because FLUENT assumes the emissivity is 1. If you are using the DO model you will also need to define the wall as diffuse, specular, or semi-transparent. See Section 11.3.16 for details.

**Defining Discrete Phase Boundary Conditions for Walls**

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the wall in the DPM section of the Wall panel. See Section 19.10 for details.

**Defining Multiphase Boundary Conditions for Walls**

If you are using the VOF model and you are modeling wall adhesion, you can specify the contact angle for each pair of phases at the wall in the Momentum section of the Wall panel. See Section 20.6.14 for details.

**Defining Boundary Conditions for User-Defined Scalars**

If you are using any user-defined scalars in your simulation, you can specify boundary conditions for them in the UDS section of the Wall panel. See the separate UDF Manual for details about user-defined scalars.

**6.13.2 Default Settings at Wall Boundaries**

The default thermal boundary condition is a fixed heat flux of zero. Walls are, by default, not moving.
6.13 Wall Boundary Conditions

6.13.3 Shear-Stress Calculation Procedure at Wall Boundaries

For no-slip wall conditions, FLUENT uses the properties of the flow adjacent to the wall/fluid boundary to predict the shear stress on the fluid at the wall. In laminar flows this calculation simply depends on the velocity gradient at the wall, while in turbulent flows one of the approaches described in Section 10.8 is used.

For specified-shear walls, FLUENT will compute the tangential velocity at the boundary.

If you are modeling inviscid flow with FLUENT, all walls use a slip condition, so they are frictionless and exert no shear stress on the adjacent fluid.

Shear-Stress Calculation in Laminar Flow

In a laminar flow, the wall shear stress is defined by the normal velocity gradient at the wall as

$$\tau_w = \mu \frac{\partial v}{\partial n} \quad (6.13-6)$$

When there is a steep velocity gradient at the wall, you must be sure that the grid is sufficiently fine to accurately resolve the boundary layer. Guidelines for the appropriate placement of the near-wall node in laminar flows are provided in Section 5.2.2.

Shear-Stress Calculation in Turbulent Flows

Wall treatments for turbulent flows are described in Section 10.8.

6.13.4 Heat Transfer Calculations at Wall Boundaries

Temperature Boundary Conditions

When a fixed temperature condition is applied at the wall, the heat flux to the wall from a fluid cell is computed as
**Boundary Conditions**

\[ q = h_f(T_w - T_f) + q_{rad} \]  \hspace{1cm} (6.13-7)

where

- \( h_f \) = fluid-side local heat transfer coefficient
- \( T_w \) = wall surface temperature
- \( T_f \) = local fluid temperature
- \( q_{rad} \) = radiative heat flux

Note that the fluid-side heat transfer coefficient is computed based on the local flow-field conditions (e.g., turbulence level, temperature, and velocity profiles), as described by Equations 6.13-14 and 10.8-5.

Heat transfer to the wall boundary from a solid cell is computed as

\[ q = \frac{k_s}{\Delta n} (T_w - T_s) + q_{rad} \]  \hspace{1cm} (6.13-8)

where

- \( k_s \) = thermal conductivity of the solid
- \( T_s \) = local solid temperature
- \( \Delta n \) = distance between wall surface and the solid cell center

**Heat Flux Boundary Conditions**

When you define a heat flux boundary condition at a wall, you specify the heat flux at the wall surface. FLUENT uses Equation 6.13-7 and your input of heat flux to determine the wall surface temperature adjacent to a fluid cell as

\[ T_w = \frac{q - q_{rad}}{h_f} + T_f \]  \hspace{1cm} (6.13-9)

where, as noted above, the fluid-side heat transfer coefficient is computed based on the local flow-field conditions. When the wall borders a solid region, the wall surface temperature is computed as

\[ T_w = \frac{(q - q_{rad})\Delta n}{k_s} + T_s \]  \hspace{1cm} (6.13-10)
6.13 Wall Boundary Conditions

**Convective Heat Transfer Boundary Conditions**

When you specify a convective heat transfer coefficient boundary condition at a wall, FLUENT uses your inputs of the external heat transfer coefficient and external heat sink temperature to compute the heat flux to the wall as

\[
q = h_f(T_w - T_f) + q_{\text{rad}} = h_{\text{ext}}(T_{\text{ext}} - T_w)
\]  

(6.13-11)

where

- \(h_{\text{ext}}\) = external heat transfer coefficient defined by you
- \(T_{\text{ext}}\) = external heat-sink temperature defined by you
- \(q_{\text{rad}}\) = radiative heat flux

Equation 6.13-11 assumes a wall of zero thickness.

**External Radiation Boundary Conditions**

When the external radiation boundary condition is used in FLUENT, the heat flux to the wall is computed as

\[
q = h_f(T_w - T_f) + q_{\text{rad}} = \epsilon_{\text{ext}}\sigma(T_\infty^4 - T_w^4)
\]  

(6.13-12)

where

- \(\epsilon_{\text{ext}}\) = emissivity of the external wall surface defined by you
- \(\sigma\) = Stefan-Boltzmann constant
- \(T_w\) = surface temperature of the wall
- \(T_\infty\) = temperature of the radiation source or sink on the exterior of the domain, defined by you
- \(q_{\text{rad}}\) = radiative heat flux to the wall from within the domain

Equation 6.13-12 assumes a wall of zero thickness.


**Boundary Conditions**

**Combined External Convection and Radiation Boundary Conditions**

When you choose the combined external heat transfer condition, the heat flux to the wall is computed as

\[
q = h_f(T_w - T_f) + q_{\text{rad}} = h_{\text{ext}}(T_{\text{ext}} - T_w) + \epsilon_{\text{ext}}\sigma(T_\infty^4 - T_w^4)
\]

(6.13-13)

where the variables are as defined above. Equation 6.13-13 assumes a wall of zero thickness.

**Calculation of the Fluid-Side Heat Transfer Coefficient**

In laminar flows, the fluid side heat transfer at walls is computed using Fourier’s law applied at the walls. **FLUENT** uses its discrete form:

\[
q = k_f \left( \frac{\partial T}{\partial n} \right)_{\text{wall}}
\]

(6.13-14)

where \( n \) is the local coordinate normal to the wall.

For turbulent flows, **FLUENT** uses the law-of-the-wall for temperature derived using the analogy between heat and momentum transfer [129]. See Section 10.8.2 for details.
6.14 Symmetry Boundary Conditions

Symmetry boundary conditions are used when the physical geometry of interest, and the expected pattern of the flow/thermal solution, have mirror symmetry. They can also be used to model zero-shear slip walls in viscous flows. This section describes the treatment of the flow at symmetry planes and provides examples of the use of symmetry. You do not define any boundary conditions at symmetry boundaries, but you must take care to correctly define your symmetry boundary locations.

At the centerline of an axisymmetric geometry, you should use the axis boundary type rather than the symmetry boundary type, as illustrated in Figure 6.16.1. See Section 6.16 for details.

6.14.1 Examples of Symmetry Boundaries

Symmetry boundaries are used to reduce the extent of your computational model to a symmetric subsection of the overall physical system. Figures 6.14.1 and 6.14.2 illustrate two examples of symmetry boundary conditions used in this way.

Figure 6.14.1: Use of Symmetry to Model One Quarter of a 3D Duct
Boundary Conditions

2 symmetry planes
(model includes a 90° sector)

Figure 6.14.2: Use of Symmetry to Model One Quarter of a Circular Cross-Section
Figure 6.14.3 illustrates two problems in which a symmetry plane would be *inappropriate*. In both examples, the problem geometry is symmetric but the flow itself does not obey the symmetry boundary conditions. In the first example, buoyancy creates an asymmetric flow. In the second, swirl in the flow creates a flow normal to the would-be symmetry plane. Note that this second example should be handled using rotationally periodic boundaries (as illustrated in Figure 6.15.1).

Figure 6.14.3: Inappropriate Use of Symmetry
6.14.2 Calculation Procedure at Symmetry Boundaries

FLUENT assumes a zero flux of all quantities across a symmetry boundary. There is no convective flux across a symmetry plane: the normal velocity component at the symmetry plane is thus zero. There is no diffusion flux across a symmetry plane: the normal gradients of all flow variables are thus zero at the symmetry plane. The symmetry boundary condition can therefore be summarized as follows:

- Zero normal velocity at a symmetry plane
- Zero normal gradients of all variables at a symmetry plane

As stated above, these conditions determine a zero flux across the symmetry plane, which is required by the definition of symmetry. Since the shear stress is zero at a symmetry boundary, it can also be interpreted as a “slip” wall when used in viscous flow calculations.
6.15 Periodic Boundary Conditions

Periodic boundary conditions are used when the physical geometry of interest and the expected pattern of the flow/thermal solution have a periodically repeating nature. Two types of periodic conditions are available in FLUENT. The first type does not allow a pressure drop across the periodic planes. (Note to FLUENT 4 users: This type of periodic boundary is referred to as a “cyclic” boundary in FLUENT 4.) The second type allows a pressure drop to occur across translationally periodic boundaries, enabling you to model “fully-developed” periodic flow. (In FLUENT 4 this is a “periodic” boundary.)

This section discusses the no-pressure-drop periodic boundary condition. A complete description of the fully-developed periodic flow modeling capability is provided in Section 8.3.

You can find the following information about periodic boundary conditions in this section:

- Section 6.15.1: Examples of Periodic Boundaries
- Section 6.15.2: Inputs for Periodic Boundaries
- Section 6.15.3: Default Settings at Periodic Boundaries
- Section 6.15.4: Calculation Procedure at Periodic Boundaries

6.15.1 Examples of Periodic Boundaries

Periodic boundary conditions are used when the flows across two opposite planes in your computational model are identical. Figure 6.15.1 illustrates a typical application of periodic boundary conditions. In this example the flow entering the computational model through one periodic plane is identical to the flow exiting the domain through the opposite periodic plane. Periodic planes are always used in pairs as illustrated in this example.
Boundary Conditions

4 tangential inlets

periodic boundaries

Figure 6.15.1: Use of Periodic Boundaries to Define Swirling Flow in a Cylindrical Vessel

6.15.2 Inputs for Periodic Boundaries

For a periodic boundary without any pressure drop, there is only one input you need to consider: whether the geometry is rotationally or translationally periodic. (Additional inputs are required for a periodic flow with a periodic pressure drop. See Section 8.3.)

Rotationally periodic boundaries are boundaries that form an included angle about the centerline of a rotationally symmetric geometry. Figure 6.15.1 illustrates rotational periodicity. Translationally periodic boundaries are boundaries that form periodic planes in a rectilinear geometry. Figure 6.15.2 illustrates translationally periodic boundaries.

You will specify translational or rotational periodicity for a periodic boundary in the Periodic panel (Figure 6.15.3), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).

(There will be an additional item in the Periodic panel for the coupled solvers, which allows you to specify the periodic pressure jump. See
6.15 Periodic Boundary Conditions

(a) Physical Domain

(b) Modeled Domain

Figure 6.15.2: Example of Translational Periodicity
Section 8.3 for details.)

If the domain is rotationally periodic, select Rotational as the Periodic Type; if it is translationally periodic, select Translational. For rotationally periodic domains, the solver will automatically compute the angle through which the periodic zone is rotated. The axis used for this rotation is the axis of rotation specified for the adjacent cell zone.

Note that there is no need for the adjacent cell zone to be moving for you to use a rotationally periodic boundary. You could, for example, model pipe flow in 3D using a nonrotating reference frame with a pie-slice of the pipe; the sides of the slice would require rotational periodicity.

You can use the Grid/Check menu item (see Section 5.5) to compute and display the minimum, maximum, and average rotational angles of all faces on periodic boundaries. If the difference between the minimum, maximum, and average values is not negligible, then there is a problem with the grid: the grid geometry is not periodic about the specified axis.

6.15.3 Default Settings at Periodic Boundaries

By default, all periodic boundaries are translational.
6.15.4 Calculation Procedure at Periodic Boundaries

FLUENT treats the flow at a periodic boundary as though the opposing periodic plane is a direct neighbor to the cells adjacent to the first periodic boundary. Thus, when calculating the flow through the periodic boundary adjacent to a fluid cell, the flow conditions at the fluid cell adjacent to the opposite periodic plane are used.

6.16 Axis Boundary Conditions

The axis boundary type must be used as the centerline of an axisymmetric geometry (see Figure 6.16.1). It can also be used for the centerline of a cylindrical-polar quadrilateral or hexahedral grid (e.g., a grid created for a structured-grid code such as FLUENT 4). You do not need to define any boundary conditions at axis boundaries.

Figure 6.16.1: Use of an Axis Boundary as the Centerline in an Axisymmetric Geometry
Calculation Procedure at Axis Boundaries

To determine the appropriate physical value for a particular variable at a point on the axis, FLUENT uses the cell value in the adjacent cell.

6.17 Fluid Conditions

A fluid zone is a group of cells for which all active equations are solved. The only required input for a fluid zone is the type of fluid material. You must indicate which material the fluid zone contains so that the appropriate material properties will be used.

If you are modeling species transport and/or combustion, you will not select a material here; the mixture material is specified in the Species Model panel when you enable the model. Similarly, you will not specify the materials for a multiphase flow here; you will choose them when you define the phases, as described in Section 20.6.8.

Optional inputs allow you to set sources or fixed values of mass, momentum, heat/temperature, turbulence, species, and other scalar quantities. You can also define motion for the fluid zone. If there are rotationally periodic boundaries adjacent to the fluid zone, you will need to specify the rotation axis. If you are modeling turbulence using one of the $k$-$\varepsilon$ models, the $k$-$\omega$ model, or the Spalart-Allmaras model, you can choose to define the fluid zone as a laminar flow region. If you are modeling radiation using the DO model, you can specify whether or not the fluid participates in radiation.

For information about porous zones, see Section 6.19.

6.17.1 Inputs for Fluid Zones

You will set all fluid conditions in the Fluid panel (Figure 6.17.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).

Defining the Fluid Material

To define the material contained in the fluid zone, select the appropriate item in the Material Name drop-down list. This list will contain all fluid materials that have been defined (or loaded from the materials database)
in the Materials panel. If you want to check or modify the properties of the selected material, you can click Edit... to open the Material panel; this panel contains just the properties of the selected material, not the full contents of the standard Materials panel.

If you are modeling species transport or multiphase flow, the Material Name list will not appear in the Fluid panel. For species calculations, the mixture material for all fluid zones will be the material you specified in the Species Model panel. For multiphase flows, the materials are specified when you define the phases, as described in Section 20.6.8.
Boundary Conditions

Defining Sources

If you wish to define a source of heat, mass, momentum, turbulence, species, or other scalar quantity within the fluid zone, you can do so by enabling the Source Terms option. See Section 6.27 for details.

Defining Fixed Values

If you wish to fix the value of one or more variables in the fluid zone, rather than computing them during the calculation, you can do so by enabling the Fixed Values option. See Section 6.26 for details.

Specifying a Laminar Zone

When you are calculating a turbulent flow using one of the \( k-\epsilon \) models, the \( k-\omega \) model, or the Spalart-Allmaras model, it is possible to “turn off” turbulence modeling (i.e., disable turbulence production and turbulent viscosity, but transport the turbulence quantities) in a specific fluid zone. This is useful if you know that the flow in a certain region is laminar. For example, if you know the location of the transition point on an airfoil, you can create a laminar/turbulent transition boundary where the laminar cell zone borders the turbulent cell zone. This feature allows you to model turbulent transition on the airfoil.

To disable turbulence modeling in a fluid zone, turn on the Laminar Zone option in the Fluid panel.

Specifying the Rotation Axis

If there are rotationally periodic boundaries adjacent to the fluid zone or if the zone is rotating, you must specify the rotation axis. To define the axis, set the Rotation-Axis Direction and Rotation-Axis Origin. This axis is independent of the axis of rotation used by any adjacent wall zones or any other cell zones. For 3D problems, the axis of rotation is the vector from the Rotation-Axis Origin in the direction of the vector given by your Rotation-Axis Direction inputs. For 2D non-axisymmetric problems, you will specify only the Rotation-Axis Origin; the axis of rotation is the \( z \)-direction vector passing through the specified point. (The \( z \) direction is normal to the plane of your geometry so that rotation occurs in the
6.18 Solid Conditions

plane.) For 2D axisymmetric problems, you will not define the axis: the rotation will always be about the z axis, with the origin at (0,0).

Defining Zone Motion

To define zone motion for a rotating or translating reference frame, select Moving Reference Frame in the Motion Type drop-down list (visible if you scroll down using the scroll bar to the right of the Rotation-Axis Origin and Direction) and then set the appropriate parameters in the expanded portion of the panel.

To define zone motion for a sliding mesh, select Moving Mesh in the Motion Type drop-down list and then set the appropriate parameters in the expanded portion of the panel. See Section 9.5 for details.

For problems that include linear, translational motion of the fluid zone, specify the Translational Velocity by setting the X, Y, and Z components. For problems that include rotational motion, specify the rotational Speed under Rotational Velocity. The rotation axis is defined as described above.

See Chapter 9 for details about modeling flows in moving reference frames.

Defining Radiation Parameters

If you are using the DO radiation model, you can specify whether or not the fluid zone participates in radiation using the Participates in Radiation option. See Section 11.3.16 for details.

6.18 Solid Conditions

A “solid” zone is a group of cells for which only a heat conduction problem is solved; no flow equations are solved. The material being treated as a solid may actually be a fluid, but it is assumed that no convection is taking place. The only required input for a solid zone is the type of solid material. You must indicate which material the solid zone contains so that the appropriate material properties will be used. Optional inputs allow you to set a volumetric heat generation rate (heat source) or a fixed value of temperature. You can also define motion for the solid
Boundary Conditions

zone. If there are rotationally periodic boundaries adjacent to the solid zone, you will need to specify the rotation axis. If you are modeling radiation using the DO model, you can specify whether or not the solid material participates in radiation.

6.18.1 Inputs for Solid Zones

You will set all solid conditions in the Solid panel (Figure 6.18.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).

![Figure 6.18.1: The Solid Panel](image)

Defining the Solid Material

To define the material contained in the solid zone, select the appropriate item in the Material Name drop-down list. This list will contain all solid materials that have been defined (or loaded from the materials database) in the Materials panel. If you want to check or modify the properties of
the selected material, you can click Edit... to open the Material panel; this panel contains just the properties of the selected material, not the full contents of the standard Materials panel.

Defining a Heat Source

If you wish to define a source of heat within the solid zone, you can do so by enabling the Source Terms option. See Section 6.27 for details.

Defining a Fixed Temperature

If you wish to fix the value of temperature in the solid zone, rather than computing it during the calculation, you can do so by enabling the Fixed Values option. See Section 6.26 for details.

Specifying the Rotation Axis

If there are rotationally periodic boundaries adjacent to the solid zone or if the zone is rotating, you must specify the rotation axis. To define the axis, set the Rotation-Axis Direction and Rotation-Axis Origin. This axis is independent of the axis of rotation used by any adjacent wall zones or any other cell zones. For 3D problems, the axis of rotation is the vector from the Rotation-Axis Origin in the direction of the vector given by your Rotation-Axis Direction inputs. For 2D non-axisymmetric problems, you will specify only the Rotation-Axis Origin; the axis of rotation is the z-direction vector passing through the specified point. (The z direction is normal to the plane of your geometry so that rotation occurs in the plane.) For 2D axisymmetric problems, you will not define the axis: the rotation will always be about the x axis, with the origin at (0,0).

Defining Zone Motion

To define zone motion for a rotating or translating reference frame, select Moving Reference Frame in the Motion Type drop-down list and then set the appropriate parameters in the expanded portion of the panel.

To define zone motion for a sliding mesh, select Moving Mesh in the Motion Type drop-down list and then set the appropriate parameters in the expanded portion of the panel. See Section 9.5 for details.
Boundary Conditions

For problems that include linear, translational motion of the fluid zone, specify the Translational Velocity by setting the X, Y, and Z components. For problems that include rotational motion, specify the rotational Speed under Rotational Velocity. The rotation axis is defined as described above.

See Chapter 9 for details about modeling flows in moving reference frames.

Defining Radiation Parameters

If you are using the DO radiation model, you can specify whether or not the solid material participates in radiation using the Participates in Radiation option. See Section 11.3.16 for details.

6.19 Porous Media Conditions

The porous media model can be used for a wide variety of problems, including flows through packed beds, filter papers, perforated plates, flow distributors, and tube banks. When you use this model, you define a cell zone in which the porous media model is applied and the pressure loss in the flow is determined via your inputs as described in Section 6.19.2. Heat transfer through the medium can also be represented, subject to the assumption of thermal equilibrium between the medium and the fluid flow, as described in Section 6.19.3.

A 1D simplification of the porous media model, termed the “porous jump,” can be used to model a thin membrane with known velocity/pressure-drop characteristics. The porous jump model is applied to a face zone, not to a cell zone, and should be used (instead of the full porous media model) whenever possible because it is more robust and yields better convergence. See Section 6.22 for details.

You can find the following information about modeling porous media in this section:

- Section 6.19.1: Limitations and Assumptions of the Porous Media Model
- Section 6.19.2: Momentum Equations for Porous Media
6.19 Porous Media Conditions

- Section 6.19.3: Treatment of the Energy Equation in Porous Media
- Section 6.19.4: Treatment of Turbulence in Porous Media
- Section 6.19.5: Effect of Porosity on Transient Scalar Equations
- Section 6.19.6: User Inputs for Porous Media
- Section 6.19.7: Solution Strategies for Porous Media
- Section 6.19.8: Postprocessing for Porous Media

6.19.1 Limitations and Assumptions of the Porous Media Model

As described in the following sections, the porous media model incorporates an empirically determined flow resistance in a region of your model defined as “porous”. In essence, the porous media model is nothing more than an added momentum sink in the governing momentum equations. As such, the following modeling assumptions and limitations should be readily recognized:

- Since the volume blockage that is present physically is not represented in the model, FLUENT uses and reports a superficial velocity inside the porous medium, based on the volumetric flow rate, to ensure continuity of the velocity vectors across the porous medium interface.

- The effect of the porous medium on the turbulence field is only approximated. See Section 6.19.4 for details.

6.19.2 Momentum Equations for Porous Media

Porous media are modeled by the addition of a momentum source term to the standard fluid flow equations. The source term is composed of two parts, a viscous loss term (Darcy, the first term on the right-hand side of Equation 6.19-1), and an inertial loss term (the second term on the right-hand side of Equation 6.19-1):
Boundary Conditions

\[ S_i = -\left( \sum_{j=1}^{3} D_{ij} \mu v_j + \sum_{j=1}^{3} C_{ij} \frac{1}{2} \rho |v_j| v_j \right) \]  

(6.19-1)

where \( S_i \) is the source term for the \( i \)th \((x, y, \text{or} z)\) momentum equation, and \( D \) and \( C \) are prescribed matrices. This momentum sink contributes to the pressure gradient in the porous cell, creating a pressure drop that is proportional to the fluid velocity (or velocity squared) in the cell.

To recover the case of simple homogeneous porous media

\[ S_i = -\left( \frac{\mu}{\alpha} v_i + C_2 \frac{1}{2} \rho |v_i| v_i \right) \]  

(6.19-2)

where \( \alpha \) is the permeability and \( C_2 \) is the inertial resistance factor, simply specify \( D \) and \( C \) as diagonal matrices with \( 1/\alpha \) and \( C_2 \), respectively, on the diagonals (and zero for the other elements).

FLUENT also allows the source term to be modeled as a power law of the velocity magnitude:

\[ S_i = -C_0 |v|^{C_1} = -C_0 |v|^{(C_1 - 1)} v_i \]  

(6.19-3)

where \( C_0 \) and \( C_1 \) are user-defined empirical coefficients.

In the power-law model the pressure drop is isotropic and the units for \( C_0 \) are SI.

Darcy’s Law in Porous Media

In laminar flows through porous media, the pressure drop is typically proportional to velocity and the constant \( C_2 \) can be considered to be zero. Ignoring convective acceleration and diffusion, the porous media model then reduces to Darcy’s Law:

\[ \nabla p = -\frac{\mu}{\alpha} \mathbf{v} \]  

(6.19-4)
6.19 Porous Media Conditions

The pressure drop that FLUENT computes in each of the three \((x,y,z)\) coordinate directions within the porous region is then

\[
\Delta p_x = \sum_{j=1}^{3} \frac{\mu}{\alpha_{xj}} v_j \Delta n_x
\]

\[
\Delta p_y = \sum_{j=1}^{3} \frac{\mu}{\alpha_{yj}} v_j \Delta n_y \tag{6.19-5}
\]

\[
\Delta p_z = \sum_{j=1}^{3} \frac{\mu}{\alpha_{zj}} v_j \Delta n_z
\]

where \(1/\alpha_{ij}\) are the entries in the matrix \(D\) in Equation 6.19-1, \(v_j\) are the velocity components in the \(x, y,\) and \(z\) directions, and \(\Delta n_x, \Delta n_y,\) and \(\Delta n_z\) are the thickness of the medium in the \(x, y,\) and \(z\) directions.

Here, the thickness of the medium \((\Delta n_x, \Delta n_y,\) or \(\Delta n_z)\) is the actual thickness of the porous region in your model. Thus if the thicknesses used in your model differ from the actual thicknesses, you must make the adjustments in your inputs for \(1/\alpha_{ij}\).

Inertial Losses in Porous Media

At high flow velocities, the constant \(C_2\) in Equation 6.19-1 provides a correction for inertial losses in the porous medium. This constant can be viewed as a loss coefficient per unit length along the flow direction, thereby allowing the pressure drop to be specified as a function of dynamic head.

If you are modeling a perforated plate or tube bank, you can sometimes eliminate the permeability term and use the inertial loss term alone, yielding the following simplified form of the porous media equation:

\[
\nabla p = - \sum_{j=1}^{3} C_{2,ij} \left( \frac{1}{2} \rho v_j |v_j| \right) \tag{6.19-6}
\]
Boundary Conditions

or when written in terms of the pressure drop in the \( x, y, z \) directions:

\[
\begin{align*}
\Delta p_x & \approx \sum_{j=1}^{3} C_{2xj} \Delta n_x \frac{1}{2} \rho v_j |v_j| \\
\Delta p_y & \approx \sum_{j=1}^{3} C_{2yj} \Delta n_y \frac{1}{2} \rho v_j |v_j| \\
\Delta p_z & \approx \sum_{j=1}^{3} C_{2zj} \Delta n_z \frac{1}{2} \rho v_j |v_j|
\end{align*}
\]

(6.19-7)

Again, the thickness of the medium (\( \Delta n_x, \Delta n_y, \) or \( \Delta n_z \)) is the thickness you have defined in your model.

6.19.3 Treatment of the Energy Equation in Porous Media

FLUENT solves the standard energy transport equation (Equation 11.2-1) in porous media regions with modifications to the conduction flux and the transient terms only. In the porous medium, the conduction flux uses an effective conductivity and the transient term includes the thermal inertia of the solid region on the medium:

\[
\frac{\partial}{\partial t} \left( \gamma \rho_f E_f + (1 - \gamma) \rho_s E_s \right) + \nabla \cdot (\bar{v}(\rho_f E_f + p)) = \\
\nabla \cdot \left[ k_{\text{eff}} \nabla T - \left( \sum_i h_i J_i \right) + (\nabla \cdot \bar{v}) \right] + S^h_f
\]

(6.19-8)

where

- \( E_f \) = total fluid energy
- \( E_s \) = total solid medium energy
- \( \gamma \) = porosity of the medium
- \( k_{\text{eff}} \) = effective thermal conductivity of the medium
- \( S^h_f \) = fluid enthalpy source term
6.19 Porous Media Conditions

**Effective Conductivity in the Porous Medium**

The effective thermal conductivity in the porous medium, \( k_{\text{eff}} \), is computed by FLUENT as the volume average of the fluid conductivity and the solid conductivity:

\[
k_{\text{eff}} = \gamma k_f + (1 - \gamma) k_s
\]  

(6.19-9)

where

- \( \gamma \) = porosity of the medium
- \( k_f \) = fluid phase thermal conductivity (including the turbulent contribution, \( k_t \))
- \( k_s \) = solid medium thermal conductivity

When this simple volume averaging is not desirable, perhaps due to the effects of the medium geometry, the effective conductivity can be computed via a user-defined function. In all cases, however, the effective conductivity is treated as an isotropic property of the medium.

**6.19.4 Treatment of Turbulence in Porous Media**

FLUENT will, by default, solve the standard conservation equations for turbulence quantities in the porous medium. In this default approach, therefore, turbulence in the medium is treated as though the solid medium has no effect on the turbulence generation or dissipation rates. This assumption may be reasonable if the medium’s permeability is quite large and the geometric scale of the medium does not interact with the scale of the turbulent eddies. In other instances, however, you may want to suppress the effect of turbulence in the medium.

If you are using one of the \( k-\epsilon \) turbulence models, the \( k-\omega \) model, or the Spalart-Allmaras model, you can suppress the effect of turbulence in a porous region by setting the turbulent contribution to viscosity, \( \mu_t \), equal to zero. When you choose this option, FLUENT will transport the inlet turbulence quantities through the medium, but their effect on the fluid mixing and momentum will be ignored. In addition, the generation of turbulence will be set to zero in the medium. This modeling strategy is enabled by turning on the Laminar Zone option in the Fluid panel. Enabling this option implies that \( \mu_t \) is zero and that generation
Boundary Conditions

of turbulence will be zero in this porous zone. Disabling the option (the default) implies that turbulence will be computed in the porous region just as in the bulk fluid flow.

6.19.5 Effect of Porosity on Transient Scalar Equations

For transient porous media calculations, the effect of porosity on the time-derivative terms is accounted for in all scalar transport equations and the continuity equation. When the effect of porosity is accounted for, the time-derivative term becomes \( \frac{\partial}{\partial t}(\gamma \rho \phi) \), where \( \phi \) is the scalar quantity (\( k \), \( \epsilon \), etc.) and \( \gamma \) is the porosity.

The effect of porosity is enabled automatically for transient calculations, and the porosity is set to 1 by default.

6.19.6 User Inputs for Porous Media

When you are modeling a porous region, the only additional inputs for the problem setup are as follows. Optional inputs are indicated as such.

1. Define the porous zone.
2. Identify the fluid material flowing through the porous medium.
3. Set the viscous resistance coefficients (\( D_{ij} \) in Equation 6.19-1, or \( 1/\alpha \) in Equation 6.19-2) and the inertial resistance coefficients (\( C_{ij} \) in Equation 6.19-1, or \( C_2 \) in Equation 6.19-2), and define the direction vectors for which they apply. Alternatively, specify the coefficients for the power-law model.
4. Specify the porosity of the porous medium.
5. Select the material contained in the porous medium (required only for models that include heat transfer).
6. Set the volumetric heat generation rate in the solid portion of the porous medium (or any other sources, such as mass or momentum). (optional)
7. Set any fixed values for solution variables in the fluid region (optional).
8. Suppress the turbulent viscosity in the porous region, if appropriate.

9. Specify the rotation axis and/or zone motion, if relevant.

Methods for determining the resistance coefficients and/or permeability are presented below. If you choose to use the power-law approximation of the porous-media momentum source term, you will enter the coefficients $C_0$ and $C_1$ in Equation 6.19-3 instead of the resistance coefficients and flow direction.

You will set all parameters for the porous medium in the Fluid panel (Figure 6.19.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).

**Defining the Porous Zone**

As mentioned in Section 6.1, a porous zone is modeled as a special type of fluid zone. To indicate that the fluid zone is a porous region, enable the Porous Zone option in the Fluid panel. The panel will expand to show the porous media inputs (as shown in Figure 6.19.1).

**Defining the Fluid Passing Through the Porous Medium**

To define the fluid that passes through the porous medium, select the appropriate fluid in the Material Name drop-down list in the Fluid panel. If you want to check or modify the properties of the selected material, you can click Edit... to open the Material panel; this panel contains just the properties of the selected material, not the full contents of the standard Materials panel.

! If you are modeling species transport or multiphase flow, the Material Name list will not appear in the Fluid panel. For species calculations, the mixture material for all fluid/porous zones will be the material you specified in the Species Model panel. For multiphase flows, the materials are specified when you define the phases, as described in Section 20.6.8.
Figure 6.19.1: The Fluid Panel for a Porous Zone
Defining the Viscous and Inertial Resistance Coefficients

The viscous and inertial resistance coefficients are both defined in the same manner. The basic approach for defining the coefficients using a Cartesian coordinate system is to define one direction vector in 2D or two direction vectors in 3D, and then specify the viscous and/or inertial resistance coefficients in each direction. In 2D, the second direction, which is not explicitly defined, is normal to the plane defined by the specified direction vector and the z direction vector. In 3D, the third direction is normal to the plane defined by the two specified direction vectors. For a 3D problem, the second direction must be normal to the first. If you fail to specify two normal directions, the solver will ensure that they are normal by ignoring any component of the second direction that is in the first direction. You should therefore be certain that the first direction is correctly specified.

In 3D, it is also possible to define the coefficients using a conical (or cylindrical) coordinate system, as described below.

The procedure for defining resistance coefficients is as follows:

1. Define the direction vectors.
   - To use a Cartesian coordinate system, simply specify the Direction-1 Vector and, for 3D, the Direction-2 Vector. Each direction will be that of the vector from (0,0) or (0,0,0) to the specified (X,Y) or (X,Y,Z). (The unspecified direction will be determined as described above.)

   For some problems in which the principal axes of the porous medium are not aligned with the coordinate axes of the domain, you may not know a priori the direction vectors of the porous medium. In such cases, the plane tool in 3D (or the line tool in 2D) can help you to determine these direction vectors.

   (a) “Snap” the plane tool (or the line tool) onto the boundary of the porous region. (Follow the instructions in Section 24.6.1 or 24.5.1 for initializing the tool to a position on an existing surface.)
Boundary Conditions

(b) Rotate the axes of the tool appropriately until they are aligned with the porous medium.

(c) Once the axes are aligned, click on the Update From Plane Tool or Update From Line Tool button in the Fluid panel. FLUENT will automatically set the Direction-1 Vector to the direction of the red arrow of the tool, and (in 3D) the Direction-2 Vector to the direction of the green arrow.

- To use a conical coordinate system (e.g., for an annular, conical filter element), follow the steps below. This option is available only in 3D cases.

(a) Turn on the Conical option.

(b) Specify the Cone Axis Vector and Point on Cone Axis. The direction of the Cone Axis Vector will be that of the vector from (0,0,0) to the specified (X,Y,Z). The Point on Cone Axis will be used by FLUENT to transform the resistances to the Cartesian coordinate system.

(c) Set the Cone Half Angle (the angle between the cone’s axis and its surface, shown in Figure 6.19.2). To use a cylindrical coordinate system, set the Cone Half Angle to 0.

\[ \theta \]

Figure 6.19.2: Cone Half Angle

For some problems in which the axis of the conical filter element is not aligned with the coordinate axes of the domain, you may not know a priori the direction vector of the cone axis and coordinates of a point on the cone axis. In such cases, the plane tool can help you to determine the cone axis vector and point coordinates. One method is as follows:
(a) Select a boundary zone of the conical filter element that is normal to the cone axis vector in the drop-down list next to the Snap to Zone button.

(b) Click on the Snap to Zone button. FLUENT will automatically “snap” the plane tool onto the boundary. It will also set the Cone Axis Vector and the Point on Cone Axis. (Note that you will still have to set the Cone Half Angle yourself.)

An alternate method is as follows:

(a) “Snap” the plane tool onto the boundary of the porous region. (Follow the instructions in Section 24.6.1 for initializing the tool to a position on an existing surface.)

(b) Rotate and translate the axes of the tool appropriately until the red arrow of the tool is pointing in the direction of the cone axis vector and the origin of the tool is on the cone axis

(c) Once the axes and origin of the tool are aligned, click on the Update From Plane Tool button in the Fluid panel. FLUENT will automatically set the Cone Axis Vector and the Point on Cone Axis. (Note that you will still have to set the Cone Half Angle yourself.)

2. Under Viscous Resistance, specify the viscous resistance coefficient $1/\alpha$ in each direction, and under Inertial Resistance, specify the inertial resistance coefficient $C_2$ in each direction. (You will need to scroll down with the scroll bar to view these inputs.) If you are using the Conical specification method, Direction-1 is the cone axis direction, Direction-2 is the normal to the cone surface (radial ($r$) direction for a cylinder), and Direction-3 is the circumferential ($\theta$) direction.

In 3D there are three possible categories of coefficients, and in 2D there are two:

- In the isotropic case, the resistance coefficients in all directions are the same (e.g., a sponge). For an isotropic case, you must
Boundary Conditions

explicitly set the resistance coefficients in each direction to the same value.

- When (in 3D) the coefficients in two directions are the same and those in the third direction are different or (in 2D) the coefficients in the two directions are different, you must be careful to specify the coefficients properly for each direction. For example, if you had a porous region consisting of cylindrical straws with small holes in them positioned parallel to the flow direction, the flow would pass easily through the straws, but the flow in the other two directions (through the small holes) would be very little. If you had a plane of flat plates perpendicular to the flow direction, the flow would not pass through them at all; it would instead move in the other two directions.

- In 3D the third possible case is one in which all three coefficients are different. For example, if the porous region consisted of a plane of irregularly-spaced objects (e.g., pins), the movement of flow between the blockages would be different in each direction. You would therefore need to specify different coefficients in each direction.

(Note the comments in Section 6.19.7 when specifying anisotropic coefficients.)

Methods for deriving viscous and inertial loss coefficients are described below.

Deriving Porous Media Inputs Based on Superficial Velocity, Using a Known Pressure Loss

When you use the porous media model, you must keep in mind that the porous cells in FLUENT are 100% open, and that the values that you specify for $1/\alpha_{ij}$ and/or $C_{2,ij}$ must be based on this assumption. Suppose, however, that you know how the pressure drop varies with the velocity through the actual device, which is only partially open to flow. The following exercise is designed to show you how to compute a value for $C_2$ which is appropriate for the FLUENT model.
Consider a perforated plate which has 25% area open to flow. The pressure drop through the plate is known to be 0.5 times the dynamic head in the plate. The loss factor, $K_L$, defined as

$$\Delta p = K_L\left(\frac{1}{2} \rho v^2_{25\%\text{open}}\right) \quad (6.19-10)$$

is therefore 0.5, based on the actual fluid velocity in the plate, i.e., the velocity through the 25% open area. To compute an appropriate value for $C_2$, note that in the FLUENT model:

1. The velocity through the perforated plate assumes that the plate is 100% open.
2. The loss coefficient must be converted into dynamic head loss per unit length of the porous region.

Noting item 1, the first step is to compute an adjusted loss factor, $K'_L$, which would be based on the velocity of a 100% open area:

$$\Delta p = K'_L\left(\frac{1}{2} \rho v^2_{100\%\text{open}}\right) \quad (6.19-11)$$

or, noting that for the same flow rate, $v_{25\%\text{open}} = 4 \times v_{100\%\text{open}}$:

$$K'_L = K_L \times \frac{v^2_{25\%\text{open}}}{v^2_{100\%\text{open}}}$$

$$= 0.5 \times \left(\frac{4}{1}\right)^2$$

$$= 8 \quad (6.19-12)$$

The adjusted loss factor has a value of 8. Noting item 2, you must now convert this into a loss coefficient per unit thickness of the perforated plate. Assume that the plate has a thickness of 1.0 mm ($10^{-3}$ m). The inertial loss factor would then be
Boundary Conditions

\[ C_2 = \frac{K L'}{\text{thickness}} = \frac{8}{10^{-3}} = 8000 \text{ m}^{-1} \quad (6.19-13) \]

Note that, for anisotropic media, this information must be computed for each of the 2 (or 3) coordinate directions.

Using the Ergun Equation to Derive Porous Media Inputs for a Packed Bed

As a second example, consider the modeling of a packed bed. In turbulent flows, packed beds are modeled using both a permeability and an inertial loss coefficient. One technique for deriving the appropriate constants involves the use of the Ergun equation [62], a semi-empirical correlation applicable over a wide range of Reynolds numbers and for many types of packing:

\[ \frac{|\Delta p|}{L} = \frac{150 \mu}{D_p^2} \frac{(1 - \epsilon)^2}{\epsilon^3} v_\infty + \frac{1.75 \rho}{D_p} \frac{(1 - \epsilon)}{\epsilon^3} v_\infty^2 \quad (6.19-14) \]

When modeling laminar flow through a packed bed, the second term in the above equation may be dropped, resulting in the Blake-Kozeny equation [62]:

\[ \frac{|\Delta p|}{L} = \frac{150 \mu}{D_p^2} \frac{(1 - \epsilon)^2}{\epsilon^3} v_\infty \quad (6.19-15) \]

In these equations, \( \mu \) is the viscosity, \( D_p \) is the mean particle diameter, \( L \) is the bed depth, and \( \epsilon \) is the void fraction, defined as the volume of voids divided by the volume of the packed bed region. Comparing Equations 6.19-4 and 6.19-6 with 6.19-14, the permeability and inertial loss coefficient in each component direction may be identified as

\[ \alpha = \frac{D_p^2 \epsilon^3}{150 (1 - \epsilon)^2} \quad (6.19-16) \]
and

\[ C_2 = \frac{3.5 (1 - \epsilon)}{D_p \epsilon^3} \quad (6.19-17) \]

Using an Empirical Equation to Derive Porous Media Inputs for Turbulent Flow Through a Perforated Plate

As a third example we will take the equation of Van Winkle et al. [175, 215] and show how porous media inputs can be calculated for pressure loss through a perforated plate with square-edged holes.

The expression, which is claimed by the authors to apply for turbulent flow through square-edged holes on an equilateral triangular spacing, is

\[ \dot{m} = C A_f \sqrt{(2 \rho \Delta p)/(1 - (A_f/A_p)^2)} \quad (6.19-18) \]

where

\[ \dot{m} = \text{mass flow rate through the plate} \]
\[ A_f = \text{the free area or total area of the holes} \]
\[ A_p = \text{the area of the plate (solid and holes)} \]
\[ C = \text{a coefficient which has been tabulated for various Reynolds-number ranges and for various } D/t \]
\[ D/t = \text{the ratio of hole diameter to plate thickness} \]

for \( t/D > 1.6 \) and for \( \text{Re} > 4000 \) the coefficient \( C \) takes a value of approximately 0.98, where the Reynolds number is based on hole diameter and velocity in the holes.

Rearranging Equation 6.19-18, making use of the relationship

\[ \dot{m} = \rho v A_p \quad (6.19-19) \]

and dividing by the plate thickness, \( \Delta x = t \), we obtain

\[ \frac{\Delta p}{\Delta x} = \left( \frac{1}{2} \rho v^2 \right) \left( \frac{1}{C^2} \right) \frac{(A_p/A_f)^2 - 1}{t} \quad (6.19-20) \]
Boundary Conditions

where \( v \) is the superficial velocity (not the velocity in the holes). Comparing with Equation 6.19-6 it is seen that, for the direction normal to the plate, the constant \( C_2 \) can be calculated from

\[
C_2 = \frac{1}{C^2} \left( \frac{A_p}{A_f} \right)^2 - 1
\]  
(6.19-21)

Using Tabulated Data to Derive Porous Media Inputs for Laminar Flow Through a Fibrous Mat

Consider the problem of laminar flow through a mat or filter pad which is made up of randomly-oriented fibers of glass wool. As an alternative to the Blake-Kozeny equation (Equation 6.19-15) we might choose to employ tabulated experimental data. Such data is available for many types of fiber [98].

<table>
<thead>
<tr>
<th>volume fraction of solid material</th>
<th>dimensionless permeability ( B ) of glass wool</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.262</td>
<td>0.25</td>
</tr>
<tr>
<td>0.258</td>
<td>0.26</td>
</tr>
<tr>
<td>0.221</td>
<td>0.40</td>
</tr>
<tr>
<td>0.218</td>
<td>0.41</td>
</tr>
<tr>
<td>0.172</td>
<td>0.80</td>
</tr>
</tbody>
</table>

where \( B = \alpha/a^2 \) and \( a \) is the fiber diameter. \( \alpha \), for use in Equation 6.19-4, is easily computed for a given fiber diameter and volume fraction.

Using the Power-Law Model

If you choose to use the power-law approximation of the porous-media momentum source term (Equation 6.19-3), the only inputs required are the coefficients \( C_0 \) and \( C_1 \). Under Power Law Model in the Fluid panel, enter the values for \( C_0 \) and \( C_1 \). Note that the power-law model can be used in conjunction with the Darcy and inertia models.

\( C_0 \) must be in SI units, consistent with the value of \( C_1 \).
6.19 Porous Media Conditions

Defining Porosity

To define the porosity, scroll down below the resistance inputs in the Fluid panel, and set the Porosity under Fluid Porosity.

The porosity, \( \gamma \), is the volume fraction of fluid within the porous region (i.e., the open volume fraction of the medium). The porosity is used in prediction of heat transfer in the medium, as described in Section 6.19.3, and in the time-derivative term in the scalar transport equations for unsteady flow, as described in Section 6.19.5. It also impacts the calculation of reaction source terms and body forces in the medium. These sources will be proportional to the fluid volume in the medium. If you want to represent the medium as completely open (no effect of the solid medium), you should set the porosity equal to 1.0 (the default). When the porosity is equal to 1.0, the solid portion of the medium will have no impact on heat transfer or thermal/reaction source terms in the medium. Note that the porosity will never impact the flow velocity in the medium, as explained in Section 6.19.2. No matter what value you set for porosity, FLUENT predicts that the flow velocity is the superficial velocity in the medium.

Defining the Porous Material

If you choose to model heat transfer in the porous medium, you must specify the material contained in the porous medium.

To define the material contained in the porous medium, scroll down below the resistance inputs in the Fluid panel, and select the appropriate solid in the Solid Material Name drop-down list under Fluid Porosity. If you want to check or modify the properties of the selected material, you can click Edit... to open the Material panel; this panel contains just the properties of the selected material, not the full contents of the standard Materials panel.

Defining Sources

If you want to include effects of the heat generated by the porous medium in the energy equation, enable the Source Terms option and set a non-zero Energy source. The solver will compute the heat generated by the
porous region by multiplying this value by the total volume of the cells comprising the porous zone. You may also define sources of mass, momentum, turbulence, species, or other scalar quantities, as described in Section 6.27.

**Defining Fixed Values**

If you want to fix the value of one or more variables in the fluid region of the zone, rather than computing them during the calculation, you can do so by enabling the **Fixed Values** option. See Section 6.26 for details.

**Suppressing the Turbulent Viscosity in the Porous Region**

As discussed in Section 6.19.4, turbulence will be computed in the porous region just as in the bulk fluid flow. If you are using one of the $k-\varepsilon$ turbulence models, the Spalart-Allmaras model, or the $k-\omega$ model, and you want to suppress the effect of turbulence in a porous region (so that turbulence generation is zero in the porous zone), turn on the **Laminar Zone** option in the **Fluid** panel.

**Specifying the Rotation Axis and Defining Zone Motion**

Inputs for the rotation axis and zone motion are the same as for a standard fluid zone. See Section 6.17.1 for details.

**6.19.7 Solution Strategies for Porous Media**

In general, you can use the standard solution procedures and solution parameter settings when your FLUENT model includes porous media. You may find, however, that the rate of convergence slows when you define a porous region through which the pressure drop is relatively large in the flow direction (e.g., the permeability, $\alpha$, is low or the inertial factor, $C_2$, is large). This slow convergence can occur because the porous media pressure drop appears as a momentum source term—yielding a loss of diagonal dominance—in the matrix of equations solved. The best remedy for poor convergence of a problem involving a porous medium is to supply a good initial guess for the pressure drop across the medium. You can supply this guess by patching a value for the pressure in the
6.19 Porous Media Conditions

fluid cells upstream and/or downstream of the medium, as described in Section 22.13.2. It is important to recall, when patching the pressure, that the pressures you input should be defined as the gauge pressures used by the solver (i.e., relative to the operating pressure defined in the Operating Conditions panel).

Another possible way to deal with poor convergence is to temporarily disable the porous media model (by turning off the Porous Zone option in the Fluid panel) and obtain an initial flow field without the effect of the porous region. With the porous media model turned off, FLUENT will treat the porous zone as a fluid zone and calculate the flow field accordingly. Once an initial solution is obtained, or the calculation is proceeding steadily to convergence, you can enable the porous media model and continue the calculation with the porous region included. (This method is not recommended for porous media with high resistance.)

Simulations involving highly anisotropic porous media may, at times, pose convergence troubles. You can address these issues by limiting the anisotropy of the porous media coefficients \(1/\alpha_{ij}\) and \(C_{2,ij}\) to two or three orders of magnitude. Even if the medium’s resistance in one direction is infinite, you do not need to set the resistance in that direction to be greater than 1000 times the resistance in the primary flow direction.

6.19.8 Postprocessing for Porous Media

The impact of a porous region on the flow field can be determined by examining either velocity components or pressure values. Graphical plots (including XY plots and contour or vector plots) or alphanumeric reports of the following variables/functions may be of interest:

- **X, Y, Z Velocity** (in the Velocity... category)
- **Static Pressure** (in the Pressure... category)

These variables are contained in the specified categories of the variable selection drop-down list that appears in postprocessing panels.

Note that thermal reporting in the porous region does not reflect the solid medium properties. The reported density, heat capacity, conductivity,
and enthalpy in the porous region are those of the fluid and do not include the effect of the solid medium.

6.20 Fan Boundary Conditions

The fan model is a lumped parameter model that can be used to determine the impact of a fan with known characteristics upon some larger flow field. The fan boundary type allows you to input an empirical fan curve which governs the relationship between head (pressure rise) and flow rate (velocity) across a fan element. You can also specify radial and tangential components of the fan swirl velocity. The fan model does not provide an accurate description of the detailed flow through the fan blades. Instead, it predicts the amount of flow through the fan. Fans may be used in conjunction with other flow sources, or as the sole source of flow in a simulation. In the latter case, the system flow rate is determined by the balance between losses in the system and the fan curve.

FLUENT also provides a connection for a special user-defined fan model that updates the pressure jump function during the calculation. This feature is described in Section 6.23.

You can find the following information about modeling fans in this section:

- Section 6.20.1: Fan Equations
- Section 6.20.2: User Inputs for Fans
- Section 6.20.3: Postprocessing for Fans

6.20.1 Fan Equations

Modeling the Pressure Rise Across the Fan

A fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the velocity through the fan. The relationship may be a constant, a polynomial, piecewise-linear, or piecewise-polynomial function, or a user-defined function.

In the case of a polynomial, the relationship is of the form
6.20 Fan Boundary Conditions

\[ \Delta p = \sum_{n=1}^{N} f_n v^{n-1} \]  
\hspace{1cm} (6.20-1)

where \( \Delta p \) is the pressure jump, \( f_n \) are the pressure-jump polynomial coefficients, and \( v \) is the magnitude of the local fluid velocity normal to the fan.

! The velocity \( v \) can be either positive or negative. You must be careful to model the fan so that a pressure rise occurs for forward flow through the fan.

You can, optionally, use the mass-averaged velocity normal to the fan to determine a single pressure-jump value for all faces in the fan zone.

**Modeling the Fan Swirl Velocity**

For three-dimensional problems, the values of the convected tangential and radial velocity fields can be imposed on the fan surface to generate swirl. These velocities can be specified as functions of the radial distance from the fan center. The relationships may be constant or polynomial functions, or user-defined functions.

! You must use SI units for all fan swirl velocity inputs.

For the case of polynomial functions, the tangential and radial velocity components can be specified by the following equations:

\[ U_\theta = \sum_{n=-1}^{N} f_n r^n; -1 \leq N \leq 6 \]  
\hspace{1cm} (6.20-2)

\[ U_r = \sum_{n=-1}^{N} g_n r^n; -1 \leq N \leq 6 \]  
\hspace{1cm} (6.20-3)

where \( U_\theta \) and \( U_r \) are, respectively, the tangential and radial velocities on the fan surface in m/s, \( f_n \) and \( g_n \) are the tangential and radial velocity polynomial coefficients, and \( r \) is the distance to the fan center.
6.20.2 User Inputs for Fans

Once the fan zone has been identified (in the Boundary Conditions panel), you will set all modeling inputs for the fan in the Fan panel (Figure 6.20.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).

![Fan Panel](image)

Figure 6.20.1: The Fan Panel

Inputs for a fan are as follows:

1. Identify the fan zone.
2. Define the pressure jump across the fan.
3. Define the discrete phase boundary condition for the fan (for discrete phase calculations).

4. Define the swirl velocity, if desired (3D only).

**Identifying the Fan Zone**

Since the fan is considered to be infinitely thin, it must be modeled as the interface between cells, rather than a cell zone. Thus the fan zone is a type of internal face zone (where the faces are line segments in 2D or triangles/quadrilaterals in 3D). If, when you read your grid into FLUENT, the fan zone is identified as an interior zone, use the Boundary Conditions panel (as described in Section 6.1.3) to change the appropriate interior zone to a fan zone.

Once the interior zone has been changed to a fan zone, you can open the Fan panel and specify the pressure jump and, optionally, the swirl velocity.

**Defining the Pressure Jump**

To define the pressure jump, you will specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, a user-defined function, or a constant value. You should also check the Zone Average Direction vector to be sure that a pressure rise occurs for forward flow through the fan. The Zone Average Direction, calculated by the solver, is the face-averaged direction vector for the fan zone. If this vector is pointing in the direction you want the fan to blow, do not select Reverse Fan Direction; if it is pointing in the opposite direction, select Reverse Fan Direction.

**Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function**

Follow these steps to set a polynomial, piecewise-linear, or piecewise-polynomial function for the pressure jump:

1. Check that the Profile Specification of Pressure-Jump option is off in the Fan panel.
2. Choose polynomial, piecewise-linear, or piecewise-polynomial in the drop-down list to the right of Pressure-Jump. (If the function type you want is already selected, you can click on the Edit... button to open the panel where you will define the function.)

3. In the panel that appears for the definition of the Pressure-Jump function (e.g., Figure 6.20.2), enter the appropriate values. These profile input panels are used the same way as the profile input panels for temperature-dependent properties. See Section 7.1.3 to find out how to use them.

![Figure 6.20.2: Polynomial Profile Panel for Pressure-Jump Definition](image)

4. Set any of the optional parameters described below. (optional)

When you define the pressure jump using any of these types of functions, you can choose to limit the minimum and maximum velocity magnitudes used to calculate the pressure jump. Turning on the Limit Polynomial Velocity Range option allows you to specify minimum and maximum velocity magnitudes. If the calculated normal velocity magnitude exceeds the Max Velocity Magnitude that you specify, the Max Velocity Magnitude value will be used instead. Similarly, if the calculated velocity is less than the specified Min Velocity Magnitude, the Min Velocity Magnitude will be substituted for the calculated value.
You also have the option to use the mass-averaged velocity normal to the fan to determine a single pressure-jump value for all faces in the fan zone. Turning on Calculate Pressure-Jump from Average Conditions enables this option.

**Constant Value**

To define a constant pressure jump, follow these steps:

1. Turn off the Profile Specification of Pressure-Jump option in the Fan panel.
2. Choose constant in the drop-down list to the right of Pressure-Jump.
3. Enter the value for $\Delta p$ in the Pressure-Jump field.

You can follow the procedure below, if it is more convenient:

1. Turn on the Profile Specification of Pressure-Jump option.
2. Select constant in the drop-down list below Pressure Jump Profile, and enter the value for $\Delta p$ in the Pressure Jump Profile field.

**User-Defined Function or Boundary Profile**

For a user-defined pressure-jump function or a function defined in a boundary profile file, you will follow these steps:

1. Turn on the Profile Specification of Pressure-Jump option.
2. Choose the appropriate function in the drop-down list below Pressure Jump Profile.

See the separate UDF Manual for information about user-defined functions, and Section 6.25 for details about boundary profile files.
Boundary Conditions

Example: Determining the Pressure Jump Function

This example shows you how to determine the function for the pressure jump. Consider the simple two-dimensional duct flow illustrated in Figure 6.20.3. Air at constant density enters the 2.0 m × 0.4 m duct with a velocity of 15 m/s. Centered in the duct is a fan.

![Diagram of a Fan Located In a 2D Duct](image)

Figure 6.20.3: A Fan Located In a 2D Duct

Assume that the fan characteristics are as follows when the fan is operating at 2000 rpm:

<table>
<thead>
<tr>
<th>$Q$ (m$^3$/s)</th>
<th>$\Delta p$ (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>0.0</td>
</tr>
<tr>
<td>20</td>
<td>175</td>
</tr>
<tr>
<td>15</td>
<td>350</td>
</tr>
<tr>
<td>10</td>
<td>525</td>
</tr>
<tr>
<td>5</td>
<td>700</td>
</tr>
<tr>
<td>0</td>
<td>875</td>
</tr>
</tbody>
</table>

where $Q$ is the flow through the fan and $\Delta p$ is the pressure rise across the fan. The fan characteristics in this example follow a simple linear relationship between pressure rise and flow rate. To convert this into a relationship between pressure rise and velocity, the cross-sectional area of the fan must be known. In this example, assuming that the duct is 1.0 m deep, this area is 0.4 m$^2$, so that the corresponding velocity values are as follows:
6.20 Fan Boundary Conditions

<table>
<thead>
<tr>
<th>( v ) (m/s)</th>
<th>( \Delta p ) (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>62.5</td>
<td>0.0</td>
</tr>
<tr>
<td>50.0</td>
<td>175</td>
</tr>
<tr>
<td>37.5</td>
<td>350</td>
</tr>
<tr>
<td>25.0</td>
<td>525</td>
</tr>
<tr>
<td>12.5</td>
<td>700</td>
</tr>
<tr>
<td>0</td>
<td>875</td>
</tr>
</tbody>
</table>

The polynomial form of this relationship is the following equation for a line:

\[
\Delta p = 875 - 14v
\]  
\( (6.20-4) \)

**Defining Discrete Phase Boundary Conditions for the Fan**

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the fan. See Section 19.10 for details.

**Defining the Fan Swirl Velocity**

If you want to set tangential and radial velocity fields on the fan surface to generate swirl in a 3D problem, follow these steps:

1. Turn on the Swirl-Velocity Specification option in the Fan panel.
2. Specify the fan’s axis of rotation by defining the axis origin (Fan Origin) and direction vector (Fan Axis).
3. Set the value for the radius of the fan’s hub (Fan Hub Radius). The default is \( 1 \times 10^{-6} \) to avoid division by zero in the polynomial.
4. Set the tangential and radial velocity functions as polynomial functions of radial distance, constant values, or user-defined functions.

! You must use SI units for all fan swirl velocity inputs.
**Boundary Conditions**

**Polynomial Function**

To define a polynomial function for tangential or radial velocity, follow the steps below:

1. Check that the Profile Specification of Tangential Velocity or Profile Specification of Radial Velocity option is off in the Fan panel.

2. Enter the coefficients $f_n$ in Equation 6.20-2 or $g_n$ in Equation 6.20-3 in the Tangential- or Radial-Velocity Polynomial Coefficients field. Enter $f_{-1}$ first, then $f_0$, etc. Separate each coefficient by a blank space. Remember that the first coefficient is for $\frac{1}{r}$.

**Constant Value**

To define a constant tangential or radial velocity, the steps are as follows:

1. Turn on the Profile Specification of Tangential Velocity or Profile Specification of Radial Velocity option in the Fan panel.

2. Select constant in the drop-down list under Tangential or Radial Velocity Profile.

3. Enter the value for $U_\theta$ or $U_r$ in the Tangential or Radial Velocity Profile field.

You can follow the procedure below, if it is more convenient:

1. Turn off the Profile Specification of Tangential Velocity or Profile Specification of Radial Velocity option in the Fan panel.

2. Enter the value for $U_\theta$ or $U_r$ in the Tangential- or Radial-Velocity Polynomial Coefficients field.
User-Defined Function or Boundary Profile

For a user-defined tangential or radial velocity function or a function contained in a boundary profile file, follow the procedure below:

1. Turn on the Profile Specification of Tangential or Radial Velocity option.

2. Choose the appropriate function from the drop-down list under Tangential or Radial Velocity Profile.

See the separate UDF Manual for information about user-defined functions, and Section 6.25 for details about boundary profile files.

6.20.3 Postprocessing for Fans

Reporting the Pressure Rise Through the Fan

You can use the Surface Integrals panel to report the pressure rise through the fan, as described in Section 26.5. There are two steps to this procedure:

1. Create a surface on each side of the fan zone. Use the Transform Surface panel (as described in Section 24.10) to translate the fan zone slightly upstream and slightly downstream to create two new surfaces.

2. In the Surface Integrals panel, report the average Static Pressure just upstream and just downstream of the fan. You can then calculate the pressure rise through the fan.

Graphical Plots

Graphical reports of interest with fans are as follows:

- Contours or profiles of Static Pressure and Static Temperature
- XY plots of Static Pressure and Static Temperature vs. position
Chapter 25 explains how to generate graphical displays of data. When generating these plots, be sure to turn off the display of node values so that you can see the different values on each side of the fan. (If you display node values, the cell values on either side of the fan will be averaged to obtain a node value, and you will not see, for example, the pressure jump across the fan.)

6.21 Radiator Boundary Conditions

A lumped-parameter model for a heat exchange element (for example, a radiator or condenser), is available in FLUENT. The radiator boundary type allows you to specify both the pressure drop and heat transfer coefficient as functions of the velocity normal to the radiator.

You can find the following information about modeling radiators in this section:

- Section 6.21.1: Radiator Equations
- Section 6.21.2: User Inputs for Radiators
- Section 6.21.3: Postprocessing for Radiators

A more detailed heat exchanger model is also available in FLUENT. See Section 6.24 for details.

6.21.1 Radiator Equations

Modeling the Pressure Loss Through a Radiator

A radiator is considered to be infinitely thin, and the pressure drop through the radiator is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient which you supply. That is, the pressure drop, $\Delta p$, varies with the normal component of velocity through the radiator, $v$, as follows:

$$\Delta p = k_L \frac{1}{2} \rho v^2$$

(6.21-1)
where $\rho$ is the fluid density, and $k_L$ is the nondimensional loss coefficient, which can be specified as a constant or as a polynomial, piecewise-linear, or piecewise-polynomial function.

In the case of a polynomial, the relationship is of the form

$$k_L = \sum_{n=1}^{N} r_n v^{n-1} \quad (6.21-2)$$

where $r_n$ are polynomial coefficients and $v$ is the magnitude of the local fluid velocity normal to the radiator.

**Modeling the Heat Transfer Through a Radiator**

The heat flux from the radiator to the surrounding fluid is given as

$$q = h(T_{HX} - T_{exit}) \quad (6.21-3)$$

where $q$ is the heat flux, $T_{HX}$ is the heat-exchanger (radiator) temperature, and $T_{exit}$ is the exit fluid temperature. The convective heat transfer coefficient, $h$, can be specified as a constant or as a polynomial, piecewise-linear, or piecewise-polynomial function.

For a polynomial, the relationship is of the form

$$h = \sum_{n=0}^{N} h_n v^n; 0 \leq N \leq 7 \quad (6.21-4)$$

where $h_n$ are polynomial coefficients and $v$ is the magnitude of the local fluid velocity normal to the radiator in m/s.

Either the actual heat flux ($q$) or the heat transfer coefficient and radiator temperature ($h, T_{HX}$) may be specified. $q$ (either the entered value or the value calculated using Equation 6.21-3) is integrated over the radiator surface area.
**Boundary Conditions**

*Calculating the Heat Transfer Coefficient*

To model the thermal behavior of the radiator, you must supply an expression for the heat transfer coefficient, $h$, as a function of the fluid velocity through the radiator, $v$. To obtain this expression, consider the heat balance equation:

$$ q = \frac{\dot{m}c_p \Delta T}{A} = h(T_{HX} - T_{exit}) \quad (6.21-5) $$

where

- $q$ = heat flux (W/m$^2$)
- $\dot{m}$ = fluid mass flow rate (kg/s)
- $c_p$ = specific heat capacity of fluid (J/kg-K)
- $h$ = empirical heat transfer coefficient (W/m$^2$-K)
- $T_{exit}$ = exit fluid temperature (K)
- $T_{HX}$ = heat exchanger (e.g., water-side) temperature (K)
- $A$ = heat exchanger frontal area (m$^2$)

Equation 6.21-5 can be rewritten as

$$ q = \frac{\dot{m}c_p(T_{exit} - T_{inlet})}{A} = h(T_{HX} - T_{exit}) \quad (6.21-6) $$

The heat transfer coefficient, $h$, can therefore be computed as

$$ h = \frac{\dot{m}c_p(T_{exit} - T_{inlet})}{A(T_{HX} - T_{exit})} \quad (6.21-7) $$

or, in terms of the fluid velocity,

$$ h = \frac{\rho v c_p(T_{exit} - T_{inlet})}{T_{HX} - T_{exit}} \quad (6.21-8) $$
6.21 Radiator Boundary Conditions

6.21.2 User Inputs for Radiators

Once the radiator zone has been identified (in the Boundary Conditions panel), you will set all modeling inputs for the radiator in the Radiator panel (Figure 6.21.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).

![Figure 6.21.1: The Radiator Panel](image)

The inputs for a radiator are as follows:

1. Identify the radiator zone.
2. Define the pressure loss coefficient.
3. Define either the heat flux or the heat transfer coefficient and radiator temperature.
4. Define the discrete phase boundary condition for the radiator (for discrete phase calculations).
Boundary Conditions

Identifying the Radiator Zone

Since the radiator is considered to be infinitely thin, it must be modeled as the interface between cells, rather than a cell zone. Thus the radiator zone is a type of internal face zone (where the faces are line segments in 2D or triangles/quadrilaterals in 3D). If, when you read your grid into FLUENT, the radiator zone is identified as an interior zone, use the Boundary Conditions panel (as described in Section 6.1.3) to change the appropriate interior zone to a radiator zone.

Once the interior zone has been changed to a radiator zone, you can open the Radiator panel and specify the loss coefficient and heat flux information.

Defining the Pressure Loss Coefficient Function

To define the pressure loss coefficient $k_L$ you can specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, or a constant value.

Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function

Follow these steps to set a polynomial, piecewise-linear, or piecewise-polynomial function for the pressure loss coefficient:

1. Choose polynomial, piecewise-linear, or piecewise-polynomial in the drop-down list to the right of Loss-Coefficient. (If the function type you want is already selected, you can click on the Edit... button to open the panel where you will define the function.)

2. In the panel that appears for the definition of the Loss-Coefficient function (e.g., Figure 6.21.2), enter the appropriate values. These profile input panels are used the same way as the profile input panels for temperature-dependent properties. See Section 7.1.3 to find out how to use them.
6.21 Radiator Boundary Conditions

Figure 6.21.2: Polynomial Profile Panel for Loss-Coefficient Definition

**Constant Value**

To define a constant loss coefficient, follow these steps:

1. Choose constant in the Loss-Coefficient drop-down list.
2. Enter the value for \( k_L \) in the Loss-Coefficient field.

**Example: Calculating the Loss Coefficient**

This example shows you how to determine the loss coefficient function. Consider the simple two-dimensional duct flow of air through a water-cooled radiator, shown in Figure 6.21.3.

The radiator characteristics must be known empirically. For this case, assume that the radiator to be modeled yields the test data shown in Table 6.21.1, which was taken with a waterside flow rate of 7 kg/min and an inlet water temperature of 400.0 K. To compute the loss coefficient, it is helpful to construct a table with values of the dynamic head, \( \frac{1}{2} \rho v^2 \), as a function of pressure drop, \( \Delta p \), and the ratio of these two values, \( k_L \) (from Equation 6.21-1). (The air density, defined in Figure 6.21.3, is 1.0 kg/m\(^3\)). The reduced data are shown in Table 6.21.2 below.
Boundary Conditions

Figure 6.21.3: A Simple Duct with a Radiator

Table 6.21.1: Airside Radiator Data

<table>
<thead>
<tr>
<th>Velocity (m/s)</th>
<th>Inlet Temp (K)</th>
<th>Exit Temp (K)</th>
<th>Pressure Drop (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.0</td>
<td>300.0</td>
<td>330.0</td>
<td>75.0</td>
</tr>
<tr>
<td>10.0</td>
<td>300.0</td>
<td>322.5</td>
<td>250.0</td>
</tr>
<tr>
<td>15.0</td>
<td>300.0</td>
<td>320.0</td>
<td>450.0</td>
</tr>
</tbody>
</table>

Table 6.21.2: Reduced Radiator Data

<table>
<thead>
<tr>
<th>v (m/s)</th>
<th>$\frac{1}{2} \rho v^2$ (Pa)</th>
<th>$\Delta p$ (Pa)</th>
<th>$k_L$</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.0</td>
<td>12.5</td>
<td>75.0</td>
<td>6.0</td>
</tr>
<tr>
<td>10.0</td>
<td>50.0</td>
<td>250.0</td>
<td>5.0</td>
</tr>
<tr>
<td>15.0</td>
<td>112.5</td>
<td>450.0</td>
<td>4.0</td>
</tr>
</tbody>
</table>
The loss coefficient is a linear function of the velocity, decreasing as the velocity increases. The form of this relationship is

\[ k_L = 7.0 - 0.2v \]  \hspace{1cm} (6.21-9)

where \( v \) is now the absolute value of the velocity through the radiator.

**Defining the Heat Flux Parameters**

As mentioned in Section 6.21.1, you can either define the actual heat flux \((q)\) in the Heat Flux field, or set the heat transfer coefficient and radiator temperature \((h, T_{HX})\). All inputs are in the Radiator panel.

To define the actual heat flux, specify a Temperature of 0, and set the constant Heat Flux value.

To define the radiator temperature, enter the value for \( T_{HX} \) in the Temperature field. To define the heat transfer coefficient, you can specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, or a constant value.

**Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function**

Follow these steps to set a polynomial, piecewise-linear, or piecewise-polynomial function for the heat transfer coefficient:

1. Choose polynomial, piecewise-linear, or piecewise-polynomial in the drop-down list to the right of Heat-Transfer-Coefficient. (If the function type you want is already selected, you can click on the Edit... button to open the panel where you will define the function.)

2. In the panel that appears for the definition of the Heat-Transfer-Coefficient function, enter the appropriate values. These profile input panels are used the same way as the profile input panels for temperature-dependent properties. See Section 7.1.3 to find out how to use them.
Boundary Conditions

Constant Value

To define a constant heat transfer coefficient, follow these steps:

1. Choose constant in the Heat-Transfer-Coefficient drop-down list.
2. Enter the value for $h$ in the Heat-Transfer-Coefficient field.

Example: Determining the Heat Transfer Coefficient Function

This example shows you how to determine the function for the heat transfer coefficient. Consider the simple two-dimensional duct flow of air through a water-cooled radiator, shown in Figure 6.21.3.

The data supplied in Table 6.21.1 along with values for the air density (1.0 kg/m$^3$) and specific heat (1000 J/kg-K) can be used to obtain the following values for the heat transfer coefficient $h$:

<table>
<thead>
<tr>
<th>Velocity (m/s)</th>
<th>$h$ (W/m$^2$-K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.0</td>
<td>2142.9</td>
</tr>
<tr>
<td>10.0</td>
<td>2903.2</td>
</tr>
<tr>
<td>15.0</td>
<td>3750.0</td>
</tr>
</tbody>
</table>

The heat transfer coefficient obeys a second-order polynomial relationship (fit to the points in the table above) with the velocity, which is of the form

$$h = 1469.1 + 126.11v + 1.73v^2$$  \hspace{1cm} (6.21-10)

Note that the velocity $v$ is assumed to be the absolute value of the velocity passing through the radiator.

Defining Discrete Phase Boundary Conditions for the Radiator

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the radiator. See Section 19.10 for details.
6.21 Radiator Boundary Conditions

6.21.3 Postprocessing for Radiators

Reporting the Radiator Pressure Drop

You can use the Surface Integrals panel to report the pressure drop across the radiator, as described in Section 26.5. There are two steps to this procedure:

1. Create a surface on each side of the radiator zone. Use the Transform Surface panel (as described in Section 24.10) to translate the radiator zone slightly upstream and slightly downstream to create two new surfaces.

2. In the Surface Integrals panel, report the average Static Pressure just upstream and just downstream of the radiator. You can then calculate the pressure drop across the radiator.

To check this value against the expected value based on Equation 6.21-1, you can use the Surface Integrals panel to report the average normal velocity through the radiator. (If the radiator is not aligned with the x, y, or z axis, you will need to use the Custom Field Function Calculator panel to generate a function for the velocity normal to the radiator.) Once you have the average normal velocity, you can use Equation 6.21-2 to determine the loss coefficient and then Equation 6.21-1 to calculate the expected pressure loss.

Reporting Heat Transfer in the Radiator

To determine the temperature rise across the radiator, follow the procedure outlined above for the pressure drop to generate surfaces upstream and downstream of the radiator. Then use the Surface Integrals panel (as for the pressure drop report) to report the average Static Temperature on each surface. You can then calculate the temperature rise across the radiator.

Graphical Plots

Graphical reports of interest with radiators are as follows:
Boundary Conditions

- Contours or profiles of Static Pressure and Static Temperature
- XY plots of Static Pressure and Static Temperature vs. position.

Chapter 25 explains how to generate graphical displays of data.

When generating these plots, be sure to turn off the display of node values so that you can see the different values on each side of the radiator. (If you display node values, the cell values on either side of the radiator will be averaged to obtain a node value, and you will not see, for example, the pressure loss across the radiator.)

6.22 Porous Jump Boundary Conditions

Porous jump conditions are used to model a thin “membrane” that has known velocity/pressure-drop characteristics. It is essentially a 1D simplification of the porous media model available for cell zones. Examples of uses for the porous jump condition include modeling pressure drops through screens and filters, and modeling radiators when you are not concerned with heat transfer. This simpler model should be used whenever possible (instead of the full porous media model) because it is more robust and yields better convergence.

The thin porous medium has a finite thickness over which the pressure change is defined as a combination of Darcy’s Law and an additional inertial loss term:

$$\Delta p = - \left( \frac{\mu}{\alpha} v + C_2 \frac{1}{2} \rho v^2 \right) \Delta m \quad (6.22-1)$$

where \(\mu\) is the laminar fluid viscosity, \(\alpha\) is the permeability of the medium, \(C_2\) is the pressure-jump coefficient, \(v\) is the velocity normal to the porous face, and \(\Delta m\) is the thickness of the medium. Appropriate values for \(\alpha\) and \(C_2\) can be calculated using the techniques described in Section 6.19.6.

User Inputs for the Porous Jump Model

Once the porous jump zone has been identified (in the Boundary Conditions panel), you will set all modeling inputs for the porous jump in the
Porous Jump panel (Figure 6.22.1), which is opened from the Boundary Conditions panel (as described in Section 6.1.4).

Identifying the Porous Jump Zone

Since the porous jump model is a 1D simplification of the porous media model, the porous-jump zone must be modeled as the interface between cells, rather than a cell zone. Thus the porous-jump zone is a type of internal face zone (where the faces are line segments in 2D or triangles/quadrilaterals in 3D). If the porous-jump zone is not identified as
Boundary Conditions

such by default when you read in the grid (i.e., if it is identified as another type of internal face zone), you can use the Boundary Conditions panel to change the appropriate face zone to a porous-jump zone.

Define → Boundary Conditions...

The procedure for changing a zone’s type is described in Section 6.1.3. Once the zone has been changed to a porous jump, you can open the Porous Jump panel (as described in Section 6.1.4) and specify the porous jump parameters listed above.

Defining Discrete Phase Boundary Conditions for the Porous Jump

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the porous jump. See Section 19.10 for details.

Postprocessing for the Porous Jump

Postprocessing suggestions for a problem that includes a porous jump are the same as for porous media problems. See Section 6.19.8.

6.23 User-Defined Fan Model

The user-defined fan model in FLUENT allows you to periodically regenerate a profile file that can be used to specify the characteristics of a fan, including pressure jump across the fan, and radial and swirling components of velocity generated by the fan.

For example, consider the calculation of the pressure jump across the fan. You can, through the standard interface, input a constant for the pressure jump, specify a polynomial that describes the pressure jump as a function of axial velocity through the fan, or use a profile file that describes the pressure jump as a function of the axial velocity or location at the fan face. If you use a profile file, the same profile will be used consistently throughout the course of the solution. Suppose, however, that you want to change the profile as the flow field develops. This would require a periodic update to the profile file itself, based upon some instructions that you supply. The user-defined fan model is designed to help you do this.
To use this model, you need to generate an executable that reads a fan profile file that is written by FLUENT, and writes out a modified one, which FLUENT will then read. The source code for this executable can be written in any programming language (Fortran or C, for example). Your program will be called and executed automatically, according to inputs that you supply through the standard interface.

6.23.1 Steps for Using the User-Defined Fan Model

To make use of the user-defined fan model, follow the steps below.

1. In your model, identify one or more interior faces to represent one or more fan zones.

   Define ➔ Boundary Conditions...

2. Input the name of your executable and the instructions for reading and writing profile files in the User-Defined Fan Model panel.

   Define ➔ User-Defined ➔ Fan Model...

3. Initialize the flow field and the profile files.

4. Enter the fan parameters using the standard Fan panel (opened from the Boundary Conditions panel).

5. Perform the calculation.

6.23.2 Example of a User-Defined Fan

Usage of the user-defined fan model is best demonstrated by an example. With this in mind, consider the domain shown in Figure 6.23.1. An inlet supplies air at 10 m/s to a cylindrical region, 1.25 m long and 0.2 m in diameter, surrounded by a symmetry boundary. At the center of the flow domain is a circular fan. A pressure outlet boundary is at the downstream end.

Solving this problem with the user-defined fan model will cause FLUENT to periodically write out a radial profile file with the current solution variables at the fan face. These variables (static pressure, pressure jump,
Boundary Conditions

Figure 6.23.1: The Inlet, Fan, and Pressure Outlet Zones for a Circular Fan Operating in a Cylindrical Domain

axial, radial, and swirling (tangential) velocity components will represent averaged quantities over annular sections of the fan. The sizes of the annular regions are determined by the size of the fan and the number of radial points to be used in the profiles.

Once the profile file is written, FLUENT will invoke an executable, which will perform the following tasks:

1. Read the profile file containing the current flow conditions at the fan.
2. Perform a calculation to compute new values for the pressure jump, radial velocity, and swirl velocity for the fan.
3. Write a new profile file that contains the results of these calculations.

FLUENT will then read the new profile file and continue with the calculation.
Setting the User-Defined Fan Parameters

Specification of the parameters for the user-defined fan begins in the User-Defined Fan Model panel (Figure 6.23.2).

In this panel, you can select the fan zone(s) on which your executable will operate under Fan Zones. In this example, there is only one fan, fan-8. If you have multiple fan zones in a simulation, for which you have different profile specifications, you can select them all at this point. Your executable will be able to differentiate between the fan zones because the zone ID for each fan is included in the solution profile file. The executable will be invoked once for each zone, and separate profile files will be written for each.

The executable file will be called on to update the profile file periodically, based on the input for the Iteration Update Interval. An input of 10, as
Boundary Conditions

shown in the panel, means that the fan executable in this example will act every 10 iterations to modify the profile file.

The number of points in the profile file to be written by FLUENT is entered under Output Profile Points. This profile file can have the same or a different number of points as the one that is written by the external executable.

Finally, the name of the executable should be entered under External Command Name. In the current example, the name of the executable is fantest.

If the executable is not located in your working directory, then you must type the complete path to the executable.

Sample User-Defined Fan Program

The executable file will be built from the Fortran program, fantest.f, which is shown below. You can obtain a copy of this subroutine and the two that it calls (to read and write profile files) by contacting your Fluent technical support engineer.

```
c  c   This program is invoked at intervals by FLUENT to
 c   read a profile-format file that contains radially
 c   averaged data at a fan face, compute new pressure-jump
 c   and swirl-velocity components, and write a new profile
 c   file that will subsequently be read by FLUENT to
 c   update the fan conditions.
 c
 c Usage: fantest < input_profile > output_profile

 integer nmax
 parameter (nmax = 900)
 integer inp ! input: number of profile points
 integer iptype ! input: profile type (0=radial, 1=point)
 real ir(nmax) ! input: radial positions
 real ip(nmax) ! input: pressure
 real idp(nmax) ! input: pressure-jump
 real iva(nmax) ! input: axial velocity
```

© Fluent Inc. November 28, 2001
After the variable declarations, which have comments on the right, the subroutine `rfanprof` is called to read the profile file, and pass the current values of the relevant variables (as defined in the declaration list) to `fantest`. A loop is done on the number of points in the profile to compute new values for:

- `idp`, the pressure jump across the fan, which in this example is a function of the axial velocity, `iva`
- `ivt`, the swirling or tangential velocity, which in this example is proportional to the radial position, `ir`
- `ivr`, the radial velocity, which in this example is set to zero

After the loop, a new profile is written by the subroutine `wfanprof`, shown below. (For more information on profile file formats, see Section 6.25.2.)

```fortran
subroutine wfanprof(unit, zoneid, ptype, n, dp, vr, vt)
```

Boundary Conditions

c writes a FLUENT profile file for input by the
c user fan model
c
integer unit ! output unit number
character*80 zoneid
integer ptype ! profile type (0=radial, 1=point)
integer n ! number of points
real r(n) ! radial position
real dp(n) ! pressure jump
real vr(n) ! radial velocity
real vt(n) ! tangential velocity
character*6 typenam

if (ptype.eq.0) then
   typenam = 'radial'
else
   typenam = 'point'
endif

write(unit,*)(((', zoneid(1:index(zoneid,'\0')-1), ',',
$   typenam, n, ')'),

write(unit,*)(r'
write(unit,100) r
write(unit,*) ')

write(unit,*)(pressure-jump'
write(unit,100) dp
write(unit,*) ')

write(unit,*)(radial-velocity'
write(unit,100) vr
write(unit,*) ')

write(unit,*)(tangential-velocity'
write(unit,100) vt
write(unit,*) ')

100 format(5(e15.8,1x))
return
end
6.23 User-Defined Fan Model

This subroutine will write a profile file in either radial or point format, based on your input for the integer \texttt{ptype}. (See Section 6.25 for more details on the types of profile files that are available.) The names that you use for the various profiles are arbitrary. Once you have initialized the profile files, the names you use in \texttt{wfanprof} will appear as profile names in the \texttt{Fan} panel.

\textit{Initializing the Flow Field and Profile Files}

The next step in the setup of the user-defined fan is to initialize (create) the profile files that will be used. To do this, first initialize the flow field with the \texttt{Solution Initialization} panel (using the velocity inlet conditions, for example), and then type the command \texttt{(update-user-fans)} in the console window. (The parentheses are part of the command, and must be typed in.)

This will create the profile names that are given in the subroutine \texttt{wfanprof}.

\textit{Selecting the Profiles}

Once the profile names have been established, you will need to visit the \texttt{Fan} panel (Figure 6.23.3) to complete the problem setup. (See Section 6.20 for general information on using the \texttt{Fan} panel.)

At this time, the \texttt{Fan Axis}, \texttt{Fan Origin}, and \texttt{Fan Hub Radius} can be entered, along with the choice of profiles for the calculation of pressure jump, tangential velocity, and radial velocity. With the profile options enabled, you can select the names of the profiles from the drop-down lists. In the panel above, the selected profiles are named \texttt{fan-8-pressure-jump}, \texttt{fan-8-tangential-velocity}, and \texttt{fan-8-radial-velocity}, corresponding to the names that were used in the subroutine \texttt{wfanprof}.

\textit{Performing the Calculation}

The solution is now ready to run. As it begins to converge, the report in the console window shows that the profile files are being written and read every 10 iterations:
Boundary Conditions

Figure 6.23.3: The Fan Panel
6.23 User-Defined Fan Model

The file \texttt{fan-8-out.prof} is written out by FLUENT and read by the executable \texttt{fantest}. It contains values for pressure, pressure jump, axial velocity, radial velocity, and tangential velocity at 20 radial locations at the site of the fan. The file \texttt{fan-8-in.prof} is generated by \texttt{fantest} and contains updated values for pressure jump and radial and tangential velocity only. It is therefore a smaller file than \texttt{fan-8-out.prof}. The prefix for these files takes its name from the fan zone with which the profiles are associated. An example of the profile file \texttt{fan-8-in.prof} is shown below. This represents the last profile file to be written by \texttt{fantest} during the convergence history.

```plaintext
((fan-8 radial 10)
 (r
  0.24295786E-01 0.33130988E-01 0.41966137E-01 0.50801374E-01 0.59635671E-01
  0.68549784E-01 0.77307909E-01 0.86142287E-01 0.94963484E-01 0.95353782E-01
 )
 (pressure-jump
  1.7612E-01 7.4618E-02 2.5194E-01 2.4538E-01 2.4569E-01
  1.6895E-01 8.3699E-02 2.0316E-01 2.0280E-01 2.1169E-01
  ... ... ... ...
))
```

© Fluent Inc. November 28, 2001

6-169
Boundary Conditions

\[
\begin{array}{cccccc}
0.10182057E+03 & 0.98394081E+02 & 0.97748657E+02 & 0.97787750E+02 & 0.97905228E+02 \\
0.98020668E+02 & 0.98138817E+02 & 0.98264198E+02 & 0.98469681E+02 & 0.98478783E+02 \\
\end{array}
\]

\[
\begin{array}{cccccc}
0.00000000E+00 & 0.00000000E+00 & 0.00000000E+00 & 0.00000000E+00 & 0.00000000E+00 \\
0.00000000E+00 & 0.00000000E+00 & 0.00000000E+00 & 0.00000000E+00 & 0.00000000E+00 \\
\end{array}
\]

\[
\begin{array}{cccccc}
0.48591572E+00 & 0.66261977E+00 & 0.83932275E+00 & 0.10160275E+01 & 0.11927314E+01 \\
0.13694369E+01 & 0.15461419E+01 & 0.17228458E+01 & 0.18992697E+01 & 0.19070756E+01 \\
\end{array}
\]

Results

A plot of the transverse velocity components at the site of the fan is shown in Figure 6.23.2. As expected, there is no radial component, and the tangential (swirling) component increases with radius.

![Velocity Vectors Colored By Velocity Magnitude (m/s)](image)

Figure 6.23.4: Transverse Velocities at the Site of the Fan

As a final check on the result, an XY plot of the static pressure as a function of \(x\) position is shown (Figure 6.23.2). This XY plot is made on a line at \(y=0.05\) m, or at about half the radius of the duct. According to the input file shown above, the pressure jump at the site of the fan...
should be approximately 97.8 Pa/m. Examination of the figure supports this finding.

Figure 6.23.5: Static Pressure Jump Across the Fan
6.24 Heat Exchanger Model

Climate control and engine cooling systems typically contain heat exchanger cores. However, for most engineering problems, it is impractical to model individual fins and tubes of a heat exchanger core. In principle, heat exchanger cores add heat and introduce a pressure drop to an air stream. In FLUENT, lumped parameter models are used to account for the pressure loss and coolant heat rejection. The coolant in the heat exchanger model is restricted to a single phase. The model can be used to compute coolant inlet temperature for a fixed heat rejection, or total heat rejection for a fixed coolant inlet temperature.

The following sections contain information about the heat exchanger model:

- Section 6.24.1: Overview of the Heat Exchanger Model
- Section 6.24.2: Heat Exchanger Model Theory
- Section 6.24.3: Using the Heat Exchanger Model
- Section 6.24.4: Postprocessing for the Heat Exchanger Model

6.24.1 Overview of the Heat Exchanger Model

In a typical heat exchanger core, the coolant temperature is stratified in the direction of coolant flow. As a result, heat rejection is not constant over the entire core. In FLUENT, the fluid zone representing the heat exchanger core is subdivided into macroscopic cells or “macros” along the coolant path (see Figure 6.24.1). The coolant inlet temperature to each macro is computed and then subsequently used to compute the heat rejection from each macro. This approach provides a realistic heat rejection distribution over the heat exchanger core.

To use the heat exchanger model, you must define a fluid zone to represent the heat exchanger core. Typically, the fluid zone is sized to the dimension of the core itself. As part of the setup procedure, you will define the coolant path, the number of macros, and the physical properties and operating conditions of the core (pressure drop parameters, heat...
exchanger effectiveness, coolant flow rate, etc.). Once you have defined the model, FLUENT will automatically set the fluid zone to a porous zone.

![Diagram of heat exchanger model with 4×2 macros](image)

**Figure 6.24.1: Core Discretized Into 4×2 Macros**

### 6.24.2 Heat Exchanger Model Theory

In FLUENT, the heat exchanger core is treated as a fluid zone with momentum and heat transfer. Pressure loss is modeled as a momentum sink in the momentum equation, and heat transfer is modeled as a heat source in the energy equation.

**Streamwise Pressure Drop**

In the heat exchanger model, pressure loss is modeled using the porous media model in FLUENT. The porous media inputs are automatically set based on your inputs to the heat exchanger model. The streamwise pressure drop can be expressed as
Boundary Conditions

\[ \Delta p = \frac{1}{2} f \rho_m U_{A_{\text{min}}}^2 \]  \hspace{1cm} (6.24-1) 

where

- \( \Delta p \) = streamwise pressure drop
- \( f \) = streamwise pressure loss coefficient
- \( \rho_m \) = mean air density
- \( U_{A_{\text{min}}} \) = air velocity at the minimum flow area

The pressure loss coefficient is computed from

\[ f = (K_e + 1 - \sigma^2) - (1 - \sigma^2 - K_c) \frac{\nu_e}{\nu_i} + 2 \left( \frac{\nu_e}{\nu_i} - 1 \right) + f_c \frac{A}{A_c} \frac{\nu_m}{\nu_i} \]  \hspace{1cm} (6.24-2) 

where

- \( \sigma \) = minimum flow to face area ratio
- \( K_c \) = entrance loss coefficient
- \( K_e \) = exit loss coefficient
- \( A \) = air-side surface area
- \( A_c \) = minimum cross-sectional flow area
- \( f_c \) = core friction factor
- \( \nu_e \) = specific volume at the exit
- \( \nu_i \) = specific volume at the inlet
- \( \nu_m \) = mean specific volume \( \equiv \frac{1}{2}(\nu_e + \nu_i) \)

You will need to specify these parameters when you set up the heat exchanger model.

In Equation 6.24-2, the core friction factor is defined as

\[ f_c = a \operatorname{Re}_{\text{min}}^b \]  \hspace{1cm} (6.24-3) 

where

- \( a \) = core friction coefficient
- \( b \) = core friction exponent
- \( \operatorname{Re}_{\text{min}} \) = Reynolds number for velocity at the minimum flow area
You will need to specify the core friction coefficient and exponent when you set up the heat exchanger model.

The Reynolds number in Equation 6.24-3 is defined as

\[ Re_{\text{min}} = \frac{\rho_m U_{\text{A}_{\text{min}}} D_h}{\mu_m} \]  

(6.24-4)

where
- \( \rho_m \) = mean air density
- \( \mu_m \) = mean air viscosity
- \( D_h \) = hydraulic diameter
- \( U_{\text{A}_{\text{min}}} \) = air velocity at the minimum flow area

For a heat exchanger core, the hydraulic diameter can be defined as

\[ D_h = 4L \left( \frac{A_c}{A} \right) \]  

(6.24-5)

Note that \( U_{\text{A}_{\text{min}}} \) can be calculated from

\[ U_{\text{A}_{\text{min}}} = \frac{U}{\sigma} \]  

(6.24-6)

where \( U \) is the air velocity, and \( \sigma \) is the minimum flow to face area ratio.

**Heat Rejection**

Heat rejection is computed for each cell within a macro and added as a source term to the energy equation for the air flow. The heat transfer for a given cell is computed from

\[ q_{\text{cell}} = \varepsilon (\dot{m} c_p)_{\text{air}} (T_{\text{in}} - T_{\text{cell}}) \]  

(6.24-7)
Boundary Conditions

where

\[ \epsilon = \text{heat exchanger effectiveness} \]

\[ (\dot{m}c_p)_{\text{air}} = \text{air capacity rate (flow rate} \times \text{specific heat}) \]

\[ T_{\text{in}} = \text{coolant inlet temperature of macro} \]

\[ T_{\text{cell}} = \text{cell temperature} \]

The heat rejection from a macro is calculated by summing the heat transfer of all the cells contained within the macro

\[ q_{\text{macro}} = \sum_{\text{all cells in macro}} q_{\text{cell}} \quad (6.24-8) \]

The total heat rejection from the heat exchanger core is computed as the sum of the heat rejection from all the macros:

\[ q_{\text{total}} = \sum_{\text{all macros}} q_{\text{macro}} \quad (6.24-9) \]

The coolant inlet temperature to each macro \((T_{\text{in}} \text{ in Equation 6.24-7})\) is computed based on the energy balance of the coolant flow. For a given macro,

\[ q_{\text{macro}} = (\dot{m}c_p)_{\text{coolant}} (T_{\text{out}} - T_{\text{in}}) \quad (6.24-10) \]

where \(T_{\text{in}}\) and \(T_{\text{out}}\) are the inlet and outlet temperatures of the coolant in the macro, respectively. The value of \(T_{\text{out}}\) then becomes the inlet temperature to the next macro.

The first macro (Macro 0) is assumed to be where the coolant enters the heat exchanger core (see Figure 6.24.1). When a fixed total heat rejection from the heat exchanger core is specified, the inlet temperature to the first macro is iteratively computed, so that Equations 6.24-7, 6.24-8, 6.24-9, and 6.24-10 are satisfied simultaneously. When a fixed inlet coolant temperature is specified, the coolant inlet temperature for Macro 0 is fixed. The heat addition is computed from Equation 6.24-7.
and the coolant inlet temperature for subsequent macros is computed from Equation 6.24-10.

**Assumptions and Restrictions**

The following assumptions are made in the heat exchanger model:

- The heat exchanger effectiveness, $\epsilon$, is defined for a complete heat exchanger, and can be applied to a small portion of the heat exchanger represented by a computational cell.
- The air capacity rate, $(\dot{m}c_p)_{air}$, is less than the coolant capacity rate.
- The cell temperature, $T_{cell}$, (i.e., the cell centroid value) can be used instead of the temperature of the fluid entering the cell.
- Flow acceleration effects are neglected in calculating the pressure loss coefficient.
- The coolant temperature must be higher than the air temperature.
- The coolant is restricted to a single phase.

**6.24.3 Using the Heat Exchanger Model**

The steps for setting up the heat exchanger model are:

1. Enable the calculation of energy in the Energy panel.

2. Specify the inputs to the heat exchanger model, using the Heat Exchanger panel (Figure 6.24.2).

   (a) Select the fluid zone representing the heat exchanger core.
   (b) Specify the dimensions of the heat exchanger core.
   (c) Specify the coolant inlet and pass-to-pass directions.
Boundary Conditions

Figure 6.24.2: The Heat Exchanger Panel

<table>
<thead>
<tr>
<th>Fluid Zone</th>
<th>fluid-14</th>
</tr>
</thead>
<tbody>
<tr>
<td>Width (m)</td>
<td>1</td>
</tr>
<tr>
<td>Height (m)</td>
<td>1</td>
</tr>
<tr>
<td>Depth (m)</td>
<td>0.25</td>
</tr>
<tr>
<td>Number of Passes</td>
<td>2</td>
</tr>
<tr>
<td>Number of Macros/Pass</td>
<td>5</td>
</tr>
<tr>
<td>View Passes</td>
<td>Draw Grid</td>
</tr>
</tbody>
</table>

Coolant Inlet Direction

<table>
<thead>
<tr>
<th>X</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y</td>
<td></td>
</tr>
<tr>
<td>Z</td>
<td></td>
</tr>
</tbody>
</table>

Pass-to-Pass Direction

<table>
<thead>
<tr>
<th>X</th>
<th>0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y</td>
<td>1</td>
</tr>
<tr>
<td>Z</td>
<td></td>
</tr>
</tbody>
</table>

- Fixed Heat Rejection
- Fixed Inlet Temperature

<table>
<thead>
<tr>
<th>Coolant Flow Rate (kg/s)</th>
<th>0.6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Specific Heat (J/kg-k)</td>
<td>4000</td>
</tr>
<tr>
<td>Heat Rejection (W)</td>
<td>8000</td>
</tr>
<tr>
<td>Initial Temperature (K)</td>
<td>340</td>
</tr>
</tbody>
</table>

Heat Exchanger Core Model: default-model

Click on the `Edit` button to adjust settings.

Set | Delete | Close | Help
(d) Define the macro grid.
(e) Specify the coolant properties and conditions.
(f) Specify the pressure drop parameters and effectiveness of the heat exchanger core.
(g) Click Set in the Heat Exchanger panel to save all the settings.
(h) Repeat the above steps for any other heat exchanger fluid zones.

**Selecting the Zone for the Heat Exchanger**

Choose the fluid zone for which you want to define a heat exchanger in the Fluid Zone drop-down list.

**Specifying the Dimensions of the Heat Exchanger Core**

Set the Width, Height, and Depth of the heat exchanger core. The Height is defined along the direction of the coolant inlet (see Figure 6.24.3), and the Width is defined along the pass-to-pass direction.

**Specifying the Coolant Inlet and Pass-to-Pass Directions**

To define the coolant direction and flow path, you will specify direction vectors for the Coolant Inlet Direction and the Pass-to-Pass Direction. Figure 6.24.3 shows these directions relative to the macros.

For some problems in which the principal axes of the heat exchanger core are not aligned with the coordinate axes of the domain, you may not know a priori the coolant inlet and pass-to-pass direction vectors. In such cases, the plane tool can help you to determine these direction vectors.

1. “Snap” the plane tool onto the boundary of the heat exchanger core. (Follow the instructions in Section 24.6.1 for initializing the tool to a position on an existing surface.)

2. Translate and rotate the axes of the tool appropriately until they are aligned with the principal directions of the heat exchanger core.
Boundary Conditions

The streamwise direction is determined by the red axis, the coolant inlet direction by the green axis, and the pass-to-pass direction by the blue axis.

3. Once the axes are aligned, click on the Update From Plane Tool button in the Heat Exchanger panel. The directional vectors will be set automatically. (Note that the Update from Plane Tool button will also set the height, width, and depth of the heat exchanger core.)

Defining the Macros

As discussed in Section 6.24.1, the fluid zone representing the heat exchanger core is split into macros. Macros are constructed based on the specified number of passes, the number of macros per pass, and the corresponding coolant inlet and pass-to-pass direction (see Figure 6.24.3). Macros are numbered from 0 to \((n - 1)\) in the direction of coolant flow, where \(n\) is the number of macros.

In the Heat Exchanger panel, specify the Number of Passes and the Number of Macros/Pass. The model will automatically scale the number of macros you specify to the depth of the heat exchanger core.

Viewing the Macros

You can view the coolant path by displaying the macros. In order to view the macros for your specified Number of Passes and Number of Macros/Pass, click on the Set button at the bottom of the panel. Then click View Passes to display it. The path of the coolant is color-coded in the display: macro 0 is red and macro \(n - 1\) is blue.

For some problems, especially complex geometries, you may want to include portions of the computational-domain grid in your macros plot as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with the macros. This is accomplished by turning on the Draw Grid option. The Grid Display panel will appear automatically when you turn on the Draw Grid option, and you can set the grid display parameters there. When you click on View Passes in
6.24 Heat Exchanger Model

Figure 6.24.3: $3 \times 4$ Macros
**Boundary Conditions**

the Heat Exchanger panel, the grid display, as defined in the Grid Display panel, will be included in the macros plot (see Figure 6.24.4).

![Grid Display With Macros](image)

**Figure 6.24.4: Grid Display With Macros**

**Specifying the Coolant Properties and Conditions**

To define the coolant properties and conditions you will specify the Coolant Flow Rate and Specific Heat \( \dot{m} \) and \( c_p \), respectively, in Equation 6.24-10), and either the Heat Rejection and Initial Temperature (when a fixed heat rejection is being modeled) or the Inlet Temperature (when a fixed inlet coolant temperature is being modeled).

- If you want FLUENT to compute the coolant inlet temperature for a specified heat rejection, follow the steps below:
  1. Enable the Fixed Heat Rejection option.
  2. Specify the Heat Rejection \( \dot{q}_{\text{total}} \) in Equation 6.24-9).
  3. Specify the Initial Temperature, which will be used by FLUENT as an initial guess for the inlet temperature \( T_{\text{in}} \) in Equations 6.24-7 and 6.24-10).
6.24 Heat Exchanger Model

- If you want FLUENT to compute the total heat rejection of the core for a given inlet coolant temperature, follow the steps below:
  1. Enable the Fixed Inlet Temperature option.
  2. Specify the Inlet Temperature ($T_{in}$ in Equations 6.24-7 and 6.24-10 for Macro 0).

Setting the Pressure Drop Parameters and Effectiveness

The pressure drop parameters and effectiveness define the heat exchanger core model. There are three ways to specify the heat exchanger core model parameters:

- Use the values in FLUENT’s default heat exchanger core model.
- Define a new heat exchanger core model with your own values.
- Read a heat exchanger core model from an external file.

The models you define will be saved in the case file.

Using the Default Heat Exchanger Model

FLUENT provides a default model for a typical heat exchanger core. To use these values, simply retain the selection of default-model in the Heat Exchanger Core Model drop-down list in the Heat Exchanger panel. (You can view the default parameters in the Heat Exchanger Model panel, as described below.)

Defining a New Heat Exchanger Core Model

If you want to define pressure drop and effectiveness parameters that are different from those in the default heat exchanger core model, you can create a new model. The steps for creating a new model are as follows:

1. Click on the Edit... button to the right of the Heat Exchanger Core Model drop-down list. This will open the Heat Exchanger Model panel (Figure 6.24.5).
Boundary Conditions

Figure 6.24.5: The Heat Exchanger Model Panel
2. Enter the name of your new model in the Name box at the top of the panel.

3. Under Air-Side Pressure Drop, specify the following parameters used in Equation 6.24-2:
   - Min Flow to Face Area Ratio ($\sigma$)
   - Entrance Loss Coefficient ($K_c$)
   - Exit Loss Coefficient ($K_e$)
   - Air Side Surface Area ($A$)
   - Min Cross Section Flow Area ($A_c$)
   and the Core Friction Coefficient and Core Friction Exponent ($a$ and $b$, respectively, in Equation 6.24-3).

4. Under Effectiveness, specify the effectiveness of the heat exchanger core ($\epsilon$ in Equation 6.24-7) as a constant or as a piecewise-linear profile containing $n$ points.
   - To specify a constant effectiveness, retain the default Number of Points (1) and specify the value in the Effectiveness field.
   - To specify a profile for effectiveness, follow the steps below:
     (a) Specify the Number of Points in your profile.
     (b) For each Point, enter the Velocity and corresponding Effectiveness.

5. Click on Change/Create. This will add your new model to the database.

Reading Heat Exchanger Parameters from an External File

You can read parameters for your heat exchanger core model from an external file. A sample file is shown below:

```
("modelname"
 (0.73 0.43 0.053 5.2 0.33 9.1 0.66)
 ((1 1.0 .6234) (2 2.0 0.5014) (3 3.5 0.3932))
```
Boundary Conditions

(4 5.0 0.3244) (5 6.5 0.2762) (6 8.0 0.2405)
(7 10.0 0.2050) (8 12.0 0.1785) (9 15.0 0.1495))

The first entry in the file is the name of the model (e.g., modelname). The second set of numbers contains the air-side pressure drop parameters:

(σ K_c K_e A A_c a b)

The third entry is a set of points that represent an effectiveness profile. Each point in the profile has the following format:

(point velocity effectiveness)

In this sample file, nine points are specified in the effectiveness profile.

To read an external heat exchanger file, you will follow these steps:

1. In the Heat Exchanger Model panel, click on the Read... button.
2. In the resulting Select File dialog box, specify the HXC Parameters File name and click OK. FLUENT will read the heat exchanger core model parameters, and add the new model to the database.

Viewing the Parameters for an Existing Core Model

To view the parameters associated with a heat exchanger core model that you have already defined, select the model name in the Database drop-down list (in the Heat Exchanger Model panel). The values for that model from the database will be displayed in the Heat Exchanger Model panel.

6.24.4 Postprocessing for the Heat Exchanger Model

To view the computed values of total heat rejection, outlet temperature, and inlet temperature for your heat exchanger core, you can use the following text command:

```
define models heat-exchanger heat-exchanger-report
```

When prompted, specify the fluid zone (e.g., 1) for which you want to compute the results.
6.25 Boundary Profiles

FLUENT provides a very flexible boundary profile definition mechanism. This feature allows you to use experimental data, data calculated by an external program, or data written from a previous solution using the Write Profile panel (as described in Section 3.9) as the boundary condition for a variable.

Information about boundary profiles is presented in the following subsections:

- Section 6.25.1: Boundary Profile Specification Types
- Section 6.25.2: Boundary Profile File Format
- Section 6.25.3: Using Boundary Profiles
- Section 6.25.4: Reorienting Boundary Profiles

6.25.1 Boundary Profile Specification Types

Four types of profiles are provided:

- Point profiles are specified by an unordered set of $n$ points: $(x_i, y_i, v_i)$, where $1 \leq i \leq n$. Profiles written using the Write Profile panel and profiles of experimental data in random order are examples of point profiles.

  FLUENT will interpolate the point cloud as needed to obtain values at the boundary faces. The interpolation method for unstructured point data is zero order. In other words, for each cell face at the boundary, the solver uses the value from the profile file located closest to the cell. Therefore, if you wish an accurate specification of an inlet profile, your profile file should contain a sufficiently high point density.

- Line profiles are specified for 2D problems by an ordered set of $n$ points: $(x_i, y_i, v_i)$, where $1 \leq i \leq n$. Zero-order interpolation is performed between the points. An example of a line profile is a profile of data obtained from an external program that calculates a boundary-layer profile.
Boundary Conditions

- Mesh profiles are specified for 3D problems by an $m$ by $n$ mesh of points: $(x_{ij}, y_{ij}, z_{ij}, v_{ij})$, where $1 \leq i \leq m$ and $1 \leq j \leq n$. Zero-order interpolation is performed between the points. Examples of mesh profiles are profiles of data from a structured mesh solution and experimental data in a regular array.

- Radial profiles are specified for 2D and 3D problems by an ordered set of $n$ points: $(r_i, v_i)$, where $1 \leq i \leq n$. The data in a radial profile are a function of radius only. Linear interpolation is performed between the points, which must be sorted in ascending order of the $r$ field. The axis for the cylindrical coordinate system is determined as follows:
  - For 2D problems, it is the $z$-direction vector through $(0,0)$.
  - For 2D axisymmetric problems, it is the $x$-direction vector through $(0,0)$.
  - For 3D problems involving a swirling fan, it is the fan axis defined in the Fan panel (unless you are using local cylindrical coordinates at the boundary, as described below).
  - For 3D problems without a swirling fan, it is the rotation axis of the adjacent fluid zone, as defined in the Fluid panel (unless you are using local cylindrical coordinates at the boundary, as described below).
  - For 3D problems in which you are using local cylindrical coordinates to specify conditions at the boundary, it is the axis of the specified local coordinate system.

6.25.2 Boundary Profile File Format

The format of the profile files is fairly simple. The file can contain an arbitrary number of profiles. Each profile consists of a header that specifies the profile name, profile type (point, line, mesh, or radial), and number of defining points, and is followed by an arbitrary number of named “fields”. Some of these fields contain the coordinate points and the rest contain boundary data.

All quantities, including coordinate values, must be specified in SI units.
because FLUENT does not perform unit conversion when reading profile files.

Parentheses are used to delimit profiles and the fields within the profiles. Any combination of tabs, spaces, and newlines can be used to separate elements.

! In the general format description below, “|” indicates that you should input only one of the items separated by |’s and “...” indicates a continuation of the list.

```plaintext
((profile1-name point|line|radial n)
 (field1-name a1 a2 ... an)
 (field2-name b1 b2 ... bn)
 .
 .
 (fieldf-name f1 f2 ... fn))

((profile2-name mesh m n)
 (field1-name a11 a12 ... a1n
  a21 a22 ... a2n
  .
  .
  .
  am1 am2 ... amn)
 .
 .
 .
 (fieldf-name f11 f12 ... f1n
  f21 f22 ... f2n
  .
  .
  .
  fm1 fm2 ... fmn))
```

Boundary profile names must have all lowercase letters (e.g., name). Uppercase letters in boundary profile names are not acceptable. Each profile
**Boundary Conditions**

of type point, line, and mesh must contain fields with names x, y, and, for 3D, z. Each profile of type radial must contain a field with name r. The rest of the names are arbitrary, but must be valid Scheme symbols. For compatibility with old-style profile files, if the profile type is missing, “point” is assumed.

**Example**

A typical usage of a boundary profile file is to specify the profile of the boundary layer at an inlet. For a compressible flow calculation, this will be done using profiles of total pressure, k, and ε. For an incompressible flow, it might be preferable to specify the inlet value of streamwise velocity, together with k and ε.

Below is an example of a profile file that does this:

```plaintext
((turb-prof point 8)
 (x
  4.00000E+00 4.00000E+00 4.00000E+00 4.00000E+00
  4.00000E+00 4.00000E+00 4.00000E+00 4.00000E+00 )
 (y
  1.06443E-03 3.19485E-03 5.33020E-03 7.47418E-03
  2.90494E-01 3.31222E-01 3.84519E-01 4.57471E-01 )
 (u
  5.47866E+00 6.59870E+00 7.05731E+00 7.40079E+00
  1.01674E+01 1.01656E+01 1.01637E+01 1.01616E+01 )
 (tke
  4.93228E-01 6.19247E-01 5.32680E-01 4.93642E-01
  6.89414E-03 6.89666E-03 6.90015E-03 6.90478E-03 )
 (eps
  1.27713E+02 6.04399E+01 3.31187E+01 2.21535E+01
)
```
6.25 Boundary Profiles

6.25.3 Using Boundary Profiles

The procedure for using a boundary profile to define a particular boundary condition is outlined below.

1. Create a file that contains the desired boundary profile, following the format described in Section 6.25.2.

2. Read the boundary profile using the Read... button in the Boundary Profiles panel (Figure 6.25.1) or the File/Read/Profile... menu item.

3. In the boundary conditions panels (e.g., the Velocity Inlet and Pressure Inlet panels), the fields defined in the profile file (and those defined in any other profile file that you have read in) will appear in the drop-down list to the right of or below each parameter for which profile specification is allowed. To use a particular profile, select it in the list.

Boundary profiles cannot be used to define volumetric source terms. If you want to define a non-constant source term, you will need to use a user-defined function. For more information on UDFs, refer to the separate UDF Manual.

Note that if you use the Boundary Profiles panel to read a file, and a profile in the file has the same name as an existing profile, the old profile will be overwritten.

Checking and Deleting Profiles

Each profile file contains one or more profiles, and each profile has one or more fields defined in it. Once you have read in a profile file, you can check which fields are defined in each profile, and you can also delete a particular profile. These tasks are accomplished in the Boundary Profiles panel (Figure 6.25.1).
To check which fields are defined in a particular profile, select the profile name in the Profile list. The available fields in that file will be displayed in the Fields list. In Figure 6.25.1, the profile fields from the profile file of Section 6.25.2 are shown.

To delete a profile, select it in the Profiles list and click on the Delete button. When a profile is deleted, all fields defined in it will be removed from the Fields list.

**Example**

For the example given in Section 6.25.2, the profiles are used for inlet values of $x$ velocity, turbulent kinetic energy, and turbulent kinetic energy dissipation rate, as illustrated in Figure 6.25.2. (The $y$ velocity is set to a constant value of zero, since it is assumed negligible. However, a profile of $y$ velocity could also be used.)

Once the profiles have been specified, the boundary conditions have been saved (OK’d), and the flow solution has been initialized, you can view the inlet profile as follows:

- For 2D calculations, open the Solution XY Plot panel. Select the
appropriate boundary zone in the Surfaces list, the variable of interest in the Y Axis Function drop-down list, and the desired Plot Direction. Ensure that the Node Values check button is turned on, and then click Plot. You should then see the inlet profile plotted. If the data plotted do not agree with your specified profile, this means that there is an error in the profile file.

- For 3D calculations, use the Contours panel to display contours on the appropriate boundary zone surface. The Node Values check button must be turned on in order for you to view the profile data. If the data shown in the contour plot do not agree with your specified profile, this means that there is an error in the profile file.

Figure 6.25.2: Example of Using Profiles as Boundary Conditions
6.25.4 Reorienting Boundary Profiles

For 3D cases only, FLUENT allows you to change the orientation of an existing boundary profile so that it can be used at a boundary positioned arbitrarily in space. This allows you, for example, to take experimental data for an inlet with one orientation and apply it to an inlet in your model that has a different spatial orientation. Note that FLUENT assumes that the profile and the boundary are planar.

Steps for Changing the Profile Orientation

The procedure for orienting the boundary profile data in the principal directions of a boundary is outlined below:

1. Define and read the boundary profile as described in Section 6.25.3.

2. In the Boundary Profiles panel, select the profile in the Profile list, and then click on the Orient... button. This will open the Orient Profile panel (Figure 6.25.3).

3. In the Orient Profile panel, enter the name of the new profile you want to create in the New Profile box.

4. Specify the number of fields you want to create using the up/down arrows next to the New Fields box. The number of new fields is equal to the number of vectors and scalars to be defined plus 1 (for the coordinates).

5. Define the coordinate field.

   (a) Enter the names of the three coordinates in the first row under New Field Names.

   (b) Select the appropriate local coordinate fields for $x$, $y$, and $z$ from the drop-down lists under Compute From.... (A selection of 0 indicates that the coordinate does not exist in the original profile; i.e., the original profile was defined in 2D.)
Figure 6.25.3: The Orient Profile Panel
6. Define the vector fields in the new profile.
   (a) Enter the names of the 3 components in the directions of the coordinate axes of the boundary under **New Field Names**.
   (b) Select the names of the 3 components of the vector in the local $x$, $y$, and $z$ directions of the boundary profile from the drop-down lists under **Compute From**.

7. Define the scalar fields in the new profile.
   (a) Enter the name of the scalar in the first column under **New Field Names**.
   (b) Click on the button under **Treat as Scalar Quantity** in the same row.
   (c) Select the name of the scalar in the first drop-down list under **Compute From**.

8. Under **Orient To**..., specify the coordinates of the origin of the local coordinate system in the **Center** field.

9. Under **Orient To**..., specify the directional vectors ($x,y,z$) for the $X'$-, $Y'$-, and $Z'$-axis. The $X'$-, $Y'$-, and $Z'$-axis are the directional vectors of the major axis, the minor axis, and the normal to the boundary. Each direction will be that of the vector from $(0,0,0)$ to the specified $(x,y,z)$.

   For some problems in which the $X'$-, $Y'$- and $Z'$- directions are not aligned with the coordinate axes of the domain, you may not know a priori the directional vectors. In such cases, the plane tool can help you to determine these directional vectors and the coordinates of the **Center** (origin):

   (a) “Snap” the plane tool onto the boundary. (Follow the instructions in Section 24.6.1 for initializing the tool to a position on an existing surface.)
   (b) Translate and rotate the axes of the tool appropriately until the axes are aligned with the principal directions of the boundary and the origin of the tool is on the centroid of the boundary.
(c) Click on the Update From Plane Tool button in the Orient Profile panel. FLUENT will automatically set the Center to the origin of the plane tool axes, the X'-axis to the direction of the green arrow of the tool, the Y'-axis to the direction of the blue arrow of the tool, and the Z'-axis to the direction of the red arrow of the tool.

10. Click the Create button in the Orient Profile panel, and your new profile will be created. Its name (which you entered in the New Profile box) will now appear in the Boundary Profiles panel and will be available for use at the desired boundary.
6.26 Fixing the Values of Variables

The option to fix values of variables in FLUENT allows you to set the value of one or more variables in a fluid or solid zone, essentially setting a boundary condition for the variables within the cells of the zone. When a variable is fixed in a given cell, the transport equation for that variable is not solved in the cell (and the cell is not included when the residual sum is computed for that variable). The fixed value is used for the calculation of face fluxes between the cell and its neighbors. The result is a smooth transition between the fixed value of a variable and the values at the neighboring cells.

You can fix values for velocity components, temperature, and species mass fractions only if you are using the segregated solver.

6.26.1 Overview of Fixing the Value of a Variable

The ability to fix the value of a variable has a wide range of applications. The velocity fixing method is often used to model the flow in stirred tanks. This approach provides an alternative to the use of a rotating reference frame (solution in the reference frame of the blade) and can be used to model baffled tanks. In both 2D and 3D geometries, a fluid cell zone may be used in the impeller regions, and velocity components can be fixed based on measured data.

Although the actual impeller geometry can be modeled and the flow pattern calculated using the sliding mesh model, experimental data for the velocity profile in the outflow region are available for many impeller types. If you do not need to know the details of the flow around the blades for your problem, you can model the impeller by fixing the experimentally-obtained liquid velocities in its outflow zone. The velocities in the rest of the vessel can then be calculated using this fixed velocity profile as a boundary condition. Figure 6.26.1 shows an example of how this method is used to model the flow pattern created by a disk-turbine in an axisymmetric stirred vessel.
Figure 6.26.1: Fixing Values for the Flow in a Stirred Tank
Variables That Can Be Fixed

The variables that can be fixed include velocity components (segregated solver only), turbulence quantities, temperature (segregated solver only), enthalpy, species mass fractions (segregated solver only), and user-defined scalars. For turbulence quantities, different values can be set depending on your choice of turbulence model. You can fix the value of the temperature in a fluid or solid zone if you are solving the energy equation. If you are using the non-premixed combustion model, you can fix the enthalpy in a fluid zone. If you have more than one species in your model, you can specify fixed values for the species mass fractions for each individual species except the last one you defined. See the separate UDF Manual for details about defining user-defined scalars.

If you are using the Eulerian multiphase model, you can fix the values of velocity components and (depending on which multiphase turbulence model you are using) turbulence quantities on a per-phase basis. See Section 20.6.14 for details about setting boundary conditions for Eulerian multiphase calculations.

6.26.2 Procedure for Fixing Values of Variables in a Zone

To fix the values of one or more variables in a cell zone, follow these steps (remembering to use only SI units):

1. In the Fluid panel or Solid panel, turn on the Fixed Values option.

2. Fix the values for the appropriate variables, noting the comments below.

   - To specify a constant value for a variable, choose constant in the drop-down list next to the relevant field and then enter the constant value in the field.

   - To specify a non-constant value for a variable, you can use a boundary profile (see Section 6.25) or a user-defined function for a boundary profile (see the separate UDF Manual). Select the appropriate profile or UDF in the drop-down list next to the relevant field.
6.26 Fixing the Values of Variables

If you specify a radial-type boundary profile (see Section 6.25.1) for temperature, enthalpy, species mass fractions, or turbulence quantities for the $k$-$\epsilon$, Spalart-Allmaras, or $k$-$\omega$ model, the local coordinate system upon which the radial profile is based is defined by the Rotation-Axis Origin and Rotation-Axis Direction for the fluid zone. See Section 6.17.1 for information about setting these parameters. (Note that it is acceptable to specify the rotation axis and direction for a non-rotating zone. This will not cause the zone to rotate; it will not rotate unless it has been explicitly defined as a moving zone.)

- If you do not want to fix the value for a variable, choose (or keep) none in the drop-down list next to the relevant field. This is the default for all variables.

**Fixing Velocity Components**

To fix the velocity components, you can specify X, Y, and (in 3D) Z Velocity values, or, for axisymmetric cases, Axial, Radial, and (for axisymmetric swirl) Swirl Velocity values. The units for a fixed velocity are m/s.

For 3D cases, you can choose to specify cylindrical velocity components instead of Cartesian components. Turn on the Local Coordinate System For Fixed Velocities option, and then specify the Axial, Radial, and/or Tangential Velocity values. The local coordinate system is defined by the Rotation-Axis Origin and Rotation-Axis Direction for the fluid zone. See Section 6.17.1 for information about setting these parameters. (Note that it is acceptable to specify the rotation axis and direction for a non-rotating zone. This will not cause the zone to rotate; it will not rotate unless it has been explicitly defined as a moving zone.)

! You can fix values for velocity components only if you are using the segregated solver.

**Fixing Temperature and Enthalpy**

If you are solving the energy equation, you can fix the temperature in a zone by specifying the value of the Temperature. The units for a fixed
Boundary Conditions

...temperature are K.

If you are using the non-premixed combustion model, you can fix the enthalpy in a zone by specifying the value of the Enthalpy. The units for a fixed enthalpy are m²/s².

If you specify a radial-type boundary profile (see Section 6.25.1) for temperature or enthalpy, the local coordinate system upon which the radial profile is based is defined by the Rotation-Axis Origin and Rotation-Axis Direction for the fluid zone. See above for details.

! You can fix the value of temperature only if you are using the segregated solver.

Fixing Species Mass Fractions

If you are using the species transport model, you can fix the values of the species mass fractions for individual species. FLUENT allows you to fix the species mass fraction for each species (e.g., h₂, o₂) except the last one you defined.

If you specify a radial-type boundary profile (see Section 6.25.1) for a species mass fraction, the local coordinate system upon which the radial profile is based is defined by the Rotation-Axis Origin and Rotation-Axis Direction for the fluid zone. See above for details.

! You can fix values for species mass fractions only if you are using the segregated solver.

Fixing Turbulence Quantities

To fix the values of k and ε in the k-ε equations, specify the Turbulence Kinetic Energy and Turbulence Dissipation Rate values. The units for k are m²/s² and those for ε are m²/s³.

To fix the value of the modified turbulent viscosity (ν̃) for the Spalart-Allmaras model, specify the Modified Turbulent Viscosity value. The units for the modified turbulent viscosity are m²/s.

To fix the values of k and ω in the k-ω equations, specify the Turbulence Kinetic Energy and Specific Dissipation Rate values. The units for k are...
6.26 Fixing the Values of Variables

m²/s² and those for ω are 1/s.

To fix the value of k, ε, or the Reynolds stresses in the RSM transport equations, specify the Turbulence Kinetic Energy, Turbulence Dissipation Rate, UU Reynolds Stress, VV Reynolds Stress, WW Reynolds Stress, UV Reynolds Stress, VW Reynolds Stress, and/or UW Reynolds Stress. The units for k and the Reynolds stresses are m²/s², and those for ε are m²/s³.

If you specify a radial-type boundary profile (see Section 6.25.1) for k, ε, ω, or ̇ Q, the local coordinate system upon which the radial profile is based is defined by the Rotation-Axis Origin and Rotation-Axis Direction for the fluid zone. See above for details. Note that you cannot specify radial-type profiles for the Reynolds stresses.

Fixing User-Defined Scalars

To fix the value of a user-defined scalar, specify the User defined scalar-n value. (There will be one for each user-defined scalar you have defined.) The units for a user-defined scalar will be the appropriate SI units for the scalar quantity. See the separate UDF Manual for information on user-defined scalars.
6.27 Defining Mass, Momentum, Energy, and Other Sources

You can define volumetric sources of mass (for single or multiple species), momentum, energy, turbulence, and other scalar quantities in a fluid zone, or a source of energy for a solid zone. This feature is useful when you want to input a known value for these sources. (For more complicated sources with functional dependency, you can create a user-defined function as described in the separate UDF Manual.) To add source terms to a cell or group of cells, you must place the cell(s) in a separate zone. The sources are then applied to that cell zone. Typical uses for this feature are listed below:

- A flow source that cannot be represented by an inlet, e.g., due to an issue of scale. If you need to model an inlet that is smaller than a cell, you can place the cell where the tiny “inlet” is located in its own fluid zone and then define the mass, momentum, and energy sources in that cell zone. For the example shown in Figure 6.27.1, you should set a mass source of \[ \dot{m} = \frac{\rho A v_i}{V} \] and a momentum source of \[ \dot{mv} = \dot{mv}_i, \] where \( V \) is the cell volume.

- Heat release due to a source (e.g., fire) that is not explicitly defined in your model. For this case, you can place the cell(s) into which the heat is originally released in its own fluid zone and then define the energy source in that cell zone.

- An energy source in a solid zone, for conjugate heat transfer applications. For this case, you can place the cell(s) into which the heat is originally released in its own solid zone and then define the energy source in that cell zone.

- A species source due to a reaction that is not explicitly included in the model. In the above example of simulating a fire, you might need to define a source for a species representing smoke generation.

Note that if you define a mass source for a cell zone, you should also define a momentum source and, if appropriate for your model, energy and turbulence sources. If you define only a mass source, that mass enters
the domain with no momentum or thermal heat. The mass will therefore have to be accelerated and heated by the flow and, consequently, there may be a drop in velocity or temperature. This drop may or may not be perceptible, depending on the size of the source. (Note that defining only a momentum, energy, or turbulence source is acceptable.)

**Sign Conventions and Units**

All positive source terms indicate sources, and all negative source terms indicate sinks. All sources must be specified in SI units.
6.27.1 Procedure for Defining Sources

To define one or more source terms for a zone, follow these steps (remembering to use only SI units):

1. In the Fluid panel or Solid panel, turn on the Source Terms option.
2. Set the appropriate source terms, noting the comments below.
   - To specify a constant source, choose constant in the drop-down list next to the source term field and then enter the constant value in the field.
   - To specify a temperature-dependent or other functional source, you can use a user-defined function (see the separate UDF Manual). Select the appropriate UDF in the drop-down list next to the relevant field.
   - If you do not want to specify a source term for a variable, choose (or keep) none in the drop-down list next to the relevant field. This is the default for all variables.
   - Remember that you should not define just a mass source without defining the other sources, as described above.
   - Since the sources you specify are defined per unit volume, to determine the appropriate value of your source term you will often need to first determine the volume of the cell(s) in the zone for which you are defining the source. To do this, you can use the Volume Integrals panel.

Mass Sources

If you have only one species in your problem, you can simply define a Mass source for that species. The units for the mass source are kg/m$^3$-s. In the continuity equation (Equation 8.2-1), the defined mass source will appear in the $S_m$ term.

If you have more than one species, you can specify mass sources for each individual species. There will be a total Mass source term as well as a source term listed explicitly for each species (e.g., $h_2$, $o_2$) except the last
6.27 Defining Mass, Momentum, Energy, and Other Sources

one you defined. If the total of all species mass sources (including the last one) is 0, then you should specify a value of 0 for the Mass source, and also specify the values of the non-zero individual species mass sources. Since you cannot specify the mass source for the last species explicitly, FLUENT will compute it by subtracting the sum of all other species mass sources from the specified total Mass source.

For example, if the mass source for hydrogen in a hydrogen-air mixture is 0.01, the mass source for oxygen is 0.02, and the mass source for nitrogen (the last species) is 0.015, you will specify a value of 0.01 in the h2 field, a value of 0.02 in the o2 field, and a value of 0.045 in the Mass field. This concept also applies within each cell if you use user-defined functions for species mass sources.

The units for the species mass sources are kg/m³-s. In the conservation equation for a chemical species (Equation 13.1-1), the defined mass source will appear in the $S_i$ term.

Momentum Sources

To define a source of momentum, specify the X Momentum, Y Momentum, and/or Z Momentum term. The units for the momentum source are N/m³. In the momentum equation (Equation 8.2-3), the defined momentum source will appear in the $\vec{F}$ term.

Energy Sources

To define a source of energy, specify an Energy term. The units for the energy source are W/m³. In the energy equation (Equation 11.2-1), the defined energy source will appear in the $S_h$ term.

Turbulence Sources

Turbulence Sources for the k-\(\epsilon\) Model

To define a source of $k$ or $\epsilon$ in the $k-\epsilon$ equations, specify the Turbulence Kinetic Energy or Turbulence Dissipation Rate term. The units for the $k$ source are kg/m-s^3 and those for $\epsilon$ are kg/m-s^4.
Boundary Conditions

The defined $k$ source will appear in the $S_k$ term on the right-hand side of the turbulent kinetic energy equation (e.g., Equation 10.4-1).

The defined $\epsilon$ source will appear in the $S_\epsilon$ term on the right-hand side of the turbulent dissipation rate equation (e.g., Equation 10.4-2).

Turbulence Sources for the Spalart-Allmaras Model

To define a source of modified turbulent viscosity, specify the Modified Turbulent Viscosity term. The units for the modified turbulent viscosity source are kg/m-s$^2$. In the transport equation for the Spalart-Allmaras model (Equation 10.3-1), the defined modified turbulent viscosity source will appear in the $S_\nu$ term.

Turbulence Sources for the $k$-$\omega$ Model

To define a source of $k$ or $\omega$ in the $k$-$\omega$ equations, specify the Turbulence Kinetic Energy or Specific Dissipation Rate term. The units for the $k$ source are kg/m-s$^3$ and those for $\omega$ are kg/m$^3$-s$^2$.

The defined $k$ source will appear in the $S_k$ term on the right-hand side of the turbulent kinetic energy equation (Equation 10.5-1).

The defined $\omega$ source will appear in the $S_\omega$ term on the right-hand side of the specific turbulent dissipation rate equation (Equation 10.5-2).

Turbulence Sources for the Reynolds Stress Model

To define a source of $k$, $\epsilon$, or the Reynolds stresses in the RSM transport equations, specify the Turbulence Kinetic Energy, Turbulence Dissipation Rate, UU Reynolds Stress, VV Reynolds Stress, WW Reynolds Stress, UV Reynolds Stress, VW Reynolds Stress, and/or UW Reynolds Stress terms. The units for the $k$ source and the sources of Reynolds stress are kg/m-s$^3$, and those for $\epsilon$ are kg/m-s$^4$.

The defined Reynolds stress sources will appear in the $S_{user}$ term on the right-hand side of the Reynolds stress transport equation (Equation 10.6-1).

The defined $k$ source will appear in the $S_k$ term on the right-hand side of Equation 10.6-23.
6.27 Defining Mass, Momentum, Energy, and Other Sources

The defined $\epsilon$ will appear in the $S_\epsilon$ term on the right-hand side of Equation 10.6-26.

**Mean Mixture Fraction and Variance Sources**

To define a source of the mean mixture fraction or its variance for the non-premixed combustion model, specify the Mean Mixture Fraction or Mixture Fraction Variance term. The units for the mean mixture fraction source are kg/m$^3$-s, and those for the mixture fraction variance source are kg/m$^3$-s.

The defined mean mixture fraction source will appear in the $S_{\text{user}}$ term in the transport equation for the mixture fraction (Equation 14.1-4).

The defined mixture fraction variance source will appear in the $S_{\text{user}}$ term in the transport equation for the mixture fraction variance (Equation 14.1-5).

If you are using the two-mixture-fraction approach, you can also specify sources of the Secondary Mean Mixture Fraction and Secondary Mixture Fraction Variance.

**P-1 Radiation Sources**

To define a source for the P-1 radiation model, specify the $P1$ term. The units for the radiation source are W/m$^3$, and the defined source will appear in the $S_G$ term in Equation 11.3-12.

Note that, if the source term you are defining represents a transfer from internal energy to radiative energy (e.g., absorption or emission), you will need to specify an Energy source of the same magnitude as the $P1$ source, but with the opposite sign, in order to ensure overall energy conservation.

**Progress Variable Sources**

To define a source of the progress variable for the premixed combustion model, specify the Progress Variable term. The units for the progress variable source are kg/m$^3$-s, and the defined source will appear in the $\rho S_c$ term in Equation 15.2-1.
Boundary Conditions

NO, HCN, and NH₃ Sources for the NOₓ Model

To define a source of NO, HCN, or NH₃ for the NOₓ model, specify the no, hcn, or nh3 term. The units for these sources are kg/m³·s, and the defined sources will appear in the S_NO, S_HCNC, and S_NH3 terms of Equations 17.1-1, 17.1-2, and 17.1-3.

User-Defined Scalar Sources

To define a source for a user-defined scalar transport equation, specify the User defined scalar-n term. (There will be one for each user-defined scalar you have defined.) The units for a user-defined scalar source will be the appropriate SI units for the scalar quantity. The source will appear in the S_φᵢ term of the scalar transport equation. See the separate UDF Manual for information on user-defined scalars.
6.28 Coupling Boundary Conditions with GT-Power

GT-Power users can define time-dependent boundary conditions in FLUENT based on information from GT-Power. During the FLUENT simulation, FLUENT and GT-Power are coupled together and information about the boundary conditions at each time step is transferred between them.

6.28.1 Requirements and Restrictions

Note the following requirements and restrictions for the GT-Power coupling:

- The flow must be unsteady.
- The compressible ideal gas law must be used for density.
- Each boundary zone for which you plan to define conditions using GT-Power must be a flow boundary of one of the following types:
  - velocity inlet
  - mass flow inlet
  - pressure inlet
  - pressure outlet

Also, a maximum of 20 boundary zones can be coupled to GT-Power.

- If a mass flow inlet or pressure inlet is coupled to GT-Power, you must select Normal to Boundary as the Direction Specification Method in the Mass-Flow Inlet or Pressure Inlet panel. For a velocity inlet, you must select Magnitude, Normal to Boundary as the Velocity Specification Method in the Velocity Inlet panel.

- Boundary conditions for the following variables can be obtained from GT-Power:
  - velocity
  - temperature
Boundary Conditions

- pressure
- density
- species mass fractions
- $k$ and $\epsilon$ (Note that it is recommended that you define these conditions in FLUENT yourself, rather than using the data provided by GT-Power, since the GT-Power values are based on a 1D model.)

- Make sure that the material properties you set in FLUENT are the same as those used in GT-Power, so that the boundary conditions will be valid for your coupled simulation.

- If your model includes species, make sure that the name of each species in GT-Power corresponds to the Chemical Formula for that species material in the Materials panel. Also, recall that FLUENT can handle a maximum of 50 species.

6.28.2 User Inputs

The procedure for setting up the GT-Power coupling in FLUENT is presented below.

1. Read in the mesh file and define the models, materials, and boundary zone types (but not the actual boundary conditions), noting the requirements and restrictions listed in Section 6.28.1.

2. Specify the location of the GT-Power data and have FLUENT use them to generate user-defined functions for the relevant boundary conditions (using the 1D Simulation Library panel, shown in Figure 6.28.1).

   Define $\rightarrow$ User-Defined $\rightarrow$ 1D Coupling...

   (a) Keep the default selection of GTpower in the 1D Library drop-down list.

   (b) Specify the name of the shared library (including the complete path, if the library is not located in the current working directory) in the Library Name field.
(c) Specify the name of the GT-Power input file in the 1D Input File Name field.

(d) Click the Start button.

When you click Start, GT-Power will start up and FLUENT user-defined functions for each boundary in the input file will be generated.

3. Set boundary conditions for all zones. For flow boundaries for which you are using GT-Power data, select the appropriate UDFs as the conditions.

Note that you must select the same UDF for all conditions at a particular boundary zone (as shown, for example, in Figure 6.28.2); this UDF contains all of the conditions at that boundary.

4. If you plan to continue the simulation at a later time, starting from the final data file of the current simulation, specify how often you want to have the case and data files saved automatically.

When FLUENT saves a data file at the specified automatic save interval, it will also save a restart file for GT-Power. This is the only
Boundary Conditions

Figure 6.28.2: Using GT-Power Data for Boundary Conditions
way to save a GT-Power restart file; if you use the File/Write/Data... menu item to save a data file manually, the restart file will not be created.

To use a GT-Power restart file to restart a FLUENT calculation, you must edit the GT-Power input data file. See the GT-Power User’s Guide for instructions.

5. Continue the problem setup and calculate a solution in the usual manner.