FEMLAB example in heat transfer

Bioprocess Laboratory
Department of Chemical Engineering
Chungnam National University

Axismmetric Transient Heat Transfer

- This model domain is 0.3 X 0.4 meters.
- Boundary conditions

  The left boundary is symmetry axis.

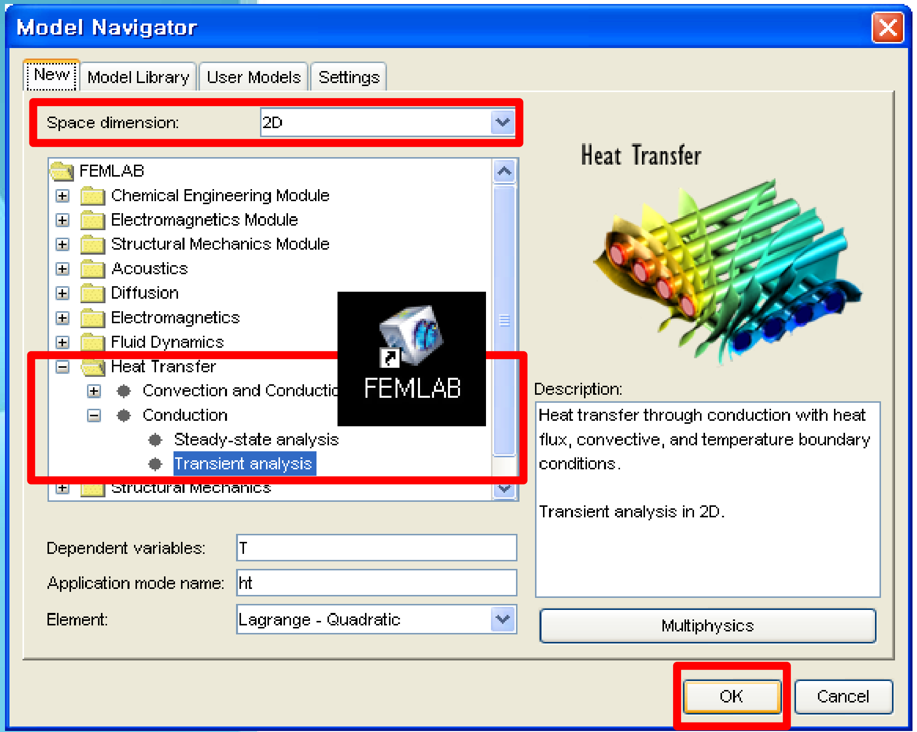
  The other boundaries have a temperature of 1000℃. The entire domain is at 0℃ at the start, which represents a step change in temperature at the boundaries.
- Material properties

  The density, ρ, is 7850 kg/m^3.

  The heat capacity is 460 J/kg·℃.

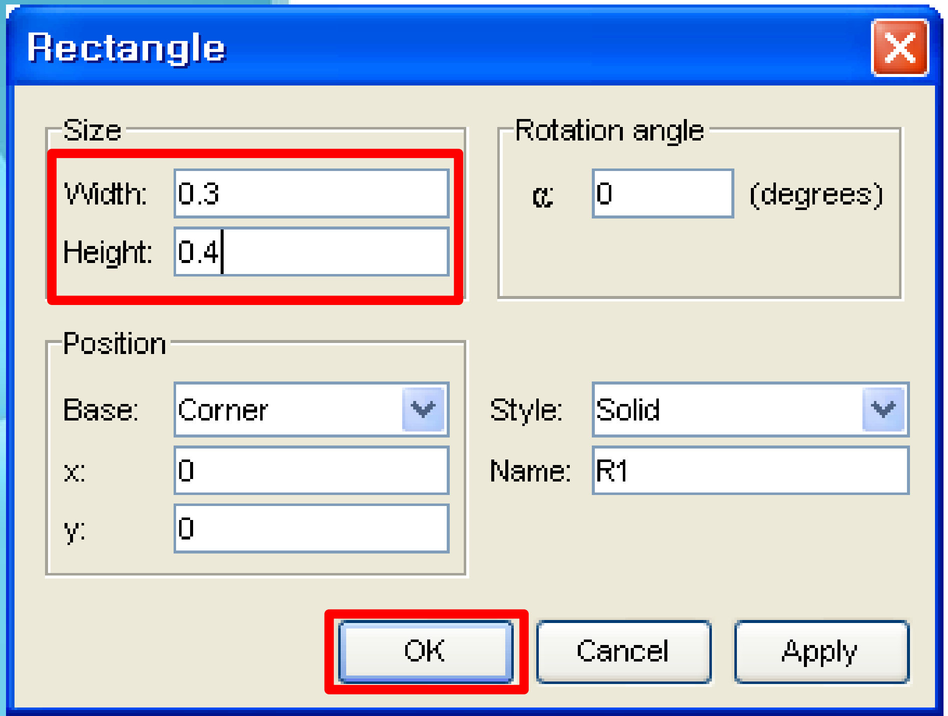
  The thermal conductivity is 52 W/m·℃.

Model navigator



1. Execute the **FEMLAB**
2. Select Axial symmetry (2D) in the **Space dimension** list.
3. Open the **Heat Transfer** folder.
4. Open the **Conduction** node.
5. Select **Transient analysis**.
6. Click **OK**.

Geometry modeling

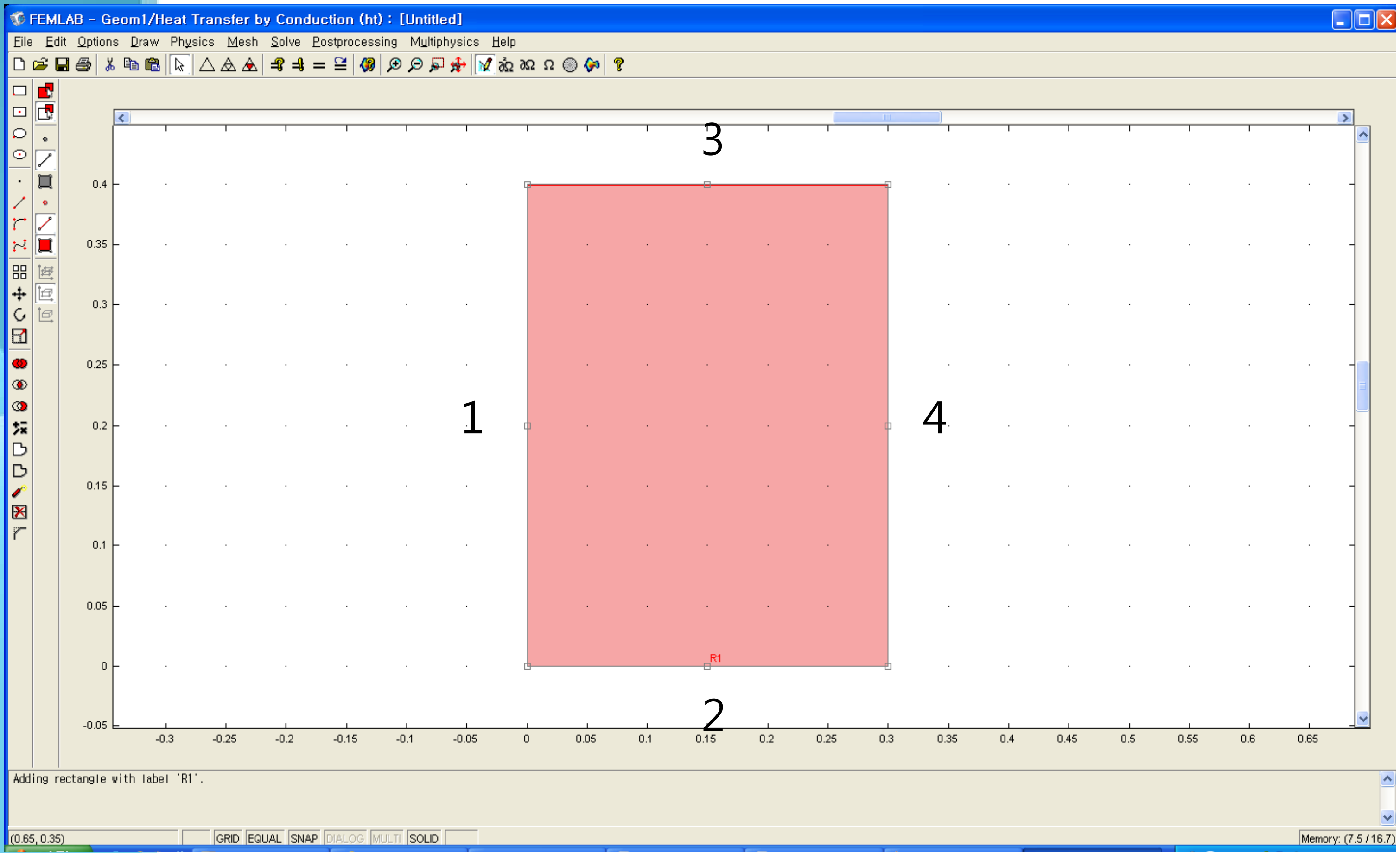


1. Go to the **Draw** menu, point to **Specify Object** and click **Rectangle**.

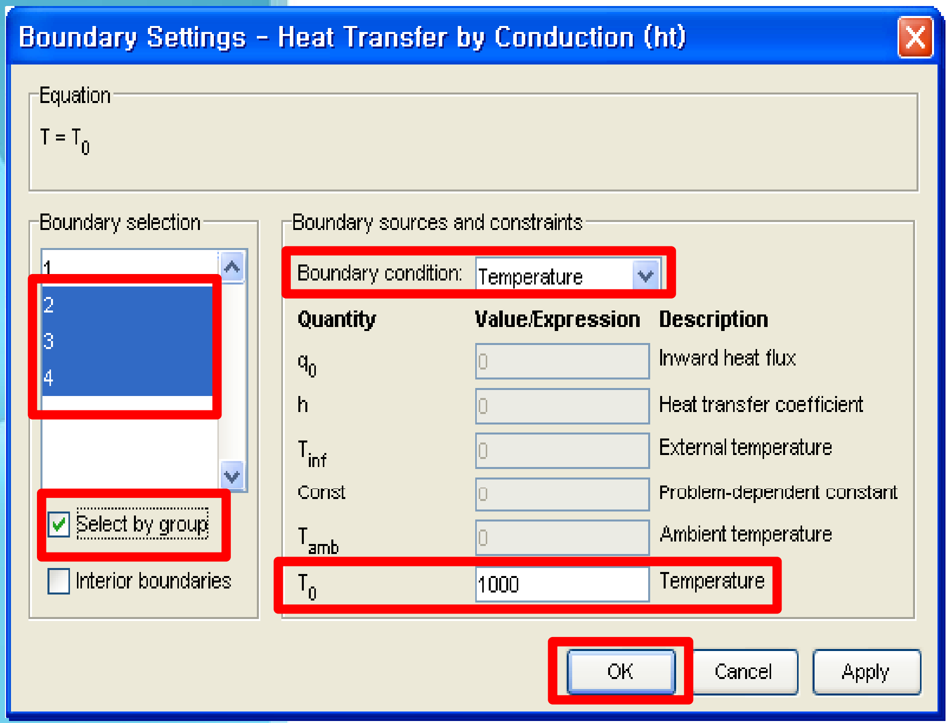
2. In the **Rectangle** dialog box go to the **Size** area and enter **0.3** in the **Width** edit field and **0.4** in the **Height** edit field.

3. Click **OK**.

Model navigator and Geometry modeling



Physics settings (Boundary settings)



1. Go to the **Physics** menu and choose **Boundary Settings**.

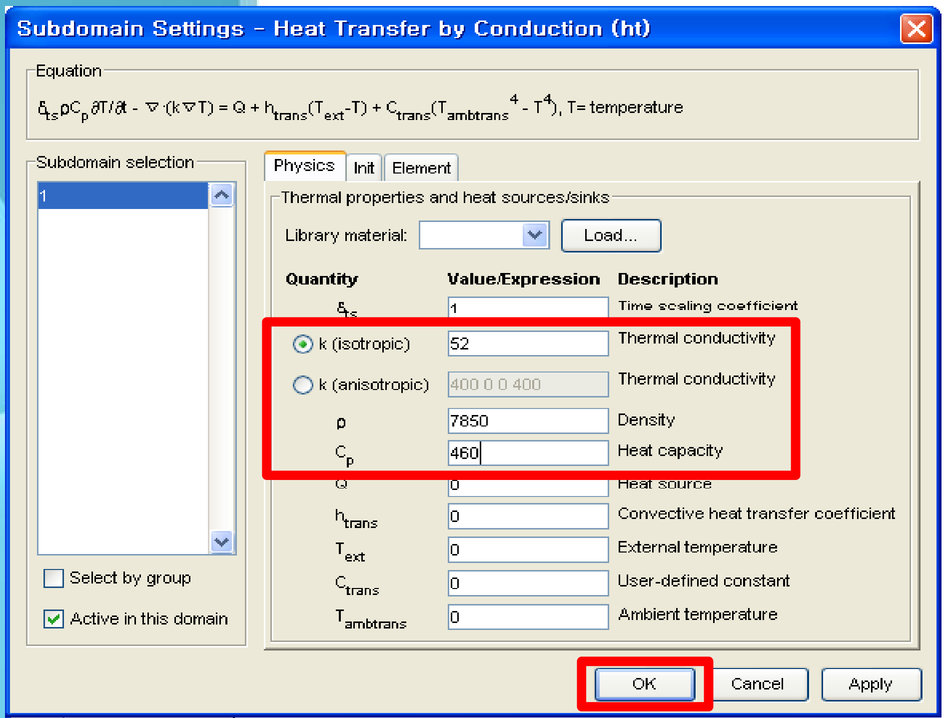
2. Select the **Select by group** check box and choose boundaries 2,3 and 4 by selecting on of them.

3. Select Temperature in the **Boundary condition** list.

4. Enter 1000 in the **Temperature** edit field.

5. Click OK.

Physics settings (Subdomain settings)

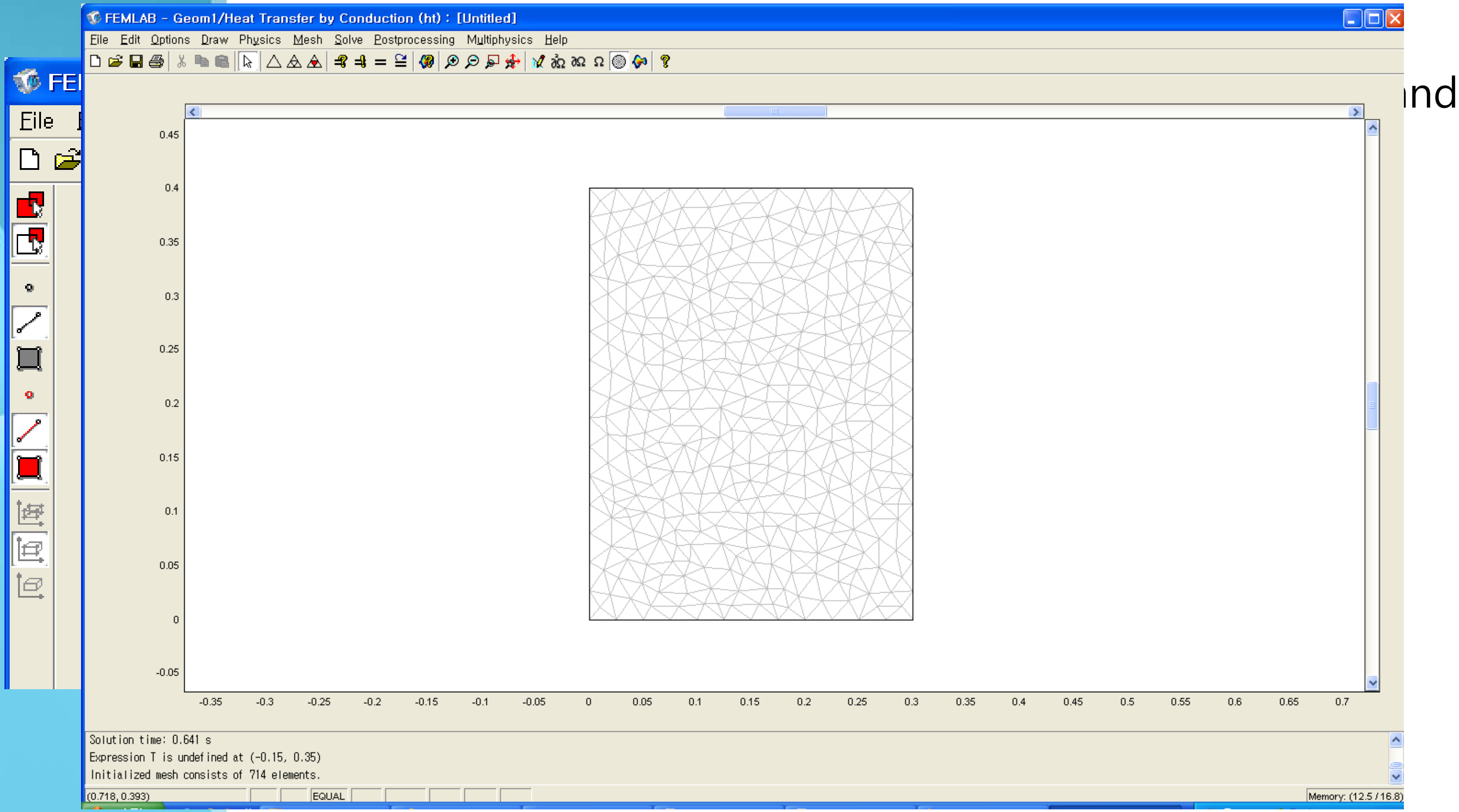


1. Go to the **Physics** menu and choose **Subdomain Settings**.

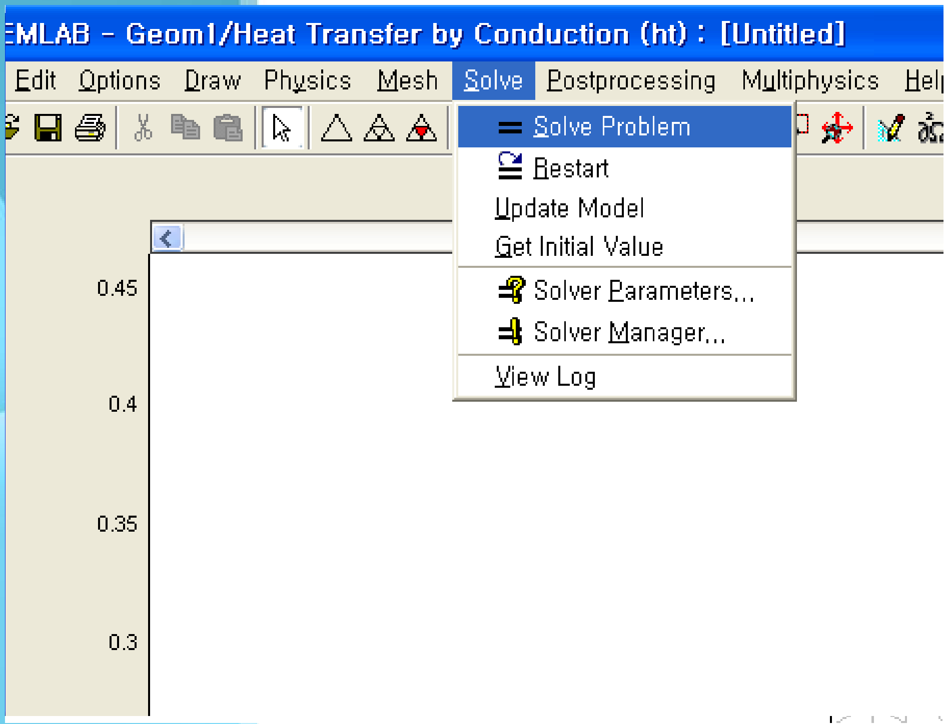
2. In the **Subdomain Settings** dialog box enter the themal properties in the domain accortding to the material properties.

3. Click **OK**.

Mesh generation



Solving the model



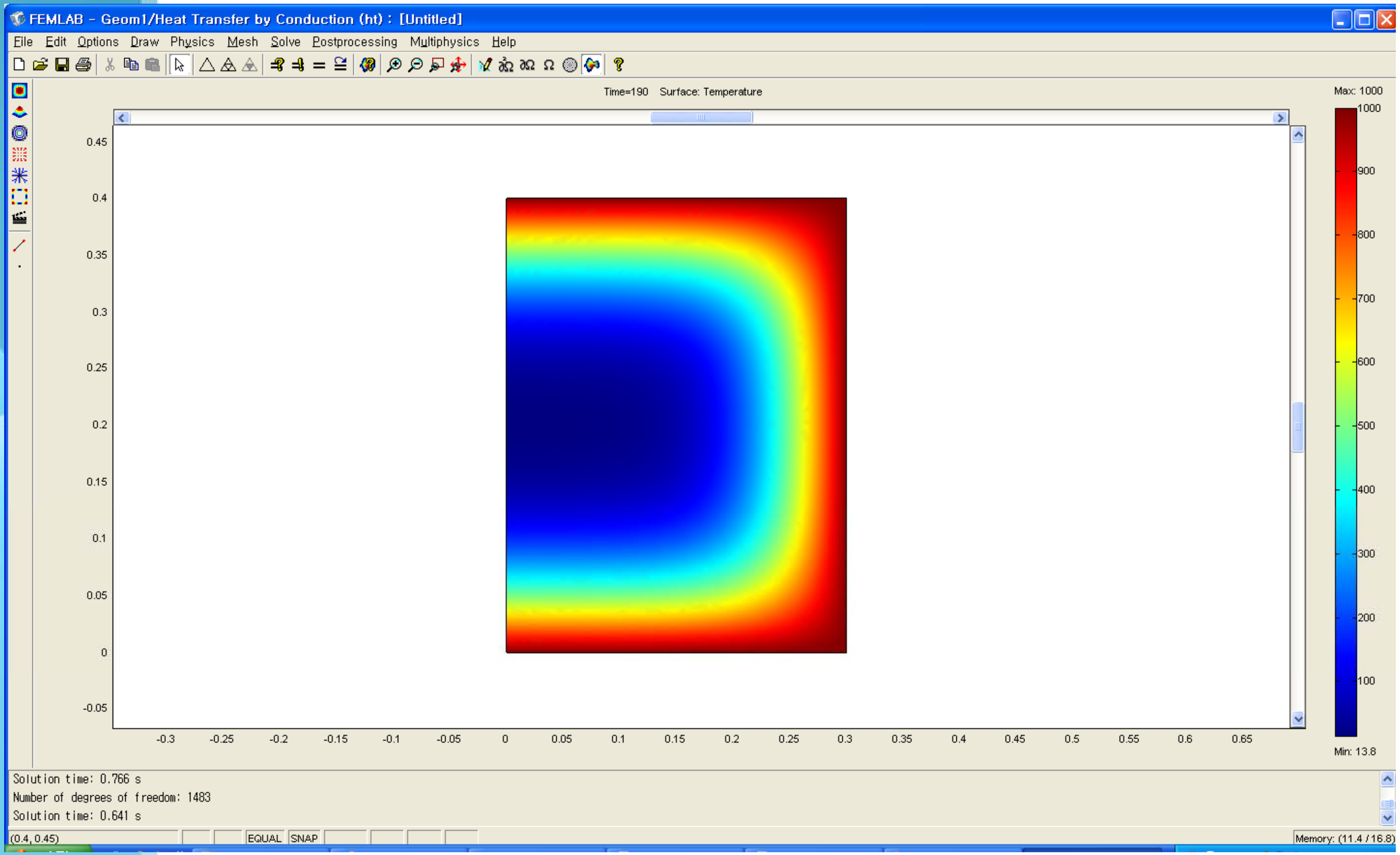
1. Go to the **Solve** menu and choose **Sover Parameters**.

2. In the **Time stepping** area in the **Solver Parameters** dialog box enter **0:10:190** in the **Times** edit field.

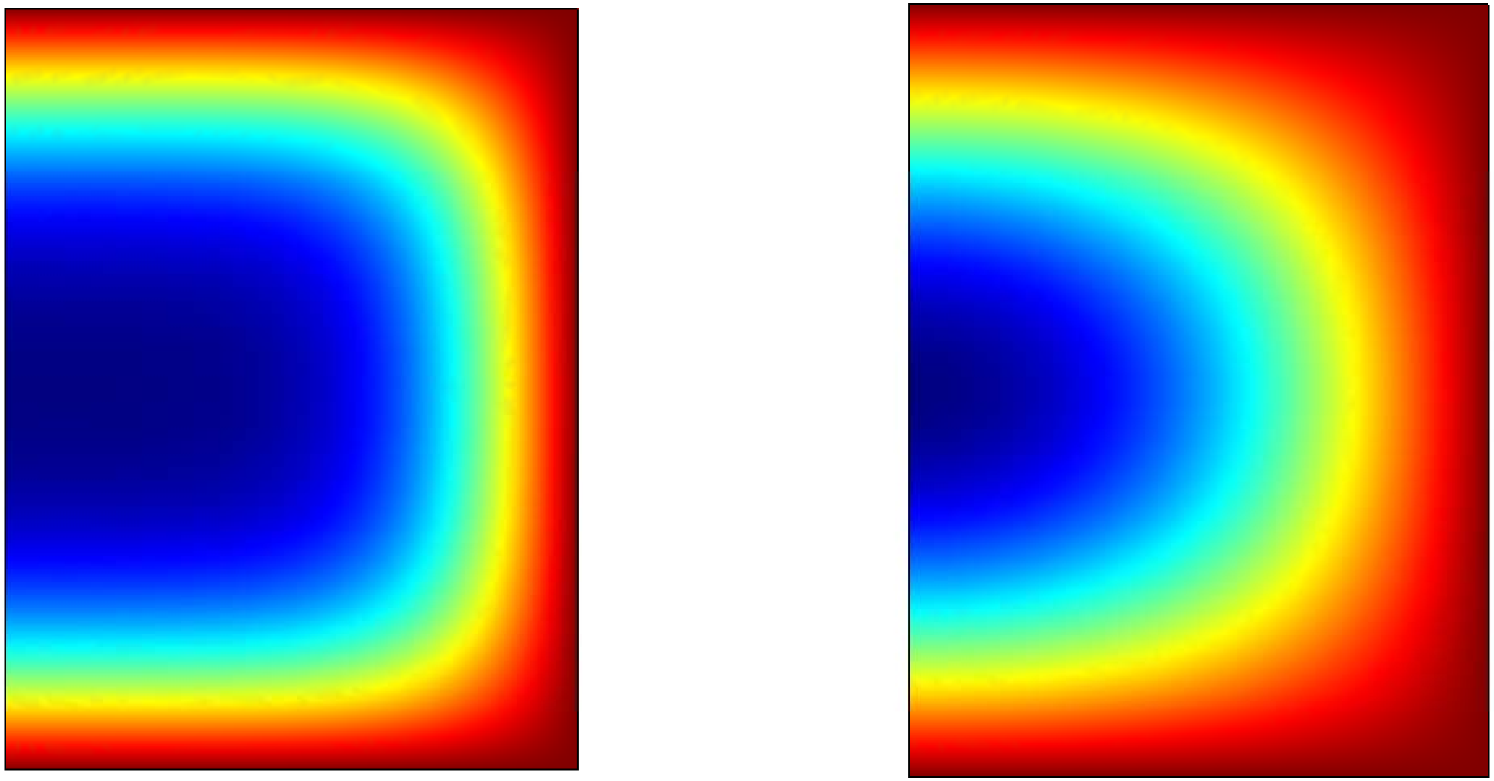
3. Click **OK**.

4. Click the **Solve** button.

Solving the model



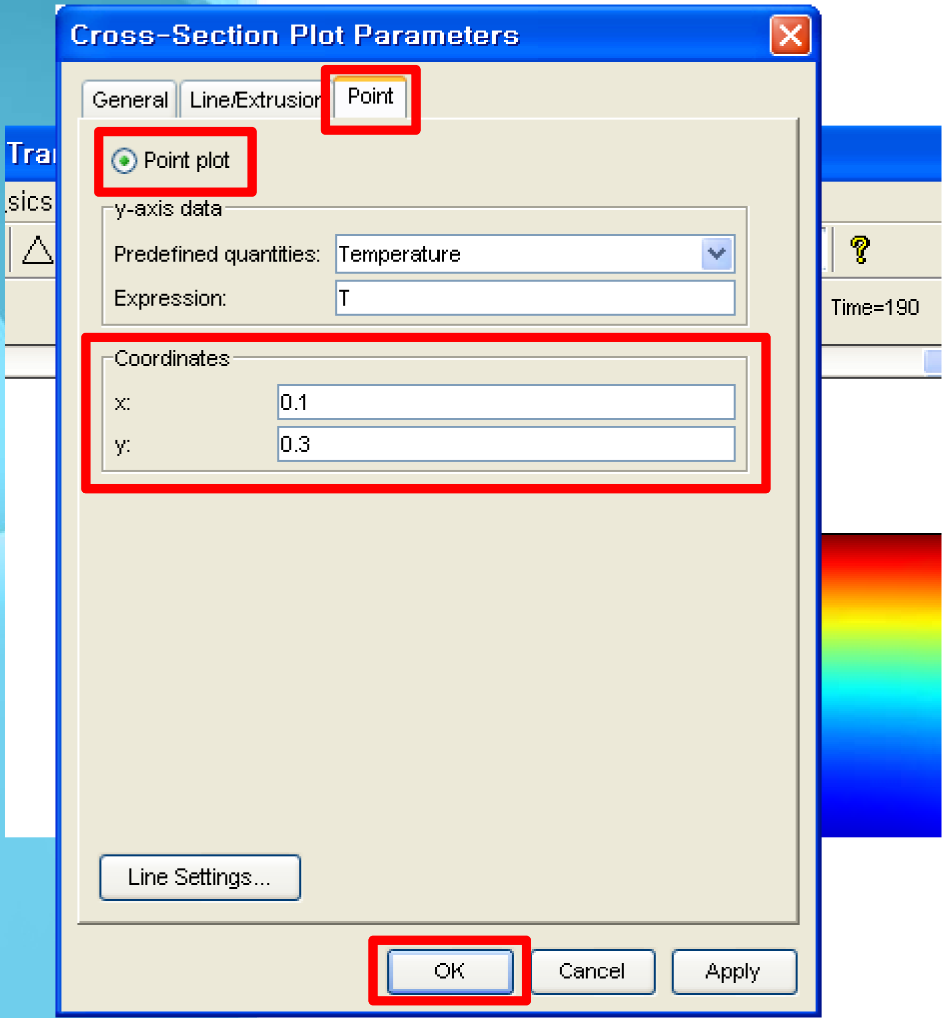
Solving the model



(a) (b)

Results of time stepping. (a)0:10:190 (b)0:10:1000

Postprocessing and visualization



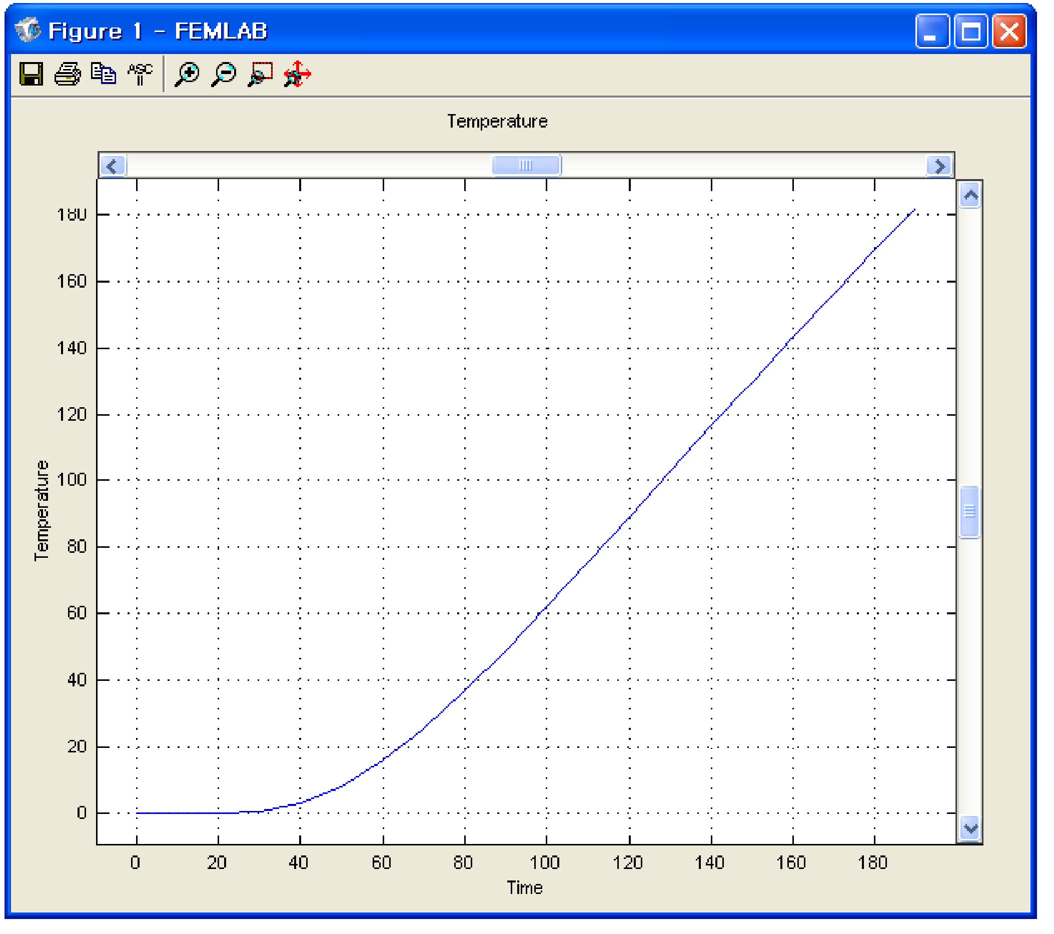
1. Go to the **Postprocessing** menu and choose **Cross-Section plot Parameters** dialog box click the **Point** tab.

2. Select the **Point plot** button.

3. Under **Coordinates** enter **0.1** in the **X** edit field and **0.3** in the **Y** edit field.

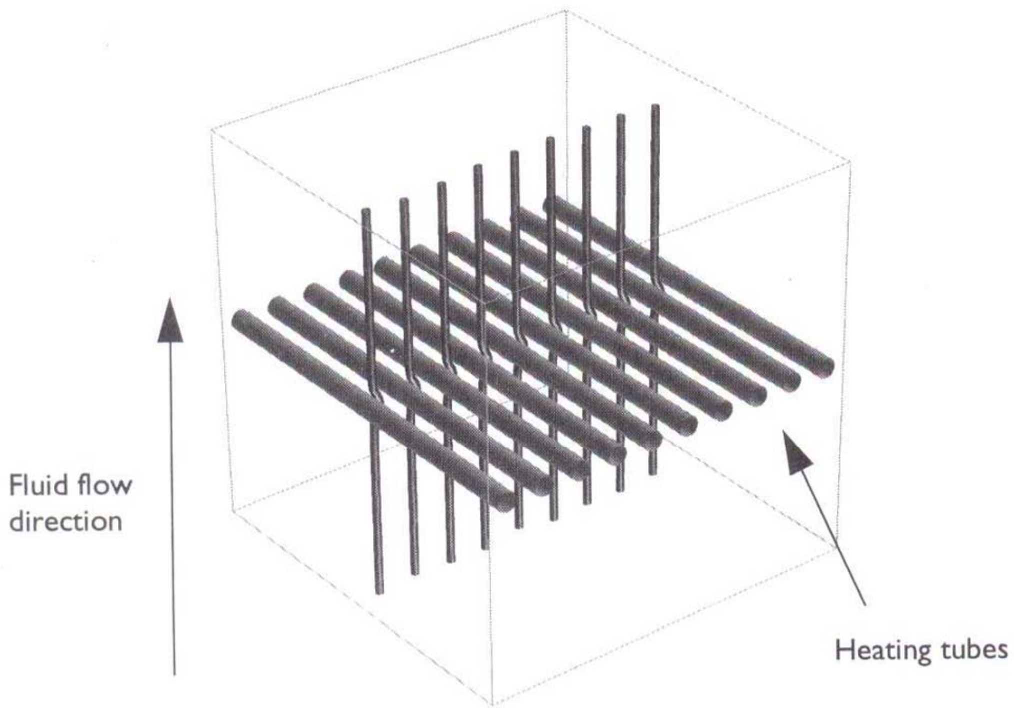
4. Click **OK**.

Postprocessing and visualization



Free convection

- This example describes a fluid problem with heat transfer in the fluid. An array of heating tubes is submerged in a vessel with fluid flow entering at the bottom.



Free convection(Partial differential equation)

- The incompressible Navier-Stoke equations consist of a momentum balance(a vector equation) and a mass conservation and incompressibility condition.

$$\rho \frac{\partial u}{\partial t} + \rho(u \cdot \nabla)u = -\nabla p + \eta \nabla^2 u + F$$

$$\nabla \cdot u = 0$$

- The equations are following variables.
- u is the velocity field.
- p is the pressure.
- F is a volume force.
- P is the fluid density.
- η is the dynamic viscosity.
- ∇ is the vector differential operator.

Free convection(continued)

- The heat equation is an energy conservation equation that says that the change in energy is equal to the heat source minus the divergence of the diffusive heat flux.

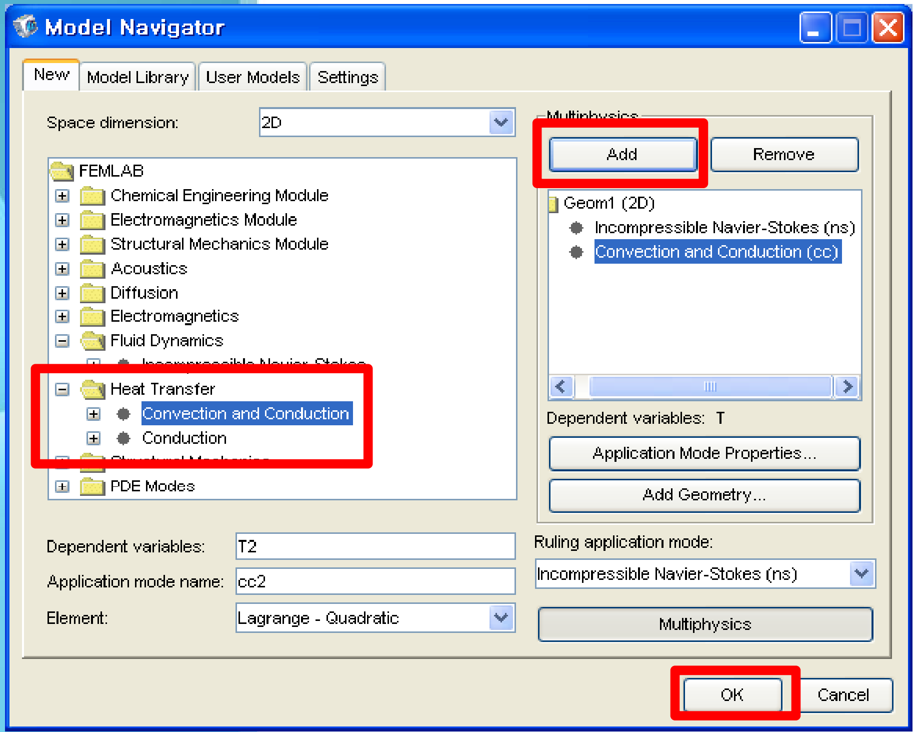
$$\rho c_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \nabla T + \rho c_p T u) = Q$$

- The equations are following variables.
- C_p is the heat capacity of the fluid.
- ρ is fluid density.
- Q is a source term.
- u is the velocity field.

Free convection(Physical constants)

Constants for modeling		
Property	Name	Expression(value)
Fluid density	rho0	1e3
Dynamic viscosity	mu	1e-3
Heat capacity	Cp	4.2e3
Thermal conductivity	kc	0.6
Volume expansion coefficient	alpha0	0.18e-3
Acceleration of gravity	g0	9.8
Inlet velocity	vin	5e-3
Inlet temperature	Tin	293
Heater temperature	Theat	303

Model navigator



1. Select 2D in the **Space dimension** list.

2. Click the **Multiphysics** button.

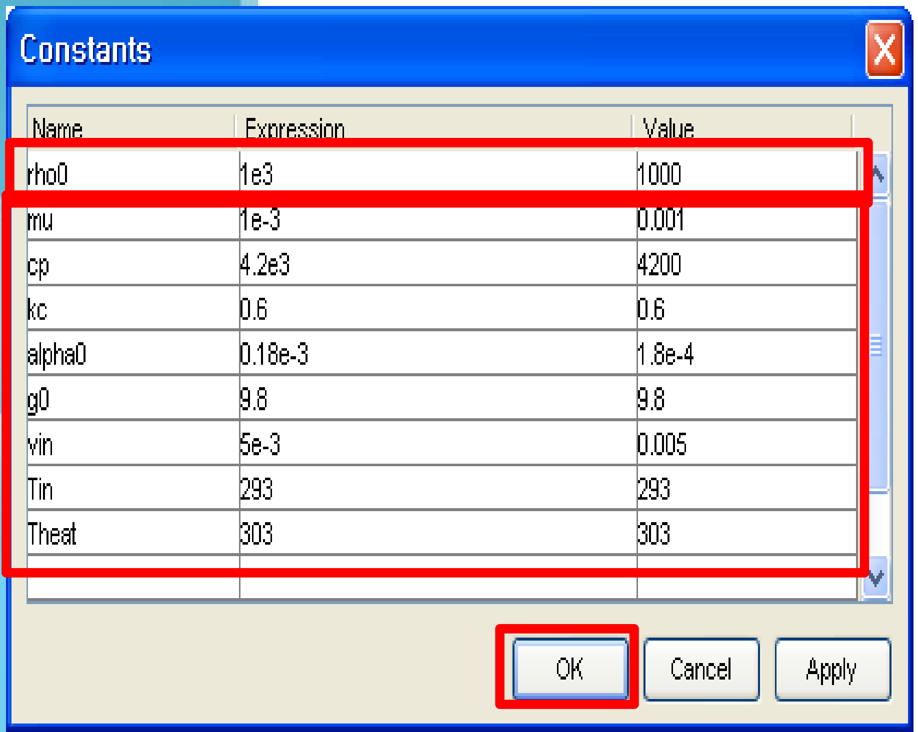
3. Open the **Fluid dynamics** and click **Incompressible Navier-Stokes.**

4. Click **Add**.

5. Open the **Heat transfer** and click **convection and conduction** and click **Add**.

6. Click **OK**.

Constants setting



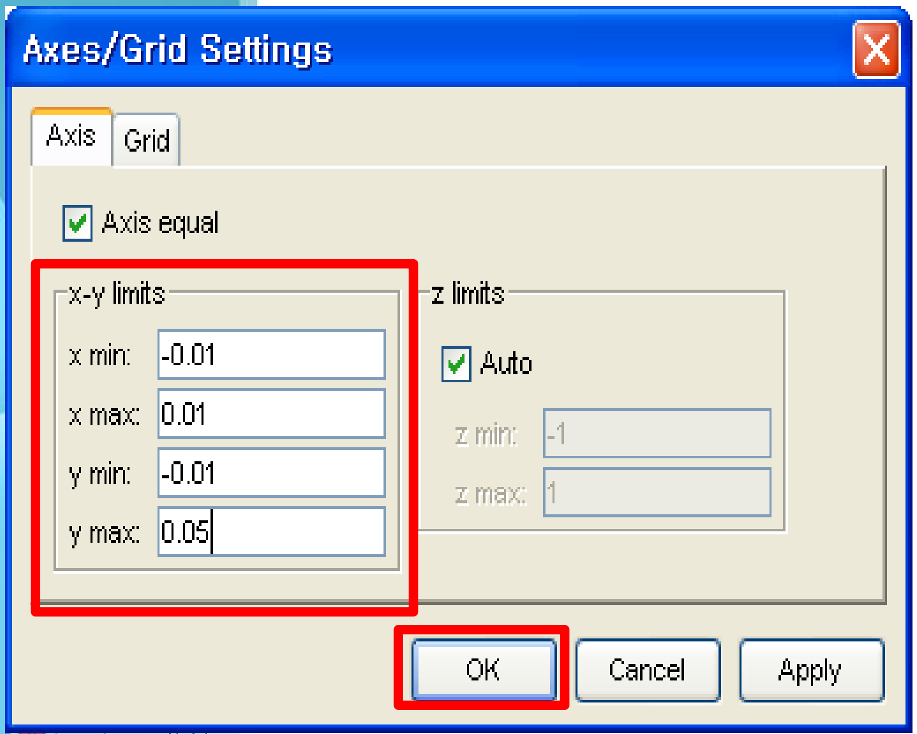
1. From the **Options** menu, choose **Constants**.

2. Enter **rho0** in the **Name** field. Move the cursor to the **Expression** field and enter **1e3**.

3. Continue by adding the remaining constants.

4. Click **OK**.

Axes/Grid setting

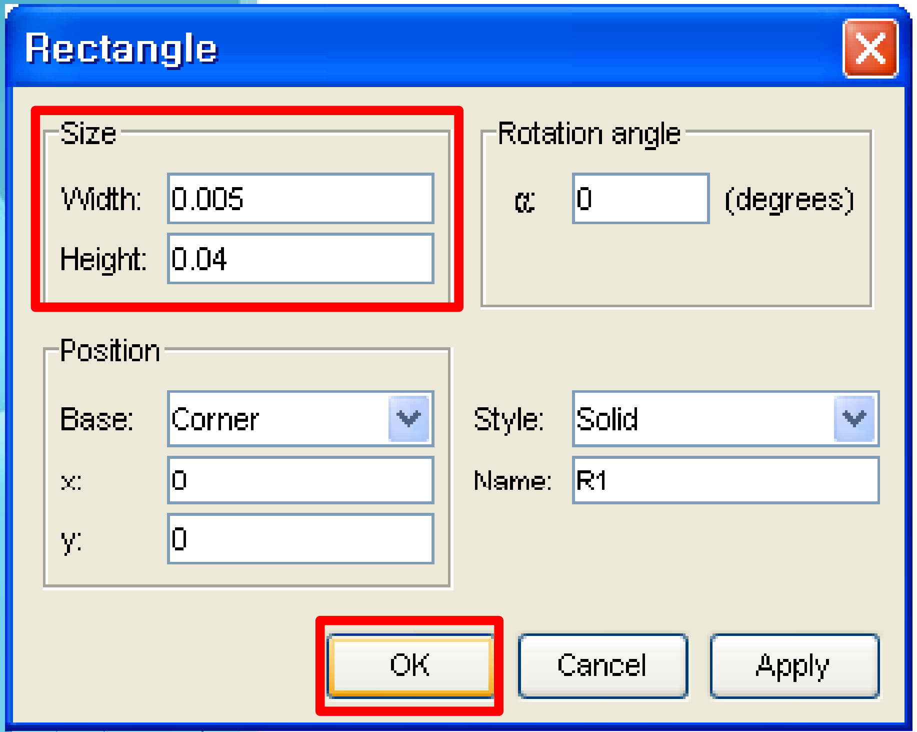


1. From **Options** menu, choose **Axes/Grid Settings**.

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

3. Click Ok.

Geometry modeling

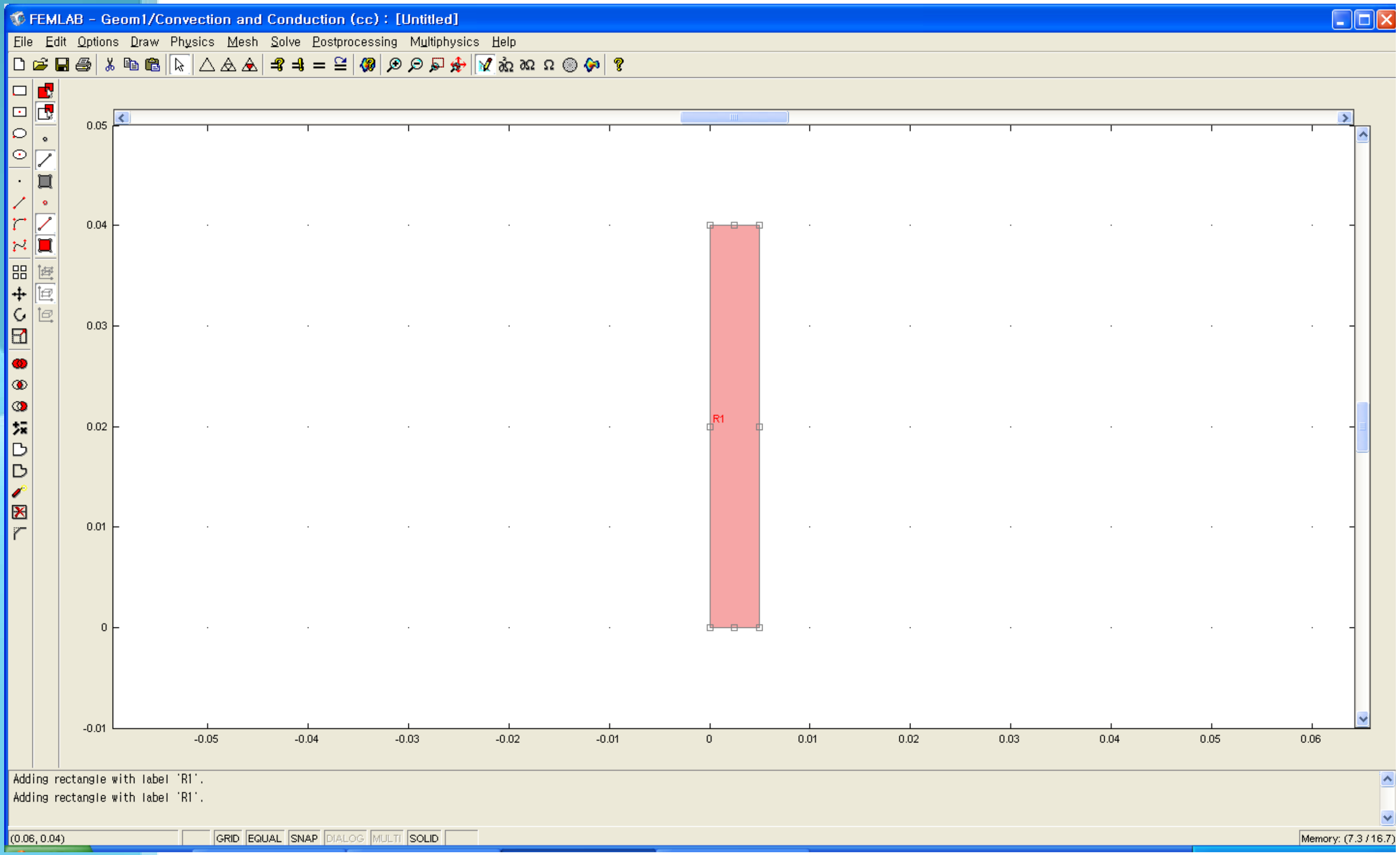


1. Go to the **Draw** menu, point to **Specify Object** and click **Rectangle**.

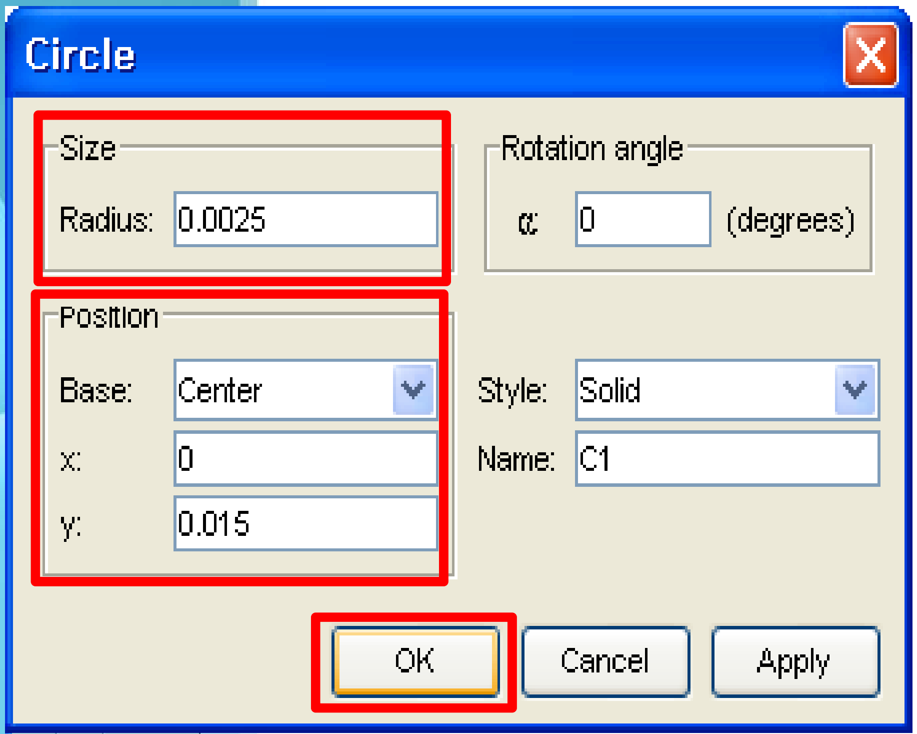
2. In the **Rectangle** dialog box go to the **Size** area and enter **0.005** in the **Width** edit field and **0.04** in the **Height** edit field.

3. Click **OK**.

Geometry modeling

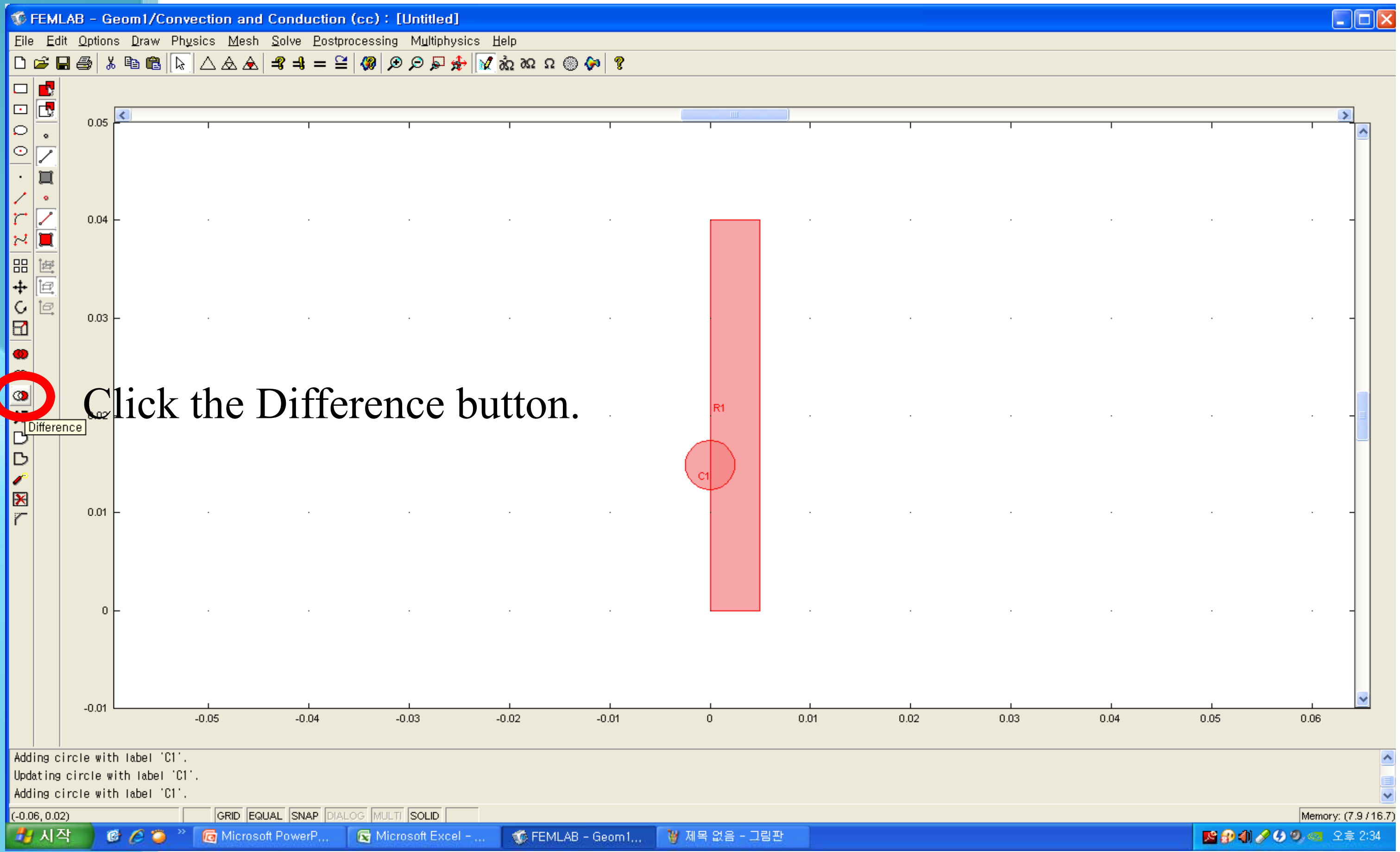


Geometry modeling



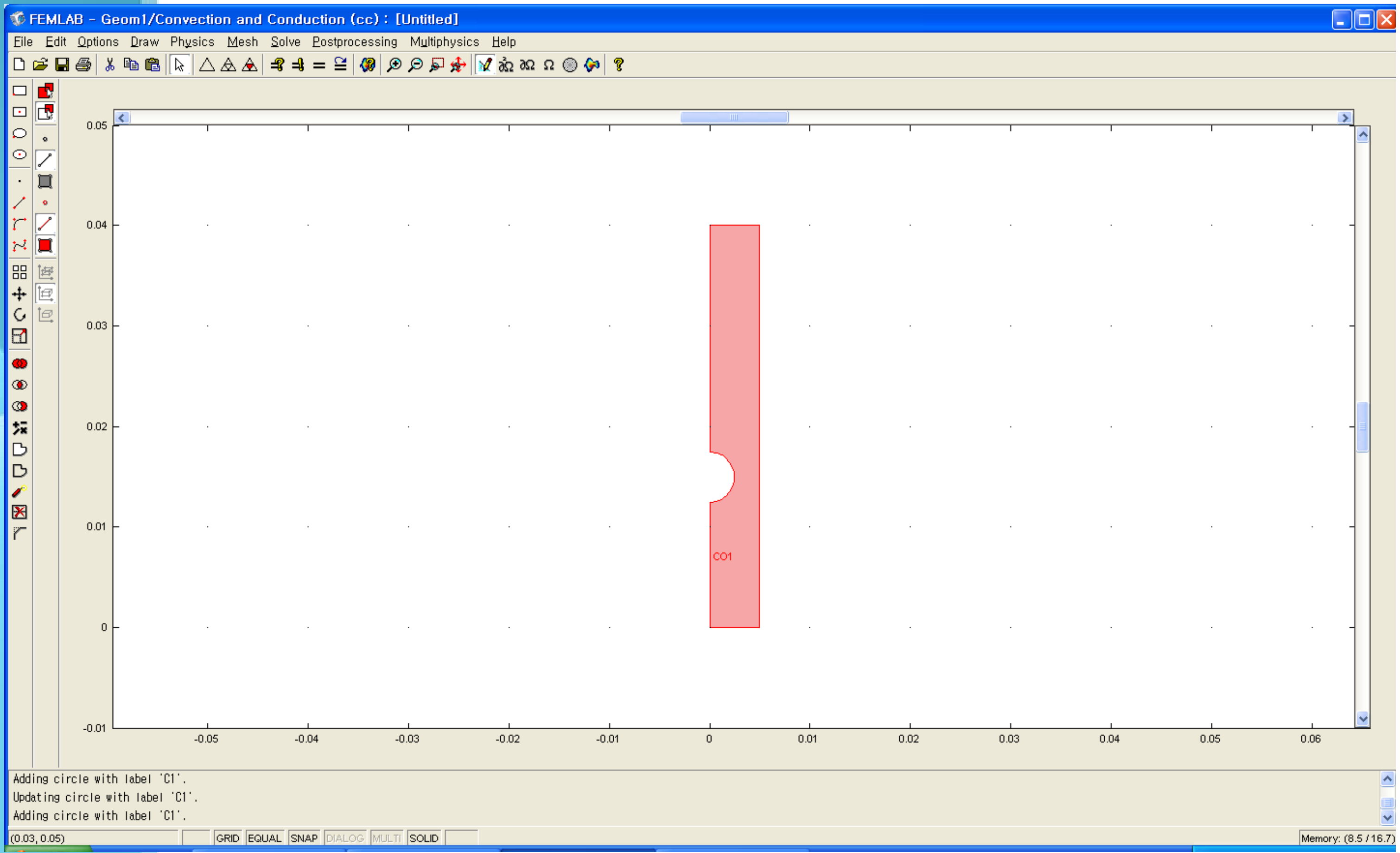
1. Go to the **Draw** menu, point to **Specify Object** and click **Circle**.
2. In the **Circle** dialog box go to the **Size** area and enter **0.0025** in the Radius edit field.
3. Go to **Position** area and enter **0.015** in the y edit field.
4. Click **OK**.

Geometry modeling

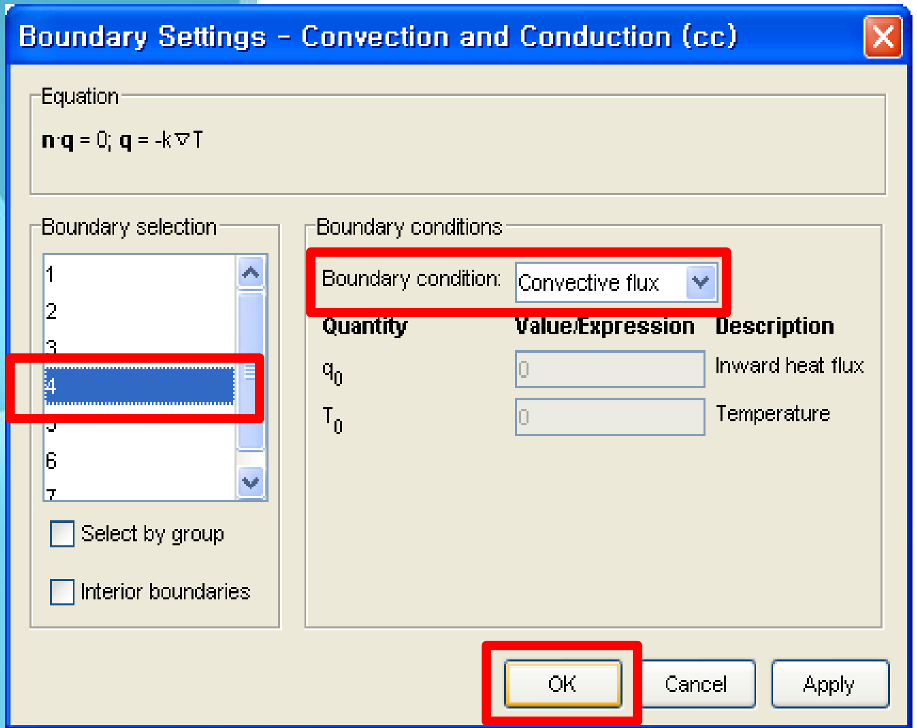


Click the Difference button.

Geometry modeling



Physics settings (Boundary settings)



1. Go to the **Physics** menu and choose **Boundary Settings**.

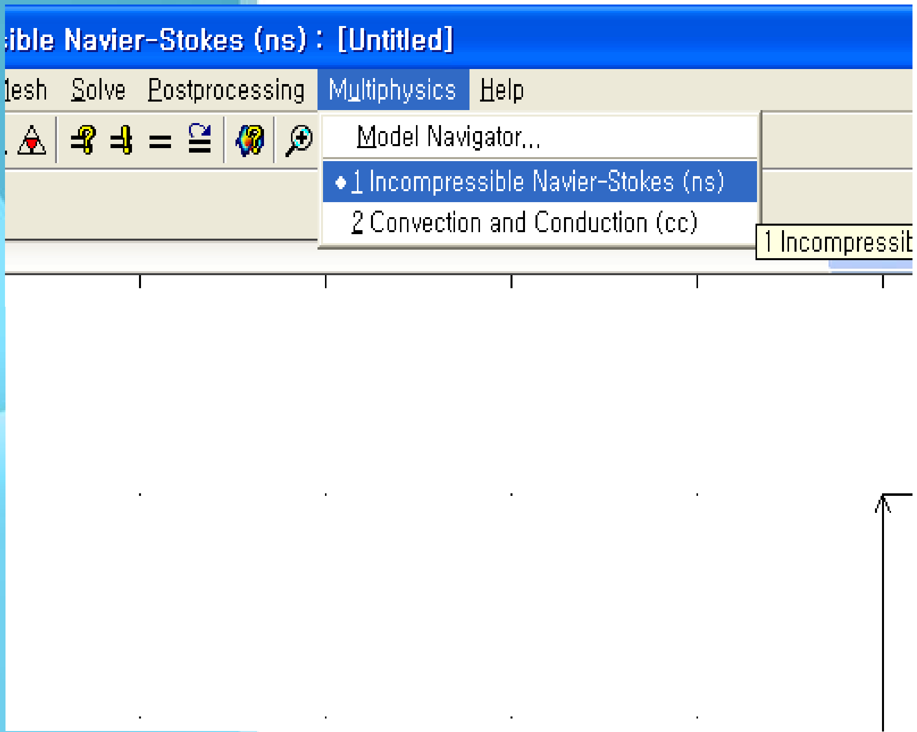
2. Select **Temperature** in the **Boundary conditions** list and enter **Tin** in the **Temperature** edit field at **boundary 2**.

3. Select **Temperature** in the **Boundary conditions** list and enter **Theat** in the **Temperature** edit field at **boundaries 6 and 7**.

4. Click the boundary 4 and select **Convective flux** in the **Boundary condition.**

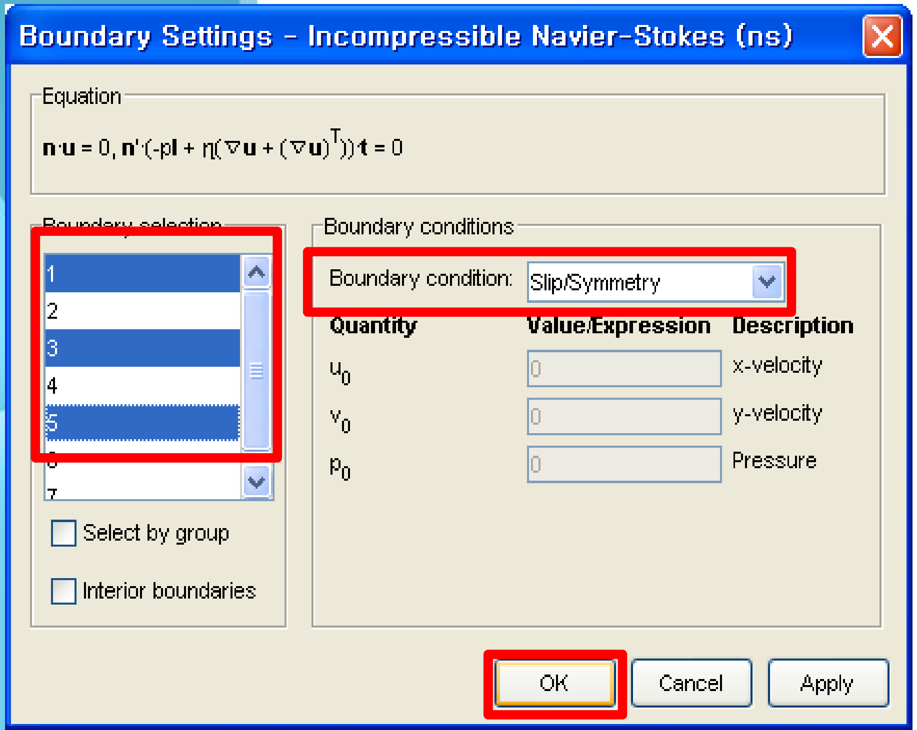
5. Click **OK**.

Physics settings (Boundary settings)



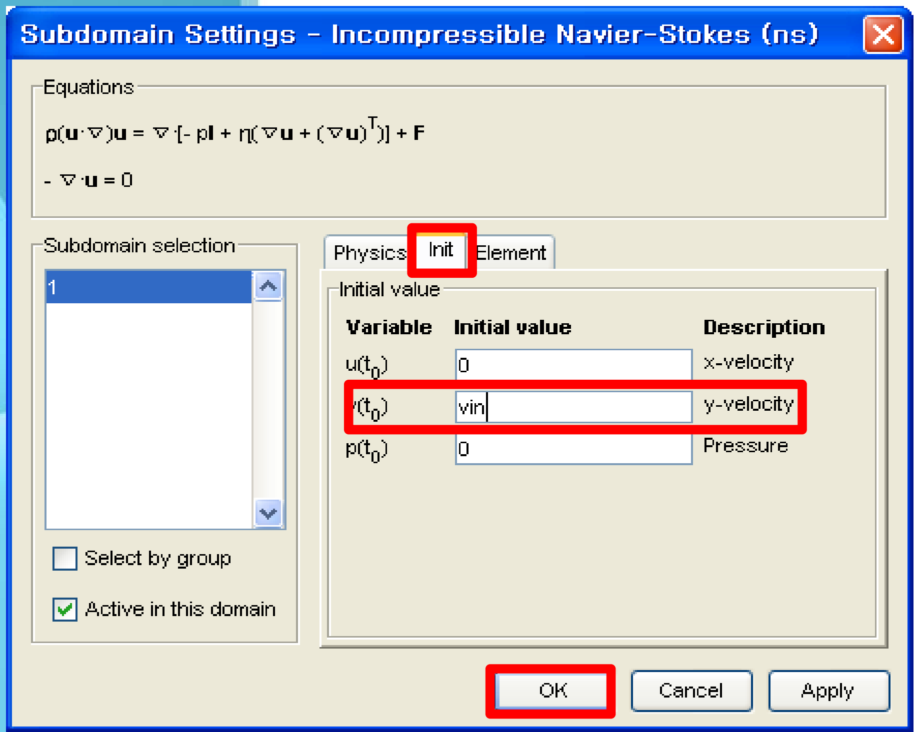
1. **Switch** to the Incompressible Navier-Stoke application mode by selecting this mode from the **Multiphysics** menu.

Physics settings (Boundary settings)



1. Select the **Physics** menu and then select **Boundary Settings**.
4. Go to the boundaries 6 and 7 and select the **No slip** boundary condition.
2. For the inflow boundary, select boundary 2 and then select the **Inflow/Outflow velocity** in the **Boundary condition** list. Enter **vin** in the **y-velocity** edit field (leave the x velocity 0).
5. Other boundaries select the **Slip/Symmetry**.
6. Click **OK**.
3. Continue by selecting the outflow boundary 4. In the **Boundary condition list**, select **Normal flow/Pressure**. Leave the pressure at 0.

Physics settings (Subdomain settings)



1. Choose **Subdomain Settings** from the **Physics** menu

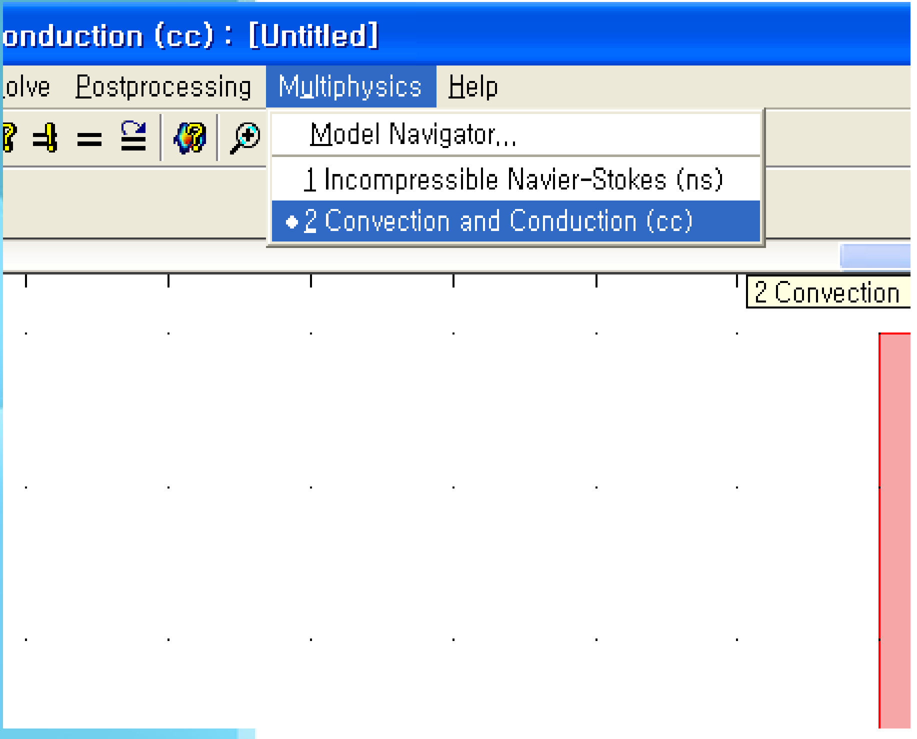
2. Select the single subdomain, number 1, and enter **rho0** and **mu** in the **Density** and **Dynamic viscosity** fields.

3. Enter **alpha0*g0*rho0*(T-Tin)** in the **volume force**, **y-dir**.

4. Click **Int** tab and set the initial value **v(t0)** to **vin**.

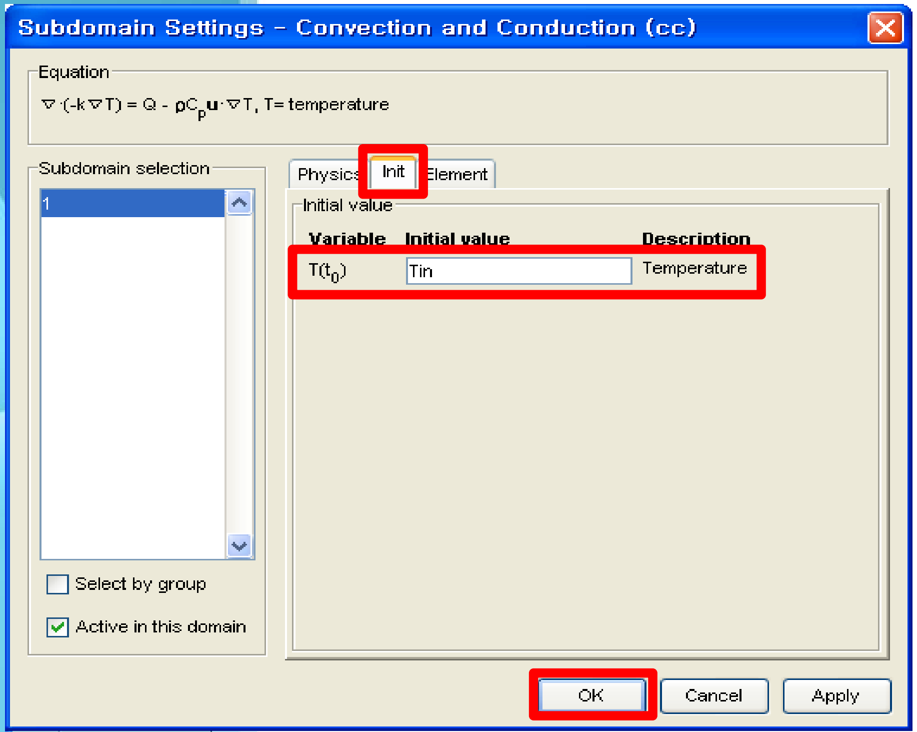
5. Click **OK**.

Physics settings (Subdomain settings)



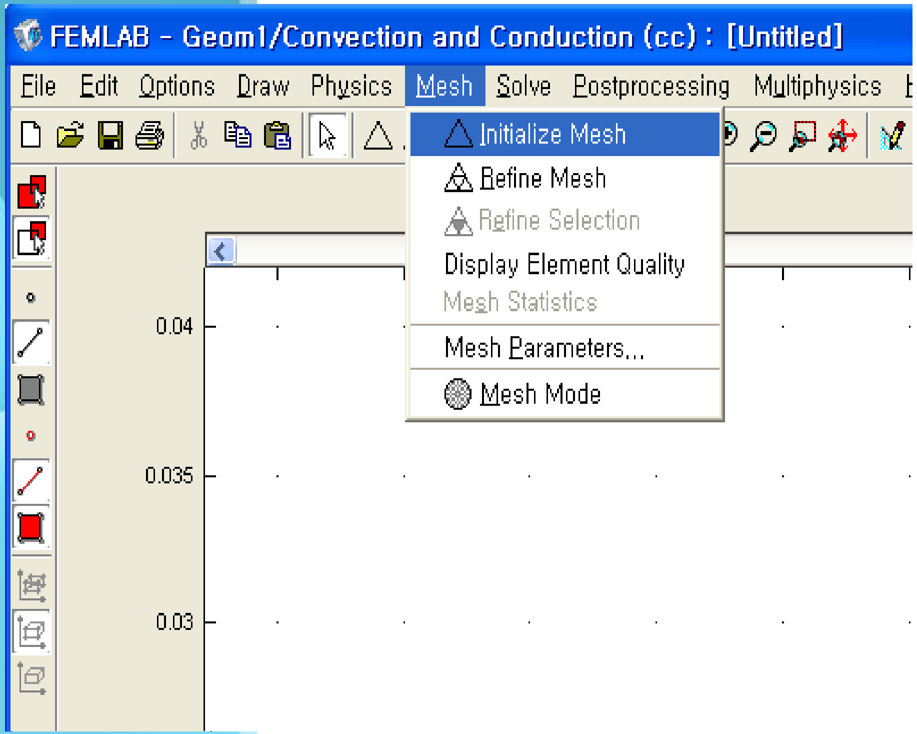
1. **Switch** to the Convection and conduction application mode by selecting this mode from the **Multiphysics** menu.

Physics settings (Subdomain settings)



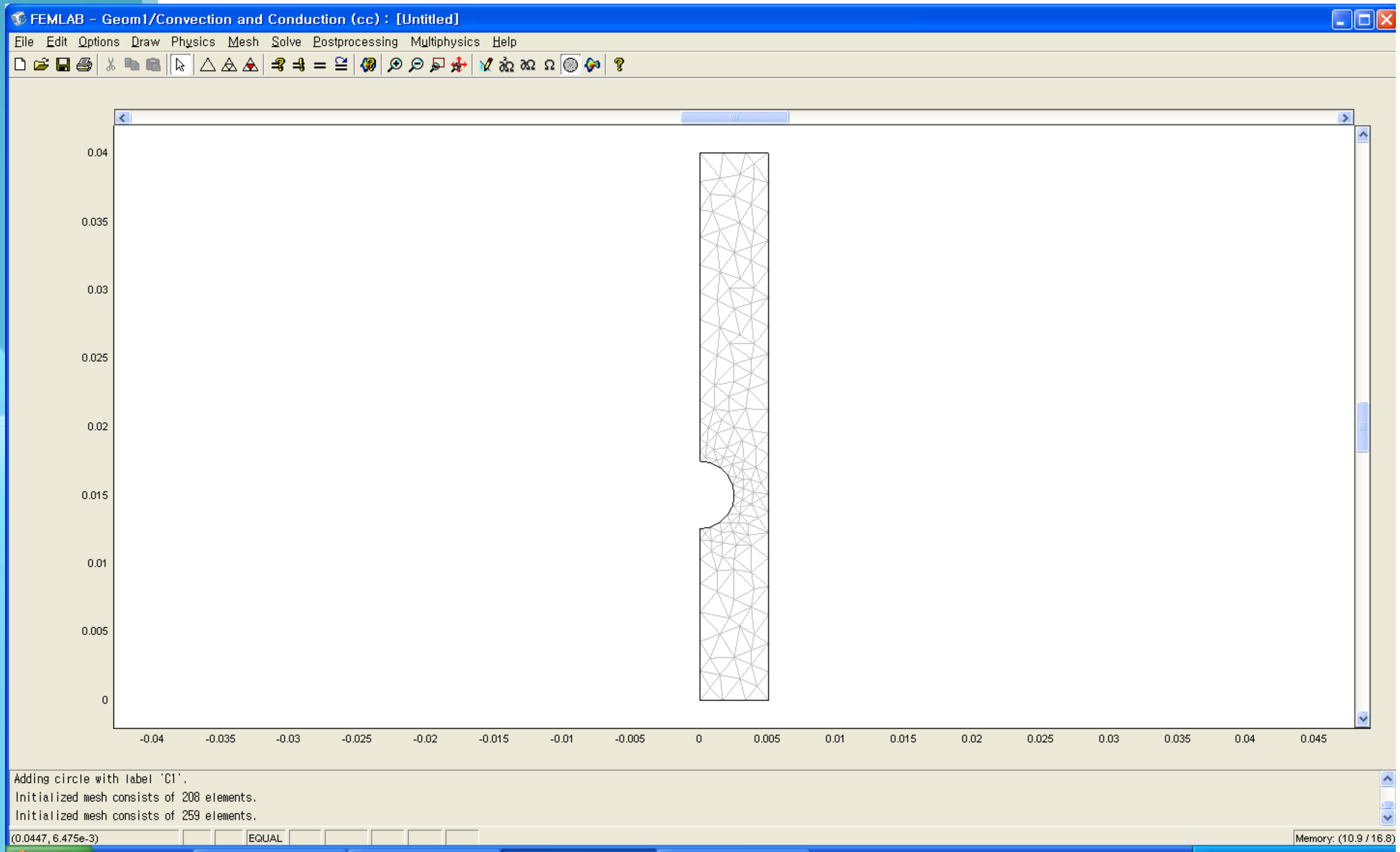
1. Choose **Subdomain Settings** from the **Physics** menu.
2. Enter **rho0**, **cp** and **kc** in the **Density**, **Heat capacity**, and **Thermal conductivity** edit fields.
3. Enter **0** in the **x-velocity** and **y-velocity** edit fields.
4. Click **Int** tab and enter **Tin** in the **T(t0)** field.
5. Click **OK**.

Mesh generation

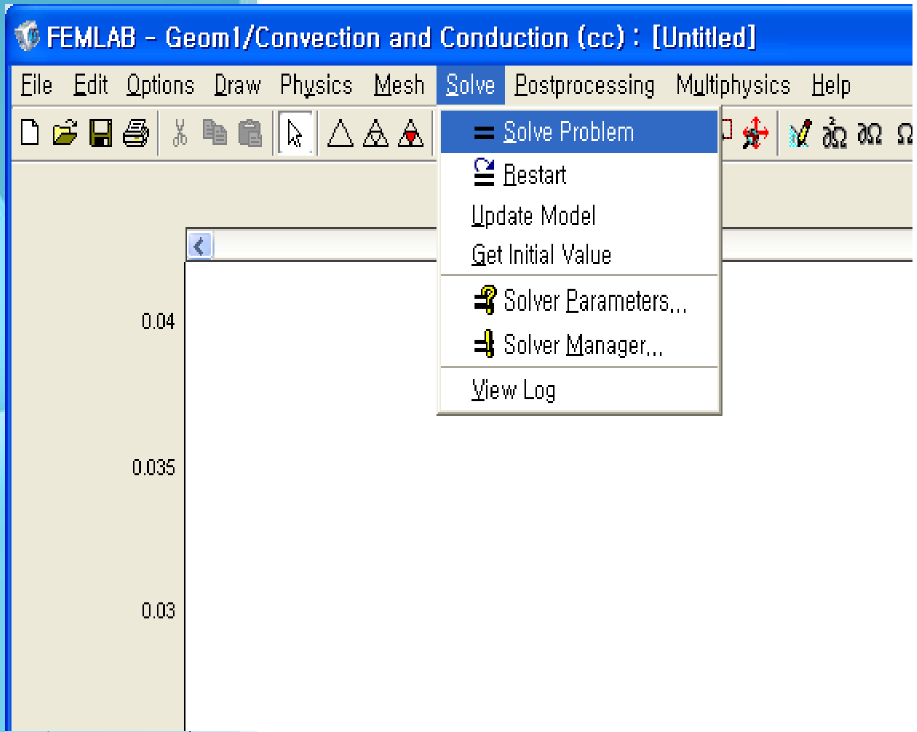


1. From the **Mesh** menu, choose **Mesh Parameters**.
2. In the **Mesh Parameters** dialog box, select **Fine** in the **Predefined mesh sizes** list.
3. Click **OK**.
4. Form the **Mesh** menu, choose **Initialize Mesh**.

Mesh generation



Solving the model



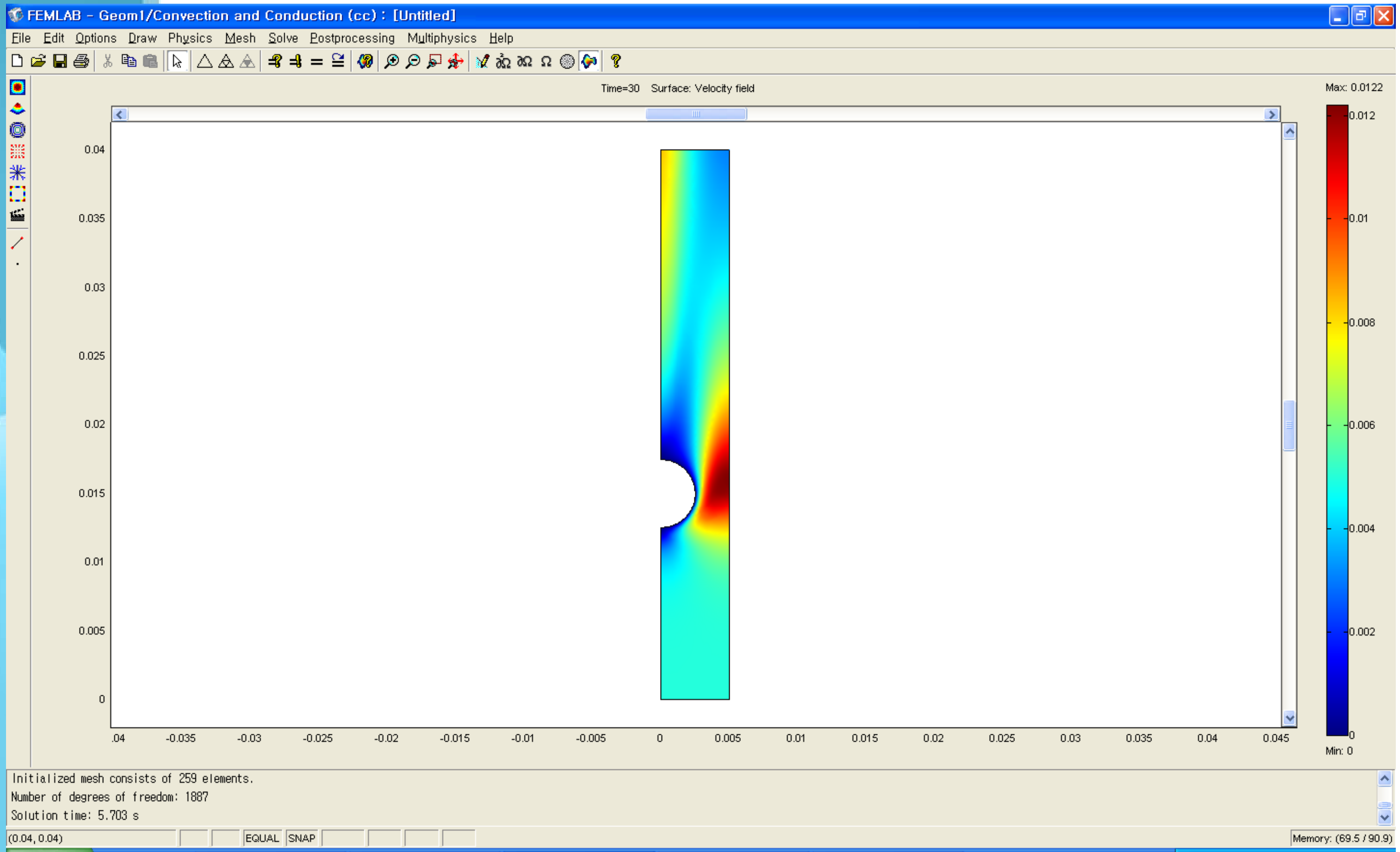
1. Go to the **Solve** menu and choose **Sover Parameters**.

2. In the **Time stepping** area in the **Solver Parameters** dialog box enter **0:1:30** in the **Times** edit field.

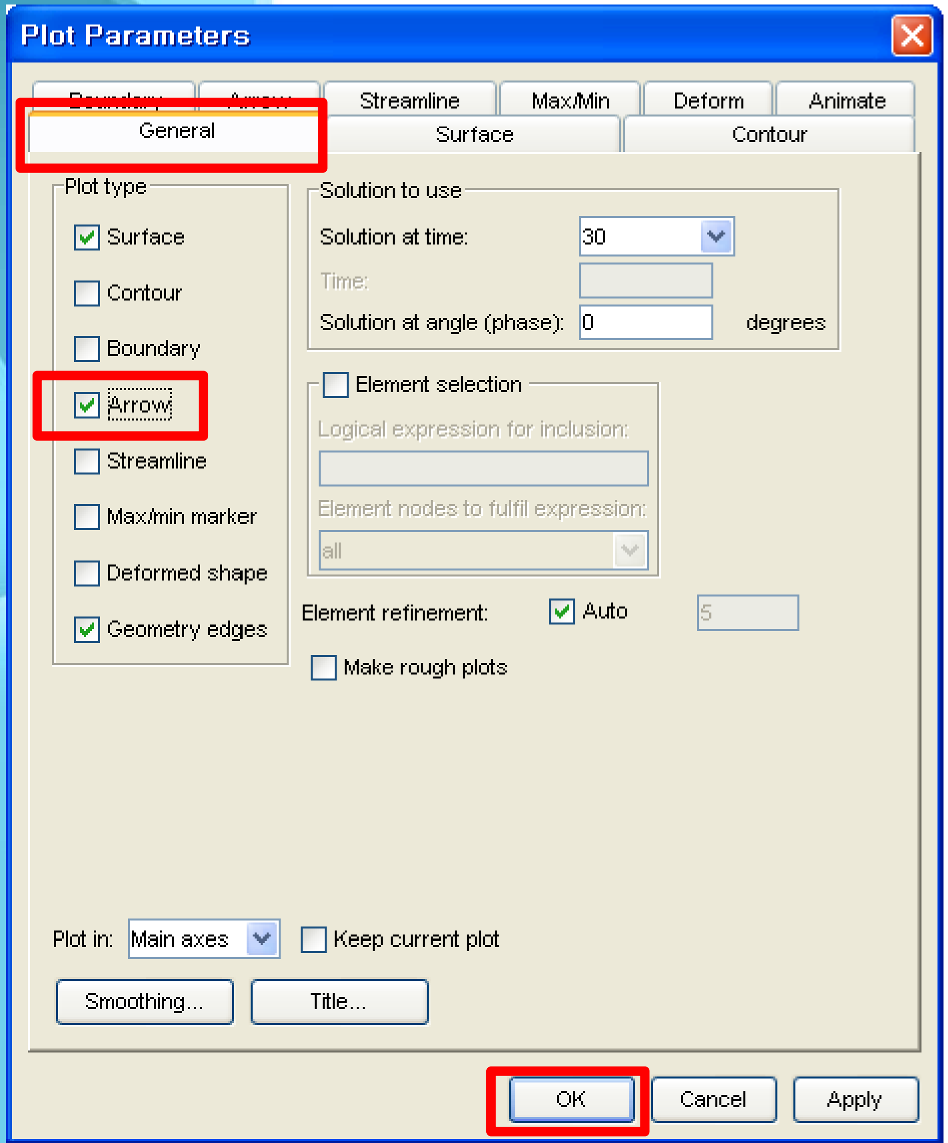
3. Click **OK**.

4. Click the **Solve** button.

Solving the model

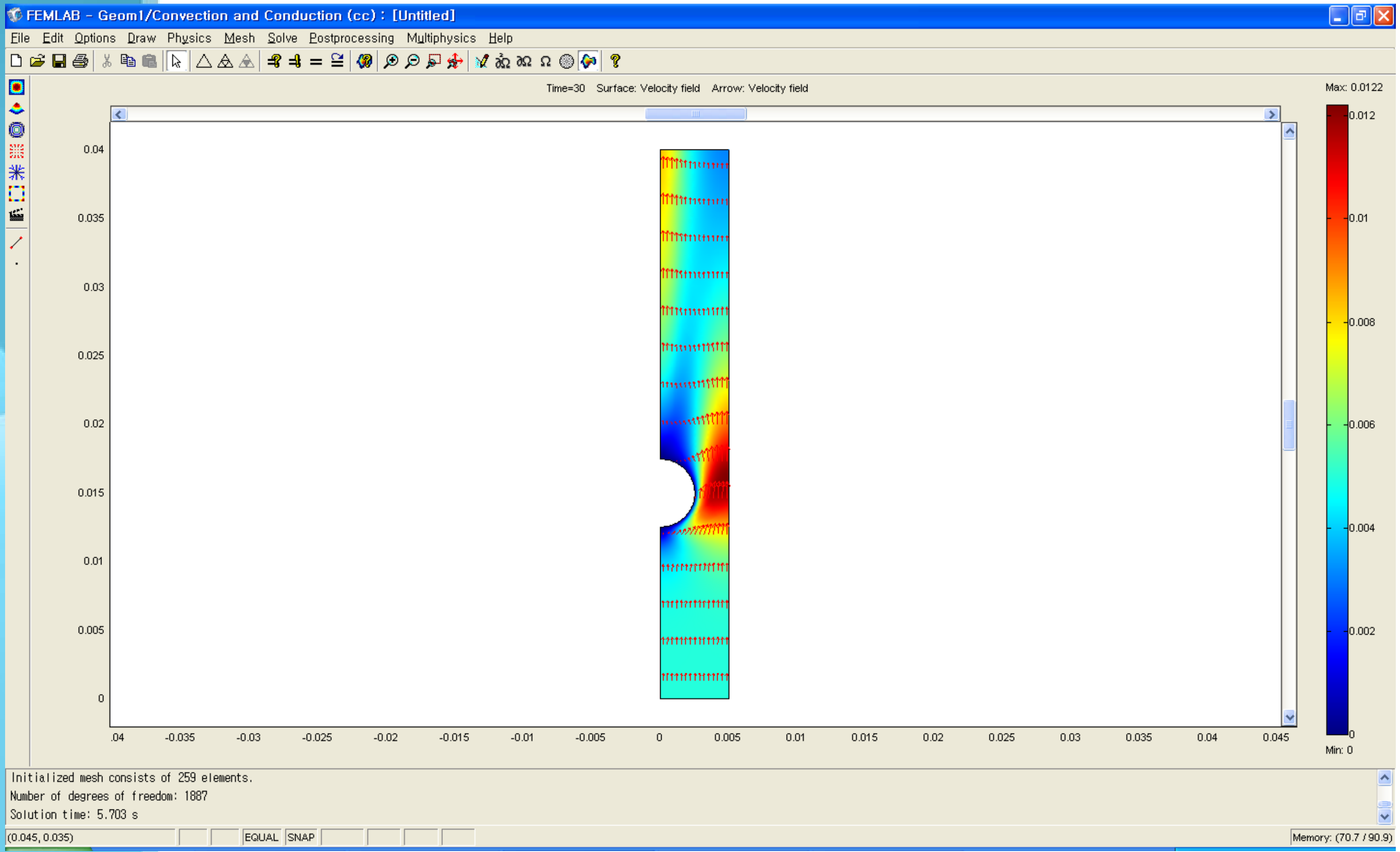


Solving the model



1. Go to the **Postprocessing** menu and choose **Plot Parameters**.
2. Check the Arrow at the plot type at the **General** tab in the **Plot Parameters** dialog box.
3. Click **OK**.

Solving the model



Solving the model