

7.2.3 Porous Media Conditions

The porous media model can be used for a wide variety of single phase and multiphase problems, including flow 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 [7.2.3](#). 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 [7.2.3](#).

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 [7.3.20](#) for details.

Limitations and Assumptions of the Porous Media Model

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 physically present is not represented in the model, by default **ANSYS 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. As a more accurate alternative, you can instruct **ANSYS FLUENT** to use the true (physical) velocity inside the porous medium. See Section [7.2.3](#) for details. In a multiphase flow system, all phases share the same porosity.
- The effect of the porous medium on the turbulence field is only approximated. See Section [7.2.3](#) for details.
- In general, the **ANSYS FLUENT** porous medium model, for both single phase and multiphase, assumes the porosity is isotropic, and it can vary with space and time.
- The Superficial Velocity Formulation and the Physical Velocity Formulation are available for multiphase porous media. See Section [7.2.3](#) for details.
- The porous media momentum resistance and heat source terms are calculated separately on each phase. See Section [7.2.3](#) for details.
- The interactions between a porous medium and shock waves are not considered.
- By default, **ANSYS FLUENT** assumes thermal equilibrium between the porous media solids and multiphase fluid flows. The solids temperature is thus estimated by phase temperatures. However, the solids temperature can also be calculated by a UDS equation (Section [9.1](#)).
- When applying the porous media model in a moving reference frame, **ANSYS FLUENT** will

either apply the relative reference frame or the absolute reference frame when you enable the **Relative Velocity Resistance Formulation**. This allows for the correct prediction of the source terms.

Momentum Equations for Porous Media

The porous media models for single phase flows and multiphase flows use the **Superficial Velocity Porous Formulation** as the default. ANSYS FLUENT calculates the the superficial phase or mixture velocities based on the volumetric flow rate in a porous region. The porous media model is described in the following sections for single phase flow, however, it is important to note the following for multiphase flow:

- In the Eulerian multiphase model ([this section](#) in the separate [Theory Guide](#)), the general porous media modeling approach, physical laws, and equations described below are applied to the corresponding phase for mass continuity, momentum, energy, and all the other scalar equations.
- The **Superficial Velocity Porous Formulation** generally gives good representations of the bulk pressure loss through a porous region. However, since the superficial velocity values within a porous region remain the same as those outside the porous region, it cannot predict the velocity increase in porous zones and thus limits the accuracy of the model.

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 [7.2-1](#)), and an inertial loss term (the second term on the right-hand side of Equation [7.2-1](#))

$$S_i = - \left(\sum_{j=1}^3 D_{ij} \mu v_j + \sum_{j=1}^3 C_{ij} \frac{1}{2} \rho |v| v_j \right) \quad (7.2-1)$$

where S_i is the source term for the i th (x , y , or z) momentum equation, $|v|$ is the magnitude of the velocity 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

$$S_i = - \left(\frac{\mu}{\alpha} v_i + C_2 \frac{1}{2} \rho |v| v_i \right) \quad (7.2-2)$$

where α 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).

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

$$S_i = -C_0|v|^{C_1} = -C_0|v|^{(C_1-1)}v_i \quad (7.2-3)$$

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:

$$\nabla p = -\frac{\mu}{\alpha} \vec{v} \quad (7.2-4)$$

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

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

$$\Delta p_y = \sum_{j=1}^3 \frac{\mu}{\alpha_{yj}} v_j \Delta n_y \quad (7.2-5)$$

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

where $1/\alpha_{ij}$ are the entries in the matrix D in Equation 7.2-1, v_j are the velocity components in the x , y , and z directions, and Δn_x , Δn_y , and Δn_z are the thicknesses of the medium in the x , y , and z directions.

Here, the thickness of the medium (Δn_x , Δn_y , or Δn_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 7.2-1 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:

$$\nabla p = - \sum_{j=1}^3 C_{2vj} \left(\frac{1}{2} \rho v_j |v| \right) \quad (7.2-6)$$

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

$$\begin{aligned} \Delta p_x &\approx \sum_{j=1}^3 C_{2xj} \Delta n_x \frac{1}{2} \rho v_j |v| \\ \Delta p_y &\approx \sum_{j=1}^3 C_{2yj} \Delta n_y \frac{1}{2} \rho v_j |v| \\ \Delta p_z &\approx \sum_{j=1}^3 C_{2zj} \Delta n_z \frac{1}{2} \rho v_j |v| \end{aligned} \quad (7.2-7)$$

Again, the thickness of the medium (Δn_x , Δn_y , or Δn_z) is the thickness you have defined in your model.

Treatment of the Energy Equation in Porous Media

ANSYS FLUENT solves the standard energy transport equation ([this equation](#) in the separate [Theory Guide](#)) 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:

$$\frac{\partial}{\partial t} (\gamma \rho_f E_f + (1-\gamma) \rho_s E_s) + \nabla \cdot (\vec{v} (\rho_f E_f + p)) = \nabla \cdot \left[k_{\text{eff}} \nabla T - \left(\sum_i h_i J_i \right) + (\bar{\tau} \cdot \vec{v}) \right] + S_f^h \quad (7.2-8)$$

where

= total fluid energy

$$E_f$$

$$E_s = \text{total solid medium energy}$$

$$\gamma = \text{porosity of the medium}$$

$$k_{\text{eff}} = \text{effective thermal conductivity of the medium}$$

$$S_f^h = \text{fluid enthalpy source term}$$

Effective Conductivity in the Porous Medium

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

$$k_{\text{eff}} = \gamma k_f + (1 - \gamma) k_s \quad (7.2-9)$$

where

$$\gamma = \text{porosity of the medium}$$

$$k_f = \text{fluid phase thermal conductivity (including the turbulent contribution, } k_t)$$

$$k_s = \text{solid medium thermal conductivity}$$

The fluid thermal conductivity k_f and the solid thermal conductivity k_s can be computed via user-defined functions.

The anisotropic effective thermal conductivity can also be specified via user-defined functions. In this case, the isotropic contributions from the fluid, γk_f , are added to the diagonal elements of the solid anisotropic thermal conductivity matrix.

Treatment of Turbulence in Porous Media

ANSYS FLUENT will, by default, solve the standard conservation equations for turbulence quantities in the porous medium. In this default approach, 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 turbulence models (with the exception of the Large Eddy Simulation (LES) model), you can suppress the effect of turbulence in a porous region by setting the turbulent

contribution to viscosity, μ_t , equal to zero. When you choose this option, **ANSYS 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 dialog box](#). Enabling this option implies that μ_t is zero and that generation 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. Refer to Section [7.2.1](#) for details about using the **Laminar Zone** option.

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 taken into account, the time-derivative term becomes $\frac{\partial}{\partial t}(\gamma\rho\phi)$, where ϕ is the scalar quantity (k , ϵ , etc.) and γ is the porosity.

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

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. Define the porous velocity formulation in the **Cell Zone Conditions** task page. (optional)
3. Identify the fluid material flowing through the porous medium.
4. Enable reactions for the porous zone, if appropriate, and select the reaction mechanism.
5. Enable the **Relative Velocity Resistance Formulation**. By default, this option is already enabled and takes the moving porous media into consideration (as described in Section [7.2.3](#)).
6. Set the viscous resistance coefficients (D_{ij} in Equation [7.2-1](#), or $1/\alpha$ in Equation [7.2-2](#)) and the inertial resistance coefficients (C_{ij} in Equation [7.2-1](#), or C_2 in Equation [7.2-2](#)), and define the direction vectors for which they apply. Alternatively, specify the coefficients for the power-law model.
7. Specify the porosity of the porous medium.
8. Select the material contained in the porous medium (required only for models that include heat transfer). Note that the specific heat capacity, C_p , for the selected material in the porous zone can only be entered as a constant value.

9. Set the volumetric heat generation rate in the non-solid portion of the porous medium (or any other sources, such as mass or momentum). (optional)
10. Set any fixed values for solution variables in the fluid region (optional).
11. Suppress the turbulent viscosity in the porous region, if appropriate.
12. 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 [7.2-3](#) instead of the resistance coefficients and flow direction.

You will set all parameters for the porous medium in the [Fluid dialog box](#) (Figure [7.2.3](#)), which is opened from the **Cell Zone Conditions** task page (as described in Section [7.1.4](#)).

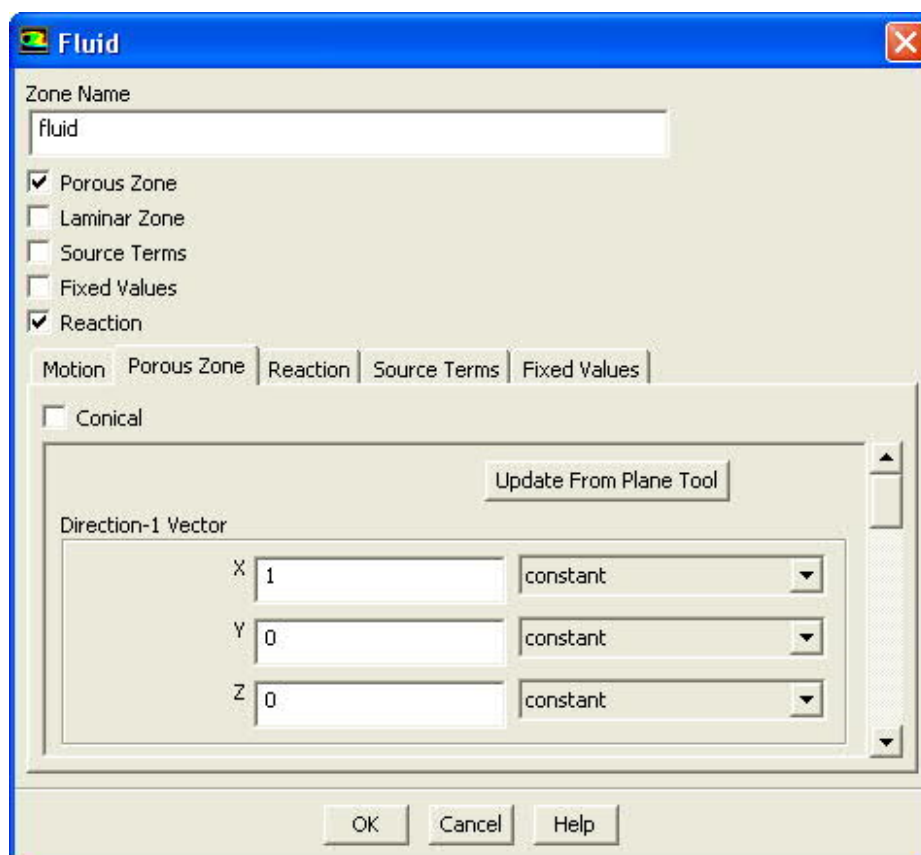


Figure 7.2.3: The **Fluid** Dialog Box for a Porous Zone

Defining the Porous Zone

As mentioned in Section [7.1](#), 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** dialog box. The dialog box will expand to show the porous media inputs (as shown in Figure [7.2.3](#)).

Defining the Porous Velocity Formulation

The **Cell Zone Conditions** task page contains a **Porous Formulation** region where you can instruct ANSYS FLUENT to use either a superficial or physical velocity in the porous medium simulation. By default, the velocity is set to **Superficial Velocity**. For details about using the **Physical Velocity** formulation, see Section [7.2.3](#).

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 dialog box](#). If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box.



If you are modeling species transport or multiphase flow, the **Material Name** list will not appear in the **Fluid** dialog box. For species calculations, the mixture material for all fluid/porous zones will be the material you specified in the [Species Model dialog box](#). For multiphase flows, the materials are specified when you define the phases, as described in Section [24.3.4](#).

Enabling Reactions in a Porous Zone

If you are modeling species transport with reactions, you can enable reactions in a porous zone by turning on the **Reaction** option in the **Fluid** dialog box and selecting a mechanism in the **Reaction Mechanism** drop-down list.

If your mechanism contains wall surface reactions, you will also need to specify a value for the **Surface-to-Volume Ratio**. This value is the surface area of the pore walls per unit volume ($\frac{A}{V}$), and can be thought of as a measure of catalyst loading. With this value, **ANSYS FLUENT** can calculate the total surface area on which the reaction takes place in each cell by multiplying $\frac{A}{V}$ by the volume of the cell. See Section [15.1.3](#) for details about defining reaction mechanisms. See Section [15.2](#) for details about wall surface reactions.

Including the Relative Velocity Resistance Formulation

Prior to **ANSYS FLUENT** 6.3, cases with moving reference frames used the absolute velocities in the source calculations for inertial and viscous resistance. This approach has been enhanced so that relative velocities are used for the porous source calculations (Section [7.2.3](#)). Using the **Relative Velocity Resistance Formulation** option (turned on by default) allows you to better predict the source terms for cases involving moving meshes or moving reference frames (MRF). This option works well in cases with non-moving and moving porous media. Note that **ANSYS FLUENT** will use the appropriate velocities (relative or absolute), depending on your case setup.

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.

You can also define the viscous and/or inertial resistance coefficients in each direction using a user-defined function (UDF). The user-defined options become available in the corresponding drop-down list when the UDF has been created and loaded into **ANSYS FLUENT**. Note that the coefficients defined in the UDF must utilize the `DEFINE_PROFILE` macro. For more information on creating and using user-defined function, see the separate UDF Manual.

If you are modeling axisymmetric swirling flows, you can specify an additional direction component for the viscous and/or inertial resistance coefficients. This direction component is always tangential to the other two specified directions. This option is available for both density-based and pressure-based solvers.

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

i Note that the viscous and inertial resistance coefficients are generally based on the superficial velocity of the fluid in the porous media.

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**. The unspecified direction will be determined as described above. These direction vectors correspond to the principle axes of the porous media.

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 [28.6.1](#) or [28.5.1](#) for initializing the tool to a position on an existing surface.)
- (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** dialog box. ANSYS 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) Set the **Cone Half Angle** (the angle between the cone's axis and its surface, shown in Figure [7.2.4](#)). To use a cylindrical coordinate system, set the **Cone Half Angle** to 0.
 - (c) Specify the **Cone Axis Vector** and **Point on Cone Axis**. The cone axis is specified as being in the direction of the **Cone Axis Vector** (unit vector), and passing through the **Point on Cone Axis**. The cone axis may or may not pass through the origin of the coordinate system.

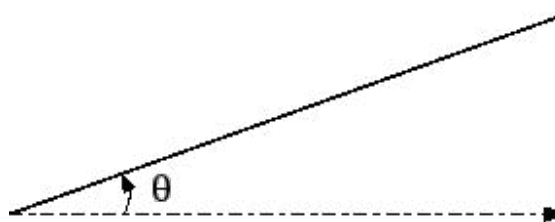


Figure 7.2.4: Cone Half Angle

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 the **Snap to Zone** button. **ANSYS 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 [28.6.1](#) 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 the **Update From Plane Tool** button in the **Fluid** dialog box. **ANSYS 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.

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.)

For porous media cases containing highly anisotropic inertial resistances, enable **Alternative Formulation** under **Inertial Resistance**. The **Alternative Formulation** option provides better stability to the calculation when your porous medium is anisotropic. The pressure loss through the medium depends on the magnitude of the velocity vector of the i th component in the medium. Using the formulation of Equation [7.2-6](#) yields the expression below:

$$S_i = \frac{1}{2} \rho C_i |v_i| v_i \quad (7.2-10)$$

Whether or not you use the **Alternative Formulation** option depends on how well you can fit your experimentally determined pressure drop data to the **ANSYS FLUENT** model. For example, if the flow through the medium is aligned with the mesh in your **ANSYS FLUENT** model, then it will not make a difference whether or not you use the formulation.

For more information about simulations involving highly anisotropic porous media, see Section [7.2.3](#).



Note that the alternative formulation is compatible only with the pressure-based solver.

If you are using the **Conical** specification method, **Direction-1** is the tangential direction of the cone, **Direction-2** is the normal to the cone surface (radial (r) direction for a cylinder), and **Direction-3** is the circumferential (θ) 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 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.

Methods for deriving viscous and inertial loss coefficients are described in the sections that follow.

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 **ANSYS FLUENT** are *100% open*, and that the values that you specify for $1/\alpha_{ij}$ and/or C_{2ij} 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 **ANSYS 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

$$\Delta p = K_L \left(\frac{1}{2} \rho v_{25\%open}^2 \right) \quad (7.2-11)$$

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 **ANSYS 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:

$$\Delta p = K'_L \left(\frac{1}{2} \rho v_{100\%open}^2 \right) \quad (7.2-12)$$

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

$$\begin{aligned} K_L' &= K_L \times \frac{v_{25\%open}^2}{v_{100\%open}^2} \\ &= 0.5 \times \left(\frac{4}{1}\right)^2 \\ &= 8 \end{aligned} \quad (7.2-13)$$

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

$$\begin{aligned} C_2 &= \frac{K_L'}{\text{thickness}} \\ &= \frac{8}{10^{-3}} = 8000 \text{ m}^{-1} \end{aligned} \quad (7.2-14)$$

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 [20], a semi-empirical correlation applicable over a wide range of Reynolds numbers and for many types of packing:

$$\frac{|\Delta p|}{L} = \frac{150\mu}{D_p^2} \frac{(1-\epsilon)^2}{\epsilon^3} v_\infty + \frac{1.75\rho}{D_p} \frac{(1-\epsilon)}{\epsilon^3} v_\infty^2 \quad (7.2-15)$$

When modeling laminar flow through a packed bed, the second term in the above equation may be dropped, resulting in the Blake-Kozeny equation [20]:

$$\frac{|\Delta p|}{L} = \frac{150\mu}{D_p^2} \frac{(1-\epsilon)^2}{\epsilon^3} v_\infty \quad (7.2-16)$$

In these equations, μ is the viscosity, D_p is the mean particle diameter, L is the bed depth, and ϵ is the void fraction, defined as the volume of voids divided by the volume of the packed bed region. Comparing Equations 7.2-4 and 7.2-6 with 7.2-15, the permeability and inertial loss coefficient in each component direction may be identified as

$$\alpha = \frac{D_p^2}{150} \frac{\epsilon^3}{(1 - \epsilon)^2} \quad (7.2-17)$$

and

$$C_2 = \frac{3.5 (1 - \epsilon)}{D_p \epsilon^3} \quad (7.2-18)$$

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. [59, 75] 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

$$\dot{m} = C A_f \sqrt{(2\rho\Delta p)/(1 - (A_f/A_p)^2)} \quad (7.2-19)$$

where

\dot{m} = mass flow rate through the plate

A_f = the free area or total area of the holes

A_p = the area of the plate (solid and holes)

C = a coefficient that has been tabulated for various Reynolds-number ranges and for various D/t

D/t = the ratio of hole diameter to plate thickness

for $t/D > 1.6$ and for $Re > 4000$ 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 7.2-19, making use of the relationship

$$\dot{m} = \rho v A_p \quad (7.2-20)$$

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

$$\frac{\Delta p}{\Delta x} = \left(\frac{1}{2} \rho v^2 \right) \frac{1}{C^2} \frac{(A_p/A_f)^2 - 1}{t} \quad (7.2-21)$$

where v is the superficial velocity (not the velocity in the holes). Comparing with Equation 7.2-6 it is seen that, for the direction normal to the plate, the constant C_2 can be calculated from

$$C_2 = \frac{1}{C^2} \frac{(A_p/A_f)^2 - 1}{t} \quad (7.2-22)$$

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 7.2-16) we might choose to employ tabulated experimental data. Such data is available for many types of fiber [35].

volume fraction of solid material	dimensionless permeability B of glass wool
0.262	0.25
0.258	0.26
0.221	0.40
0.218	0.41
0.172	0.80

where $B = \alpha/a^2$ and a is the fiber diameter. α , for use in Equation 7.2-4, is easily computed for a given fiber diameter and volume fraction.

Deriving the Porous Coefficients Based on Experimental Pressure and Velocity Data

Experimental data that is available in the form of pressure drop against velocity through the porous component, can be extrapolated to determine the coefficients for the porous media. To effect a pressure drop across a porous medium of thickness, Δn , the coefficients of the porous media are determined in the manner described below.

If the experimental data is:

Velocity (m/s)	Pressure Drop (Pa)
20.0	28.4
50.0	487.0
80.0	1432.0
110.0	2964.0

then an xy curve can be plotted to create a trendline through these points yielding the following equation

$$\Delta p = 0.28296v^2 - 4.33539v \quad (7.2-23)$$

where Δp is the pressure drop and v is the velocity.

i Although the best fit curve may yield negative coefficients, it should be avoided when using the porous media model in ANSYS FLUENT.

Note that a simplified version of the momentum equation, relating the pressure drop to the source term, can be expressed as

$$\nabla p = S_i \quad (7.2-24)$$

or

$$\Delta p = -S_i \Delta n \quad (7.2-25)$$

Hence, comparing Equation 7.2-23 to Equation 7.2-2, yields the following curve coefficients:

$$0.28296 = C_2 \frac{1}{2} \rho \Delta n \quad (7.2-26)$$

with $\rho = 1.225 \text{ kg/m}^3$, and a porous media thickness, Δn , assumed to be 1m in this example, the inertial resistance factor, $C_2 = 0.462$.

Likewise,

$$-4.33539 = \frac{\mu}{\alpha} \Delta n \quad (7.2-27)$$

with $\mu = 1.7894 \times 10^{-5}$, the viscous inertial resistance factor, $\frac{1}{\alpha} = -242282$.

Note that this same technique can be applied to the porous jump boundary condition. Similar to the case of the porous media, you have to take into account the thickness of the medium Δn .

i Your experimental data can be plotted in an xy curve, yielding an equation that is equivalent to Equation 7.3-69. From there, you can determine the permeability α and the pressure jump coefficient C_2 .

Using the Power-Law Model

If you choose to use the power-law approximation of the porous-media momentum source term (Equation [7.2-3](#)), the only inputs required are the coefficients C_0 and C_1 . Under **Power Law Model** in the **Fluid** dialog box, enter the values for **C0** and **C1**. Note that the power-law model can be used in conjunction with the Darcy and inertia models.

C0 must be in SI units, consistent with the value of **C1**.

Defining Porosity

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

You can also define the porosity using a user-defined function (UDF). The user-defined option becomes available in the corresponding drop-down list when the UDF has been created and loaded into **ANSYS FLUENT**. Note that the porosity defined in the UDF must utilize the `DEFINE_PROFILE` macro. For more information on creating and using user-defined function, see the separate UDF Manual.

The porosity, γ , is the volume fraction of fluid within the porous region (i.e., the open volume fraction of the medium). The porosity is used in the prediction of heat transfer in the medium, as described in Section [7.2.3](#), and in the time-derivative term in the scalar transport equations for unsteady flow, as described in Section [7.2.3](#). 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.

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** dialog box, 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 **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box. In the **Edit Material** dialog box, you can define the non-isotropic thermal conductivity of the porous material using a user-defined function (UDF). The user-defined option becomes available in the corresponding drop-down list when the UDF has been created and loaded into **ANSYS FLUENT**. Note that the non-isotropic thermal conductivity defined in the UDF must utilize the `DEFINE_PROPERTY` macro. For more information on creating and using user-defined function, see the separate UDF Manual.

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 [7.2.5](#).

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 [7.2.4](#) for details.

Suppressing the Turbulent Viscosity in the Porous Region

As discussed in Section [7.2.3](#), turbulence will be computed in the porous region just as in the bulk fluid flow. If you are using one of the turbulence models (with the exception of the Large Eddy Simulation (LES) Model), and you want the turbulence generation to be zero in the porous zone, turn on the **Laminar Zone** option in the **Fluid** dialog box. Refer to Section [7.2.1](#) for more information about suppressing turbulence generation.

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 [7.2.1](#) for details.

Modeling Porous Media Based on Physical Velocity

As stated in Section [7.2.3](#), by default **ANSYS FLUENT** calculates the superficial velocity based on volumetric flow rate. The superficial velocity in the governing equations can be represented as

$$\vec{u}_{\text{superficial}} = \gamma \vec{u}_{\text{physical}} \quad (7.2-28)$$

where γ is the porosity of the media defined as the ratio of the volume occupied by the fluid to the total volume.

The superficial velocity values within the porous region remain the same as those outside of the porous region. This limits the accuracy of the porous model where there should be an increase in velocity throughout the porous region. For more accurate simulations of porous media flows, it becomes necessary to solve for the true, or physical velocity throughout the flowfield, rather than the superficial velocity.

ANSYS FLUENT allows the calculation of the physical velocity using the **Porous Formulation**, available in the **Cell Zone Conditions** task page. By default, the **Superficial Velocity** option is turned on.

Single Phase Porous Media

Using the physical velocity formulation, and assuming a general scalar ϕ , the governing equation in an isotropic porous media has the following form:

$$\frac{\partial(\gamma\rho\phi)}{\partial t} + \nabla \cdot (\gamma\rho\vec{u}\phi) = \nabla \cdot (\gamma\Gamma\nabla\phi) + \gamma S_\phi \quad (7.2-29)$$

Assuming isotropic porosity and single phase flow, the volume-averaged mass and momentum conservation equations are as follows:

$$\frac{\partial(\gamma\rho)}{\partial t} + \nabla \cdot (\gamma\rho\vec{v}) = 0 \quad (7.2-30)$$

$$\frac{\partial(\gamma\rho\vec{v})}{\partial t} + \nabla \cdot (\gamma\rho\vec{v}\vec{v}) = -\gamma\nabla p + \nabla \cdot (\gamma\vec{\tau}) + \gamma\vec{B}_f - \left(\frac{\mu}{\alpha} + \frac{C_{2\rho}}{2}|\vec{v}|\right)\vec{v} \quad (7.2-31)$$

The last term in Equation [7.2-31](#) represents the viscous and inertial drag forces imposed by the pore walls on the fluid.

Note that even when you solve for the physical velocity in Equation [7.2-31](#), the two resistance coefficients can still be derived using the superficial velocity as given in Section [7.2.3](#). **ANSYS FLUENT** assumes that the inputs for these resistance coefficients are based upon well-established empirical correlations that are usually based on superficial velocity. Therefore, **ANSYS FLUENT** automatically converts the inputs for the resistance coefficients into those that are compatible with the physical velocity formulation.



Note that the inlet mass flow is also calculated from the superficial velocity. Therefore, for the same mass flow rate at the inlet and the same resistance coefficients, for either the physical or superficial velocity formulation you should obtain the same pressure drop across the porous media zone.



Multiphase Porous Media

You can simulate porous media multiphase flows using the **Physical Velocity Porous Formulation** to solve the true or physical velocity field throughout the entire flow field, including both porous and non-porous regions. In this approach, assuming a general scalar in the q^{th} phase, ϕ_q , the governing equation in an isotropic porous medium takes on the following form:

$$\frac{\partial(\gamma\alpha_q\rho_q\phi_q)}{\partial t} + \nabla \cdot (\gamma\alpha_q\rho_q\vec{v}_q\phi_q) = \nabla \cdot (\gamma\Gamma_q\nabla\phi_q) + \gamma S_{\phi,q} \quad (7.2-32)$$

Here γ is the porosity, which may vary with time and space; ρ_q is the phase density; α_q is the volume fraction; \vec{v}_q is the phase velocity vector; $S_{\phi,q}$ is the source term; and Γ_q is the diffusion coefficient.

The general scalar equation [7.2-32](#) applies to all other transport equations in the Eulerian multiphase model, such as the granular phase momentum and energy equations, turbulence modeling equations, and the species transport equations.

Assuming isotropic porosity and multiphase flows, the governing equations in the q^{th} phase, [this equation](#) in the separate [Theory Guide](#), [this equation](#) in the separate [Theory Guide](#), and [this equation](#) in the separate [Theory Guide](#) take the general forms described below.

The Continuity Equation

$$\frac{\partial}{\partial t}(\gamma\alpha_q\rho_q) + \nabla \cdot (\gamma\alpha_q\rho_q\vec{v}_q) = \gamma \sum_{p=1}^n (\dot{m}_{pq} - \dot{m}_{qp}) + \gamma S_q \quad (7.2-33)$$

The Momentum Equation

$$\frac{\partial}{\partial t}(\gamma\alpha_q\rho_q\vec{v}_q) + \nabla \cdot (\gamma\alpha_q\rho_q\vec{v}_q\vec{v}_q) = -\gamma\alpha_q\nabla p + \nabla \cdot (\gamma\vec{\tau}_q) + \gamma\alpha_q\rho_q\vec{g} +$$

$$\gamma \sum_{p=1}^n (\vec{R}_{pq} + \dot{m}_{pq}\vec{v}_{pq} - \dot{m}_{qp}\vec{v}_{qp}) + \gamma(\vec{F}_q + \vec{F}_{\text{lift},q} + \vec{F}_{\text{vm},q}) + \alpha_q \left(\frac{\mu}{K} + \frac{C_2\rho}{2}|\vec{v}_q| \right) \vec{v}_q \quad (7.2-34)$$

where the last term in Equation [7.2-34](#) is the momentum resistance (sink) source in a porous medium. It consists of two parts: a viscous loss term, and an inertial loss term. The parameter K is the permeability, and C_2 is the inertial resistance factor. Both K and C_2 are functions of $(1 - \gamma)$.

When $\gamma = 1$, the flow is non-porous and the two loss terms disappear. Details about the user inputs related to the momentum resistance sources can be found in Section [7.2.3](#).

The Energy Equation

$$\frac{\partial}{\partial t}(\gamma\alpha_q\rho_q h_q) + \nabla \cdot (\gamma\alpha_q\rho_q\vec{v}_q h_q) = -\gamma\alpha_q \frac{\partial p_q}{\partial t} + \gamma\vec{\tau}_q : \nabla\vec{v}_q - \nabla \cdot (\gamma\vec{q}_q) + \gamma S_q +$$

$$\gamma \sum_{p=1}^n (Q_{pq} + \dot{m}_{pq}h_{pq} - \dot{m}_{qp}h_{qp}) + Q_{sp} \quad (7.2-35)$$

where Q_{sp} is the heat transfer between the solids surface and the phase q in a porous medium.

Assuming only convective heat transfer, we then have

$$Q_{sp} = (1 - \gamma)\alpha_q h_{q,eff}(T_s - T_q) \quad (7.2-36)$$

where $h_{q,eff}$ is the effective convective heat transfer coefficient, and T_s is the solids surface temperature in the porous medium. It is governed by the heat conduction equation:

$$\frac{\partial}{\partial t}(\rho_s h_s) + \nabla \cdot (\vec{v}_s \rho_s h_s) = \nabla \cdot (k_s \nabla T) - \sum_{p=1}^n Q_{sp} \quad (7.2-37)$$

Equation [7.2-37](#) can be solved as a user-defined scalar (UDS) equation, as described in Section [9.1](#). By default, ANSYS FLUENT assumes that the overall heat transfer between the multiphase fluid and the

solids is in equilibrium. Therefore, instead of solving Equation [7.2-37](#) to obtain the solids surface temperature, we have

$$\sum_{p=1}^n Q_{sp} = 0 \quad (7.2-38)$$

and then

$$T_s = \frac{\sum_{p=1}^n \alpha_p h_{p,eff} T_p}{\sum_{p=1}^n \alpha_p h_{p,eff}} \quad (7.2-39)$$

Solution Strategies for Porous Media

In general, you can use the standard solution procedures and solution parameter settings when your **ANSYS 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, α , 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 fluid cells upstream and/or downstream of the medium, as described in Section [26.9.2](#). 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 dialog box](#)).

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 dialog box](#)) and obtain an initial flow field without the effect of the porous region. With the porous media model turned off, **ANSYS 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 $C_{2i,j}$) 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.

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 dialog boxes.

Note that thermal reporting in the porous region is defined as follows:

$$k_{\text{eff}} = \gamma k_s + (1 - \gamma)k_f \quad (7.2-40)$$

where

γ = porosity of the medium

k_f = fluid phase thermal conductivity (including the turbulent contribution, k_t)

k_s = solid medium thermal conductivity



For porous media involving surface reactions, you can display/report the surface reaction rates using the **Arrhenius Rate of Reaction-n** in the **Reactions...** category of the variable selection drop-down list.

Previous: [7.2.2 Solid Conditions](#)

Up: [7.2 Cell Zone Conditions](#)

Next: [7.2.4 Fixing the Values](#)

Release 12.0 © [ANSYS, Inc.](#) 2009-01-29

[Index](#) [Contents](#) [Previous](#) [Up](#) [Next](#)