MEMS in Chemical Engineering Module

Bioprocess Laboratory
Department of Chemical Engineering
Chungnam National University

Heat balance

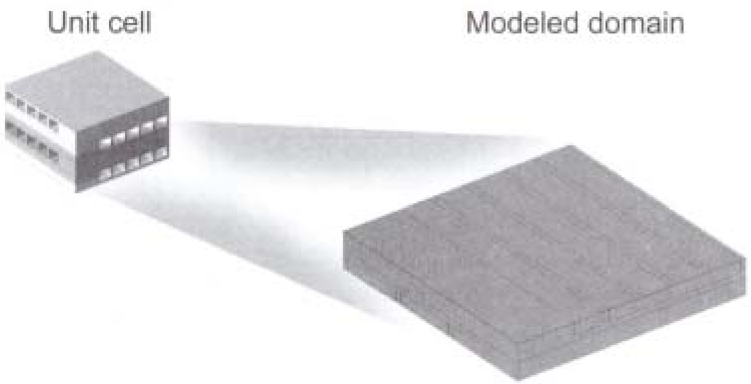
- The general equations for the heat balance

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

Parameter	Meaning
C_p	Heat capacity
T	Temperature
k	Thermal conductivity
Q	Heat sink or source
u	Velocity profile

A 3D Model of a MEMS heat exchanger

- This model deals with a micro heat exchanger made of stainless steel.
- These type of heat exchangers are found in lab-on-a-chip devices in biotechnology and in microreactors.
- Heat exchanger is of cross flow configuration and can consist of about 20 unit cells.
- At the model, modeled domain size is 800μm X 800μm X 60μm.



$$\nabla \cdot (-k\nabla T + \rho C_p T u) = 0 \quad at\ steady\ state$$

A 3D Model of a MEMS heat exchanger

- In the channels, channel flow is fully developed laminar flow.
- For both the hot and cold streams, the velocity component in the z-direction is set to zero.
- For the cold stream, the y-component of the velocity is zero while the x-component is given by the expression below

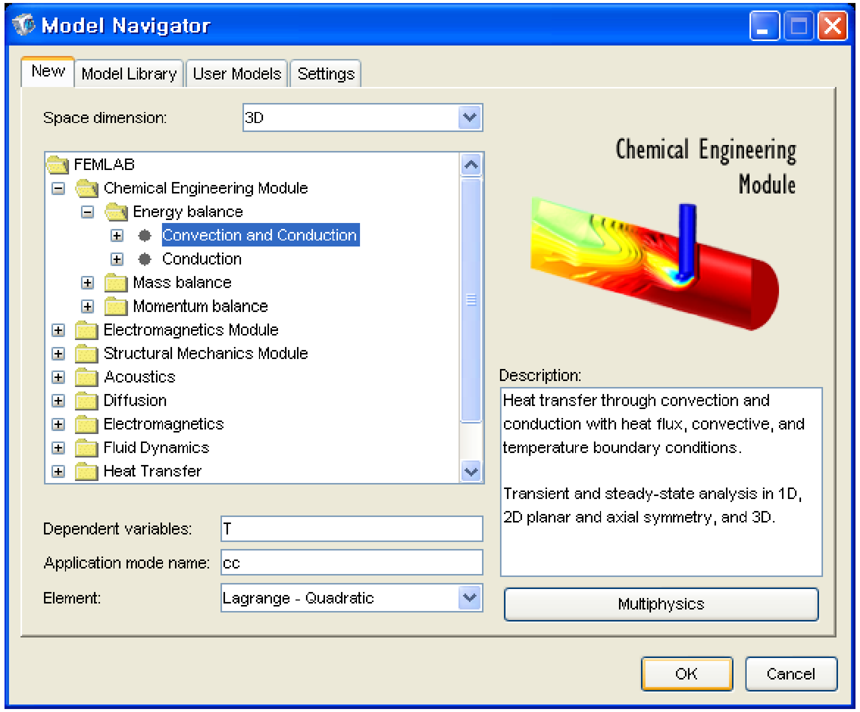
$$u = 16u_{\max} \frac{(z - z_0)(z_1 - z)}{(z_1 - z_0)^2} \frac{(y - y_0)(y_1 - y)}{(y_1 - y_0)^2}$$

- The velocity component in the hot stream is zero in the x-direction while the y-component is given by the following expression

$$v = 16v_{\max} \frac{(z - z_0)(z_1 - z)}{(z_1 - z_0)^2} \frac{(x - x_0)(x_1 - x)}{(x_1 - x_0)^2}$$

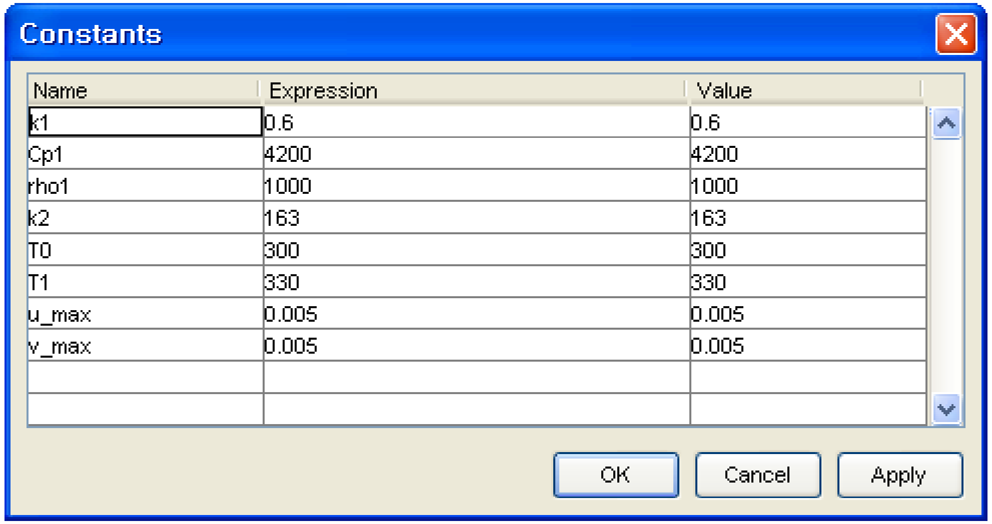
$$T = T_{cold} \qquad T = T_{hot}$$

Model navigator



1. Start FEMLAB.
2. Select **Space dimension** 3D.
3. Select the **Chemical Engineering Module**, **Energy Balance**, **Convection and Conduction mode**.
4. Click OK.

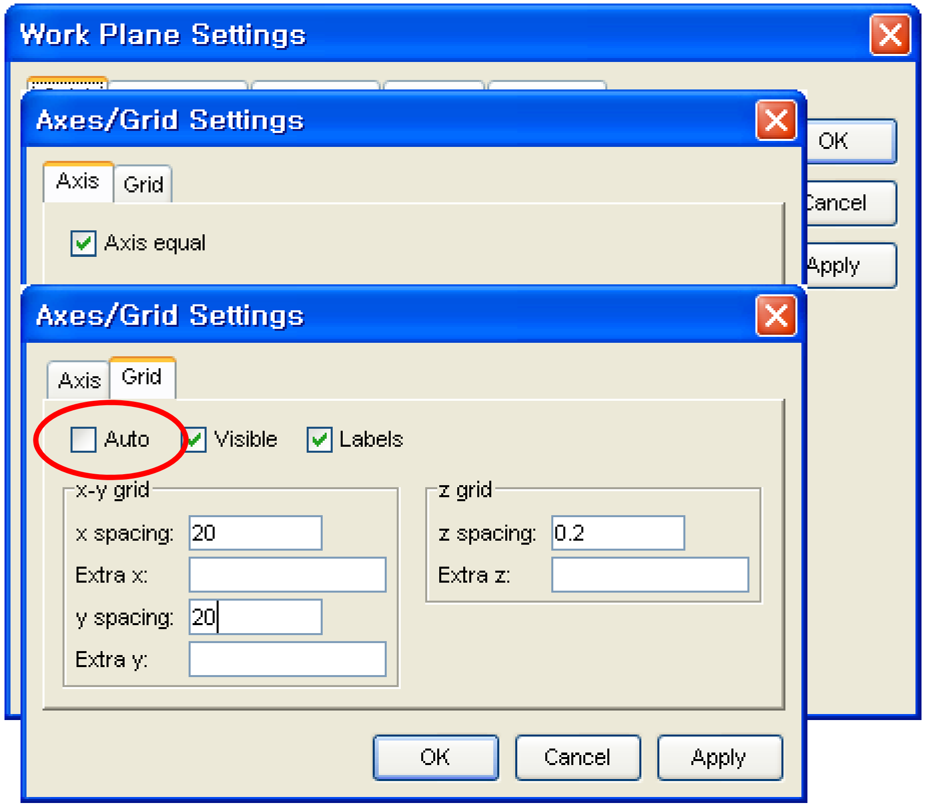
Options and settings



Name	Expression	Value
k1	0.6	0.6
Cp1	4200	4200
rho1	1000	1000
k2	163	163
T0	300	300
T1	330	330
u_max	0.005	0.005
v_max	0.005	0.005

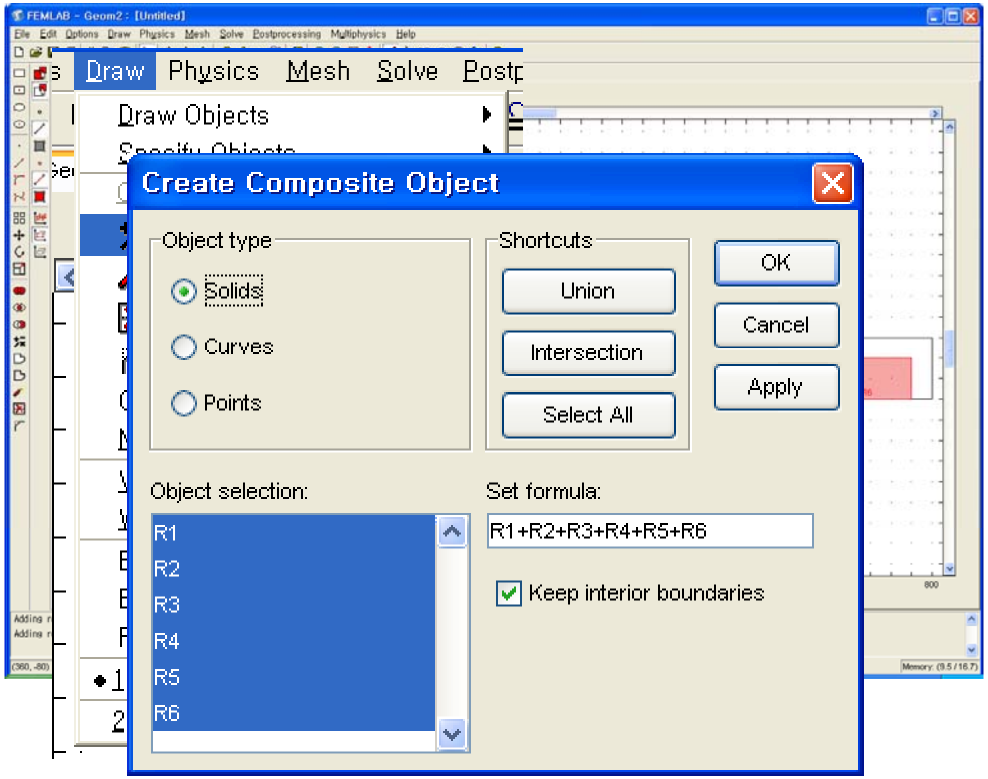
1. Select **Constants** in the **Options** menu.
2. Define the constants according to the figure below.

Geometry modeling



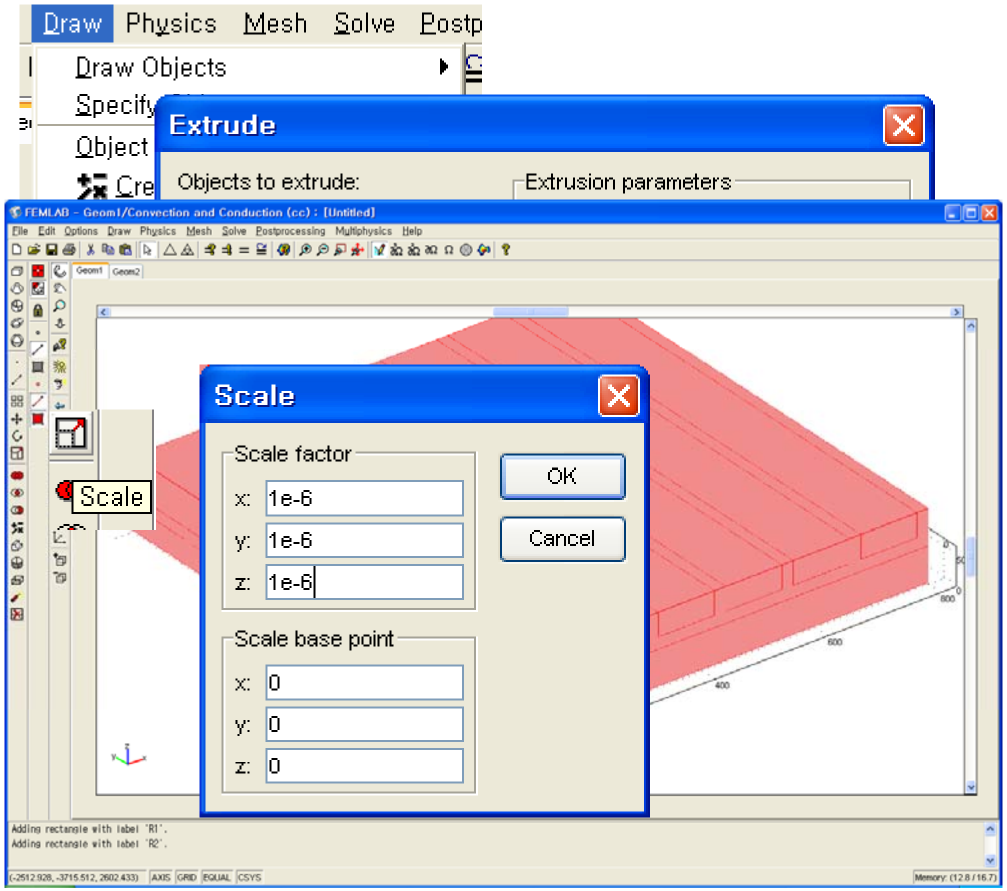
1. Select **Work Plane Settings** from the **Draw** menu.
2. Click the **Quick** tab and select y-z plane.
3. Click **OK**.
4. Open **Axes/Grid Settings** from the **Options** menu.
5. Clear the **Auto** check box.
6. Type the axis values according to the figure below.
7. Go to the **Grid** page.
8. Clear the **Auto** check box.
9. Set x spacing and y spacing to 20.
10. Click **OK**.

Geometry modeling



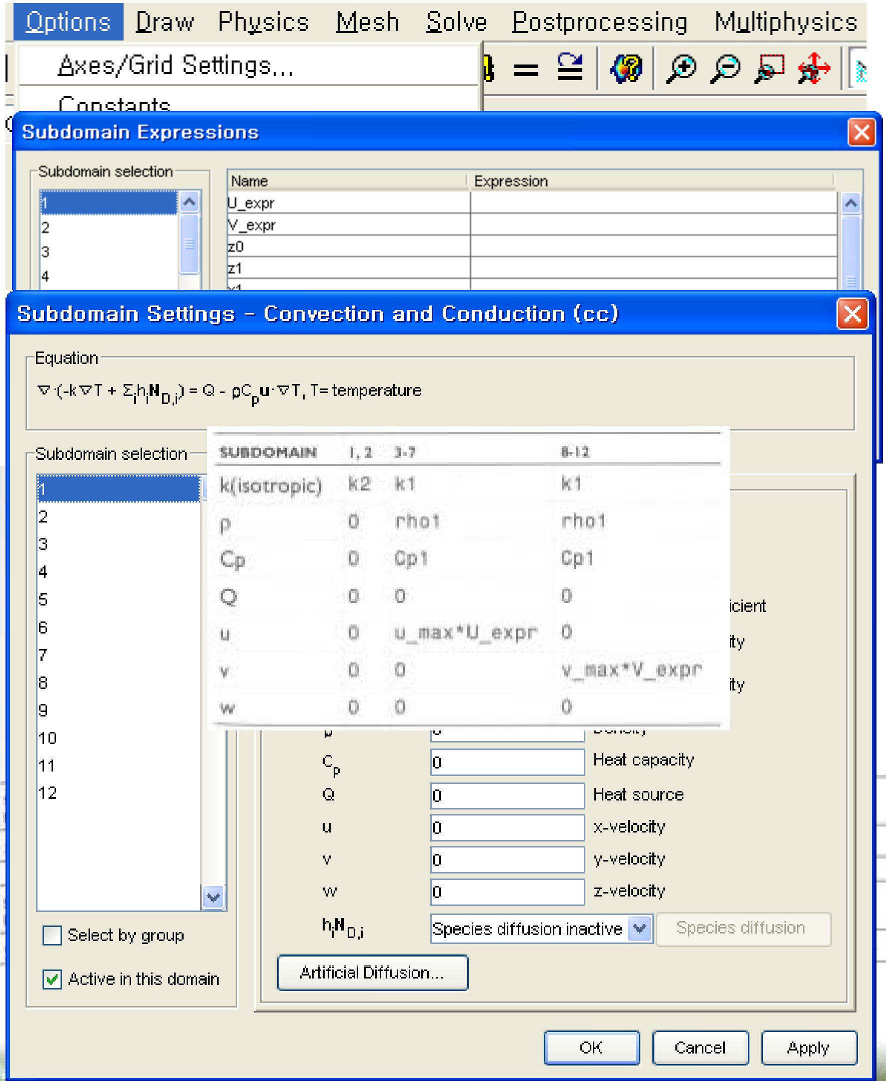
1. Click the **Rectangle/Square** button and click the coordinates (0, 0) and (800, 60).
2. Make another rectangle with corners in (200, 0) and (300, 40).
3. Click the **Array** button in the **Draw** toolbar.
4. Set x displacement to 120 and **Array** size x to 5.
5. Click **OK**.
6. Select all geometry objects by pressing **Ctrl + A**.
7. Open the **Create Composite Object** window.
8. Enter R1+R2+R3+R4+R5+R6 in the **Set formula** edit field.
9. Make sure that the **Keep interior boundaries** check box is selected.
10. Click **OK**.

Geometry modeling



1. From the **Draw** menu, select **Extrude**.
2. Set distance to 800 and click **OK**.
3. Make a copy of the **3D** object by pressing **Ctrl + C**.
4. Paste the copy with **Ctrl + V**.
5. Click the **Rotate** button in the **Draw** toolbar.
6. Set **Rotation angle** to 180.
7. Set **Point** on rotation axis according to : x : 0, y : 0, z : 60
8. Specify the **Rotation axis direction vector** according to : x : 1, y : 1, z : 0
9. Click **OK**.
10. Select all geometry objects by pressing **Ctrl + A**.
11. Click the **Scale** button.
12. Set **Scale factor** to 1e-6 in all directions.
13. Click **OK**.

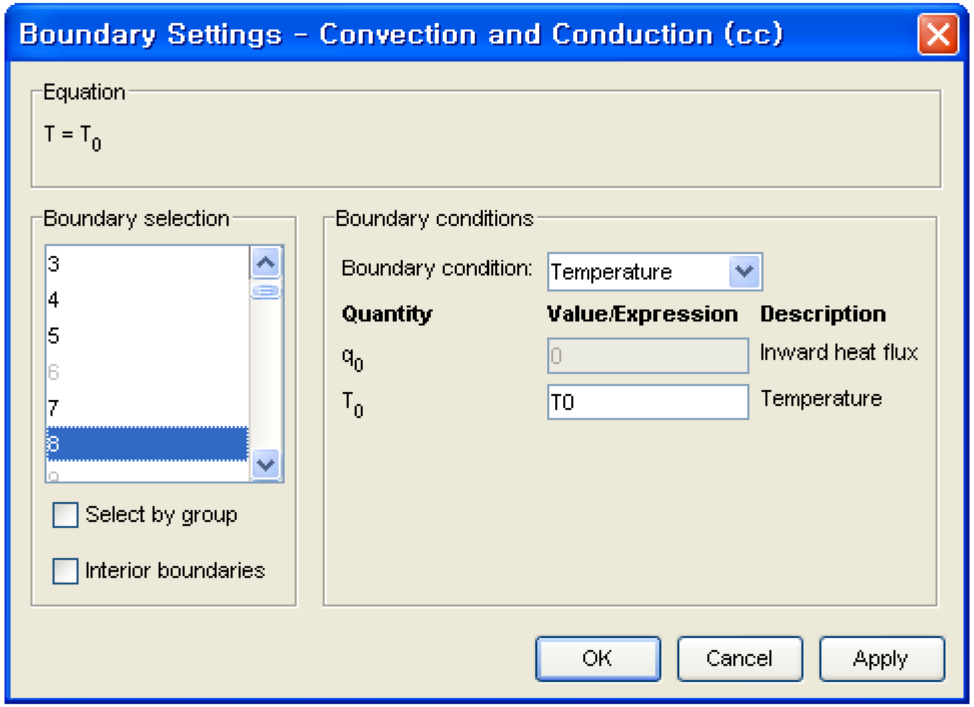
Physics settings (subdomain)



SUBDOMAIN	1, 2	3-7	8-12
k(isotropic)	k2	k1	k1
ρ	0	rho1	rho1
Cp	0	Cp1	Cp1
Q	0	0	0
u	0	u_max*U_expr	0
v	0	0	v_max*V_expr
w	0	0	0

1. Select **Expressions**, **Subdomain Expressions** in the **Options** menu.
2. Specify expression according to the table below.
3. Click **OK**.
4. Select **Subdomain Settings** from the **Physics** menu.
5. Enter subdomain preperties according to the table below.
6. Click **OK**.

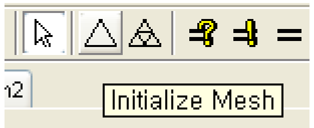
Physics settings (boundary)

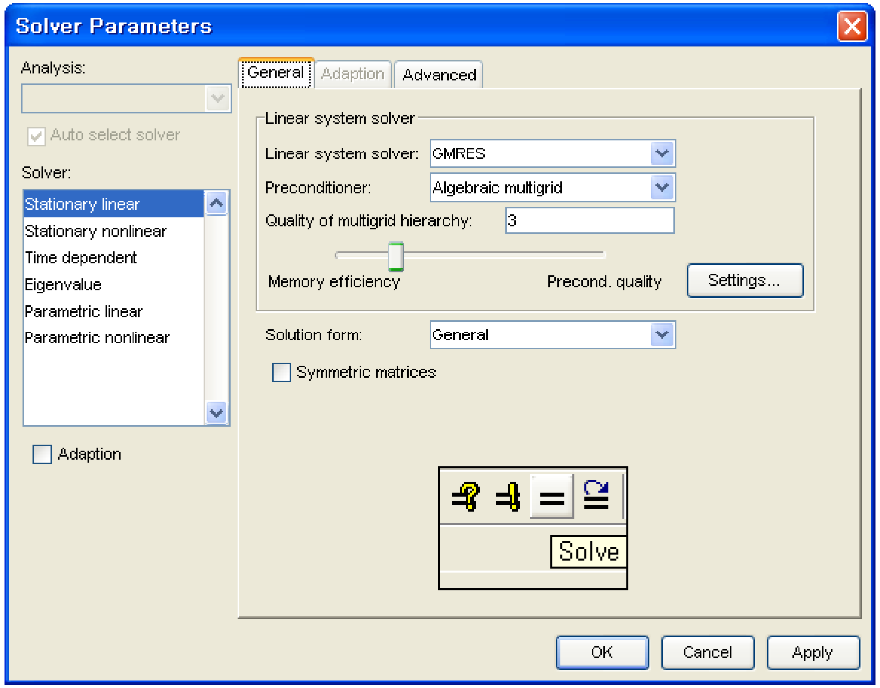


1. Select **Boundary Settings** from the **Physics** menu.
2. Enter boundary conditions according to :
3. Click **OK**.

BOUNDARY	8, 14, 20, 26, 32	41, 48, 55, 62, 69	44, 51, 58, 65, 72, 77, 78, 79, 80, AND 81	ALL OTHER
Type	Temperature	Temperature	Convective flux	Thermal insulation
T	T0	T1	-	-

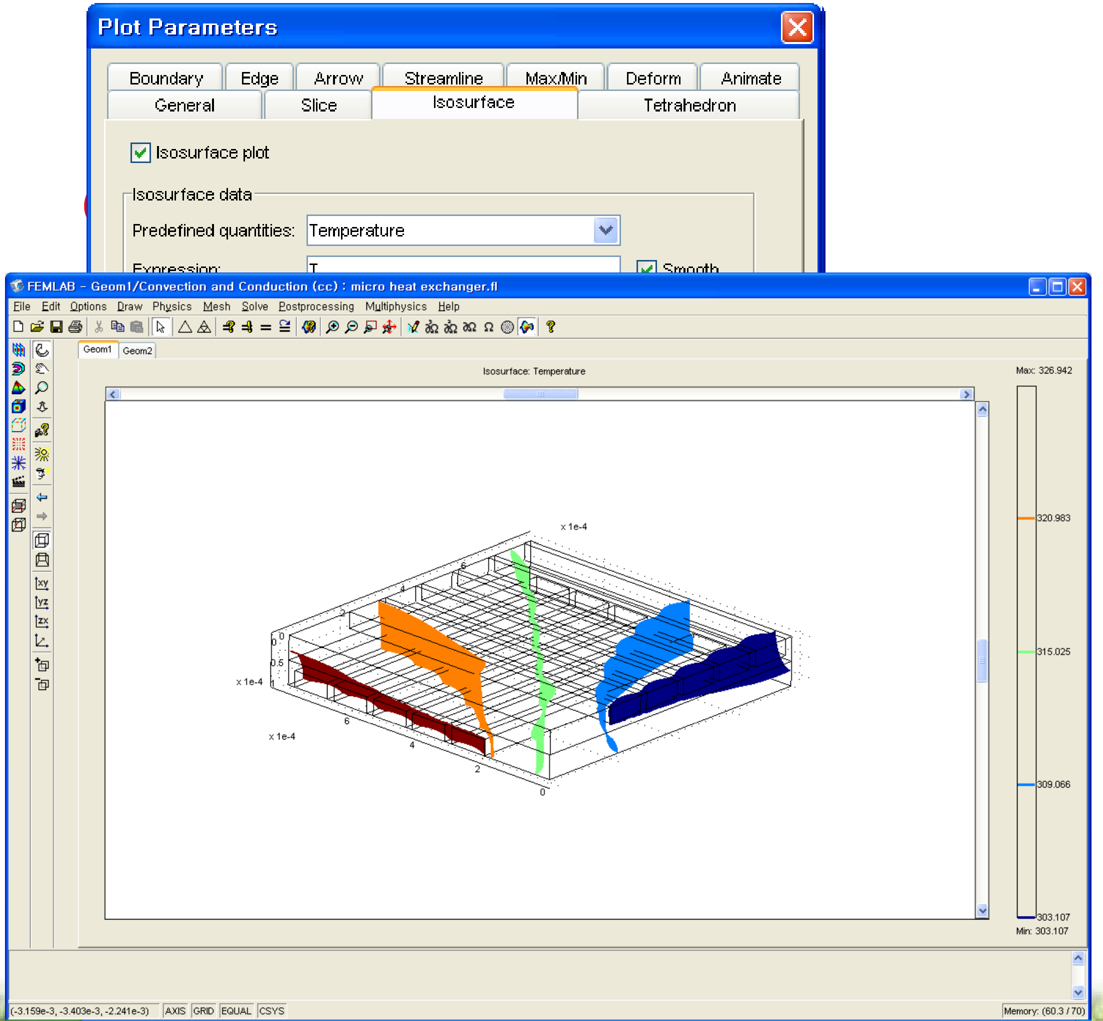
Solving the model





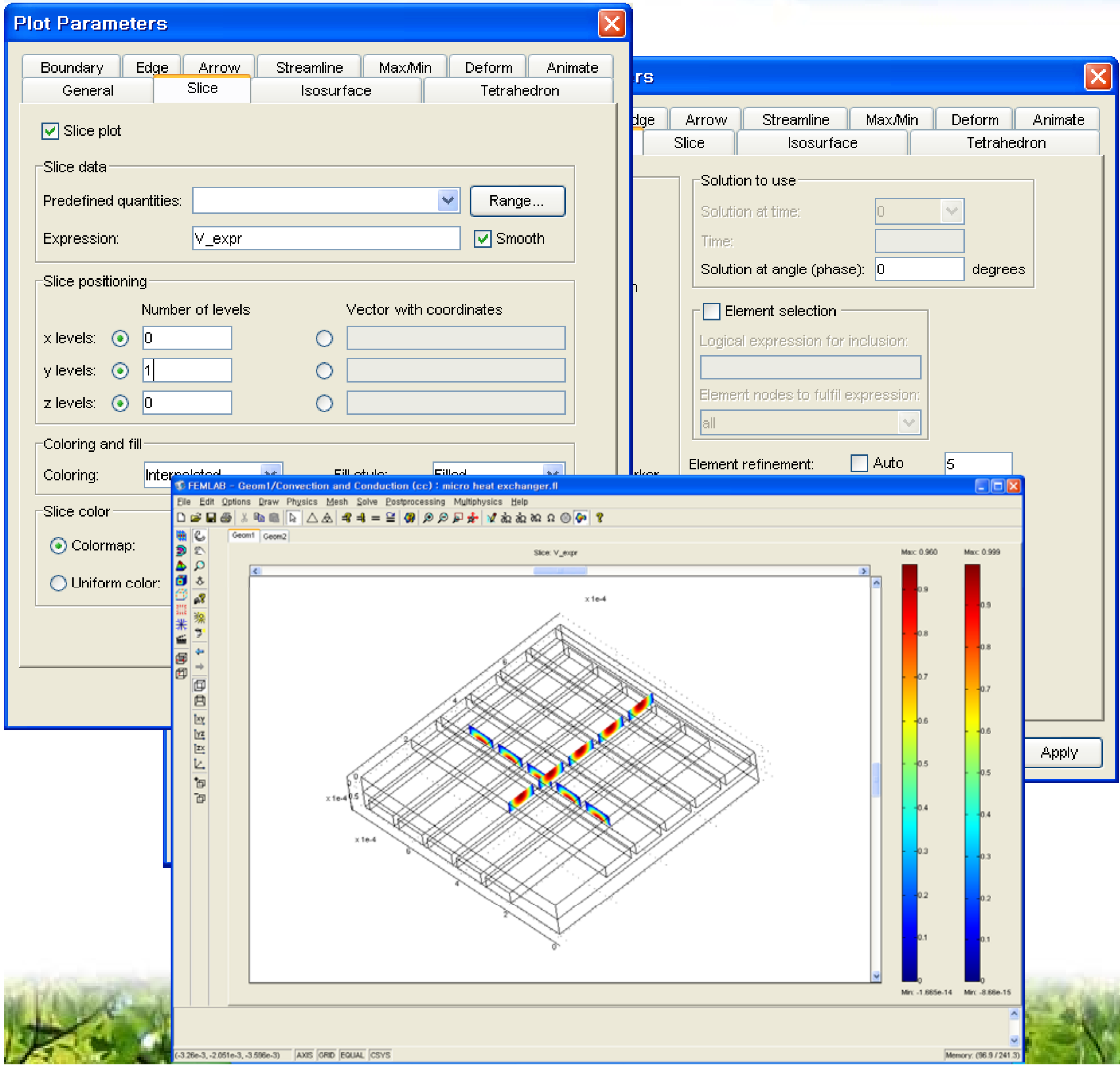
1. Initialize the mesh by pressing the **Initialize Mesh** button in the **Main** toolbar.
2. Select **Solver Parameters** from the **Solve** menu.
3. Select **Stationary linear** from the **Solver** list.
4. In the **General** page, set **Linear system solver** to **GMRES**.
5. Set **Preconditioner** to **Algebraic multigrid**.
6. Click **OK**.
7. Click the **Solve** button in the **Main** toolbar.

Postprocessing and visualization



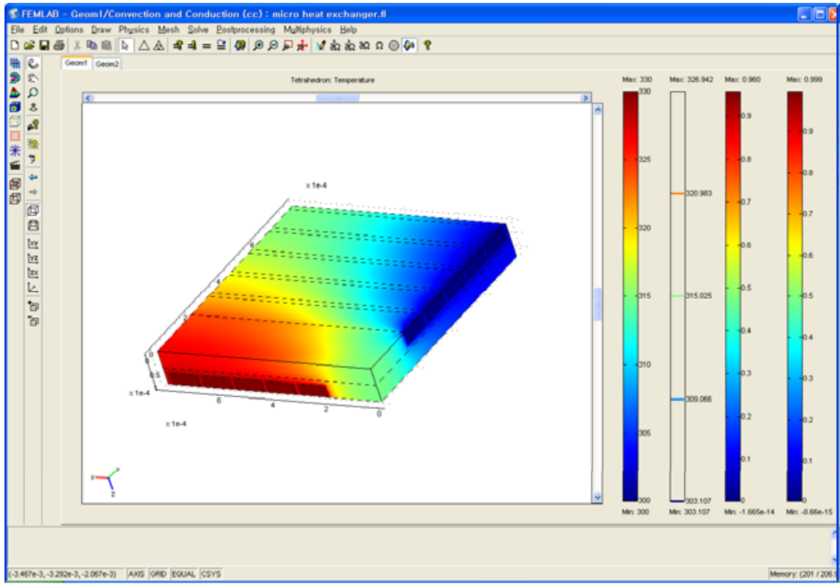
1. Select **Plot Parameters** from the **Postprocessing** menu.
2. Clear the **Slice** plot and select the **Isosurface** plot in the **General** page.
3. Clear the **Auto** option for **Element refinement**.
4. Set **Element refinement** to 5.
5. Go to the **Isosurface** window and set **Predefined quantities** to Temperature.
6. Click **OK**.

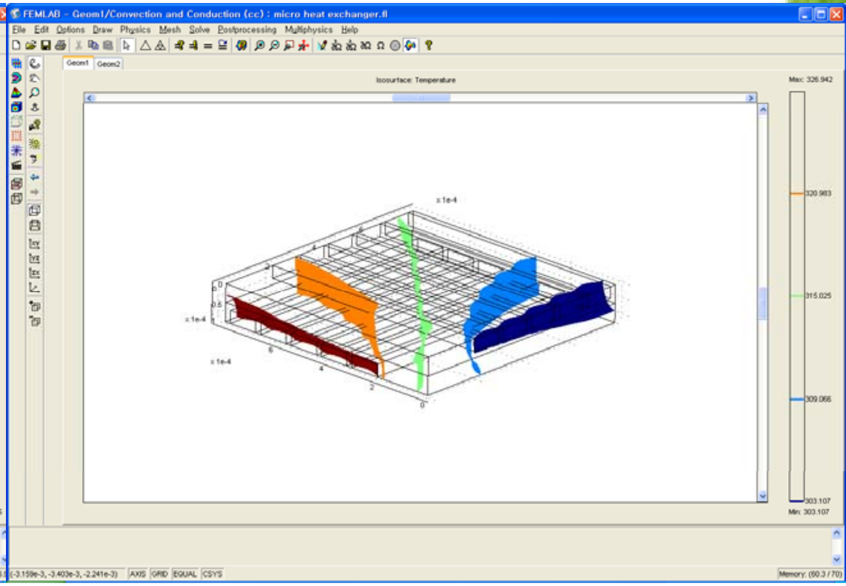
Postprocessing and visualization

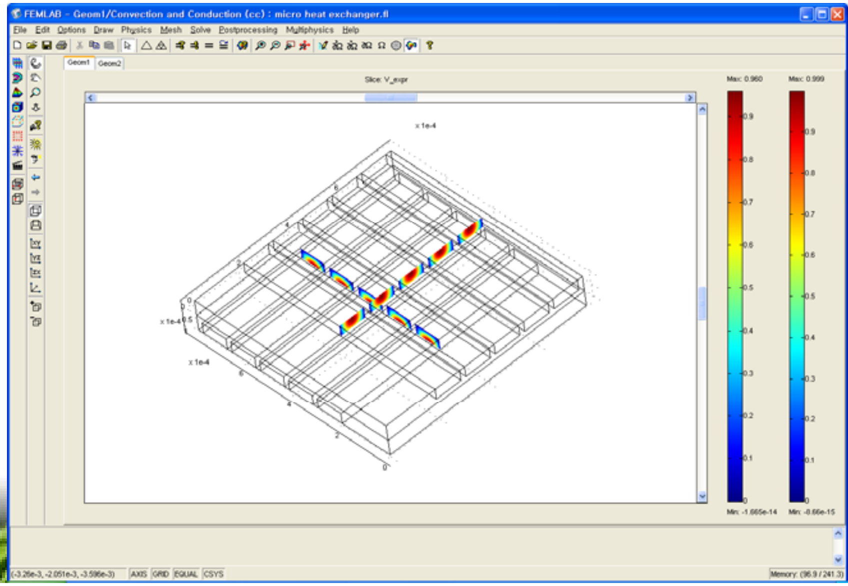


1. Clear the **Isosurface** plot and select the **Slice** plot in the **General** page.
2. Click the **Slice** tab and set **Slice** data expressions to U_expr.
3. Set x-levels to 1 and y-levels and z-levels to 0.
4. Click **Apply**.
5. On the **General** page, check the **Keep current plot** check box.
6. Go to the **Slice** window and set **Slice** data expression to V_expr.
7. Set y-levels to 1 and x-levels and z-levels to 0.
8. Click **OK**.

Results


FEMLAB - Geom1/Convection and Conduction (cc) : micro heat exchanger.fl
Tetrahedron: Temperature


FEMLAB - Geom1/Convection and Conduction (cc) : micro heat exchanger.fl
Isosurface: Temperature


FEMLAB - Geom1/Convection and Conduction (cc) : micro heat exchanger.fl
Slice: V_expr

Conclusions

- The influence of the convective term in the flow channels is clearly seen in the isothermal surfaces.
- We can see the temperature differences in the cold and hot streams at the position of the outlets.