Tutorial: Temperature Dependent Viscosity

Introduction

This tutorial examines the flow of liquid metal through a two dimensional channel. The viscosity of the liquid metal is modeled as a function of the temperature using a user-defined function (UDF).

This tutorial demonstrates how to do the following:

- Interpret the UDF.
- Use the UDF for specifying the user-defined property.
- Postprocess the resulting data.

Prerequisites

This tutorial is written with the assumption that you have completed Tutorial 1 from ANSYS FLUENT 12.0 Tutorial Guide, and that you are familiar with the ANSYS FLUENT navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

For more details about UDFs, see ANSYS FLUENT 12.0 UDF Manual.

Problem Description

The problem considered in this tutorial is shown schematically in Figure 1. As the symmetry condition is imposed at the centerline, only half the channel is modeled, The wall of the channel is split into two parts: `wall-2`, which has a temperature of 280 K and `wall-3`, which has a temperature of 290 K. The temperature-dependent viscosity of the liquid metal will respond to this change in wall temperature.

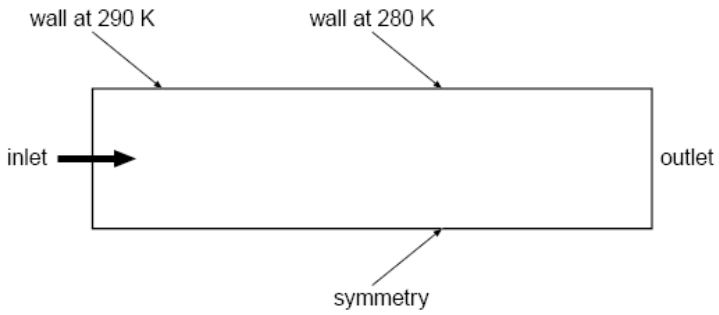


Figure 1: Schematic of the Problem

Strategy

The function `cell_viscosity` is defined on a cell using `DEFINE_PROPERTY`. Two real variables are introduced:

- `temp` = the value of `C_T` (cell, thread).
- `mu` = laminar viscosity (computed by the function).

The value of the tempertaure is checked and based upon its range, the appropriate value of `mu` is computed. At the end of the function, the computed value for `mu` is returned to the solver.

The molecular viscosity of the liquid metal will be defined as a function of temperature which is given as follows:

$$\mu = \left\{ \begin{array}{cc} 5.5 \times 10^{-3} \text{ kg/m-s} & \text{T} > 288 \text{ K} \\ 143.2135 - 0.49725 \text{ T kg/m-s} & 286 \leq \text{T} \leq 288 \text{ K} \\ \text{T} < 286 \text{ K} & 1 \text{ kg/m-s} \end{array} \right\} \tag{1}$$

where,

T = temperature of the fluid (K)

μ = molecular viscosity of the fluid (kg/m-s).

Setup and Solution

Preparation

1. Copy the files (`user-vis.msh` and `viscosity.c`) to your working folder.

2. Use FLUENT Launcher to start the 2D version of ANSYS FLUENT.

   For more information about FLUENT Launcher *see Section* 1.1.2, Starting ANSYS FLUENT Using FLUENT Launcher *in* ANSYS FLUENT *12.0 User's Guide.*

3. Enable Double-Precision in the Options list.

4. Click the UDF Compiler tab and ensure that the Setup Compilation Environment for UDF is enabled.

   The path to the .bat file which is required to compile the UDF will be displayed as soon as you enable Setup Compilation Environment for UDF.

   If the UDF Compiler *tab does not appear in the* FLUENT Launcher *dialog box by default, click the* Show More >> *button to view the additional settings.*

   The Display Options *are enabled by default. Therefore, after you read in the mesh, it will be displayed in the embedded graphics window.*

Step 1: Mesh

1. Read the mesh file (`user-vis.msh`).

   File → Read → Mesh...

   As the mesh file is read, ANSYS FLUENT *will report the progress in the console.*

Step 2: General Settings

1. Retain the default solver settings.

   General

2. Check the mesh (see Figure 2).

   General → Check

   ANSYS FLUENT *will perform various checks on the mesh and will report the progress in the console. Make sure the minimum volume reported is a positive number.*

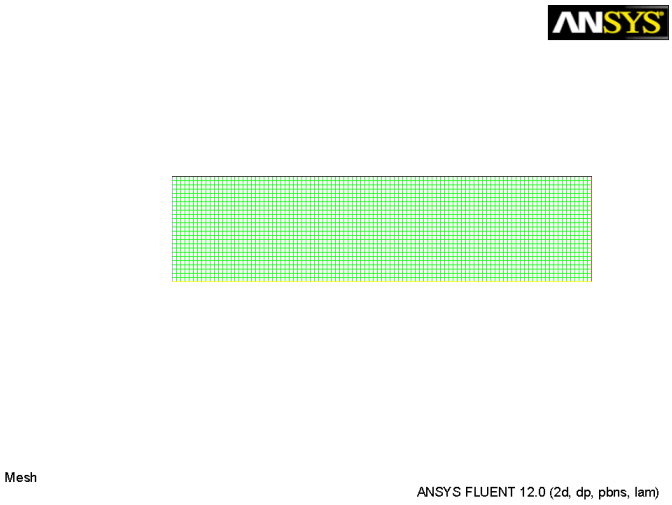


Figure 2: Mesh Display

Step 3: Models

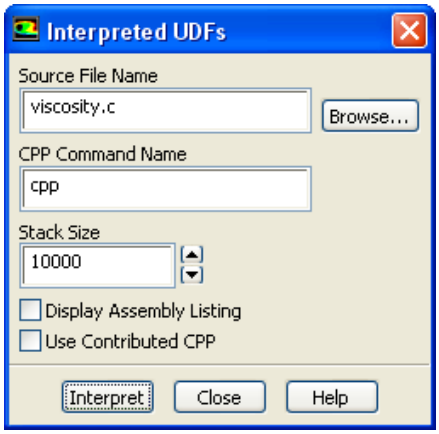
Enable energy equation.

Models → Energy → Edit...

Step 4: User-Defined Functions

Interpret the UDF.

Define → User-Defined → Functions → Interpreted...



1. Click the Browse... button to open the Select File dialog dialog box.
2. Select the file viscosity.c.
3. Specify the C preprocessor to be used for CPP Command Name.

If you want to use the C preprocessor that ANSYS, Inc. has supplied, you can enable the Use Contributed CPP *option.*

4. Retain the default value of 10000 for Stack Size.

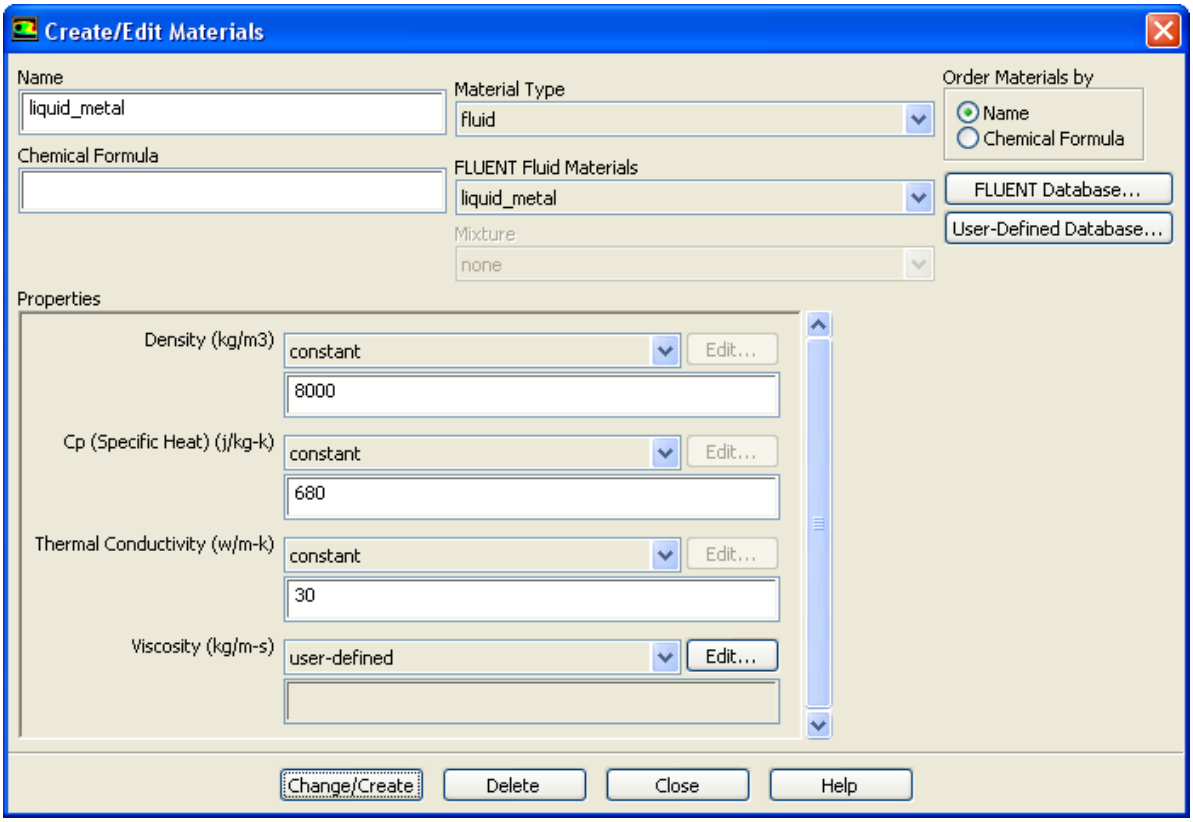
   The Stack Size *should be* 10000 *unless the number of local variables in your function causes the stack to overflow. Its value should be set to a number that is greater than the number of local variables used.*

5. Click Interpret and close the Interpreted UDFs dialog box.

Step 5: Materials

Modify the fluid material.

Materials → Fluid → Create/Edit...



1. Change Name to `liquid_metal`.
2. Enter 8000 kg/m^3 for Density.
3. Enter 680 j/kg-k for Cp (Specific Heat).
4. Enter 30 w/m-k for Thermal Conductivity.

5. Select user-defined from the Viscosity drop-down list and click OK to close User-Defined Functions dialog box.

   A Question *dialog box opens, asking whether to overwrite* air. *Click* Yes.

6. Click Change/Create and close Create/Edit Materials dialog box.

Step 6: Boundary Conditions

1. Set the boundary conditions for wall-2.

   Boundary Conditions → wall-2 → Edit...

   (a) Click Thermal tab and select Temperature from the Thermal Conditions list.

   (b) Enter 280 K for Temperature.

   (c) Click OK to close the Wall dialog box.

2. Set the boundary conditions for wall-3.

   Boundary Conditions → wall-3 → Edit...

   (a) Click Thermal tab and select Temperature from the Thermal Conditions list.

   (b) Enter 290 K for Temperature.

   (c) Click OK to close the Wall dialog box.

3. Set the boundary conditions for velocity-inlet-6.

   Boundary Conditions → velocity-inlet-6 → Edit...

   (a) Select Components from the Velocity Specification Method drop-down list.

   (b) Enter `0.001` m/s for X-Velocity.

   (c) Click Thermal tab and enter 290 K for Temperature.

   (d) Click OK to close the Velocity Inlet dialog box.

4. Set the boundary conditions for pressure-outlet-7.

   Boundary Conditions → pressure-outlet-7 → Edit...

   (a) Click Thermal tab and enter 290 K for Backflow Total Temperature.

   (b) Click OK to close the Pressure Outlet dialog box.

Step 7: Solution

1. Initialize the flow field from velocity-inlet-6.

   Solution Initialization

2. Start the calculation for 300 iterations (see Figure 3).

   Run Calculation

   The solution converges in approximately 200 iterations.

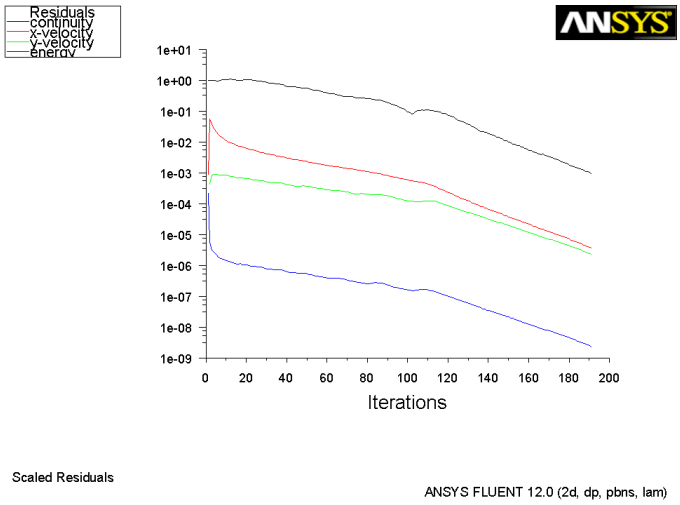


Figure 3: Scaled Residuals

Step 8: Postprocessing

1. Display filled contours of molecular viscosity.

   Graphics and Animations → Contours → Set Up...

   (a) Enable Filled in the Options list.

   (b) Select Properties... and Molecular Viscosity from Contours of drop-down list.

   (c) Click Display (Figure 4) and close the Contours dialog box.

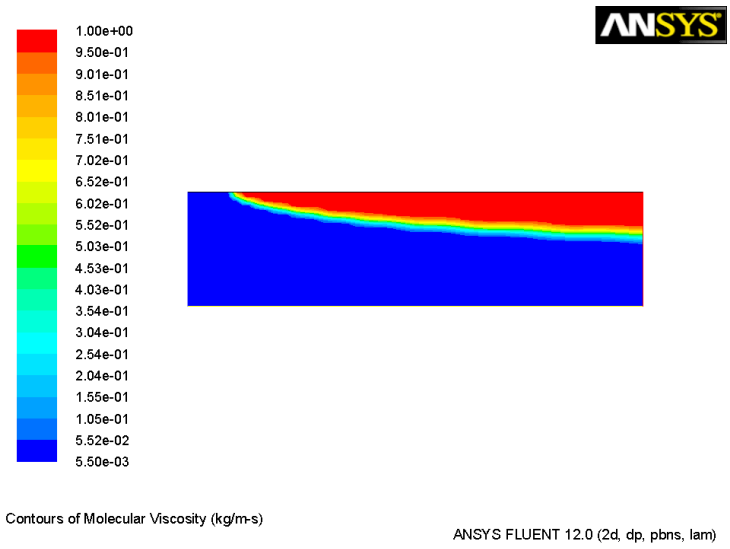


Figure 4: Contours of Molecular Viscosity

Appendix

The UDF (`viscosity.c`) is used to specify the temperature-dependent molecular viscosity in each cell. You can open the UDF in a separate editor and view its contents to understand the structure and function.

The contents of the UDF file are as follows:

```
/************************************************************/
/*                                                          */
/* User-Defined Function for temperature-dependent viscosity */
/*                                                          */
/* Author: Liz Marshall                                     */
/*                                                          */
/************************************************************/

#include "udf.h"


DEFINE_PROPERTY(user_vis, cell, thread)

{

  float temp, mu;

  temp = C_T(cell, thread);

  {

/* If the temperature is high, use a small, constant viscosity */

  if ( temp > 288. )
      mu = 5.5e-3;

  else if ( temp >= 286. )
      mu = 143.2135 - 0.49725 * temp;

  else
      mu = 1.0;
}
  return mu;
}
```

The above equations are applied to each and every cell which is associated to the thread (in this case a fluid zone). This UDF will be called from the Create/Edit Materials dialog box where the user-defined is specified for Viscosity.

Results

Figure 4 shows that when the warmer fluid enters the channel from the left and encounters the cooler wall further on, its viscosity increases according to the user-defined viscosity relation.

Summary

This tutorial demonstrated the use of UDFs for specifying a user-defined property.

Note: *This capability is available only for viscosity and thermal conductivity (density and specific heat cannot be specified through UDFs).*

Extra: *When you are comfortable with this exercise, you can try modifying the UDF to specify temperature-dependent thermal conductivity. You can copy the source code for the viscosity UDF and change the appropriate lines to calculate thermal conductivity. You need to access the thermal conductivity UDF in the* Create/Edit Materials *dialog box in the same way as viscosity.*