# Tutorial: Position Dependent Porous Media

## Introduction

The purpose of this tutorial is to illustrate the modeling of a position dependent porous plug in a two dimensional channel flow with the help of a user-defined function (UDF).

## Prerequisites

This tutorial assumes that you are familiar with the FLUENT interface and that you have a good understanding of the basic setup and solution procedures. Some steps will not be shown explicitly. For more information on UDFs, refer to the FLUENT 6.3 UDF Manual.

## Problem Description

The problem considered in this tutorial is shown schematically in Figure 1.

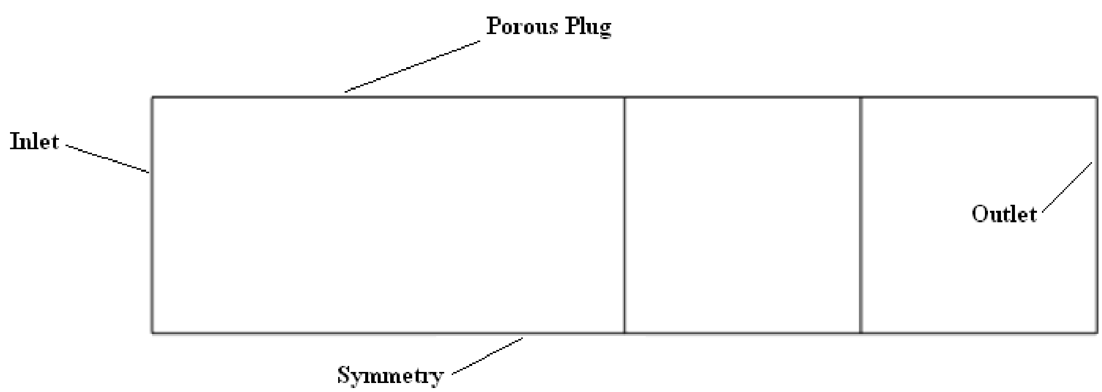



Figure 1: Schematic of the Problem

The domain is divided into two fluid zones. In the fluid cell zone, fluid-2, an X-momentum source term is included for each cell using a UDF. This source term models the effect of a porous plug on the flow in the X-direction and is given by:

$$S_x = -\frac{1}{2} C \rho y |u| u \tag{1}$$

where $C = 100$ is a model constant.

Equation source terms are added using the `DEFINE_SOURCE` macro.

The finite-volume solver of FLUENT expects the source term to be *linearized* according to the following convention:

$$S_\phi = A + B\phi = \underbrace{\left( S^* - \left( \frac{\partial S_\phi}{\partial \phi} \right)^* \phi^* \right)}_{A} + \underbrace{\left( \frac{\partial S_\phi}{\partial \phi} \right)^*}_{B} \phi \tag{2}$$

where the superscript $*$ represents the value at the previous iteration. $B$ (called `dS[eqn]` in the UDF macro) can be coded explicitly by using currently known value of $\phi$, and the entire $S$ is returned by `DEFINE_SOURCE`. For the simple source of X-momentum equation in this problem,

$$B = \frac{\partial S_x}{\partial u} = -C \rho y |u|. \tag{3}$$

See the UDF source code `porous_plug.c` for further details.

## Preparation

1. Copy the files `porous_plug.msh` and `porous_plug.c` to your working folder.

2. Start the 2D (2d) version of FLUENT.

## Setup and Solution

### Step 1: Grid

1. Read the mesh file (`porous_plug.msh`).
2. Check the grid.

   Grid ⟶Check
3. Display the grid (Figure 2).

   Display ⟶Grid...

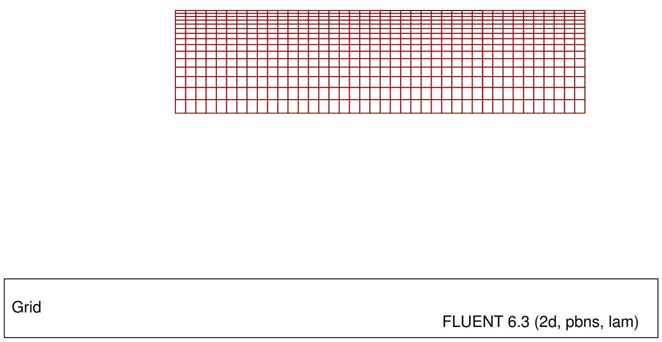



Figure 2: Grid Display

### Step 2: Models

1. Retain the default solver settings.
2. Select the standard k-epsilon (2 eqn) turbulence model with standard wall functions.

   Define ⟶ Models ⟶Viscous...

### Step 3: Materials

*You will use the default fluid properties of air for this problem. Hence, you need not make any changes to the material properties.*

### Step 4: Compile the UDF

*The UDF can be compiled as well as interpreted. In this tutorial you will use the compiled option.*

1. View the UDF.

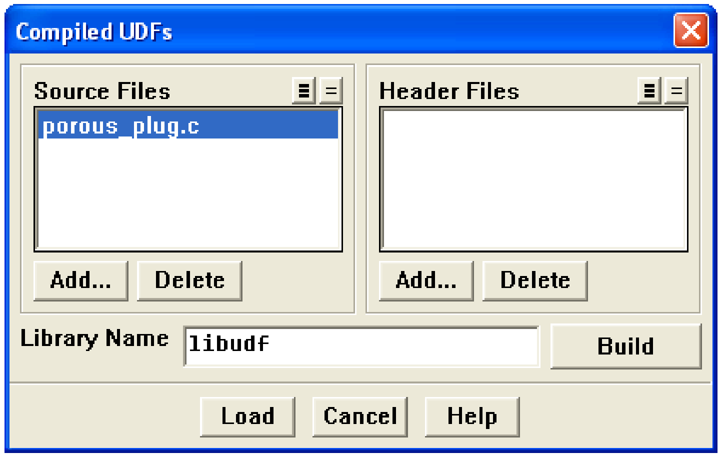
   In a separate editor, view the UDF `porous_plug.c`, and familiarize yourself with its contents to understand the structure and function of the UDF.

   The contents of the UDF file are as follows:

```
/*******************************************************************/
/*                                                                 */
/* Position-Dependent Porous Media                                 */
/*                                                                 */
/*******************************************************************/

#include "udf.h"

DEFINE_SOURCE(xmom_source, cell, thread, dS, eqn)
{
const real c2=100.0;
real x[ND_ND];
real con, source;

C_CENTROID(x, cell, thread);
con = c2*0.5*C_R(cell, thread)*x[1];
source = - con*fabs(C_U(cell, thread))*C_U(cell, thread);
dS[eqn] = - 2.*con*fabs(C_U(cell, thread));

return source;
}
```

2. Compile the UDF.

   Define ⟶ User-Defined ⟶ Functions ⟶Compiled...




   (a) Click the Add... button in the Source Files section to open the Select File dialog panel.

   (b) Select the file, porous_plug.c.

   (c) Click Build.

   *A* Warning *dialog box will be displayed, warning you to make sure that the UDF source files are in the same folder that contains the case and data files. Click* OK *to close the* Warning *dialog box.*

   *You can view the compilation history in the 'log' file that is saved in your working folder.*

   (d) Click Load to load the library.

*For more information on compiled UDFs, refer to the* FLUENT *6.3 UDF Manual.*

### Step 5: Boundary Conditions

Define ⟶Boundary Conditions...

1. Set the boundary conditions for the fluid-2 zone.
   (a) Enable Source Terms.
   (b) Click the Source Terms tab, and click the Edit... button for X Momentum (n/m3) to open the X Momentum (n/m3) sources panel.
      i. Select udf xmom_source::libudf from the drop-down list.
      ii. Click OK to close the X Momentum (n/m3) sources panel.
   (c) Click OK to close the Fluid panel.
2. Set the boundary conditions for the velocity-inlet-1 zone.
   (a) Enter 1 m/s for Velocity Magnitude.
   (b) Select Intensity and Hydraulic Diameter from the Specification Method drop-down list in the Turbulence group box.
   (c) Enter 5 % for Turbulence Intensity and 4 m for Hydraulic Diameter.
   (d) Retain the default values for the other parameters.
   (e) Click OK to close the Velocity Inlet panel.
3. Set the boundary conditions for the pressure-outlet-1 zone.
   (a) Entr 0 Pascal for Gauge Pressure.
   (b) Select Intensity and Viscosity Ratio from the Specification Method drop-down list in the Turbulence group box.
   (c) Enter 5 % for Backflow Turbulence Intensity and 10 for Backflow Turbulent Viscosity Ratio.
   (d) Retain the default values for the other parameters.
   (e) Click OK to close the Pressure Outlet panel.
4. Retain the default settings for the other zones.
5. Close the Boundary Conditions panel.

### Step 6: Solution

1. Initialize the flow field from velocity-inlet-1.

   Solve ⟶ Initialize ⟶Initialize...

2. Enable the plotting of residuals during calculation.

   Solve ⟶ Monitors ⟶Residual...

3. Start by requesting 100 iterations (Figure 3).

   Solve ⟶Iterate...

   *The solution converges in approximately 30 iterations.*

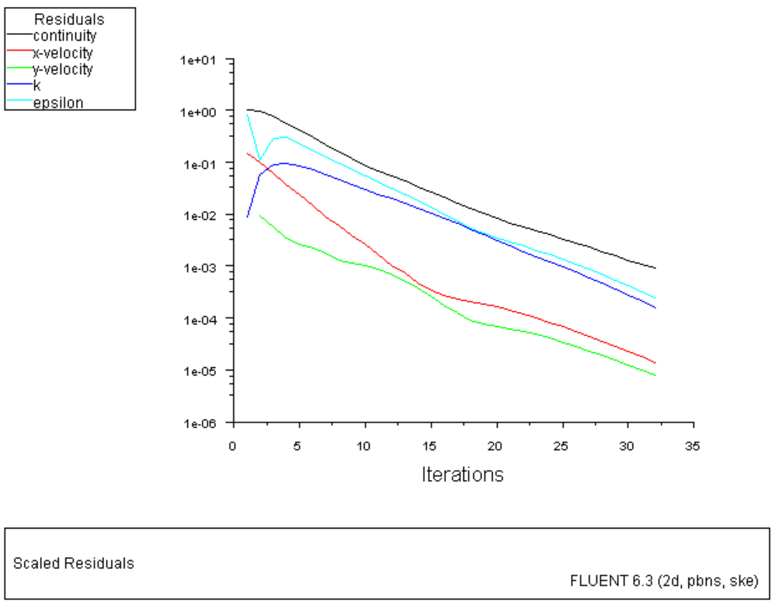



Figure 3: Scaled Residuals

### Step 7: Postprocessing

1. Display velocity vectors (Figure 4).

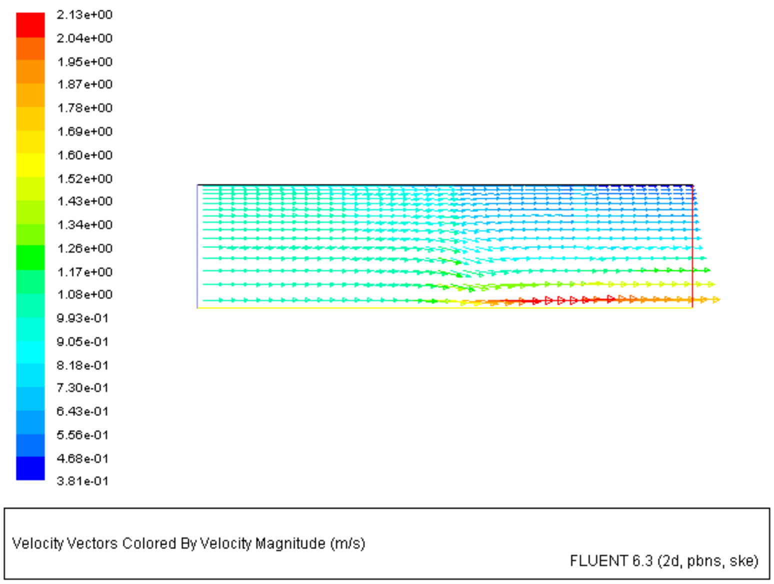



Figure 4: Velocity Vectors Colored by Velocity Magnitude

## Results

Figure 4 shows that the fluid flows preferentially near the bottom of the channel due to the effect of the source term on the axial momentum of the fluid.

## Summary

This tutorial demonstrated the use of UDFs for specifying source terms. User-defined source terms can be a useful way of introducing additional physics into the simulation, and can usually be modeled as sources or sinks of mass, momentum, energy, species, etc.

**Extra:** *When you are comfortable with the present exercise, you can try modifying the UDF to specify momentum sources for both the X and Y momentum equations. The present UDF can be copied and suitably modified for the Y momentum equation. Both functions will then reside in the UDF file, and be accessible to the solver upon compilation.*