Tutorial 11. Use of User-Defined Scalars and User-Defined Memories for Modeling Ohmic Heating

Introduction

The purpose of this tutorial is to illustrate the use of user-defined scalars (UDS) and user defined memories (UDM) for modeling the electric resistance heating of fluids.

Ohmic heating is an advanced food processing method used to heat liquid foods, where electricity is passed through the liquid food itself. Through this process, the electrical energy is converted to heat energy. Conventional food processing heating methods can damage food quality due to the relatively slow energy transfer rate and significant temperature gradients associated with conduction and convection driven heat transfer. In comparison, Ohmic heating uniformly heats the entire mass, ensuring a product of better quality.

In this tutorial you will learn how to:

- Use the UDS for modeling the electrical current continuity equation.
- Use the UDM for storing the data at each cell center.
- Use the source terms to model the volumetric heating.
- Setup the solver and perform iterations.
- Check the convergence.
- Examine the results.
- Perform postprocessing of UDS and UDM.

Prerequisites

This tutorial assumes that you have little experience with FLUENT but are familiar with the interface.

Problem Description

Consider a 2D ohmic heater with water as a working fluid. The fluid passes through the serpentine duct as shown in Figure 11.1. The opposite walls of the duct are maintained at different electrical potential. The electrical current continuity equation (solved using the UDS) is given in terms of the electric potential (ϕ) as follows:

$$\nabla \cdot (\sigma \nabla \phi) = 0$$

where, σ is electrical conductivity.

The current density vector (J) is related to the electric potential distribution as follows:

$$J = -\sigma \nabla \phi$$

Heat generated due to the dissipation of electric energy is calculated using Ohm's law and stored in User Memory 0. The volumetric rate of heat generation (q) is calculated as:

$$q = \frac{J \cdot J}{\sigma}$$

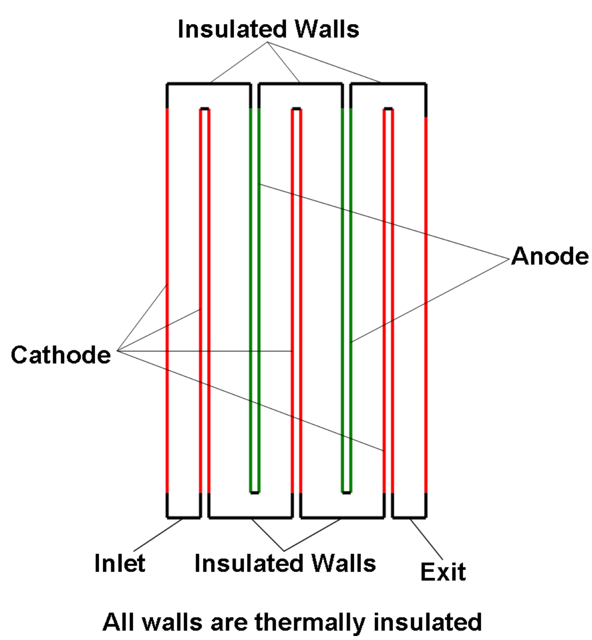


Figure 11.1: Problem Schematic

© Fluent Inc. August 22, 2006

Preparation

1. Copy the mesh file, `ohmic_heater.msh`, and the directory, `libudf`, to your working directory.
2. Start the 2D double precision solver of FLUENT.

Setup and Solution

Step 1: Grid

1. Read the mesh file, `ohmic_heater.msh`.

   File ⟶ Read ⟶ Case...

   FLUENT *will read the mesh file and report the progress in the console.*

2. Check the grid.

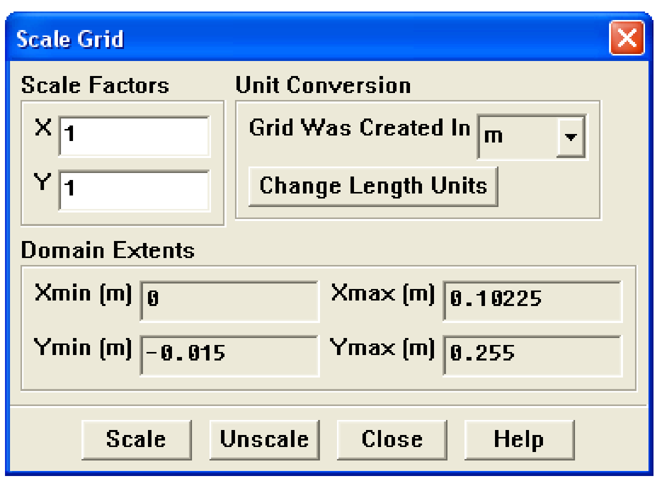
   Grid ⟶ Check

   This procedure checks the integrity of the mesh. Make sure the reported minimum volume is a positive number.

3. Check the scale of the grid.

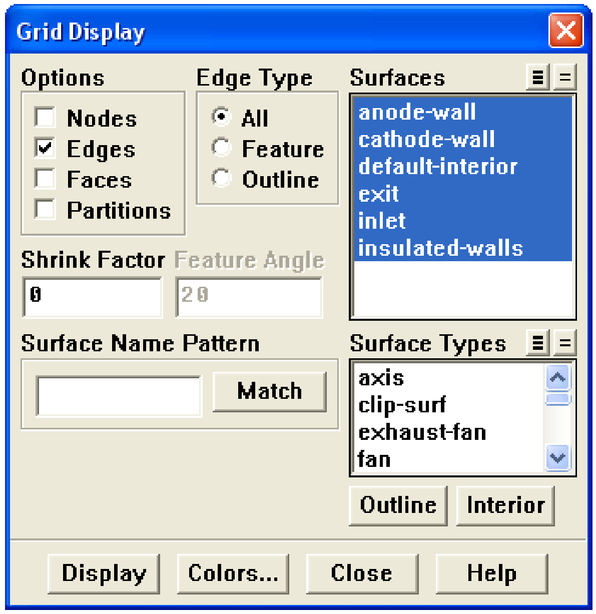
   Grid ⟶ Scale...

   Check the domain extents to see if they correspond to the actual physical dimensions. If they do not, then the grid has to be scaled with proper units. In this case, there is no need to scale the grid.



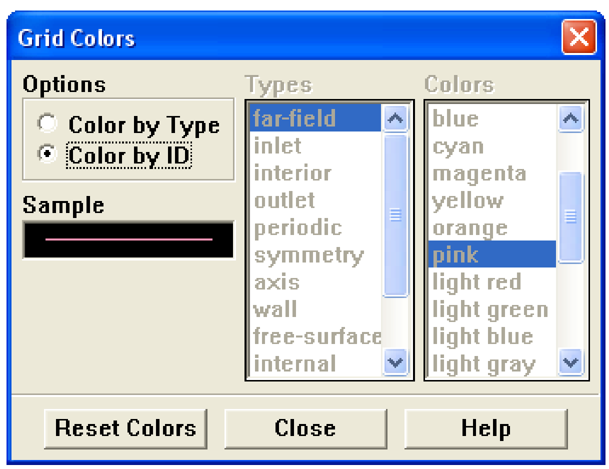
4. Display the grid (Figure 11.2).

   Display →Grid...



   (a) Click Colors....

   The Grid Colors *panel opens.*



   i. Under Options, enable Color by ID.

   ii. Close the panel.

   (b) In the Grid Display panel, click Display.

© Fluent Inc. August 22, 2006

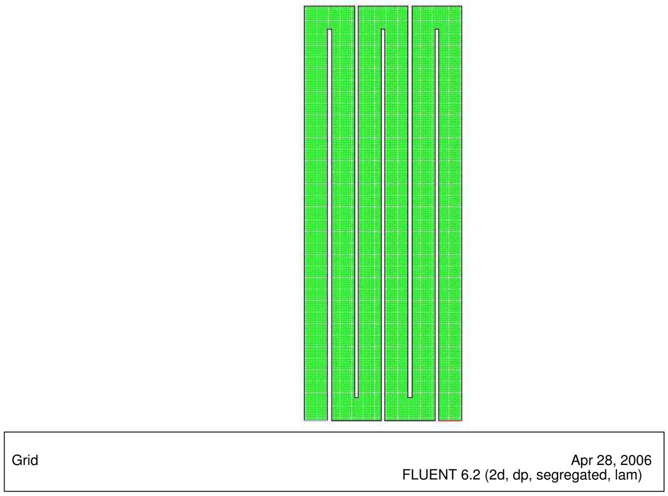
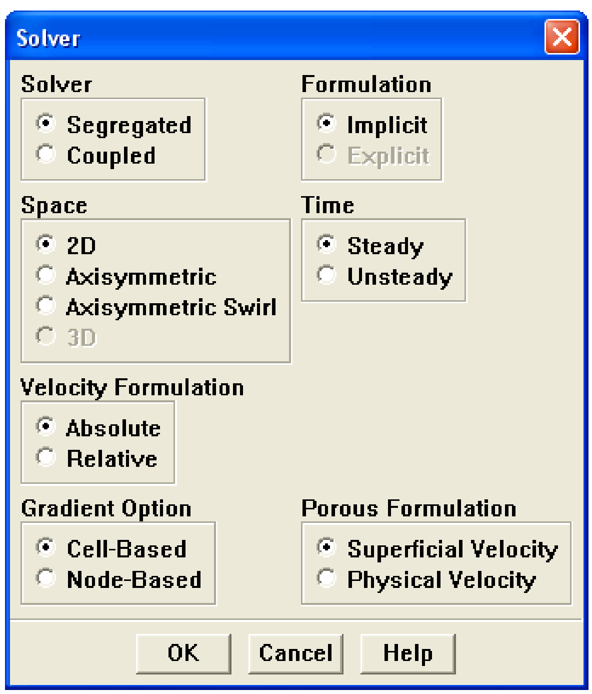


Figure 11.2: Grid Display

Step 2: Models

1. Retain the default solver settings.

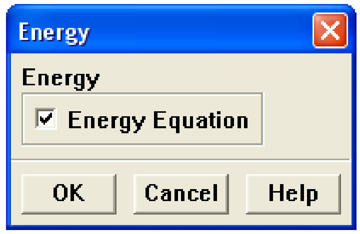
   Define ⟶ Models ⟶Solver...



The problem is to be solved in steady state with 2D laminar conditions.

2. Enable heat transfer by activating the energy equation.

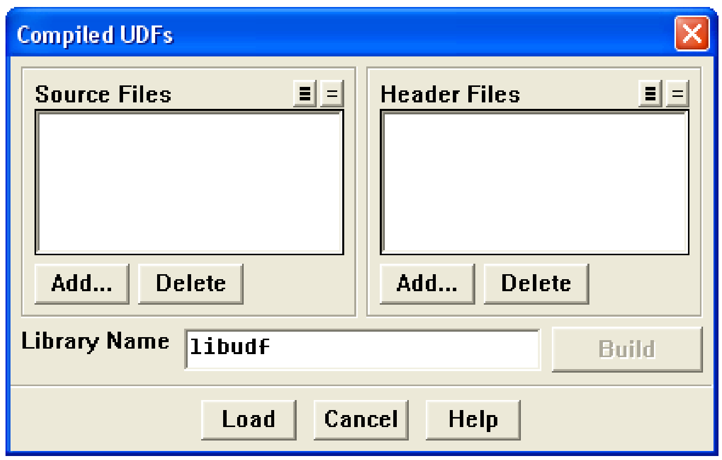
   Define ⟶ Models ⟶ Energy...



Step 3: UDF Library

1. Load the UDF library.

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



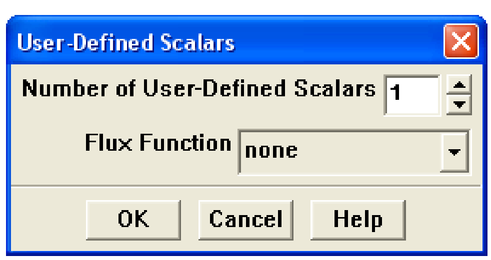
   (a) Click **Load** to load the UDF library.

   The heat generated due to ohmic heating is calculated in this UDF. A compiled UDF library named `libudf` *is created for this purpose.*

Step 4: Define UDS

1. Include UDS in the case setup.

   Define ⟶ User-Defined ⟶ Scalars...



 © Fluent Inc. August 22, 2006

(a) Increase Number of User-Defined Scalars to 1.

(b) Keep the default selection of none in the Flux Function drop-down list.

Flux Function *defines the convection flux for UDS transport. In this case, it is assumed that current convection is negligible, therefore, no need to specify any function.*

(c) Click OK in the User Defined Scalars panel.

An information dialog box pops up with the message Available material properties or methods have changed. Please confirm the property values before continuing.

(d) Click OK to close the dialog box.

Since the UDS is enabled, UDS diffusivity will be required. You will set it in Step 6.

By enabling this feature, FLUENT *solves the transport equation for an arbitrary UDS. The UDS equation is solved in the same way as* FLUENT *solves transport equation for any other scalar (e.g., temperature, species mass fraction).*

Step 5: Define UDM

1. Specify appropriate UDM.

Define ⟶ User-Defined ⟶ Memory...

(a) Increase Number of User-Defined Memory Locations to 1.

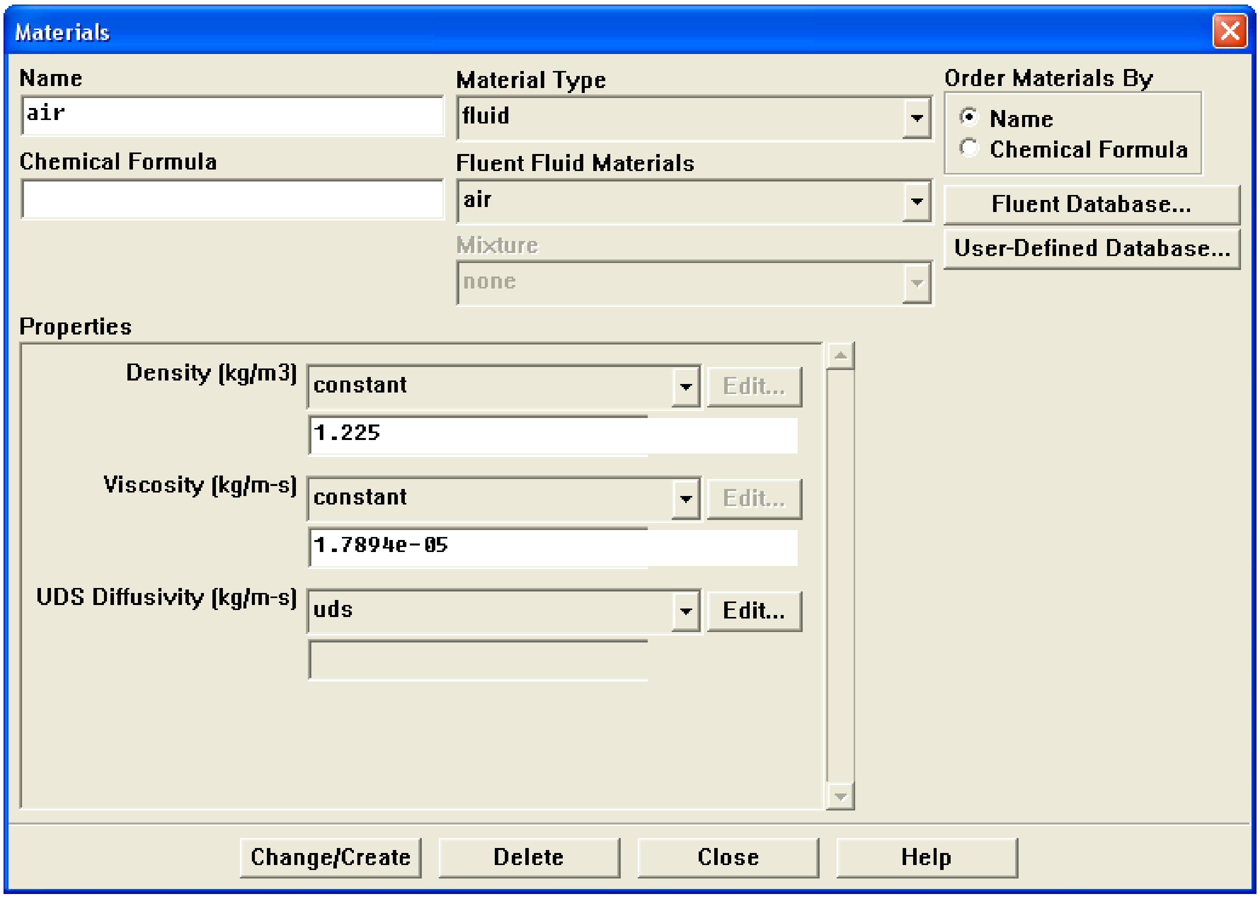
UDMs can store the variables at each cell center and face. These stored values can be used for postprocessing or by other UDFs. In this tutorial, the dissipated electric energy is stored in UDM. The UDM is used for postprocessing the distribution of the volumetric heat source and also for defining a source for the energy equation.

(b) Click OK.

Step 6: Materials

1. Add liquid water to the list of fluid materials by copying it from the materials database.

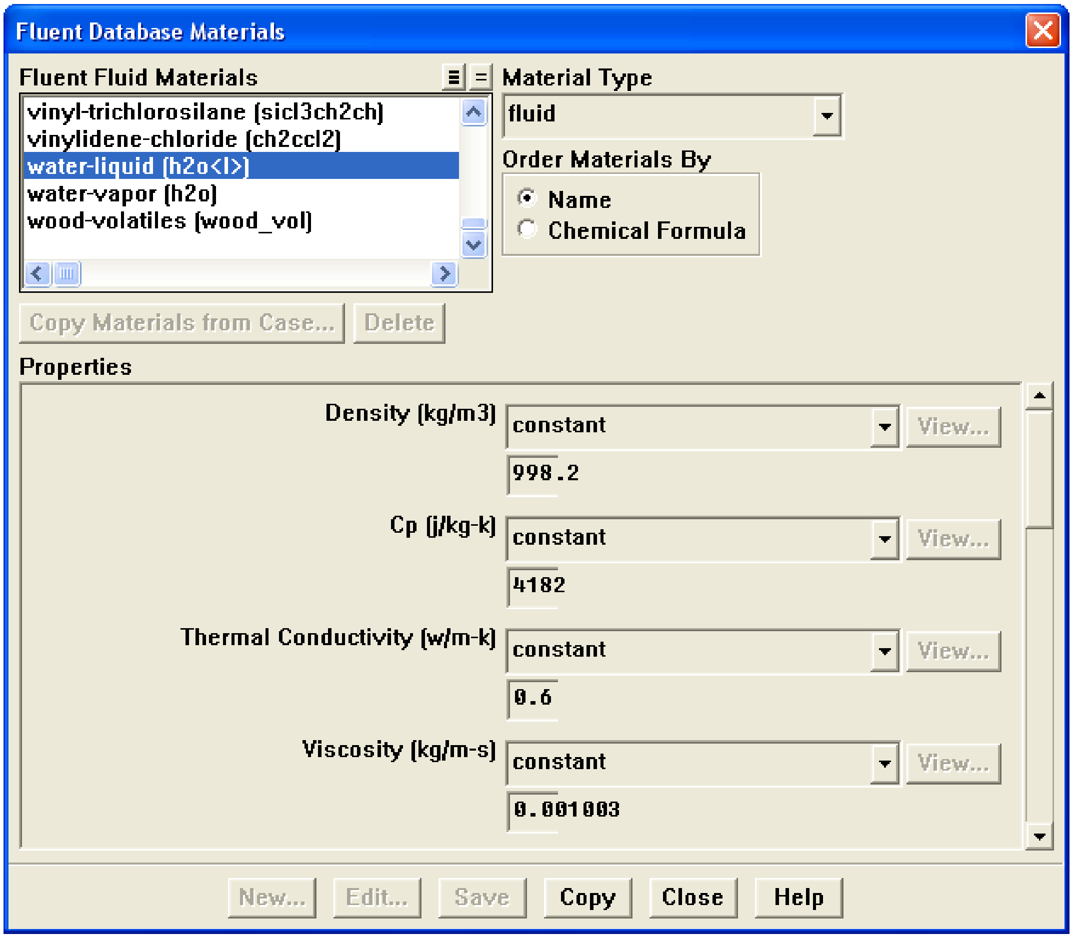
   Define ⟶Materials...



   (a) Click Fluent Database....

   Fluent Database Materials *panel opens.*

 © Fluent Inc. August 22, 2006



i. Select water-liquid ($h2o < l >$) from the Fluent Fluid Materials list.

Hint: *Scroll-down to view* water-liquid ($h2o < l >$).

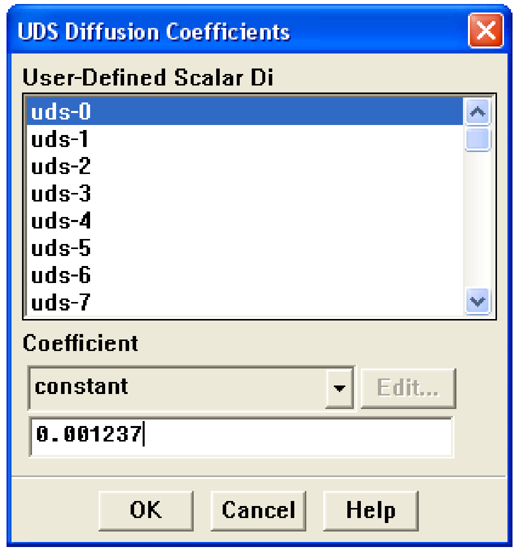
ii. Click Copy and close the panel.

(b) Define the diffusion coefficient for current density equation i.e., `UDS-0`.

i. In the Materials panel, select water-liquid ($h2o < l >$) from the Fluent Fluid Materials drop-down list.

ii. Under Properties, click Edit... to the right of UDS Diffusivity.

The UDS Diffusion Coefficients *panel opens.*



A. Select uds-0 in the User-Defined Scalar Di list.

B. Keep the default selection of constant option under Coefficient drop-down list and set the value to 0.001237.

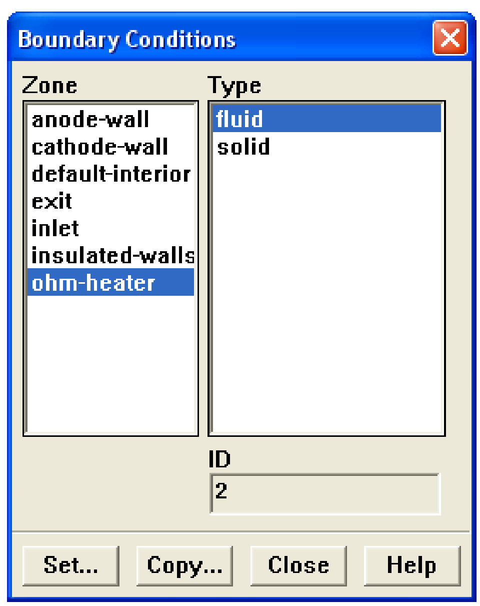
C. Click OK to close the panel.

(c) Click Change/Create and close the Materials panel.

Step 7: Boundary Conditions

1. Set the boundary condition for `ohm-heater`.

   Define ⟶Boundary Conditions...



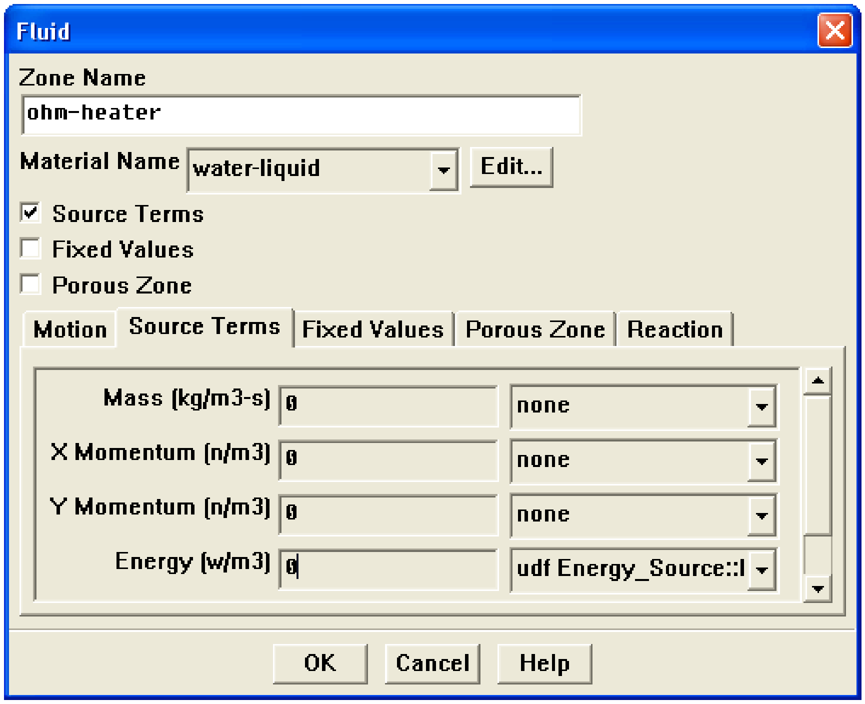
© Fluent Inc. August 22, 2006

(a) Under Zone, select ohm-heater.

The Type *will be reported as* fluid.

(b) Click Set....

The Fluid *panel opens.*



(c) In the Material Name drop-down list, select water-liquid.

(d) Enable Source Terms and click the Source Terms tab.

(e) Select udf Energy_Source::libudf in the Energy drop-down list.

(f) Click OK to close the panel.

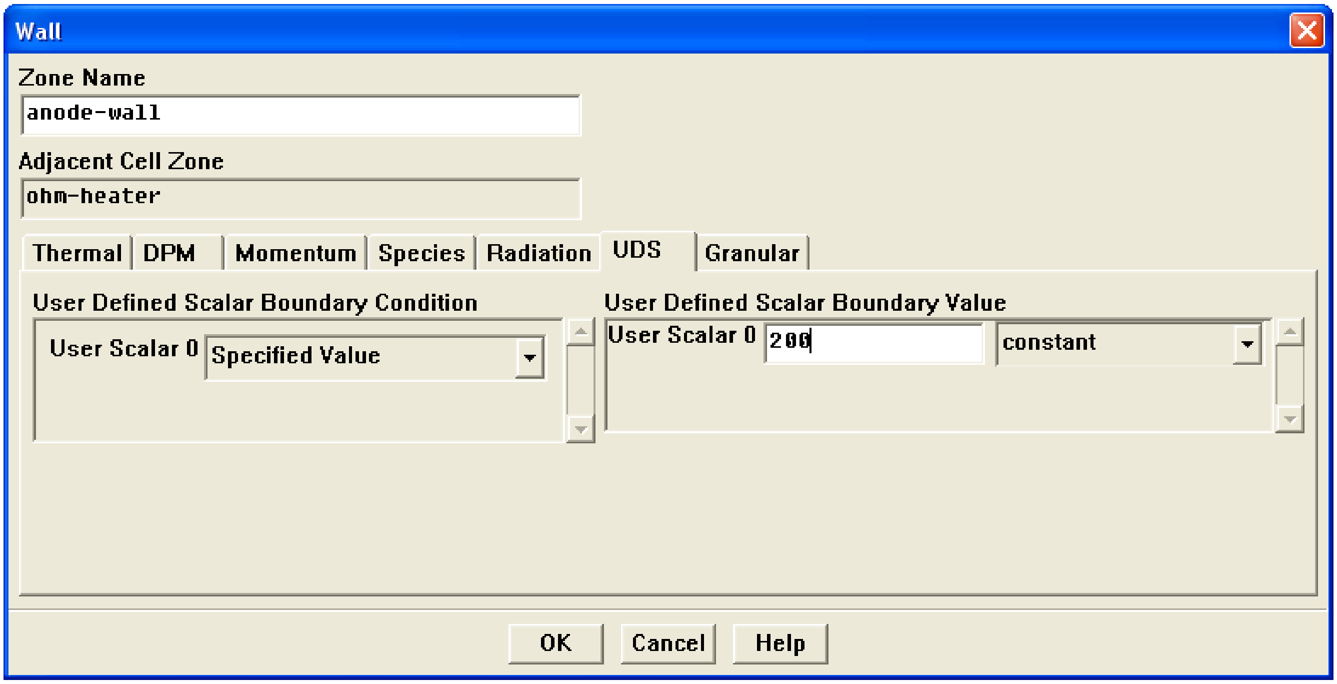
2. Set the boundary conditions for `anode-wall`.

(a) Under Zone, select anode-wall.

The Type *will be reported as* wall.

(b) Click Set....

The Wall *panel opens.*



i. Keep the default conditions under Thermal and Momentum tabs.

ii. Click the UDS tab.

iii. Select Specified Value in the User Scalar 0 drop-down list.

iv. Under User Defined Scalar Boundary Value, set User Scalar 0 to 200.

v. Click OK to close the panel.

3. Set the boundary conditions for `cathode-wall`.

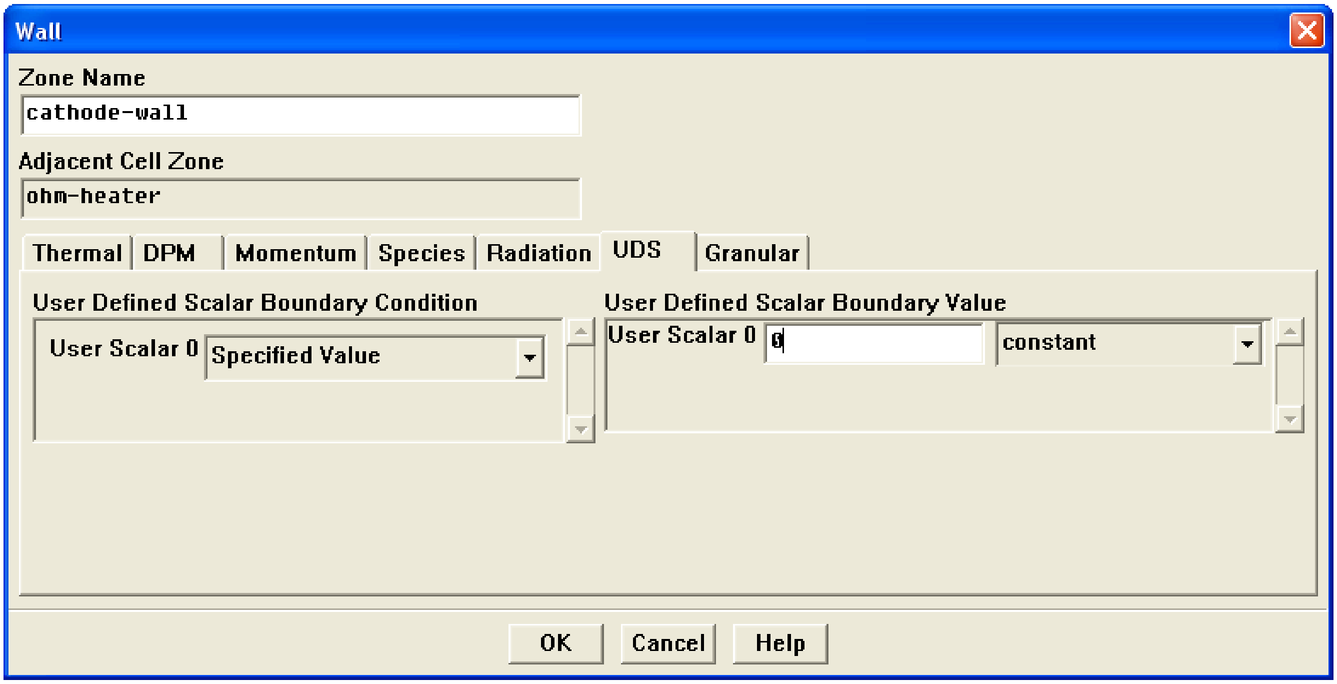
(a) Under Zone, select `cathode-wall`.

The Type *will be reported as* wall.

© Fluent Inc. August 22, 2006

(b) Click Set....

The Wall *panel opens.*



i. Keep the default conditions under Thermal and Momentum tabs.

ii. Click the UDS tab.

iii. Select Specified Value in the User Scalar 0 drop-down list.

iv. Under User Defined Scalar Boundary Value, keep 0 for User Scalar 0.

v. Click OK to close the panel.

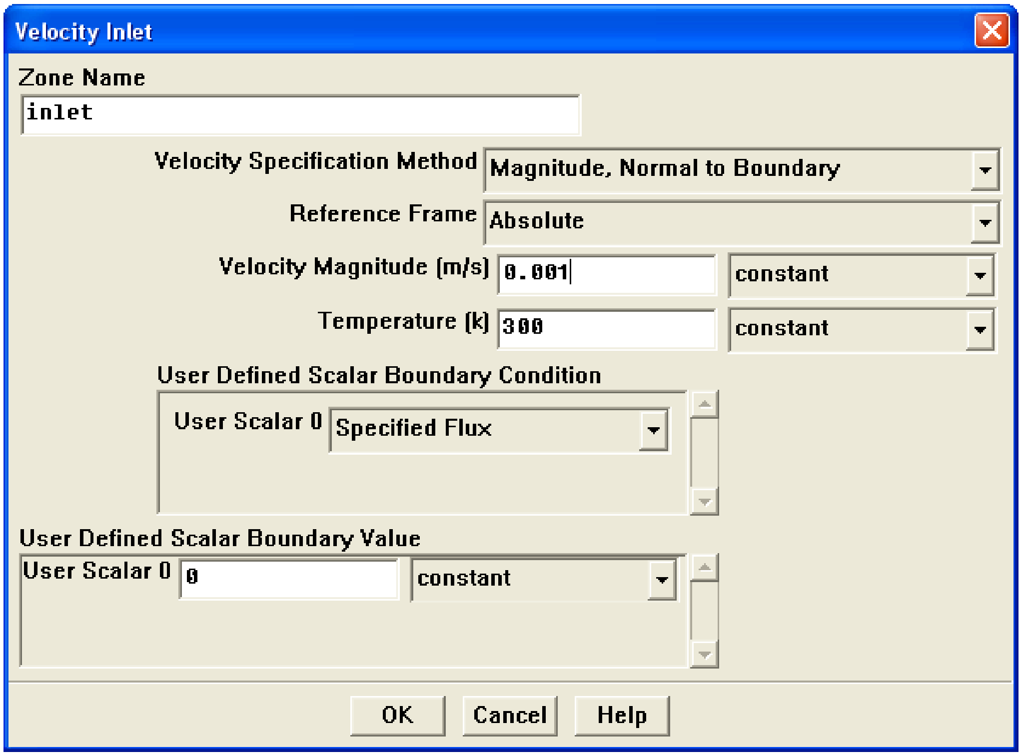
4. Set the boundary conditions for inlet.

(a) Under Zone, select `inlet`.

The Type *will be reported as* velocity-inlet.

(b) Click Set....

The Velocity Inlet *panel opens.*



i. Set Velocity Magnitude to `0.001`.

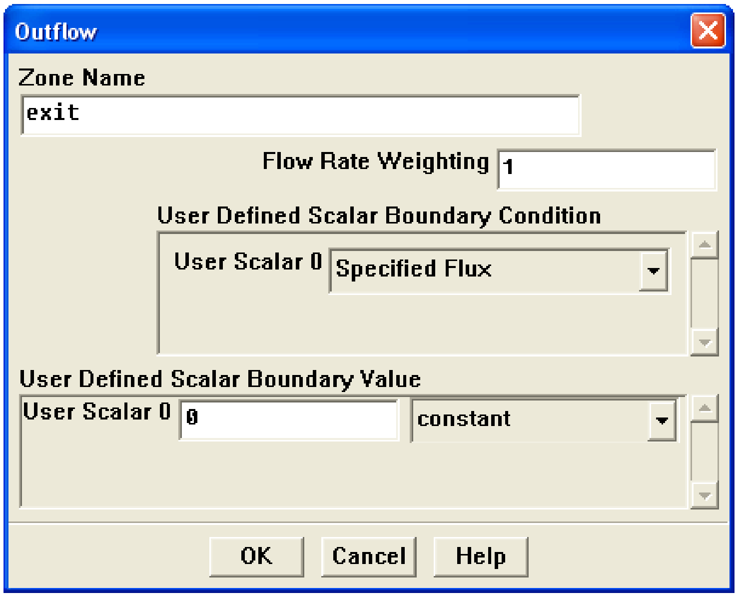
ii. Set Temperature to 300.

iii. Keep the default boundary conditions for UDS.

iv. Click OK to close the panel.

5. Keep the default boundary condition for `exit`.

© Fluent Inc. August 22, 2006

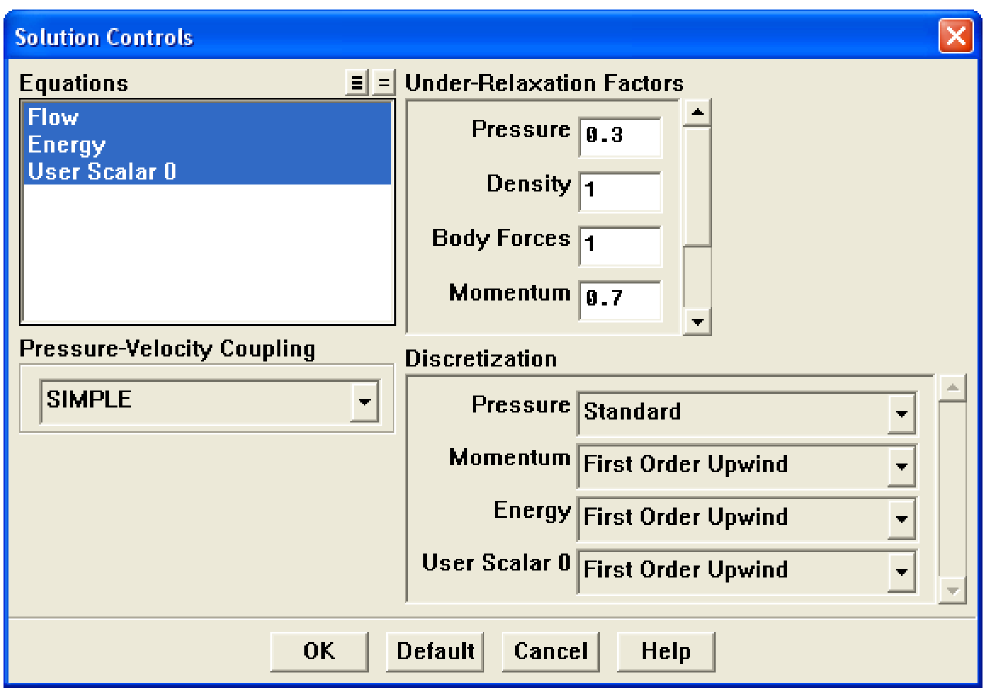


6. Close the Boundary Conditions panel.

Step 8: Solution

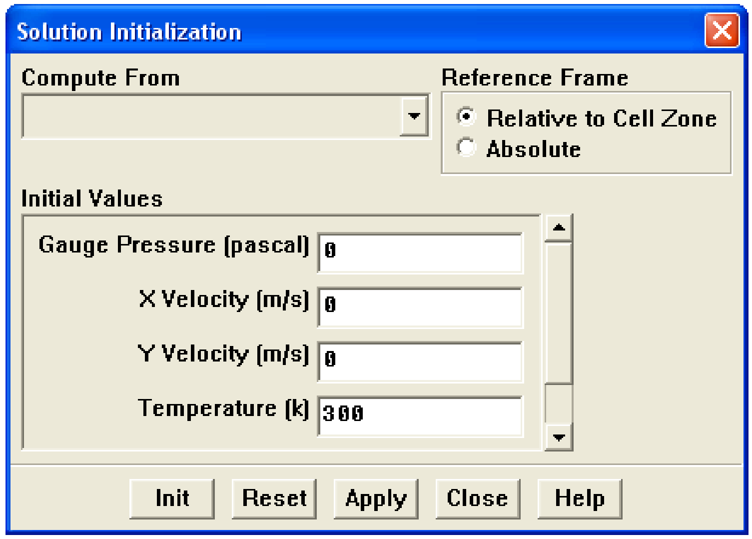
1. Keep the default solution settings.

   Solve ⟶ Controls ⟶ Solution...



2. Initialize the flow.

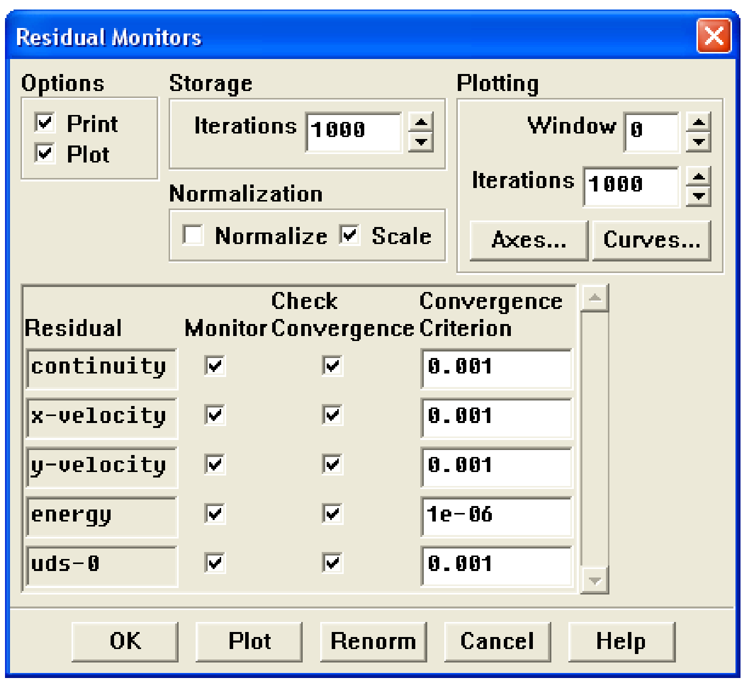
   Solve ⟶ Initialize ⟶ Initialize...



   (a) Click Init and close the panel.

3. Enable the plotting of residuals during the calculation.

   Solve ⟶ Monitors ⟶ Residuals...



   (a) Under Options, enable Plot.

© Fluent Inc. August 22, 2006

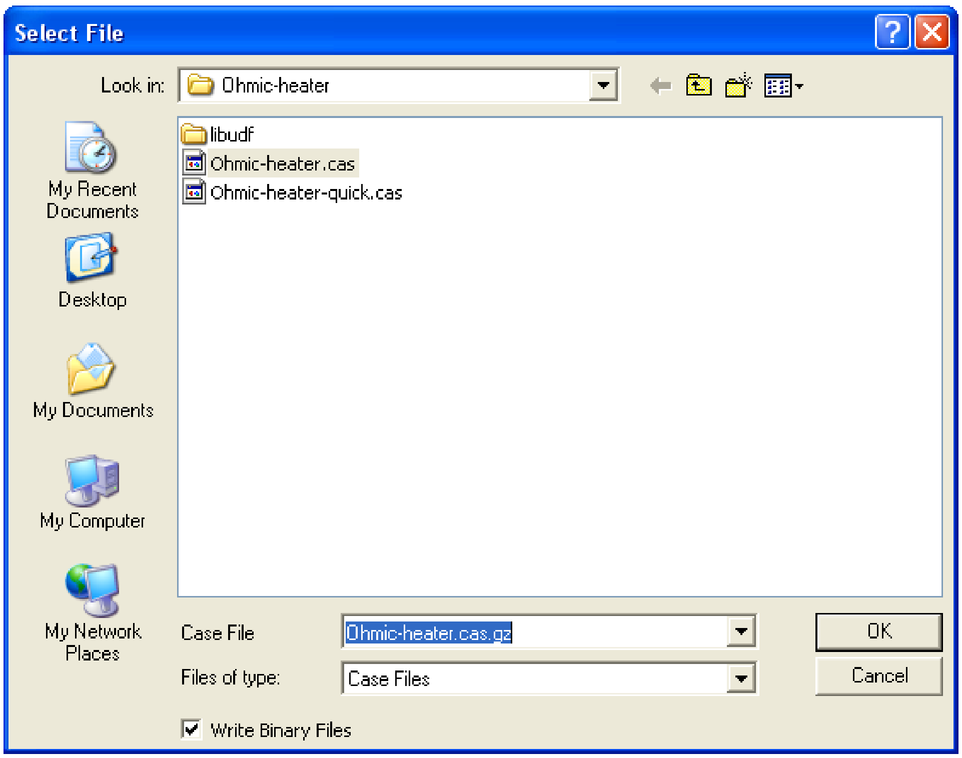
(b) Click OK to accept the settings and close the panel.

By default, all variables will be monitored and checked to determine the convergence of the solution.

4. Save the case file (`ohmic-heater.cas.gz`).

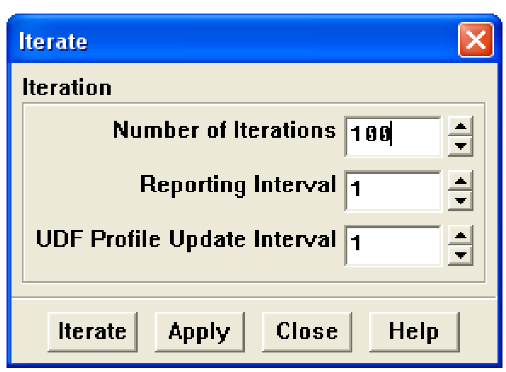
File ⟶ Write ⟶ Case...

Retain the default Write Binary Files *option so that you can write a binary file. The* `.gz` *option will save compressed files on both* Windows *and* **UNIX** *platforms.*



5. Start the calculation by requesting 100 iterations.

Solve ⟶ Iterate...



(a) Set Number of Iterations to 100.

(b) Click Iterate.

The solution converges in about 33 iterations with default convergence criteria. The number of iterations required for convergence varies according to the platform used. Also, the residual values are different for different computers, therefore, the residual plot that you will get may not be exactly the same as Figure 11.3.

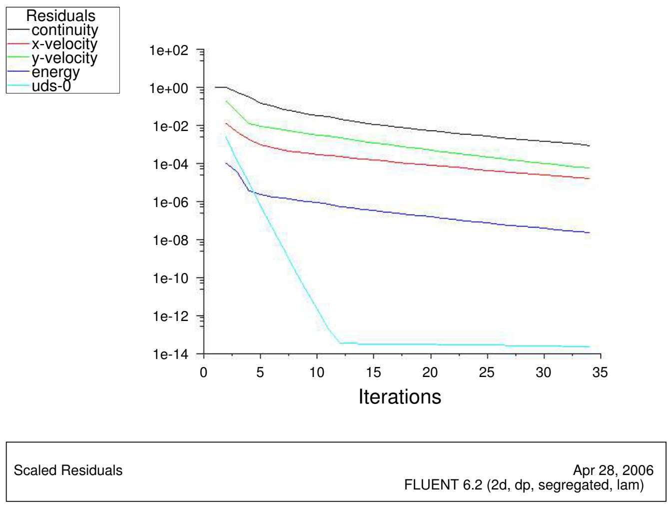


Figure 11.3: Scaled Residuals

© Fluent Inc. August 22, 2006

Step 9: Check for Convergence

There are no universal metrics for judging convergence. The unconverged results may be very misleading. The residual definitions that are useful for one class of problem are sometimes not suitable for other classes of problems. Therefore, it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities and checking for mass and energy balances.

There are three methods to check the convergence:

- Monitoring the residuals.

  Convergence occurs when the convergence criterion for each variable is reached. The default criterion is that each residual will be reduced to a value less than 1e-3, except the energy residual, for which the default criterion is 1e-6. These criteria are useful for a wide range of problems, but at times, it may be required to tighten these criteria, based on the validity of other convergence checks.

- Overall mass, momentum, energy and scalar balances are obtained.

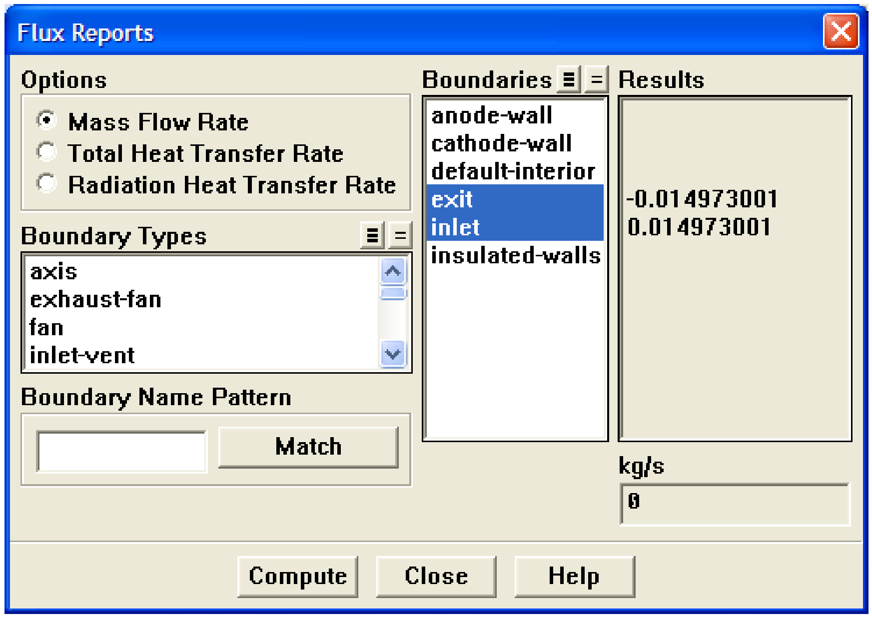
  Check the overall mass, momentum, energy and scalar balances in the Flux Reports *panel. The net imbalance should be less than 0.2% of the net flux through the domain.*

- When the solution no longer changes with iterations.

  Sometimes the residuals may not fall below the convergence criteria set in the case setup. However, monitoring the representative flow variables through iterations may show that the residuals have stagnated and do not change with further iterations. This could also be considered a convergence check.

1. Check global mass balance.

   $\boxed{\text{Report}}$ $\longrightarrow$ Fluxes...



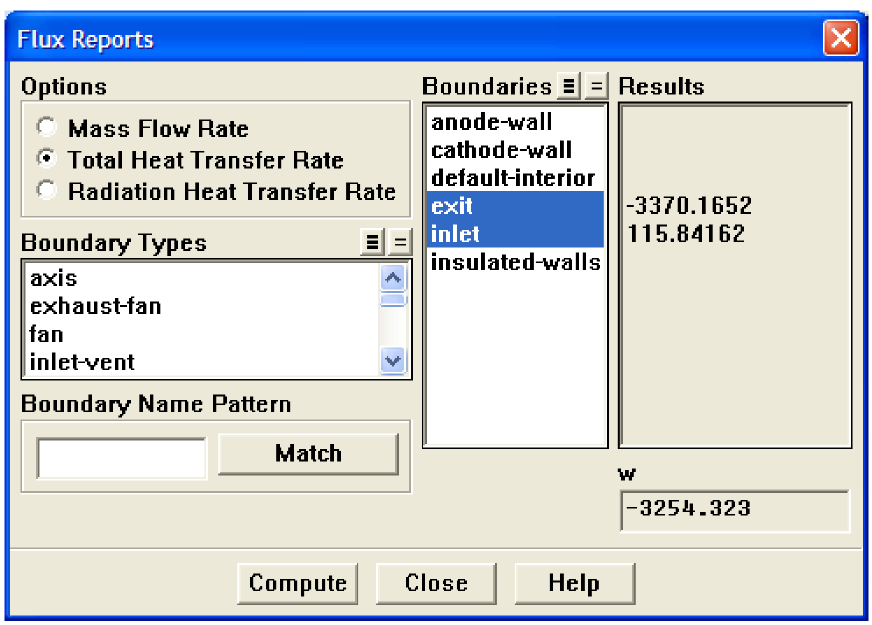
(a) Under Options, keep the default selection of Mass Flow Rate.

(b) In the Boundaries list, select `exit` and `inlet`.

(c) Click Compute.

For the specified convergence criterion, the mass imbalance is 0 kg/s. Now you will check the energy balance.

2. Check the energy balance.



(a) Under Options, enable Total Heat Transfer Rate.

© Fluent Inc. August 22, 2006

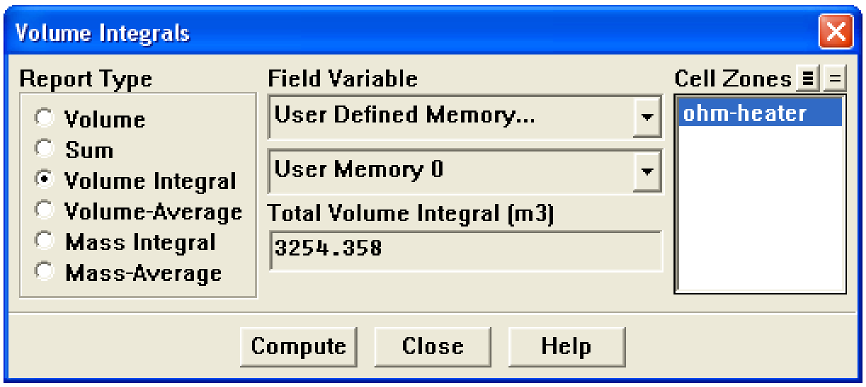
(b) In the Boundaries list, select `exit` and `inlet`.

(c) Click Compute.

For the energy equation the imbalance is -3254.323 W. This imbalance is due to the volumetric heating of the fluid. For energy conservation, this value must balance with the energy source. Now, integrate the energy source over the entire volume.

3. Check the volume integral.

Report ⟶Volume Integrals...



(a) Under Report Type, enable Volume Integral.

(b) Select User Defined Memory... and User Memory 0 in the Field Variable drop-down lists.

Heat generated due to the dissipation of electric energy is calculated by Ohms law and is stored in User Memory 0.

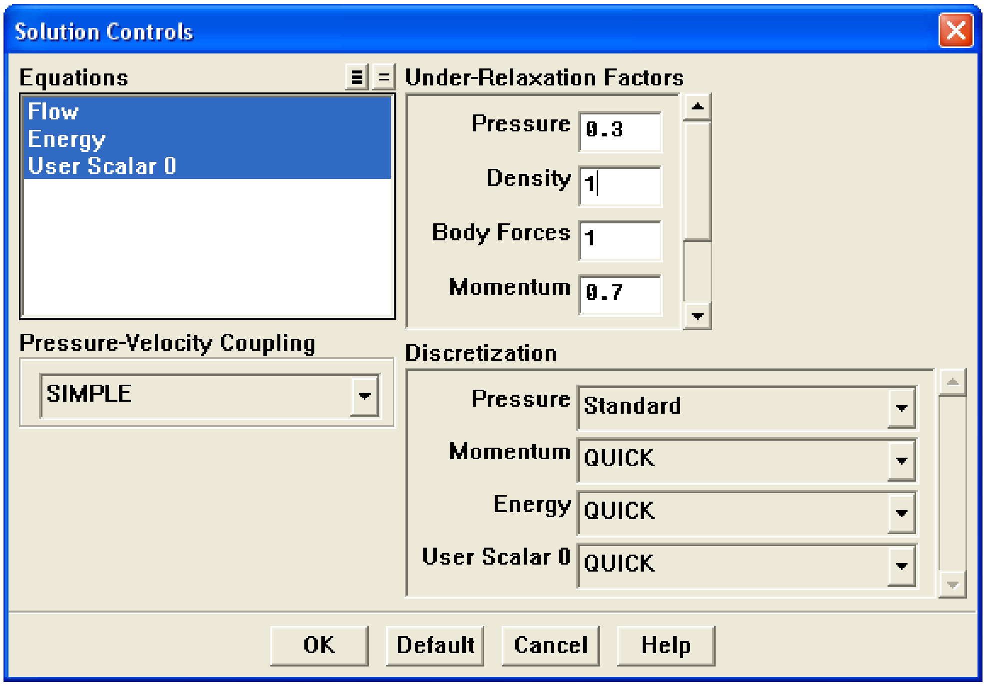
(c) In the Cell Zones list, select `ohm-heater`.

(d) Click Compute.

This is the total heat generated due to ohmic heating. This value is close to the change in the enthalpy of the fluid while passing through the heater. The net imbalance is 1.075e-5% which is well below the desired limit. In some cases, first order schemes and default convergence criteria may not provide the desired mass, momentum, energy, and scalar balances. In such cases, a better match can be obtained by selecting higher order schemes.

4. Select the higher order schemes.

   Solve ⟶ Controls ⟶ Solution...



   (a) Under Discretization, select QUICK in the Momentum, Energy, and User Scalar 0 drop-down lists.

   (b) Click OK.

   The QUICK *discretization scheme applies to quad/hex and hybrid meshes (not applied to tri mesh). It is useful for rotating/swirling flows and is 3rd-order accurate when used with a uniform mesh. In general, however, a second-order scheme should be sufficient and the* QUICK *scheme will not provide any significant improvement in comparison.*

5. Save the case file (`ohmic-heater-quick.cas.gz`).

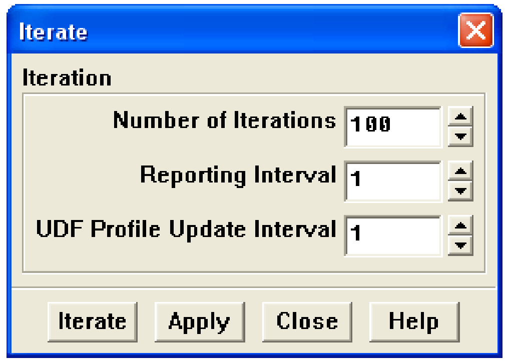
   File ⟶ Write ⟶ Case...

   Keep the Write Binary Files *(default) option on so that a binary file will be written.*

 © Fluent Inc. August 22, 2006

6. Start the calculation by requesting 100 iterations.

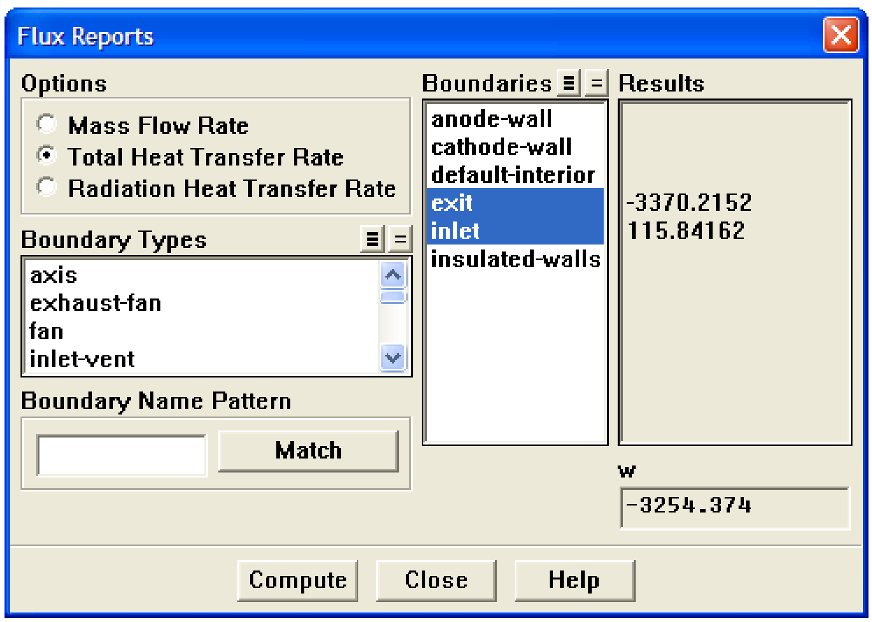
   Solve ⟶Iterate...



   The solution converges after approximately 57 iterations.

7. Check the energy balance again.

   Report ⟶Fluxes...



   (a) Under Options, enable Total Heat Transfer Rate.

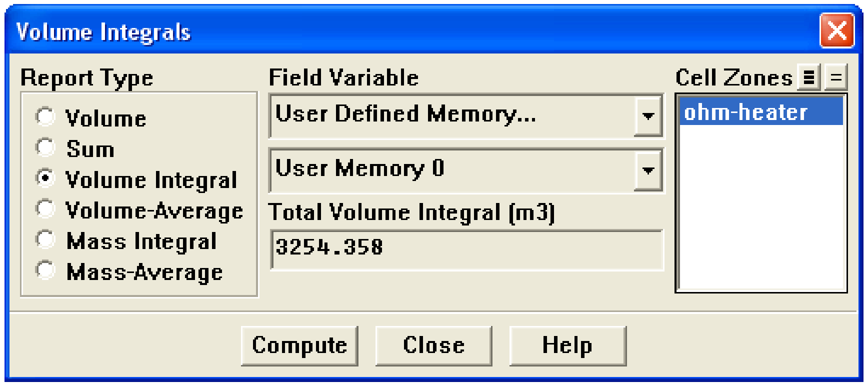
   (b) In the Boundaries list, select `exit` and `inlet`.

   (c) Click Compute.

   Now, integrate the energy source over the entire volume.

8. Check the volume integral.

   Report ⟶Volume Integrals...



(a) Under Report Type, enable Volume Integral.

(b) Select User Defined Memory... and User Memory 0 in the Field Variable drop-down lists.

(c) In the Cell Zones list, select `ohm-heater`.

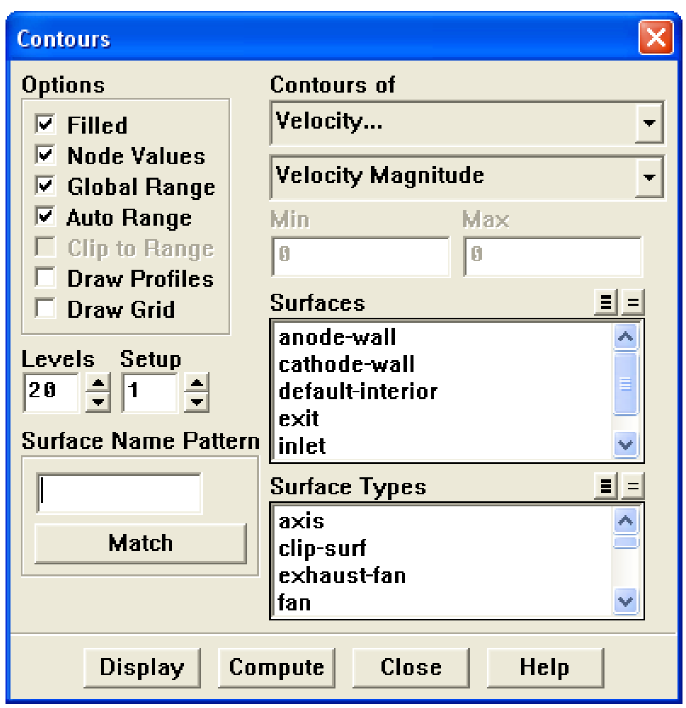
(d) Click Compute.

The net imbalance has changed from 1.075e-5% to 4.9e-6%.

Step 10: Postprocessing

1. Display contours of velocity magnitude (Figure 11.4).

Display →Contours...



© Fluent Inc. August 22, 2006

(a) Select Velocity... and Velocity Magnitude in the Contours of drop-down lists.

(b) Select Filled under Options.

(c) Click Display.

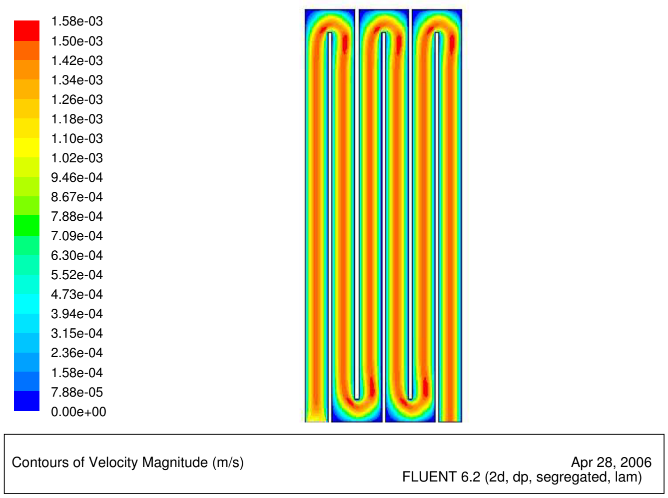
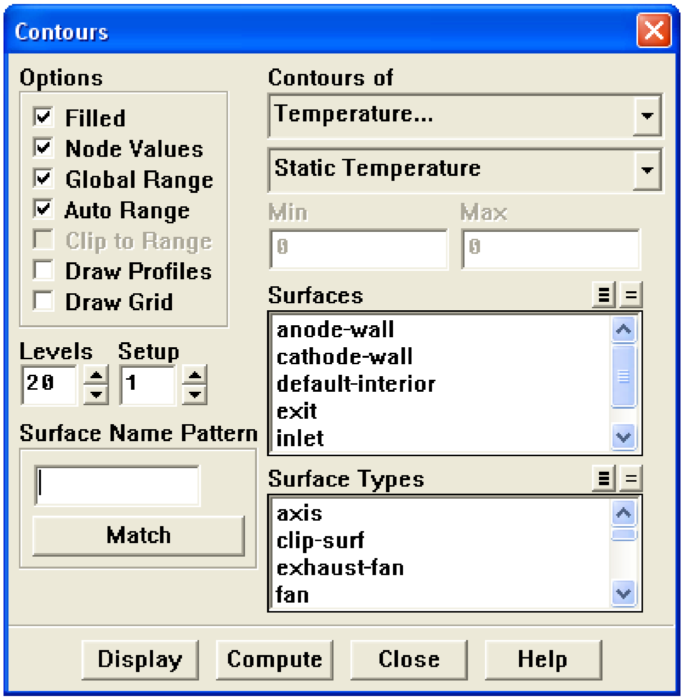


Figure 11.4: Velocity Distribution

Right-click on a point in the domain to display the value of the corresponding contour in the console.

2. Display filled contours of temperature (Figure 11.5).

   Display ⟶Contours...



   (a) Select Temperature... and Static Temperature in the Contours of drop-down lists.

   (b) Click Display.

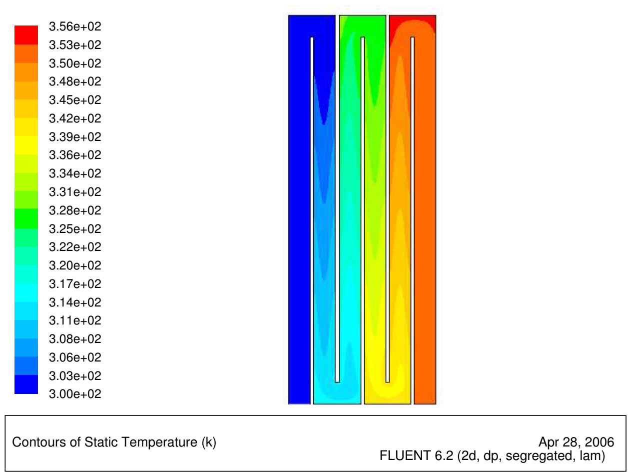


Figure 11.5: Contours of Static Temperature

© Fluent Inc. August 22, 2006

3. Display filled contours of electric potential (Figure 11.6).

   (a) Select User Defined Scalars... and User Scalar 0 in the Contours of drop-down lists.

   (b) Click Display.

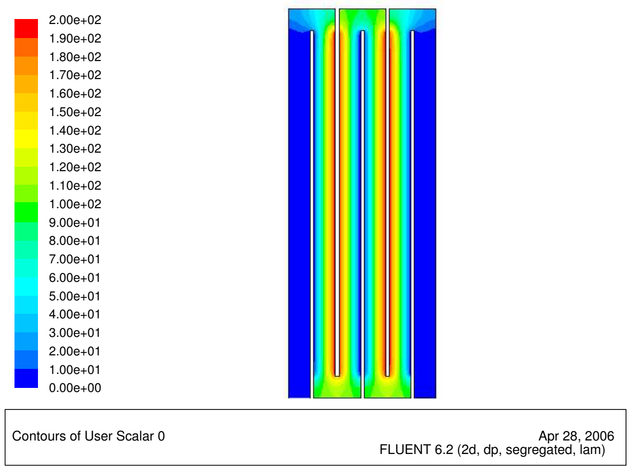


Figure 11.6: Contours of Electric Potential

4. Display filled contours of energy source (Figure 11.7).
   (a) Select User Defined Memory... and User Memory 0 in the Contours of drop-down lists.
   (b) Click Display.

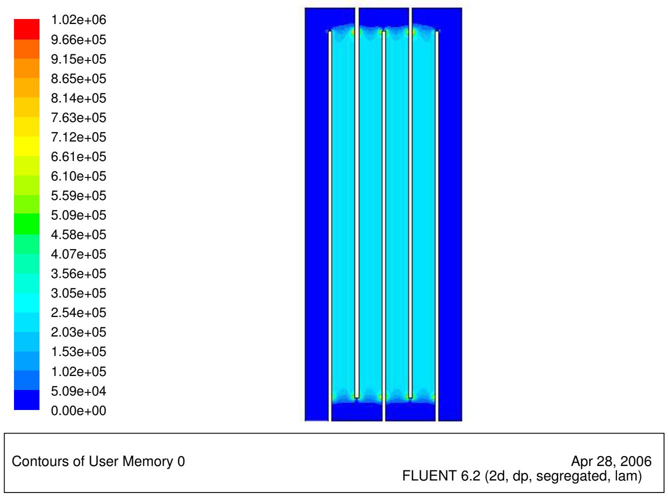


Figure 11.7: Contours of Energy Source Distribution

Summary

FLUENT UDS and UDM capabilities are illustrated for predicting the electric potential field. UDF is used for calculating the dissipation of electric energy into heat energy.

References

FLUENT 6.2 User's Guide:

http://www.fluentusers.com/fluent/doc/ori/html/ug/main_pre.htm

Exercises/Discussions

1. What will be the effect on exit temperature and maximum temperature in each of following situations:
   (a) Electrical conductivity is defined as a function of temperature.
   (b) Thermal conductivity is defined as a function of temperature.

 © Fluent Inc. August 22, 2006

2. What will be the effect on the pumping power requirement in each of following situations:
   (a) Thermal conductivity is defined as a function of temperature.
   (b) Electrical conductivity is defined as a function of temperature.
   (c) Viscosity is defined as a function of temperature.

Links for Further Reading

http://www.fsid.cvut.cz/ zitny/zitny/ohmic.htm

© Fluent Inc. August 22, 2006