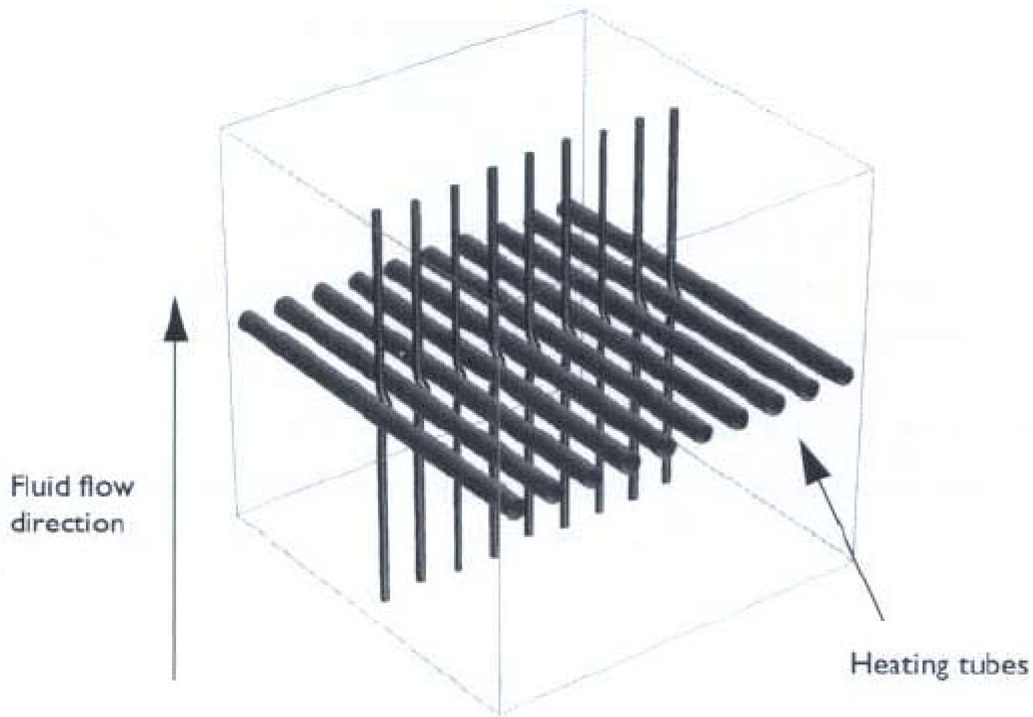
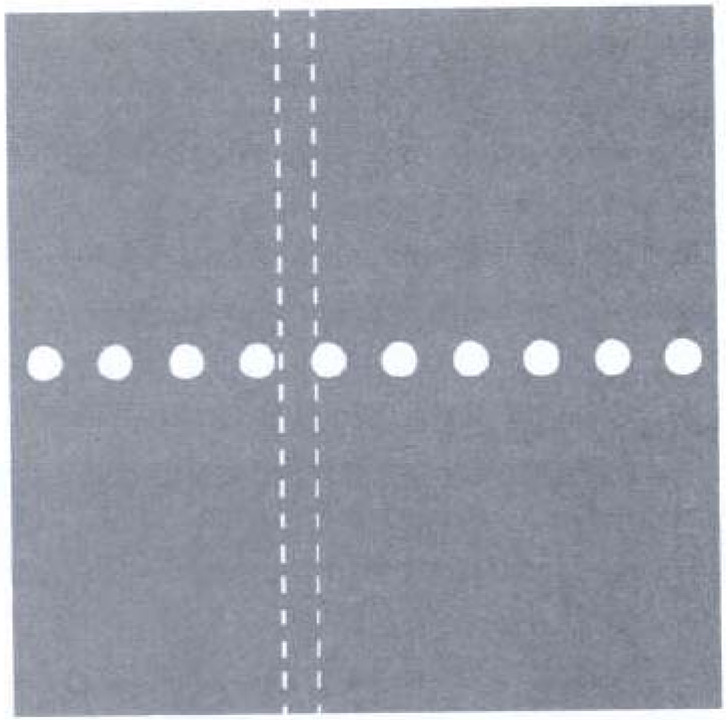
FEMLAB starting example

Example geometry



2D model



GOVERNING EQUATIONS

This is a multiphysics model because it involves more than one kind of physics. The incompressible Navier-Stokes equations from fluid dynamics works together with a heat transfer equation. There are four unknown field variables (dependent variables):

- The velocity field components, u and v
- The pressure, p
- The temperature, T

They are all related through bidirectional multiphysics couplings.

Navier-Stokes Eqn

$$\begin{cases} \rho\dfrac{\partial \mathbf{u}}{\partial t} + \rho(\mathbf{u}\cdot\nabla)\mathbf{u} = -\nabla p + \eta\nabla^2\mathbf{u} + \mathbf{F} \\ \nabla\cdot\mathbf{u} = 0 \end{cases}$$

with the following variables:

- $\mathbf{u}$ is the velocity field.
- p is the pressure.
- $\mathbf{F}$ is a volume force.
- ρ is the fluid density.
- η is the dynamic viscosity.
- ∇ is the vector differential operator.

Heat Eqn

$$\rho c_p \frac{\partial T}{\partial t} + \nabla \cdot (-k\nabla T + \rho c_p T\mathbf{u}) = Q$$

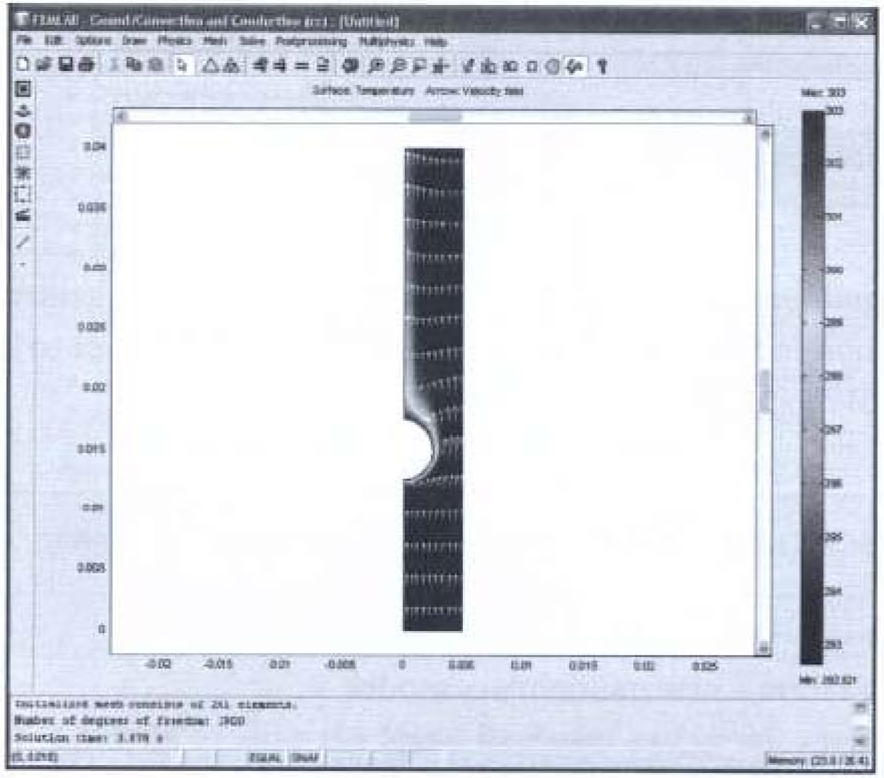
where c_p is the heat capacity of the fluid, and ρ is fluid density. The expression within the brackets is the heat flux vector. Q represents a source term. The heat flux vector contains a diffusive and a convective term, where the latter is proportional to the velocity field, $\mathbf{u}$. The velocity field comes from the incompressible Navier-Stokes equation. This means that we have a one-way multiphysics coupling from the fluid field to the energy transport by convection.

FEMLAB modelling

To build a model in FEMLAB using the above equations, use two application modes: the Incompressible Navier-Stokes application mode for fluid flow, and the Convection and Conduction application mode for heat transfer. The multiphysics couplings enter directly into the physics settings in the application modes.

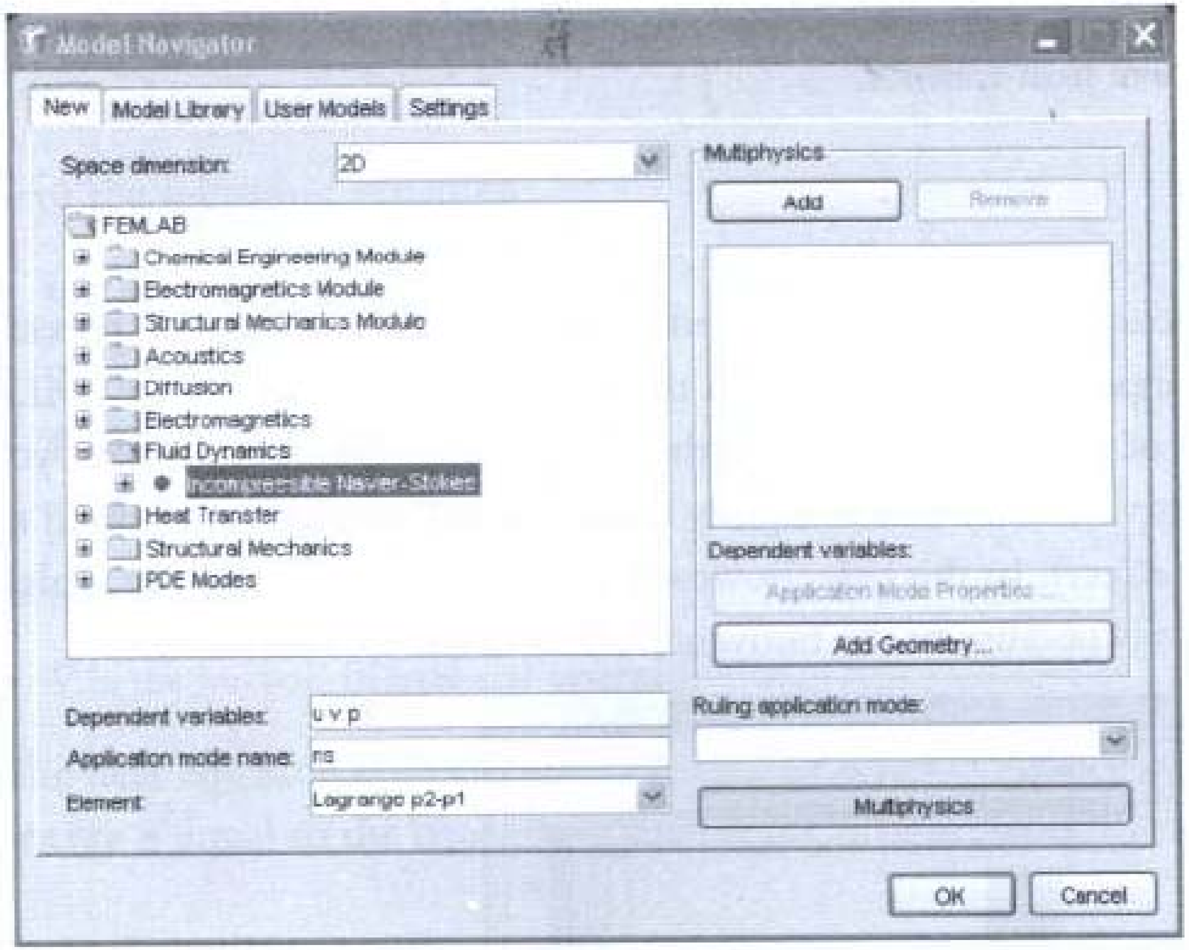
In this model, the equations are coupled through the $\mathbf{F}$ and Q terms. First, add free convection to the momentum balance with the *Boussinesq approximation*. This approximation ignores variations in density with temperature, except that the variations give rise to a buoyancy force lifting the fluid. This force enters the $\mathbf{F}$ term in the incompressible Navier-Stokes equations.

Velocity and temperatue fields



MODEL NAVIGATOR

Starting FEMLAB opens the **Model Navigator**:



Use the following steps to create a new multiphysics model:

1 Click the **New** tab, and check that `2D` is selected in the **Space dimension** list. The space dimension must always be selected first, because the available application modes vary depending on the space dimension.

2 Click the **Multiphysics** button to open the section where you can create a multiphysics model.

3 In the application modes list on the left, open the **Fluid Dynamics** folder and click `Incompressible Navier-Stokes`.

   The default application mode name is `ns`, and the default names of the dependent variables are `u`, `v`, and `p`, for the velocity components in the x- and y-direction and the pressure. The default finite elements for the Incompressible Navier-Stokes application mode are mixed second-order and first-order triangular *Lagrange elements*, `Lagrange p2-p1` elements.

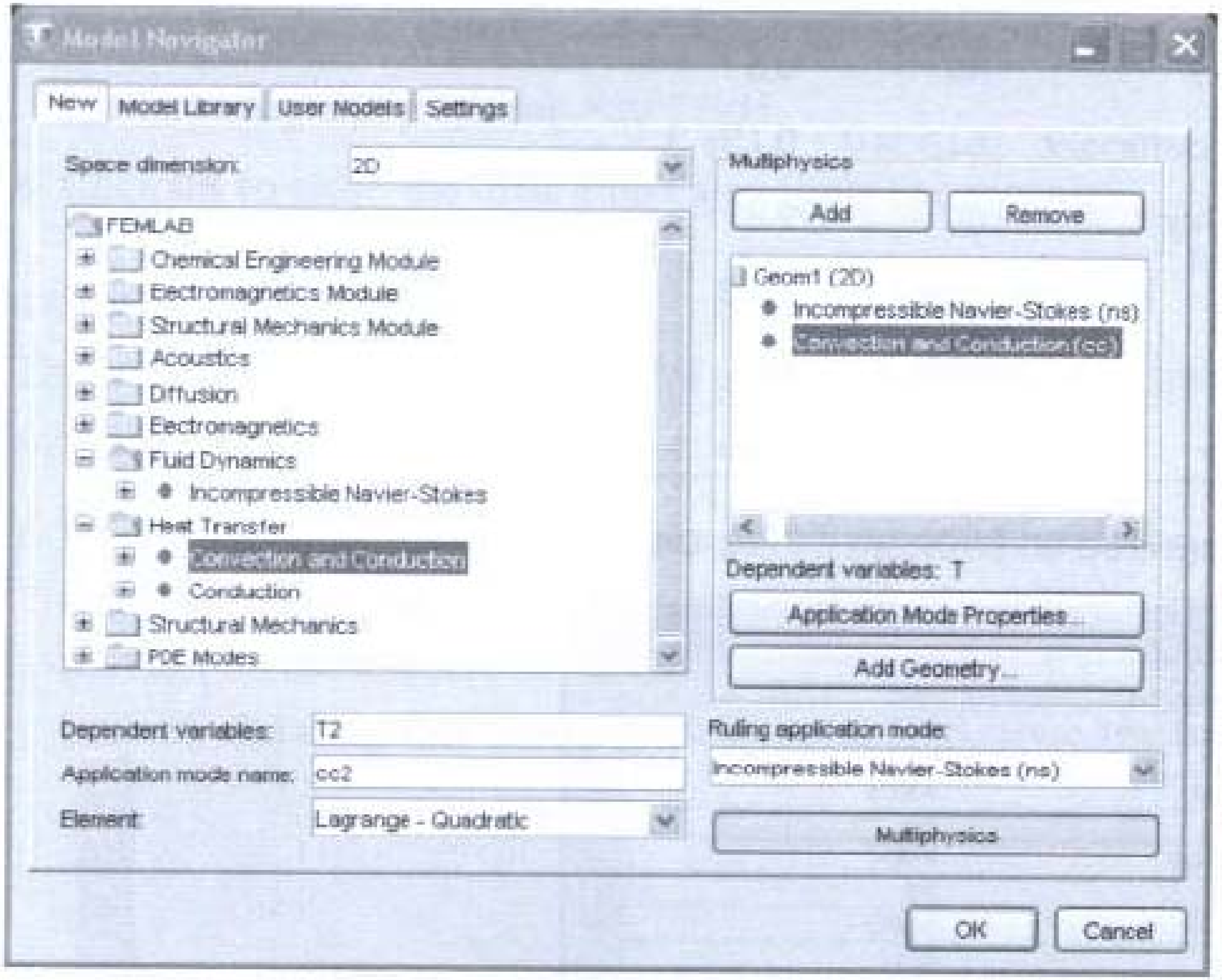
4 Click **Add**. This accepts the settings in the **Application mode name, Dependent variables**, and **Element** edit fields and transfers the Incompressible Navier-Stokes application mode to the list of application modes in **Multiphysics** section.

5 In the application modes list, open the **Heat Transfer** folder and click `Convection and Conduction`.

6 Click **Add** again to add the Convection and Conduction application mode to the model with the default variable name, `T`, and element type (second-order Lagrange elements).

7 Click **OK** to close the **Model Navigator** and create a new model.

At this point, FEMLAB opens up in the Convection and Conduction application mode. You can always find out which application mode that is active. Its name appears in the title bar of the FEMLAB window.


Model Navigator
New
Model Library
User Models
Settings
Space dimension:
2D
FEMLAB
Chemical Engineering Module
Electromagnetics Module
Structural Mechanics Module
Acoustics
Diffusion
Electromagnetics
Fluid Dynamics
Incompressible Navier-Stokes
Heat Transfer
Convection and Conduction
Conduction
Structural Mechanics
PDE Modes
Multiphysics
Add
Remove
Geom1 (2D)
Incompressible Navier-Stokes (ns)
Convection and Conduction (cc)
Dependent variables: T
Application Mode Properties...
Add Geometry...
Dependent variables:
T2
Application mode name:
cc2
Element:
Lagrange - Quadratic
Ruling application mode:
Incompressible Navier-Stokes (ns)
Multiphysics
OK
Cancel

OPTIONS AND SETTINGS

Later in this model you will need the fluid properties of the water, the temperatures at the inlet and on the surface of the heating tubes, and the inlet velocity. It is convenient to enter this data as *constants* in the **Constants** dialog box. In this model, all values are given in SI units.

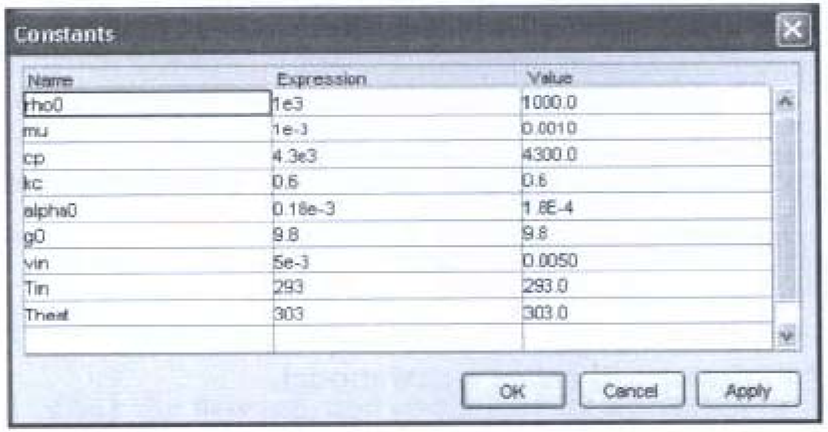
1 From the **Options** menu, choose **Constants**.

2 First add the density of the fluid:

   Enter `rho0` in the **Name** field. Press the **Tab** key to move the cursor to the **Expression** field and enter `1e3`. The value is saved when you press **Enter**, click **Apply**, or otherwise leave the **Expression** field.

3 Continue by adding the remaining properties

4 When you have entered all constants, click **OK**.

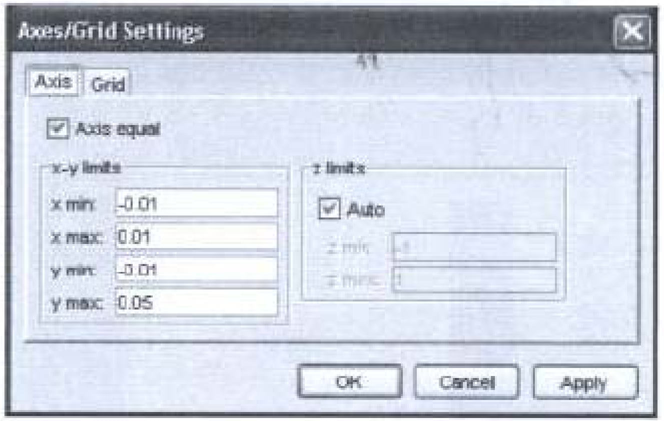


Name	Expression	Value
rho0	1e3	1000.0
mu	1e-3	0.0010
cp	4.3e3	4300.0
kc	0.6	0.6
alpha0	0.18e-3	1.8E-4
g0	9.8	9.8
vin	5e-3	0.0050
Tin	293	293.0
Theat	303	303.0

Model Size

1 From **Options** menu, choose **Axes/Grid Settings**

2 In the **Axes/Grid Settings** dialog box, enter `-0.01`, `0.01`, `-0.01`, and `0.05` in the **x min**, **x max**, **y min**, and **y max** edit fields, respectively.

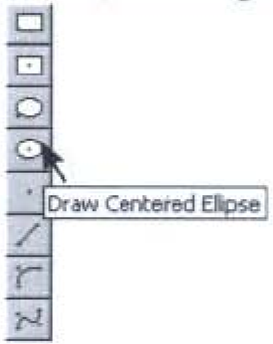


3 Click the **Grid** tab and then click to clear the **Auto** check box. Enter `0.005` in both the **x spacing** and **y spacing** edit fields.

4 Click **OK** to close the dialog box and apply the settings.

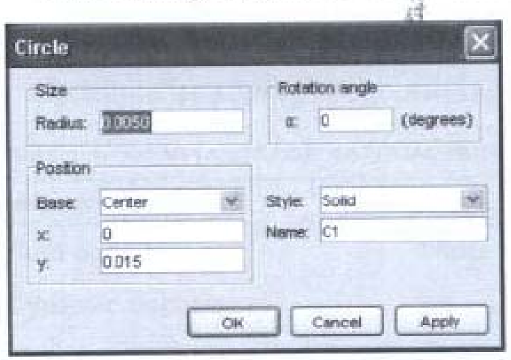
Model Geometry

1 Draw a rectangle of width 0.005 and height 0.04, with the lower left corner at the origin. Click the **Draw Rectangle** toolbar button. It is the first button on the draw toolbar, on the left side of the drawing area. Then click at (0,0) using the left mouse button, and drag the mouse to (0.005,0.04). Release the button.

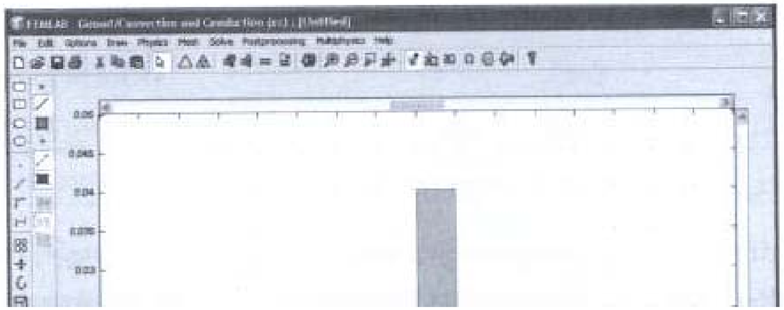


2 Draw a circle with radius 0.005 centered at (0,0.015). Click the fourth button on the Draw toolbar, **Draw Centered Ellipse**. Then, using the *right* mouse button, click at (0,0.015) and drag the mouse in any direction, keeping the button down, until the circle has a radius of 0.005. Using the right mouse button constrains the ellipse to a circle.

3 The desired radius of the circle is not 0.005, however, but is 0.0025. To fix this, double-click the circle object or select **Object Properties** from the **Draw** menu. In the **Circle** dialog box, enter `0.0025` as **Radius** and click **OK**.



4 Select both the circle and the rectangle. Either draw a rubber-band box around both them, or press the shortcut key **Ctrl+A** to select all objects.



Physics Setting

Boundary Conditions

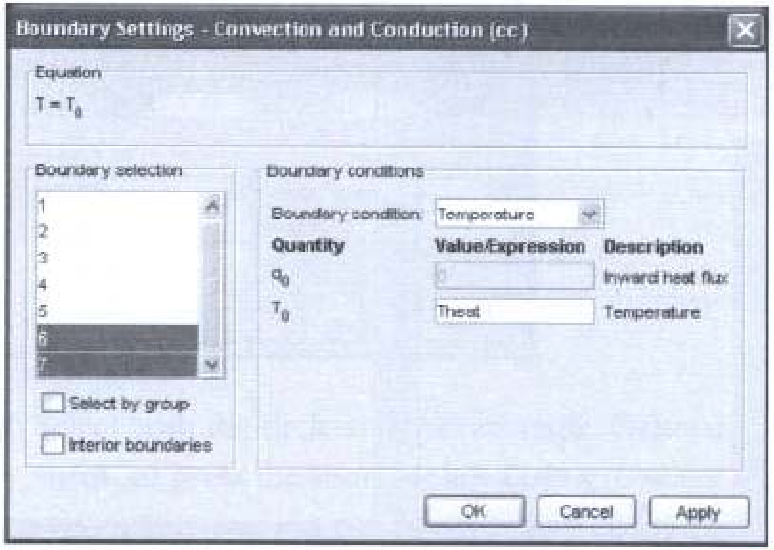
Specify the boundary conditions, first for the Convection and Conduction mode and then for the Incompressible Navier-Stokes mode.

1 From the **Physics** menu, choose **Boundary Settings**. This opens the **Boundary Setting** dialog box and transfers FEMLAB to the *boundary selection mode*. The contents of this dialog box varies depending on the application mode.

2 Start by insulating all boundaries. This is the default boundary condition, so you do not need to change anything.

3 Select the inflow boundary by clicking at the corresponding edge in the geometry or by selecting boundary number 2 in the **Boundary selection** list. Select `Temperature` in the **Boundary conditions** list and enter `Tin` in the **Temperature** edit field.

4 Select the heater that exists along boundaries 6 and 7 from the **Boundary selection** list. Then select `Temperature` in the **Boundary conditions** list and enter `Theat` in the **Temperature** edit field.



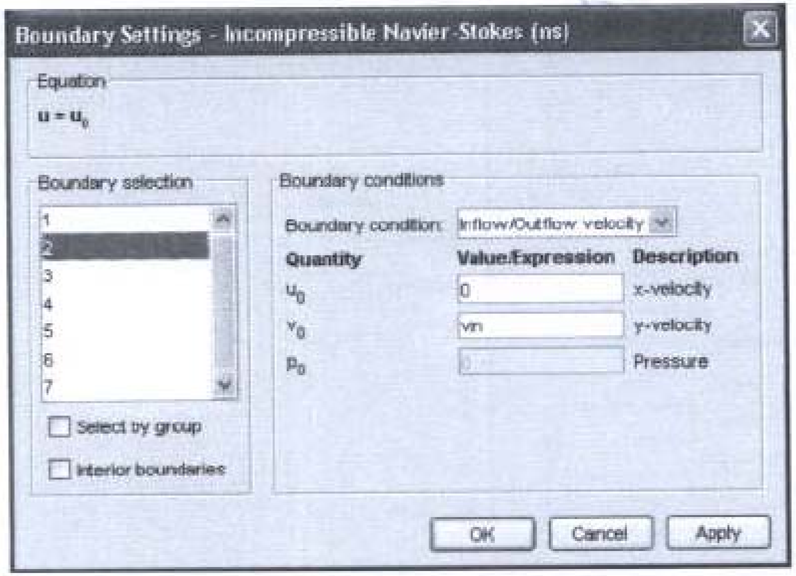
5 Click the outflow boundary (boundary number 4) and select `Convective flux` in the **Boundary conditions** list.

6 Click **Apply** to confirm the settings for the Convection and Conduction application mode.

7 Switch to the Incompressible Navier-Stokes application mode by selecting this mode from the **Multiphysics** menu. You can now open another **Boundary Settings** dialog box from the **Physics** menu.

8 Press **Ctrl+A** to select all boundaries in the list and then select the **Slip/symmetry** condition from the **Boundary conditions** list.

9 For the inflow boundary, select boundary 2 and then choose **Inflow/Outflow velocity** in the **Boundary conditions** list. Enter `vin` in the **y-velocity** edit field (leave the *x* velocity at `0`).



10 Continue by selecting the outflow boundary (boundary number 4). In the **Boundary conditions** list, select **Normal flow/Pressure**. Leave the pressure at `0`.

11 Finally select boundaries 6 and 7 and then select the **No slip** boundary condition. Click **OK** to confirm your choices and close the dialog box.

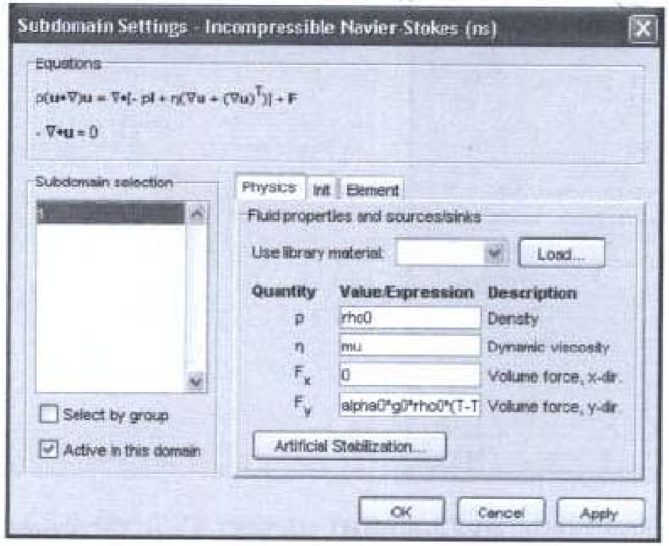
Subdomain Setting

In the **Subdomain Settings** dialog box you can also set initial values. The Navier-Stokes equations are nonlinear and therefore benefit from an educated guess as an initial solution to the nonlinear solver. In some cases, a good initial value might be necessary for the model to converge. The initial values are also used as initial conditions for the time-dependent solver.

Depending on the application mode, the coefficients in the governing equations can be interpreted as material properties, forces, and sources and sinks. You specify all of them in the **Subdomain Settings** dialog box. This time start with the Incompressible Navier-Stokes application mode and continue on to the Convection and Conduction application mode.

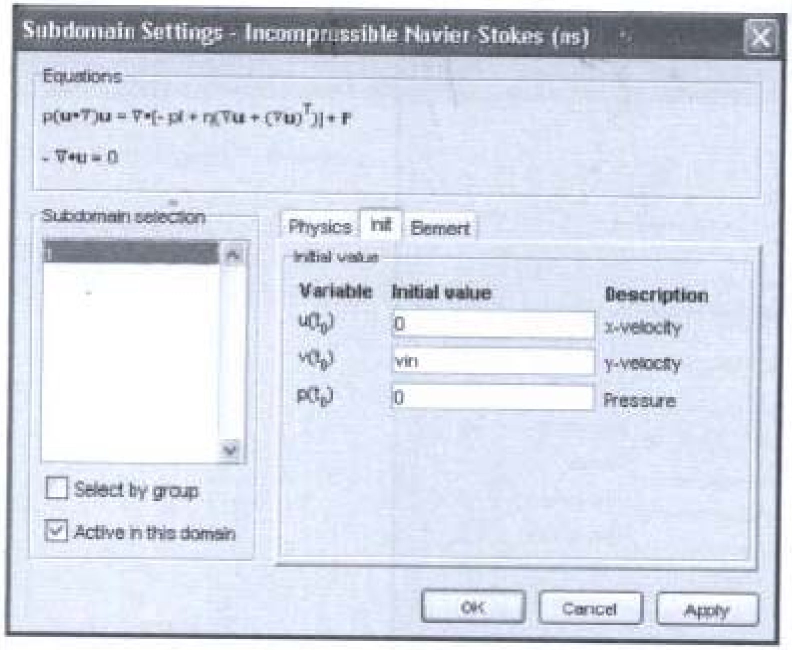
1 Choose **Subdomain Settings** from the **Subdomain** menu to open the **Subdomain Settings** dialog box and put the user interface in the subdomain selection mode.

2 Select the single subdomain, number 1, and use the already defined constants to set the density and the viscosity. Enter `rho0` and `mu` in the **Density** and **Dynamic viscosity** fields, respectively.



3 To model the effect of temperature on the density of the fluid as a buoyancy force in the y direction, enter `alpha0*g0*rho0*(T-Tin)` in the **Volume force, y-dir.** edit field (F_y). Remember that T is the dependent variable from the Convection and Conduction application mode. Notice that an extended edit field displays the entire expression.

1 Click the **Init** tab in the **Subdomain Settings** dialog box. Set the initial value **v(t_0)** to `vin`.

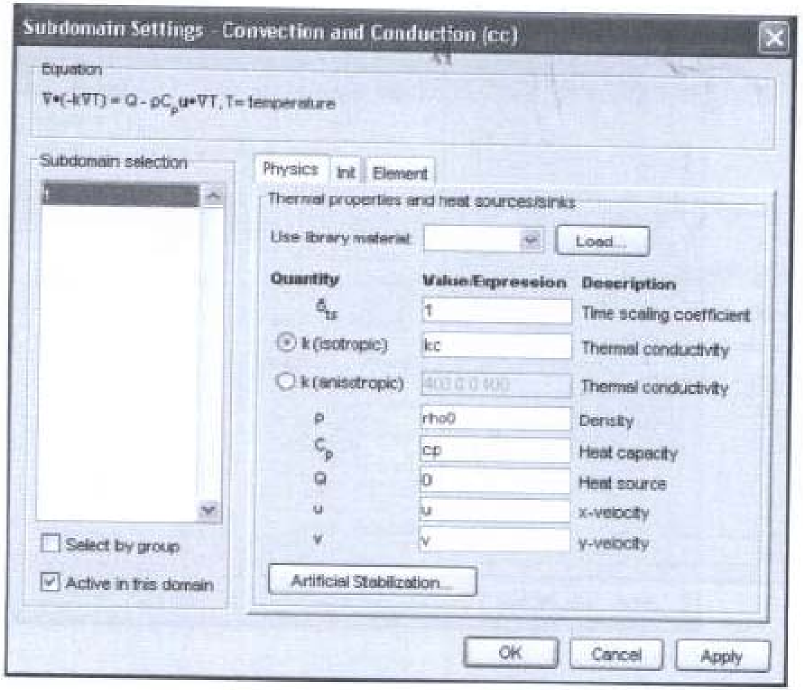


2 Click **OK**.

Continue with the subdomain settings for the Convection and Conduction application mode:

1 Use the **Multiphysics** menu to switch to the Convection and Conduction application mode.

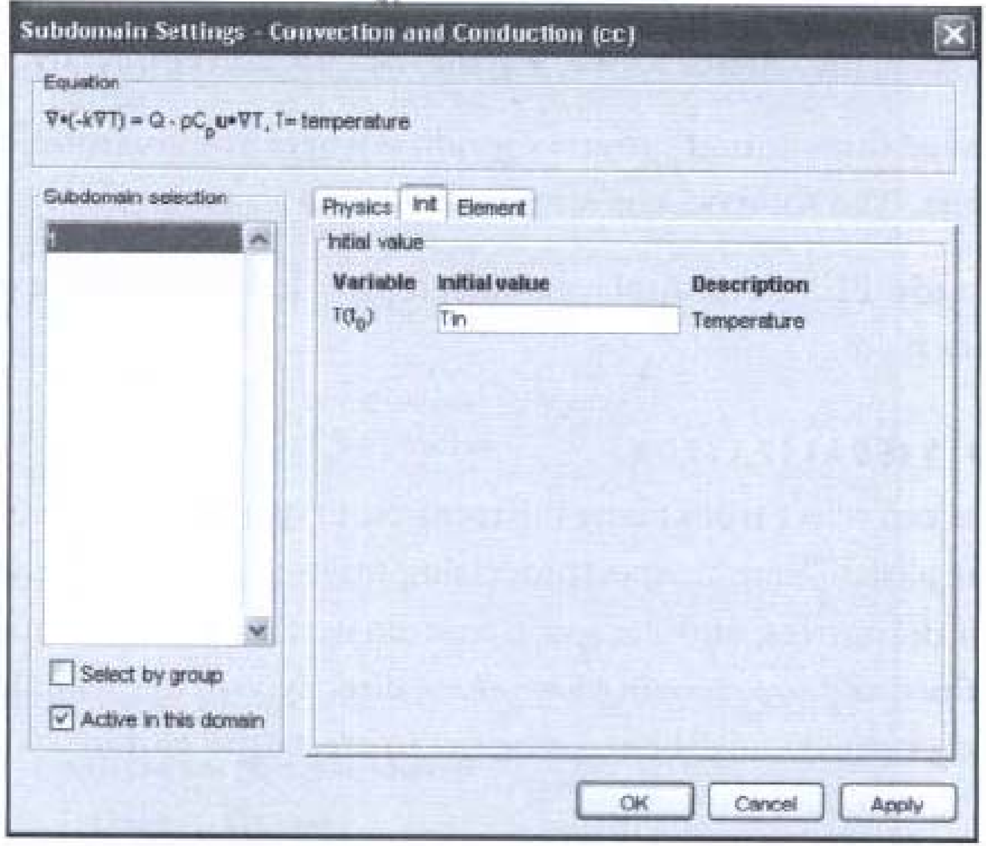
2 Open the **Subdomain Settings** dialog box for the Convection and Conduction application mode.



3 Enter `rho0`, `cp`, and `kc` in the **Density**, **Heat capacity**, and **Thermal conductivity** (isotropic) edit fields, respectively.

4 Enter `u` and `v` in the **x-velocity** and **y-velocity** edit fields, respectively. They are the dependent variables for the velocity components from the Incompressible Navier-Stokes application mode, which account for the convective transport of heat.

5 The temperature also needs an initial value. Use the inlet temperature as the starting temperature in the entire domain. Click the **Init** tab and enter `Tin` in the **T(t_0)** field. Then click **OK** to confirm all settings and close the dialog box.



Mesh Generation

To create a default mesh, enter *mesh mode* directly or click the **Initial Mesh** button. If you need a different mesh resolution or require the mesh to be denser in some parts of the geometry than in others, you can use the settings in the **Mesh Parameters** dialog box and other commands in the **Mesh** menu.

If you trust the default settings, you can proceed directly to solving the model. FEMLAB then creates a mesh when you click the **Solve** button. It is, however, good practice to inspect the mesh before solving, because the mesh density and quality affect the solution time, convergence, and accuracy.

In this model, use a predefined setting for a fine mesh:

1. From the **Mesh** menu, choose **Mesh Parameters**.
2. In the **Mesh Parameters** dialog box, select `Fine` in the **Predefined mesh sizes** list.
3. Click **OK**.
4. Initialize the default mesh clicking the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

The required nonlinear solver is the default solver, so you start the solver directly.

Click the **Solve** button. Notice the solution progress window where you can monitor and stop the solution process. The solution converges quickly in this case.

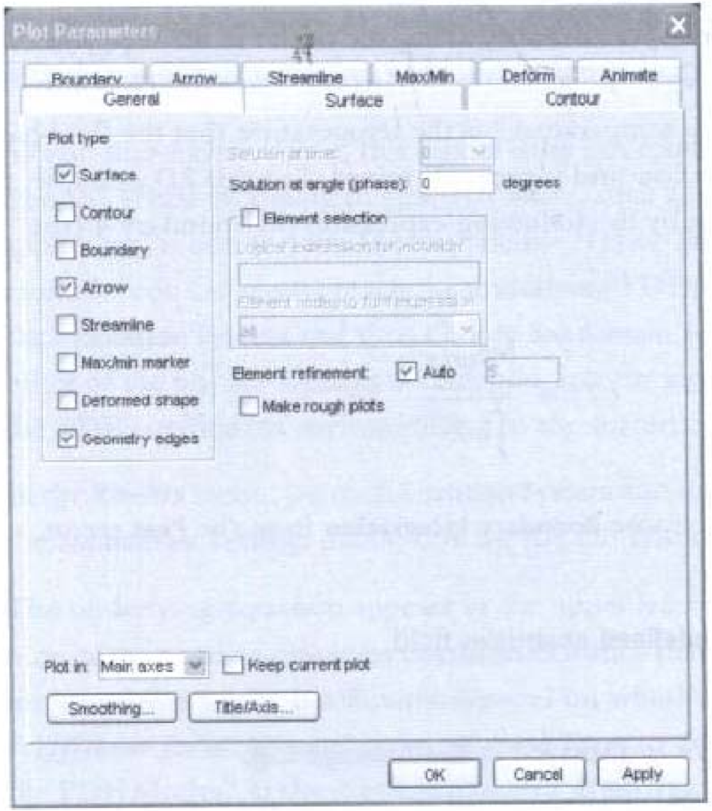
As soon as the solution is ready, FEMLAB displays a default plot. In this case, you get a surface plot of the velocity field.

POSTPROCESSING AND VISUALIZATION

In *postprocessing mode* you can select from many different plot types and set parameters for the different plots. Using the postprocessing utilities you can visualize the solution variables, their derivatives, and the space coordinates. Many frequently used expressions are predefined as *postprocessing variables*, directly available from lists

Plotting

1 Click the **Plot Parameters** button on the main toolbar. This opens the **Plot Parameters** dialog box on the **General** page.



2 To obtain a surface plot with overlaid arrows, select the **Surface** and **Arrow** check boxes.

3 Click the **Surface** tab.

4 Under **Surface data**, select `Temperature` in the **Predefined quantities** list.

5 Click the **Arrow** tab.

6 Select `Velocity field` in the **Predefined quantities** list.

7 Under **Arrow positioning**, enter 10 and 15 in the **x points** and **y points** edit fields, respectively.

8 Finally, under **Arrow parameters**, click the **Color** button to choose a suitable arrow color, for example, white.

9 Click **OK**.

Integrating to Find the Mean Temperature

Because both the velocity and temperature vary along the outlet at the top, you mus use additional postprocessing to get the mean temperature at the exit. The bulk temperature, or the "cup mixing temperature" is the temperature that the fluid has i it is collected in a cup at the outflow and is properly mixed. For this 2D example, th cup mixing temperature is given by the following expression on boundary 4 (the outlet):

$$\langle T \rangle = \frac{\int Tv\,dx}{\int v\,dx}$$

1. To obtain the denominator, choose **Boundary Integration** from the **Post** menu.
2. Select boundary 4.
3. Select `y-velocity` in the **Predefined quantities** field.
4. Click **Apply**.
5. The values of the denominator is displayed in the message log:

   `Value of integral: 2.5e-5, Expression: v, Boundary: 4.`

   Copy the value by selecting it and pressing **Ctrl+C**.
6. In the **Boundary Integration** dialog box, type `T*v/`. Then paste the denominator at the end of the expression for the integrand to form `T*v/2.5e-5` in the **Expression** edit field. This will compute the resulting mean temperature.
7. Click **OK**. The result is displayed in the message log.

You should get a mean temperature of roughly 293.75 K, that is, a temperature rise of approximately 0.75 K between inlet and outlet.

SAVING THE MODEL

To save the model to a file:

1. In the **File** menu, choose **Save**.
2. Browse to select a file or enter a file name for the model file.
3. Click **Save**.

The default format is a FEMLAB model file with extension `.fl`. Use this format to quickly save and open FEMLAB models.

Physics Menu

In the **Physics** menu, point to **Equation System** and then **Subdomain Settings**. This opens the **Subdomain Settings** dialog box for the full system of equations.

The underlying equation appears in the upper left corner, and the edit fields and tabs now correspond to equation coefficients rather than material properties. The interpretation of the coefficients depend on whether FEMLAB currently uses the *coefficient form*, *general form*, or *weak form* to describe a generic PDE. See "Using the PDE Modes" in the *Application and Modeling Guide*. Multiphysics problems are normally based on the general form, and it is this formulation that you can see at the top of the dialog box.

FEMLAB normally solves multiphysics problems directly as a coupled PDE system. In general form, symbolic differentiation provides the exact *stiffness matrix* or *Jacobian* of the complete system. You can see the result on the **c**, **a**, **al**, and **be** tabs.

There is also an extra tab, **Weak**, where you can specify additional terms to equations on the weak PDE form. Weak coefficients are available on all levels: subdomains, boundaries, edges, and points. Use them to implement, for instance, explicit streamline diffusion, weak constraints, line sources, and point sources, respectively.

Variables List

To find the application mode variables, click the right-most tab, **Variables**. The **Variables** page lists their names and how they are defined. Edit the expressions to change the predefined expressions