Boundary Layer Flows

Modified by Robert P. Hesketh, Chemical Engineering, Rowan University Fall 2006

Laminar Boundary Layer flow

Boundary layer theory for laminar flow is developed in Geankoplis section 3.10 page 209 and the solution procedure is given in Cutlip and Shacham problem 5.18 page 204. Construct a Comsol simulation to obtain a numerical solution that can be compared with the approximate results presented in the above 2 texts. Use Geankoplis problem 3.10-1 as a basis for this problem. In this problem the plate has a length of 0.305 meters in the x-direction and water is flowing past the plate at 20°C at $\upsilon_\infty = 0.914 \text{ m/s}$. The thickness of the boundary layer is defined as the velocity at $\upsilon \cong 0.99\upsilon_\infty$ The figure below is from Cutlip & Shacham page 204.

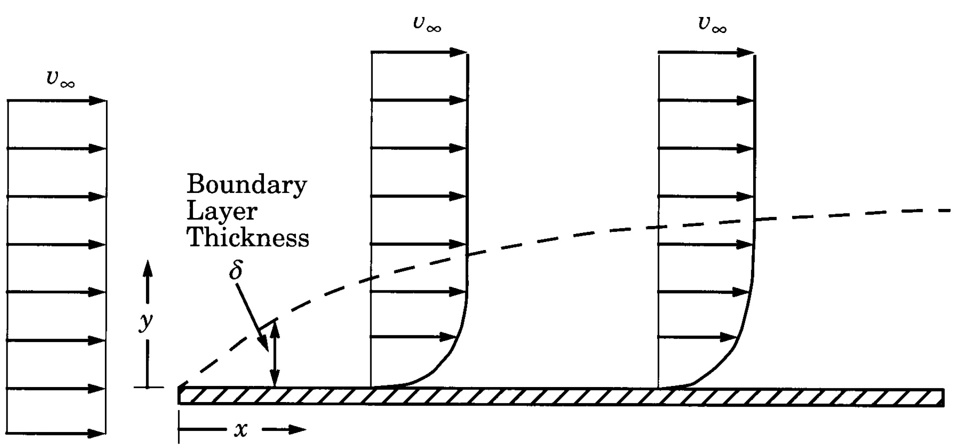


Figure 5–10 Laminar Boundary Layer for Flow Past a Flat Plate

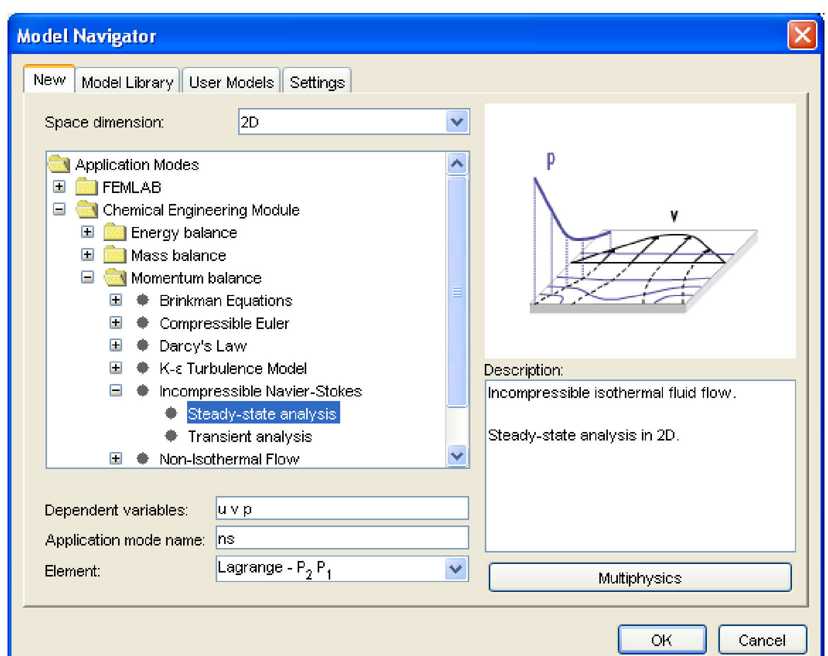
Calculate the boundary layer thickness given as the result of Blasius's approximate solution

$$\delta = \frac{5.0x}{\sqrt{N_{\mathrm{Re},x}}} = 5.0\sqrt{\frac{\mu x}{\rho \upsilon_\infty}}$$

From this result you will see

Modeling using the Graphical User Interface

1. Start COMSOL
2. In the **Model Navigator** , click the **New** page
3. Select **Chemical Engineering Module**, **Momentum Balance**, **Incompressible Navier-Stokes**, Steady-state analysis
4. Click **OK.**



Options and Settings

Define the following constants in the **Constants** dialog box in the **Option** menu. Note that eta is COMSOL's notation for viscosity.

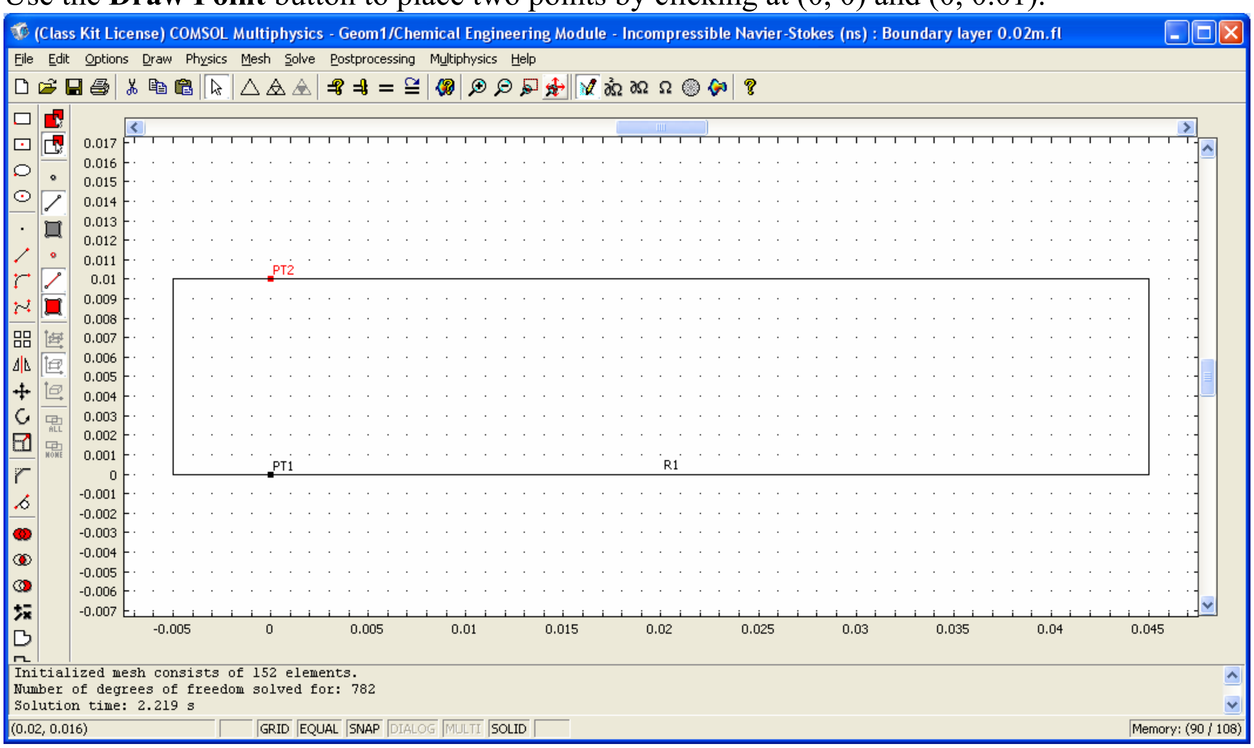
NAME	EXPRESSION
rho	1e3 kg/m^3
eta	1e-3 kg/(m s)
v0	0.914 m/s

Geometry Modeling

1. Press the **Shift** key and click the **Rectangle/Square** button.
2. Type in values that will give the horizontal dimension of 0.05 m and the vertical dimension of 0.01m. Have the rectangle start at an x value of -0.005. We will use the section form x=-0.005 to x=0 to obtain a uniform velocity profile before encountering the horizontal plate.

OBJECT DIMENSIONS	EXPRESSION
Width	0.05
Height	0.01
x-position	-0.005
y-position	0

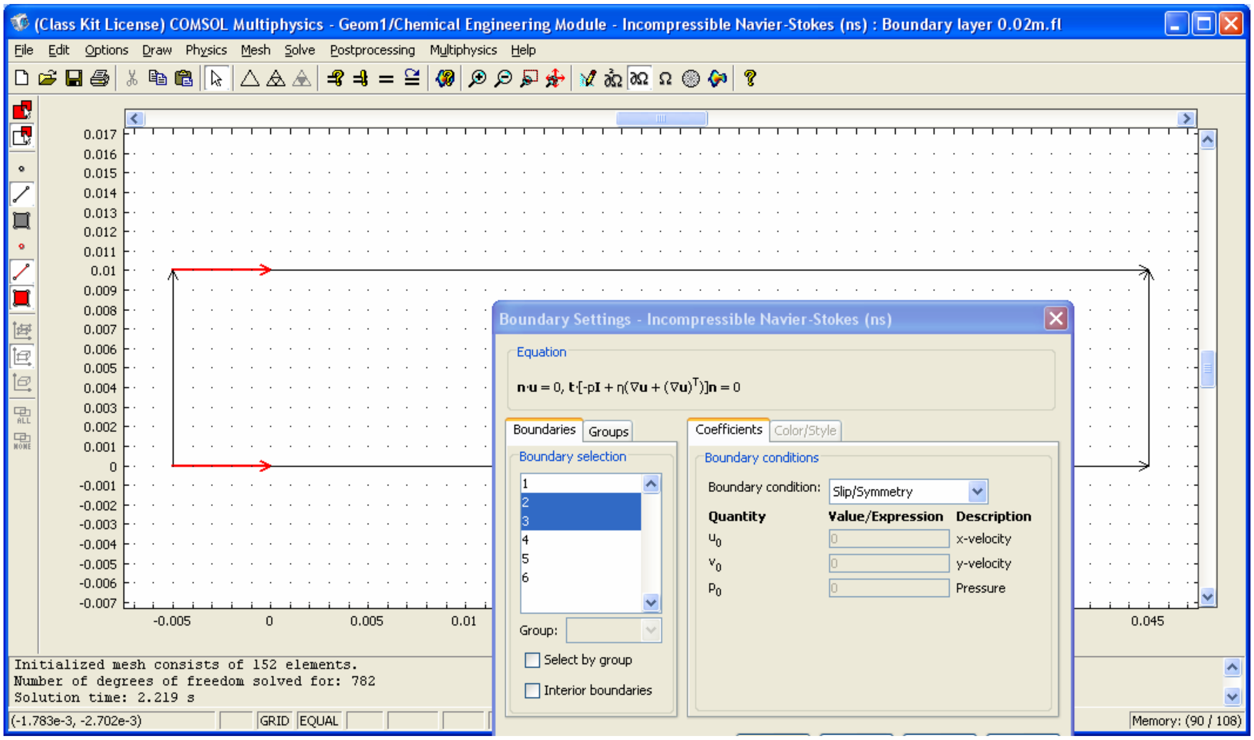
3. Click the **Zoom Extents** button in the Main toolbar.
4. Use the **Draw Point** button to place two points by clicking at (0, 0) and (0, 0.01).



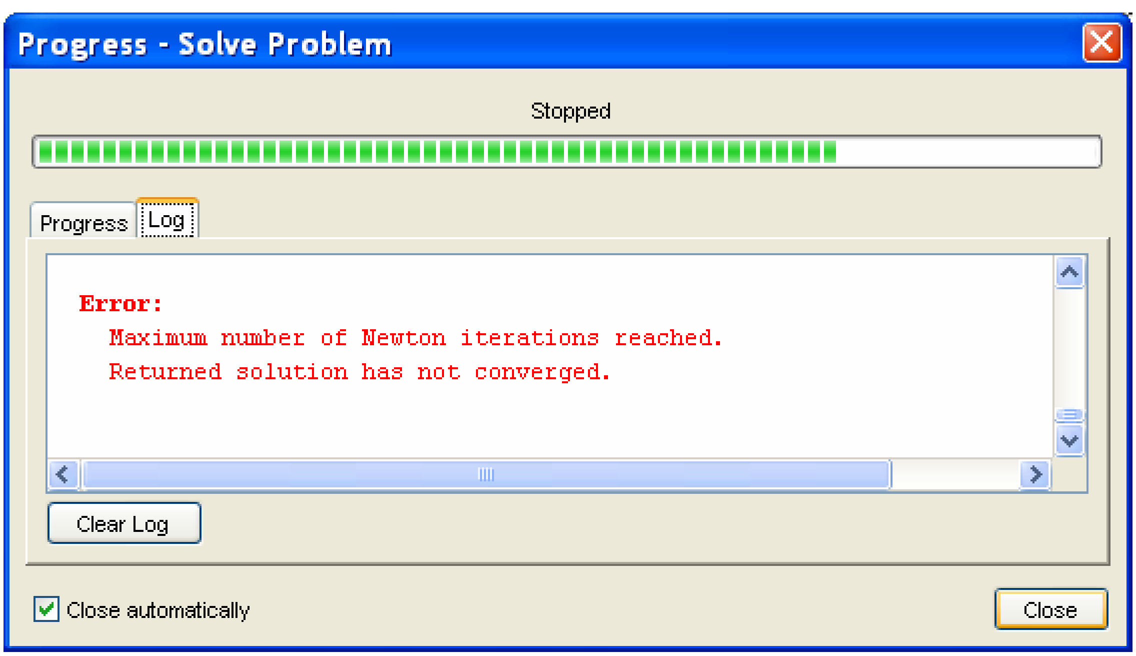
5. Now specify the sub-domain settings using your constants defined above.

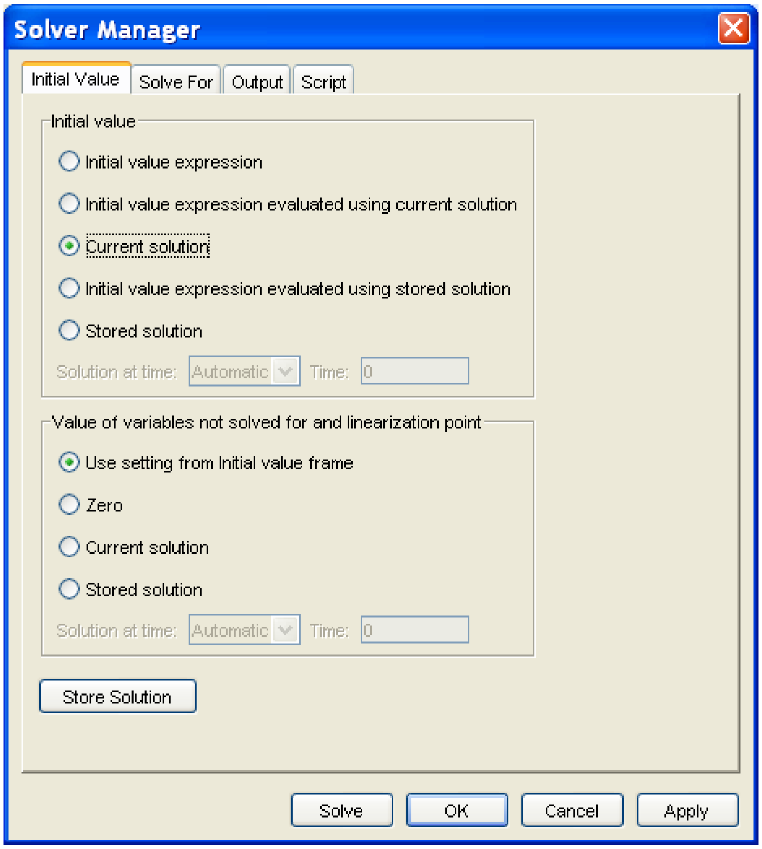
.

6. Next specify the boundary conditions. For the boundary defined by $-0.005 \le x \le 0$ specify a slip boundary condition as shown in the figure below:



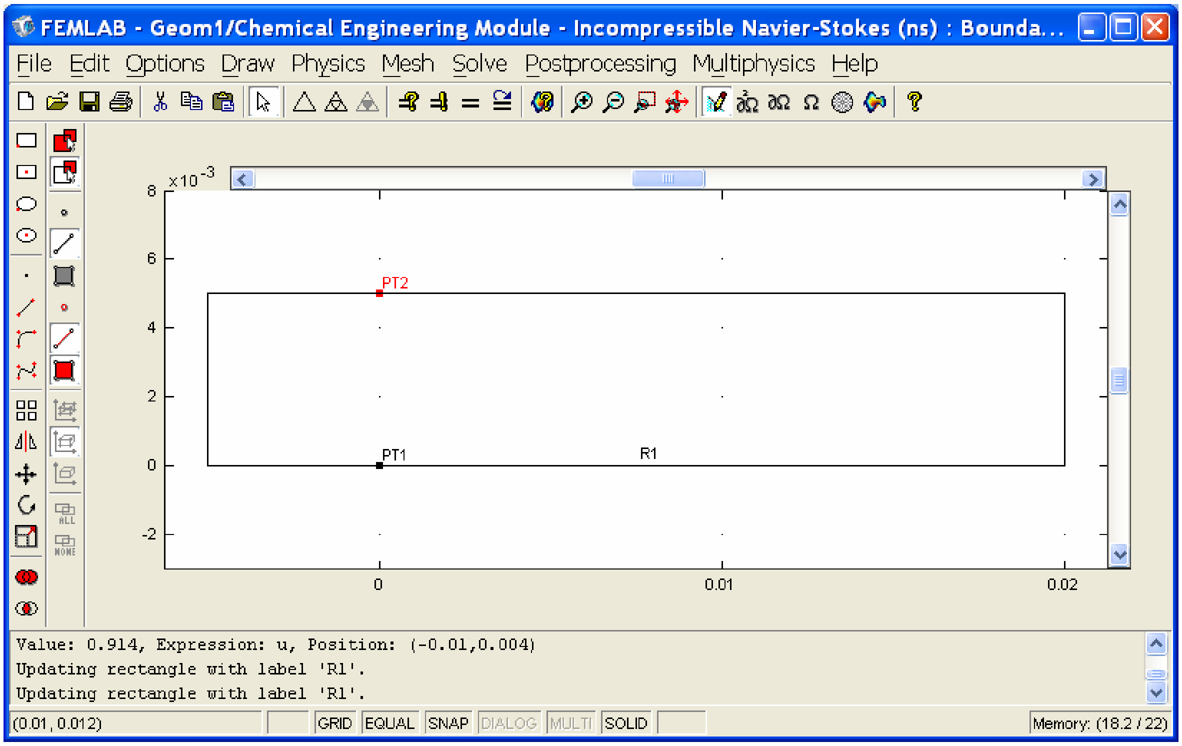
7. Specify an inflow velocity and use the outflow pressure term. For the plate specify no-slip and for far away from the plate we will assume slip.
8. Now solve this model after creating a grid. If you get the below error message then restart the problem from the current solution using Solver



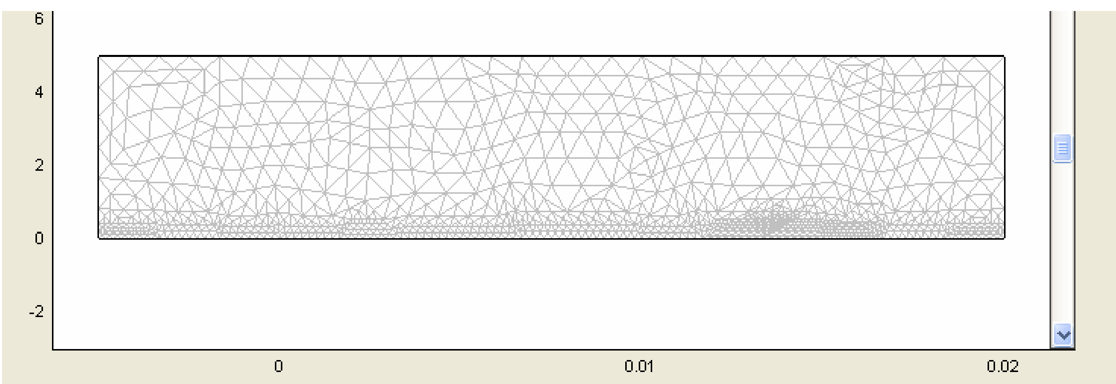


9. Look and see where the change in velocity is present!
10. Now make the rectangle smaller and move PT2 to the border using Draw, Object Properties:

OBJECT DIMENSIONS	EXPRESSION
Width	0.025
Height	0.005
x-position	-0.005
y-position	0

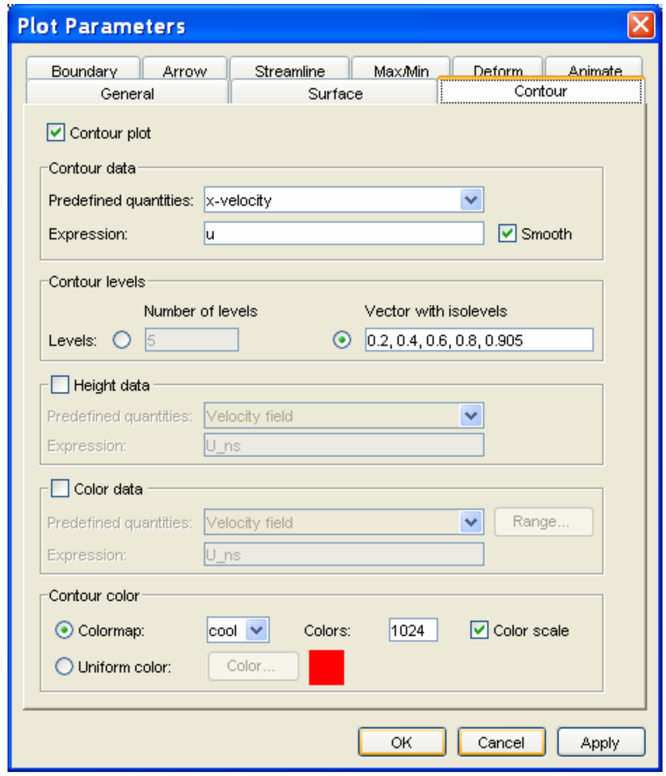


11. Calculate the boundary layer thickness $\delta = \frac{5.0x}{\sqrt{N_{\mathrm{Re},x}}} = 5.0\sqrt{\frac{\mu x}{\rho \upsilon_\infty}}$ where x = 0.02 m. Then increase the griding within the boundary layer region.



12. Solve and make set your values for the contours such that one of the values corresponds to the boundary layer thickness definition.
13. Make a cross-section plot at the x=0.02 m showing the velocity as a function of distance from the wall.
14. Add arrows to your contour plot showing the magnitude and direction of the velocity vectors.

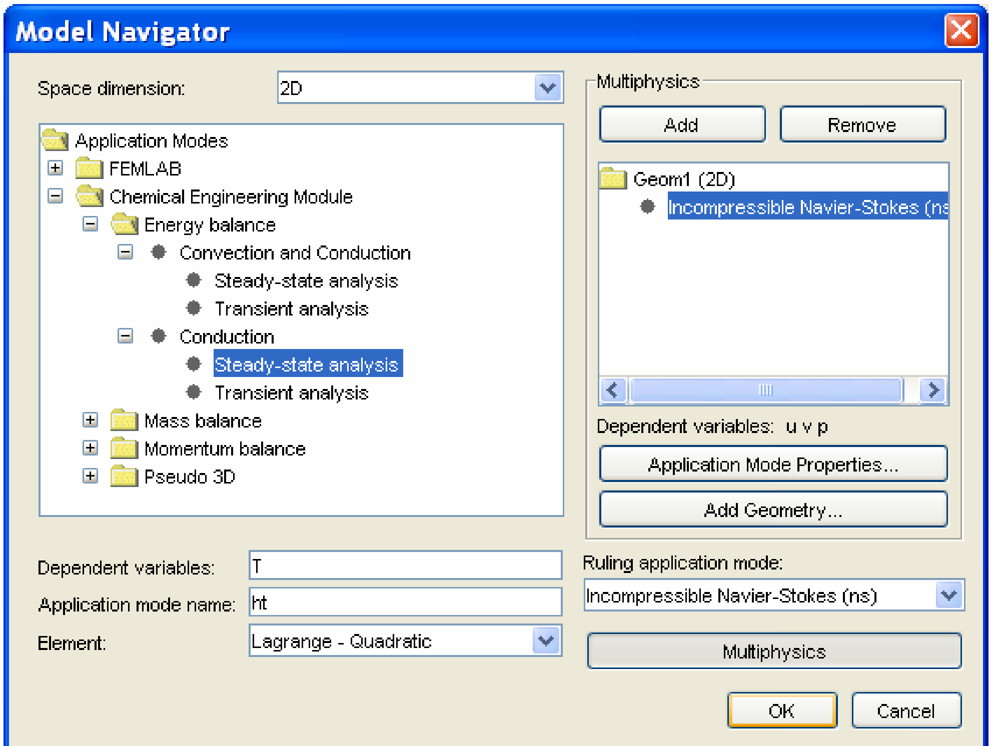
15. Make a contour plot showing (default solution) and add arrows to your plot showing the magnitude and direction of the velocity vectors. (Go to Plot Parameters and select the Arrow tab). Next make a cross-section plot of the velocity. Using excel compare the COMSOL and analytical results by plotting them on the same graph.

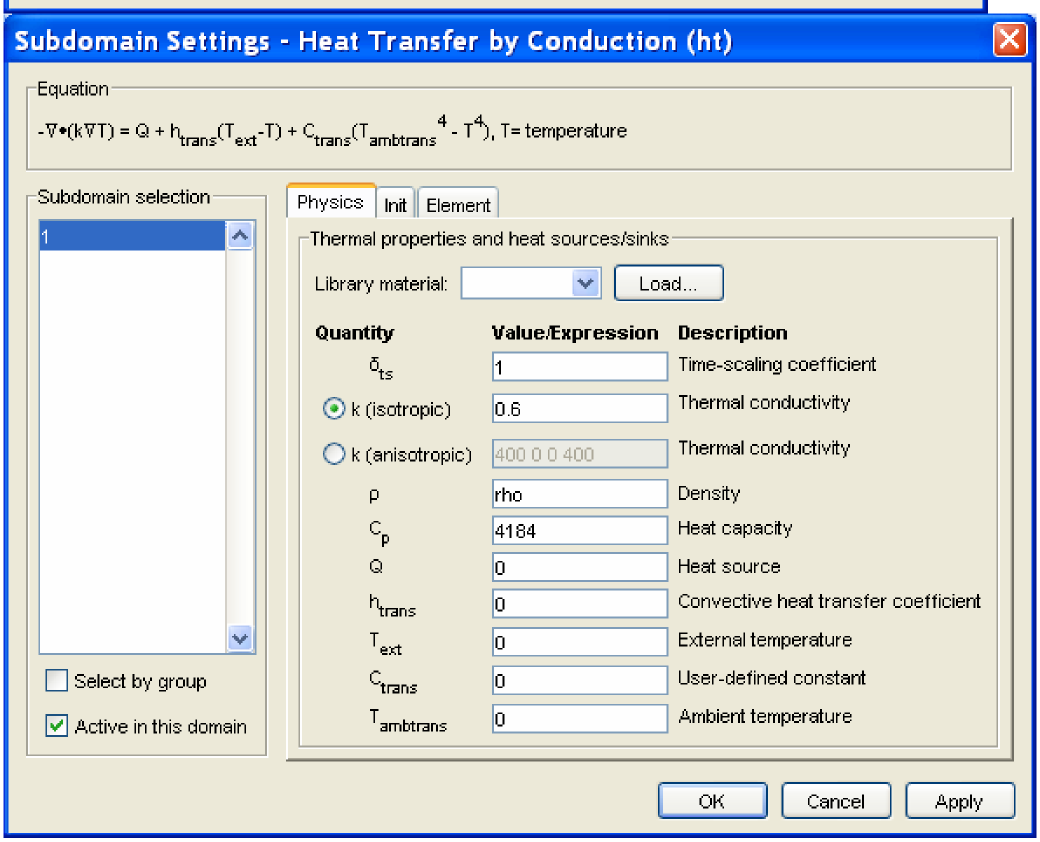


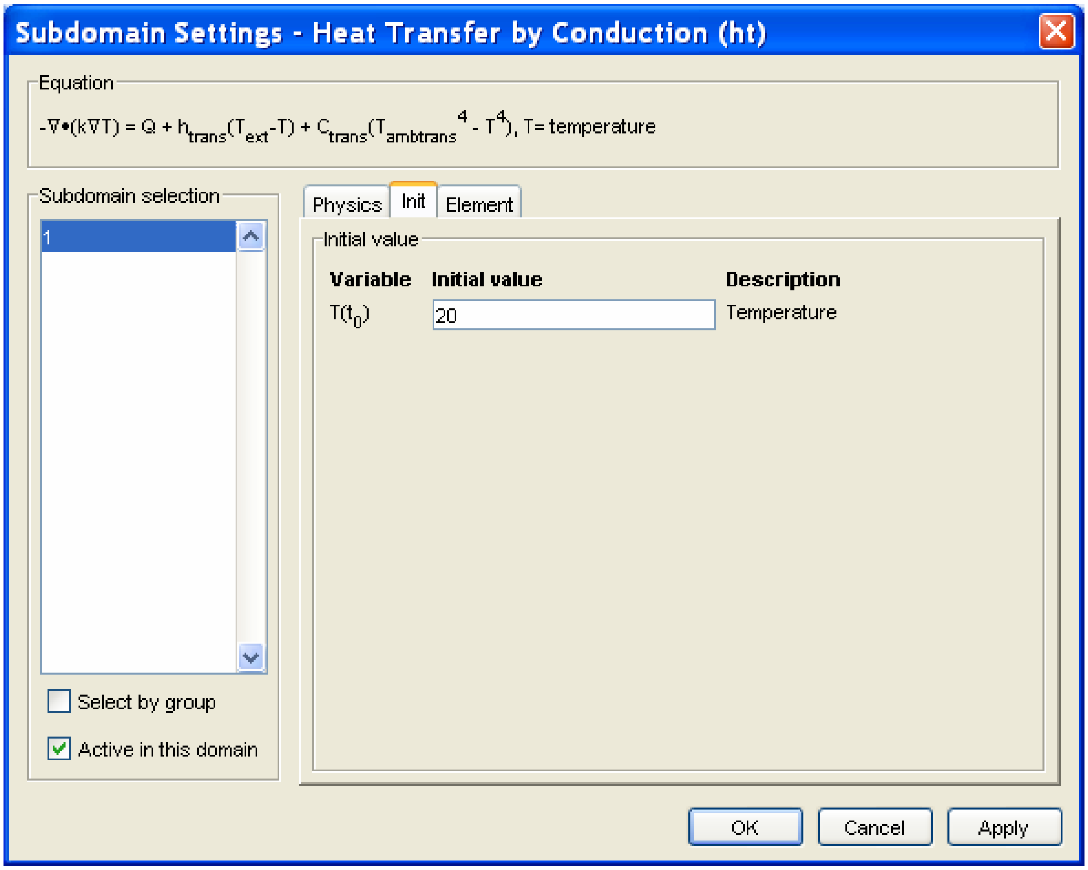
Thermal Boundary Layer

Now add an energy balance.

Now look at the temperature boundary layer by specifying a temperature of the plate as 40°C and the water temperature of 20°C. The thermal conductivity of water can be approximated at 0.6 W/(m K) and the heat capacity is 4,184 J/kg K.


Model Navigator
Space dimension:
2D
Application Modes
FEMLAB
Chemical Engineering Module
Energy balance
Convection and Conduction
Steady-state analysis
Transient analysis
Conduction
Steady-state analysis
Transient analysis
Mass balance
Momentum balance
Pseudo 3D
Dependent variables:
T
Application mode name:
ht
Element:
Lagrange - Quadratic
Multiphysics
Add
Remove
Geom1 (2D)
Incompressible Navier-Stokes (ns
Dependent variables: u v p
Application Mode Properties...
Add Geometry...
Ruling application mode:
Incompressible Navier-Stokes (ns)
Multiphysics
OK
Cancel


Subdomain Settings - Heat Transfer by Conduction (ht)
Equation
-∇•(k∇T) = Q + h_trans(T_ext-T) + C_trans(T_ambtrans^4 - T^4), T= temperature
Subdomain selection
1
Select by group
Active in this domain
Physics
Init
Element
Thermal properties and heat sources/sinks
Library material:
Load...
Quantity
Value/Expression
Description
δ_ts
1
Time-scaling coefficient
k (isotropic)
0.6
Thermal conductivity
k (anisotropic)
400 0 0 400
Thermal conductivity
ρ
rho
Density
C_p
4184
Heat capacity
Q
0
Heat source
h_trans
0
Convective heat transfer coefficient
T_ext
0
External temperature
C_trans
0
User-defined constant
T_ambtrans
0
Ambient temperature
OK
Cancel
Apply



Submit:

1. **Boundary Layer thickness calculation**
2. Contour Plots (Basic Solution) Add arrows on top of plot to signify velocity
    2.1. large rectangle
    2.2. small rectangle
3. **Cross Section Plots:**
    3.1. large rectangle
    3.2. small rectangle