Femlab tutorial

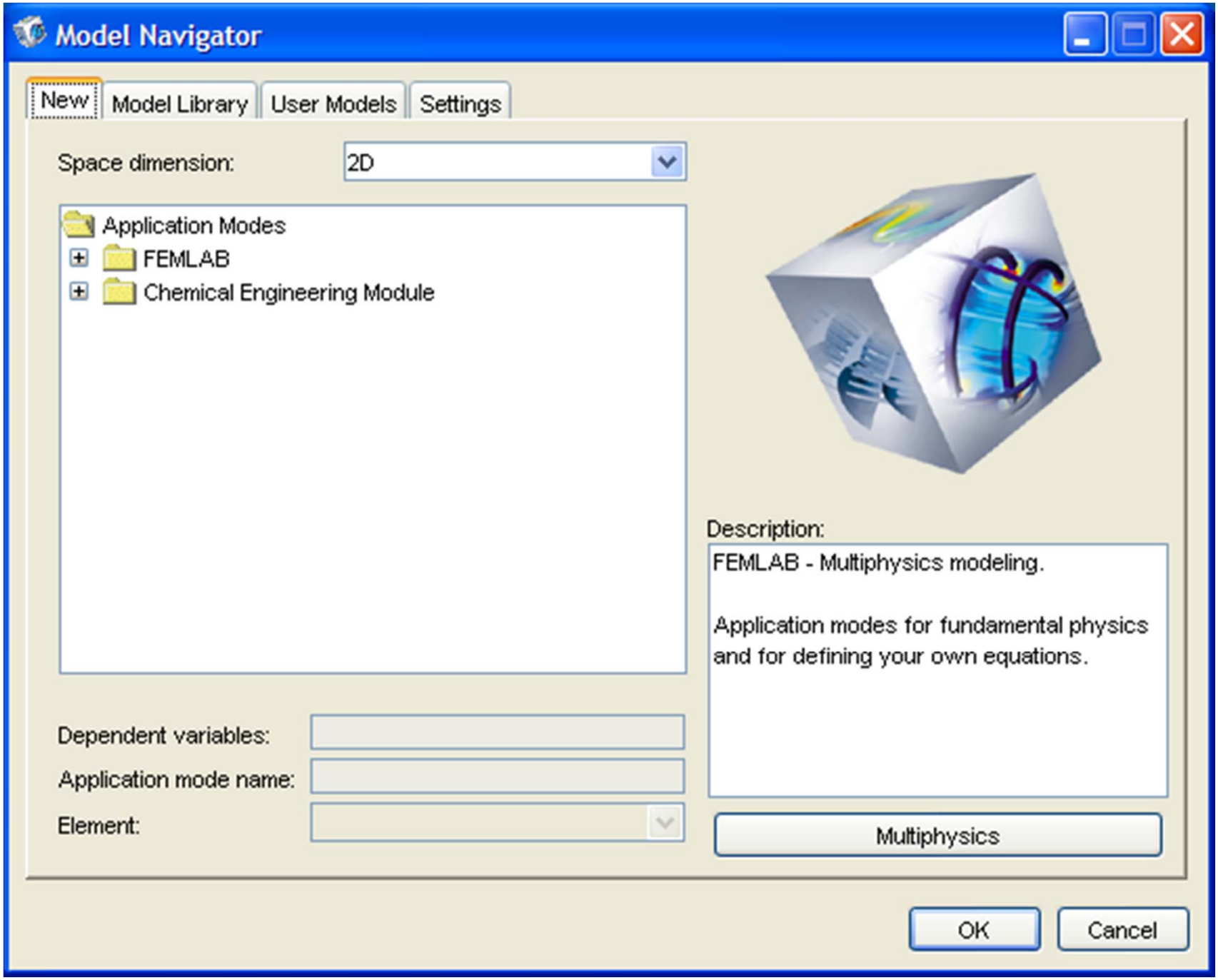
STARTING UP FEMLAB (ON A WINDOWS MACHINE IN E-CALC) AND SELECTING THE MODELS

- The software is installed locally in C:/FEMLAB31 with the path to license file already set-up.
- To start the program

  Start → Run → C:/FEMLAB31/bin/femlab

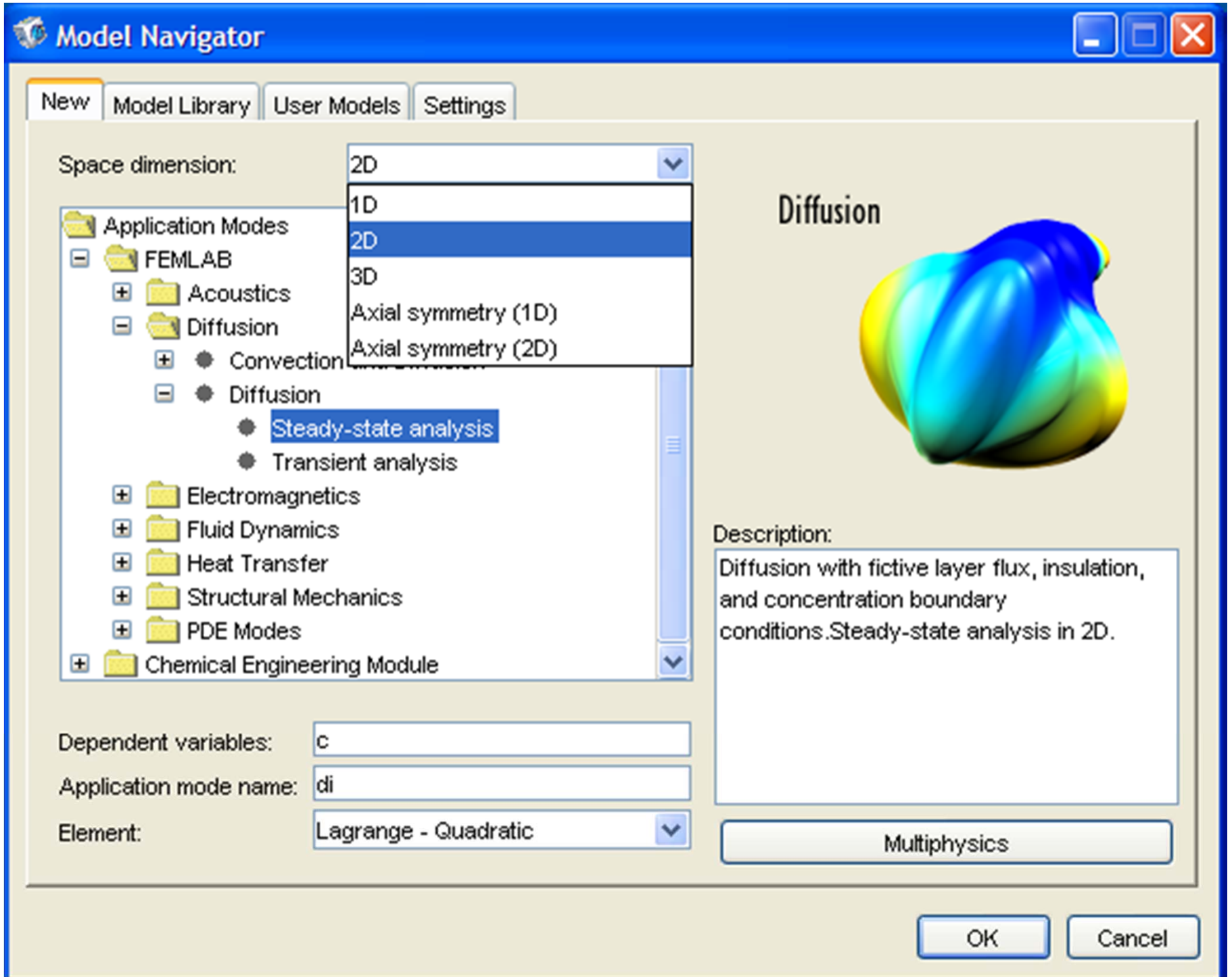
  (note the upper and lower cases)
- A GUI (graphical user interface) called as the "Model Navigator" will show up. This is the place where you select the modeling equation.

- Femlab has options to choose the governing equations from
  - Acoustics
  - Diffusion (with or without convection)
  - Electromagnetics (electrostatics, magnetics)
  - Fluid dynamics (Navier-Stokes eqn)
  - Heat transfer (conduction; with or without convection)
  - Structural mechanics (stress, strain)
- PDE modes (Classical equations like convection-diffusion, heat conduction, Helmholtz, Laplace, Poisson, Schrödinger, wave equations as well as a generalized partial differential equation in weak / coefficient / general form

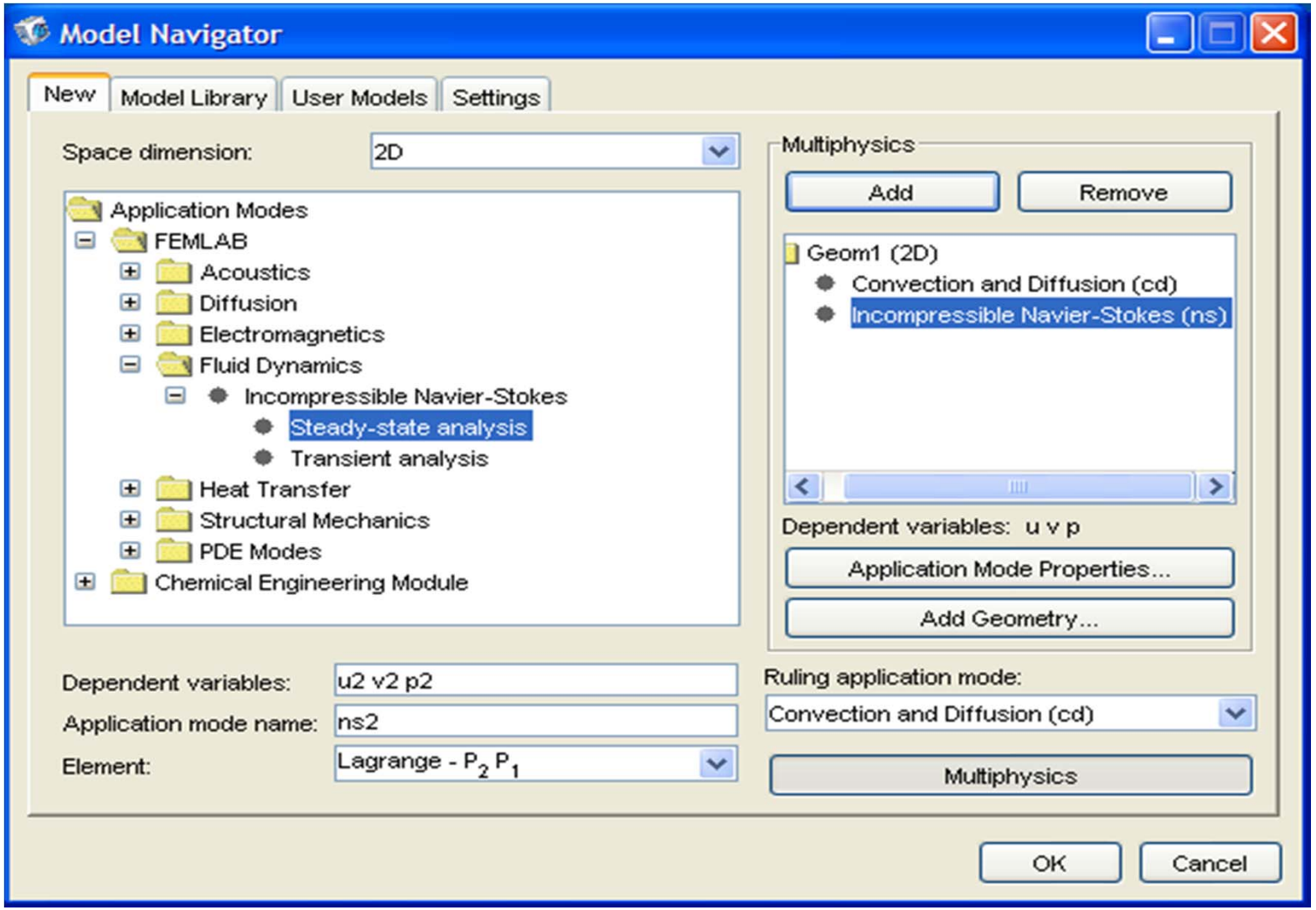

Model Navigator
New
Model Library
User Models
Settings
Space dimension:
2D
Application Modes
FEMLAB
Chemical Engineering Module
Description:
FEMLAB - Multiphysics modeling.
Application modes for fundamental physics and for defining your own equations.
Dependent variables:
Application mode name:
Element:
Multiphysics
OK
Cancel

Chemical Engineering Module

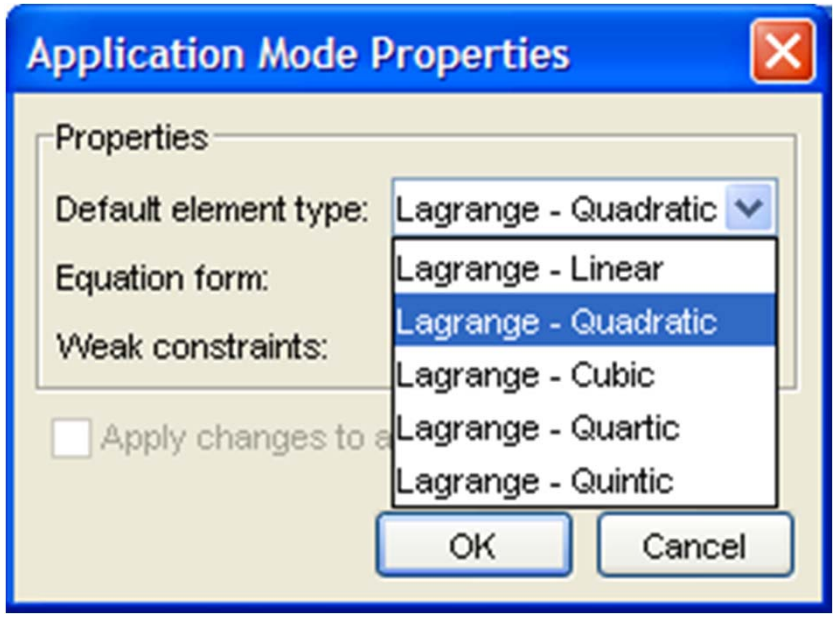
- The ‘Chemical Engineering Module’ comprises of special models built in for chemical engineering applications based on the equations mentioned above.
- The description of each of the equations can be seen in the dialog window on the bottom right corner.
- Once an equation is selected – the dependent variables are listed at the bottom of the screen. One can define here if a 1D / 2D / 3D version of the equation is to be used


Model Navigator
New
Model Library
User Models
Settings
Space dimension:
2D
1D
2D
3D
Axial symmetry (1D)
Axial symmetry (2D)
Application Modes
FEMLAB
Acoustics
Diffusion
Convection
Diffusion
Steady-state analysis
Transient analysis
Electromagnetics
Fluid Dynamics
Heat Transfer
Structural Mechanics
PDE Modes
Chemical Engineering Module
Diffusion
Description:
Diffusion with fictive layer flux, insulation, and concentration boundary conditions.Steady-state analysis in 2D.
Dependent variables:
c
Application mode name:
di
Element:
Lagrange - Quadratic
Multiphysics
OK
Cancel

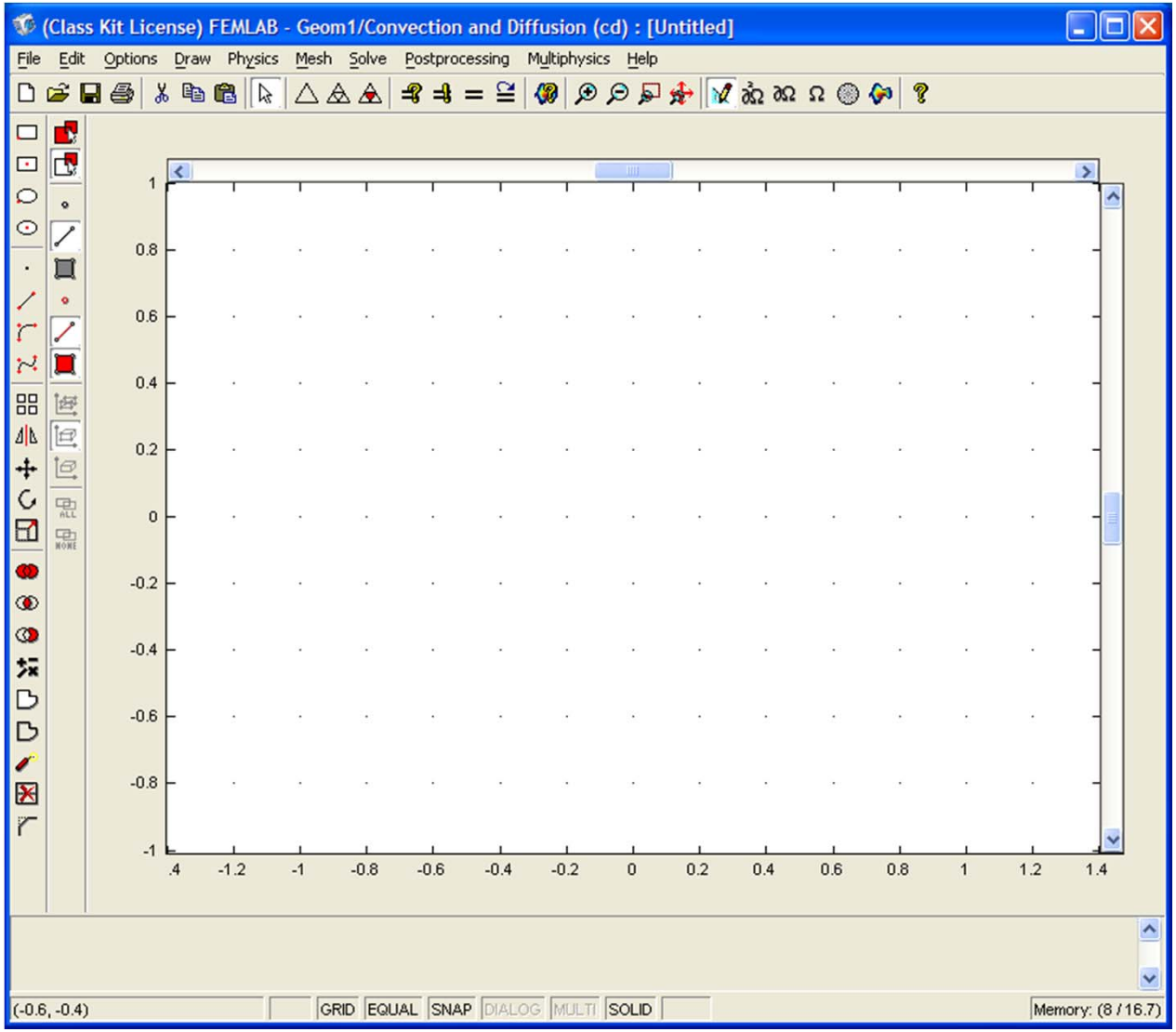
If multiple governing equations are to used one can click on the “Multiphysics” tab and then add multiple models as shown here



Femlab is a finite element based solver.
The type of elements can be changed by clicking on
the “Application Mode Properties” tab. usually;
the default selection is a quadratic element.
The higher order of the element can be used to improve the accuracy
of the solution but it should be noted that the computational cost/
i.e. cpu time may increase.

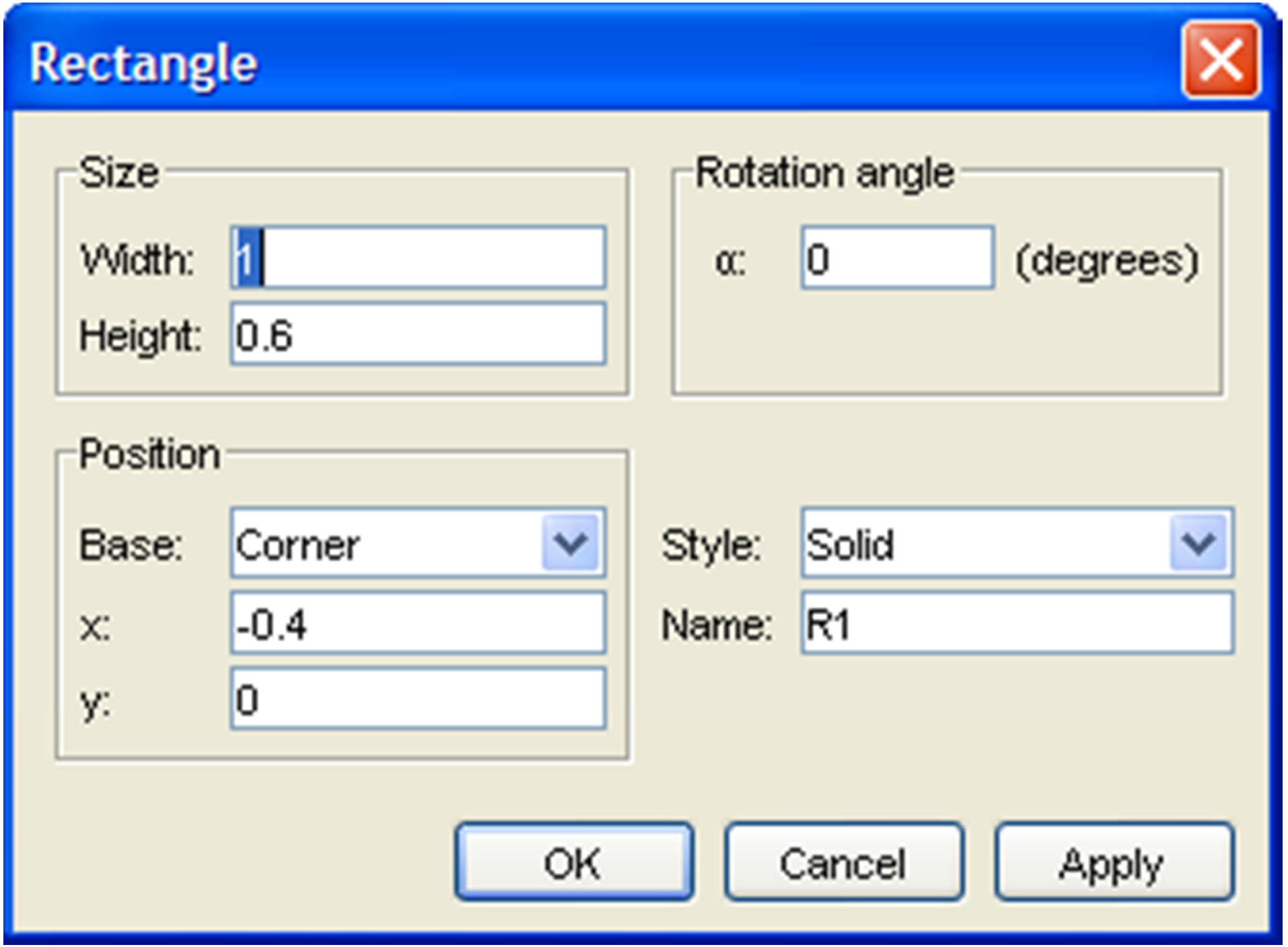


Having completed the model selection. Click “OK”. This will bring up



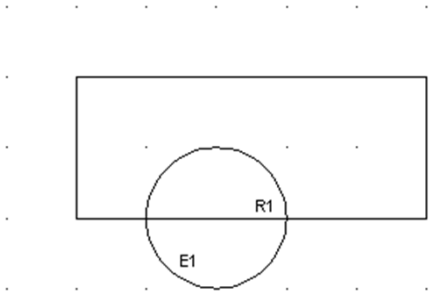
CREATING THE GEOMETRY, MESH AND DEFINING THE BOUNDARY CONDTIONS

- The most commonly used geometrical features are shown on the toolbar to the left of the drawing board. All these are also available for selection in the “Draw” menu.
- The size and location of an object can either be specified through the window or one could draw the object of desired size in on the screen.

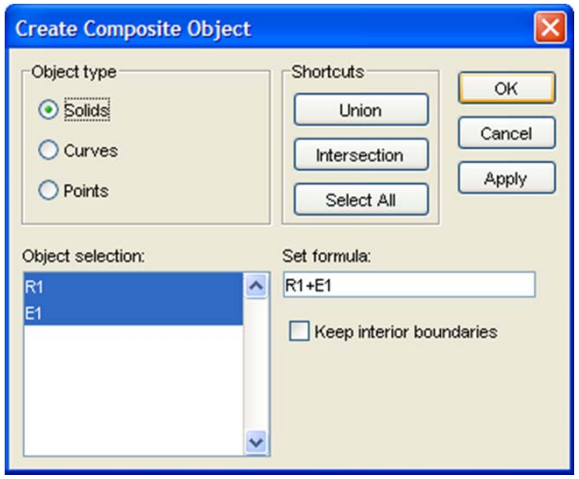

Rectangle
Size
Width:
1
Height:
0.6
Rotation angle
α:
0
(degrees)
Position
Base:
Corner
x:
-0.4
y:
0
Style:
Solid
Name:
R1
OK
Cancel
Apply

•Boolean operations can be performed to create complex objects as well.

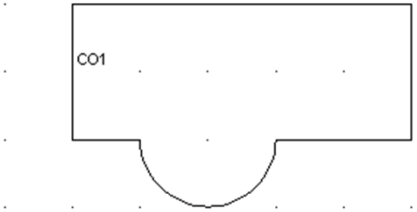
a. Draw multiple objects



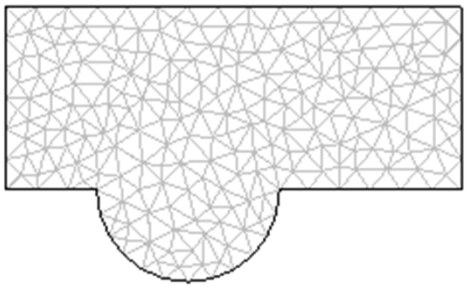
b. Specify the Boolean operations to be performed



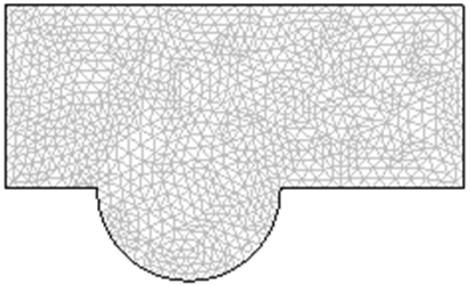
c. The result is a complex geometry



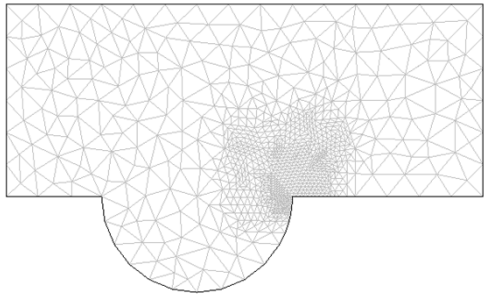
- The next step is to initialize the mesh using the option in the "Mesh" menu. For advanced meshing – parameters can be tuned using the Mesh → Mesh Parameters option. Usually the default ones are fine and additional refining of the mesh can be done for the entire geometry or a desired region



Initial mesh



Refining entire domain



Refining a region

TEST EXAMPLES (Flow in parallel plate or pipe)

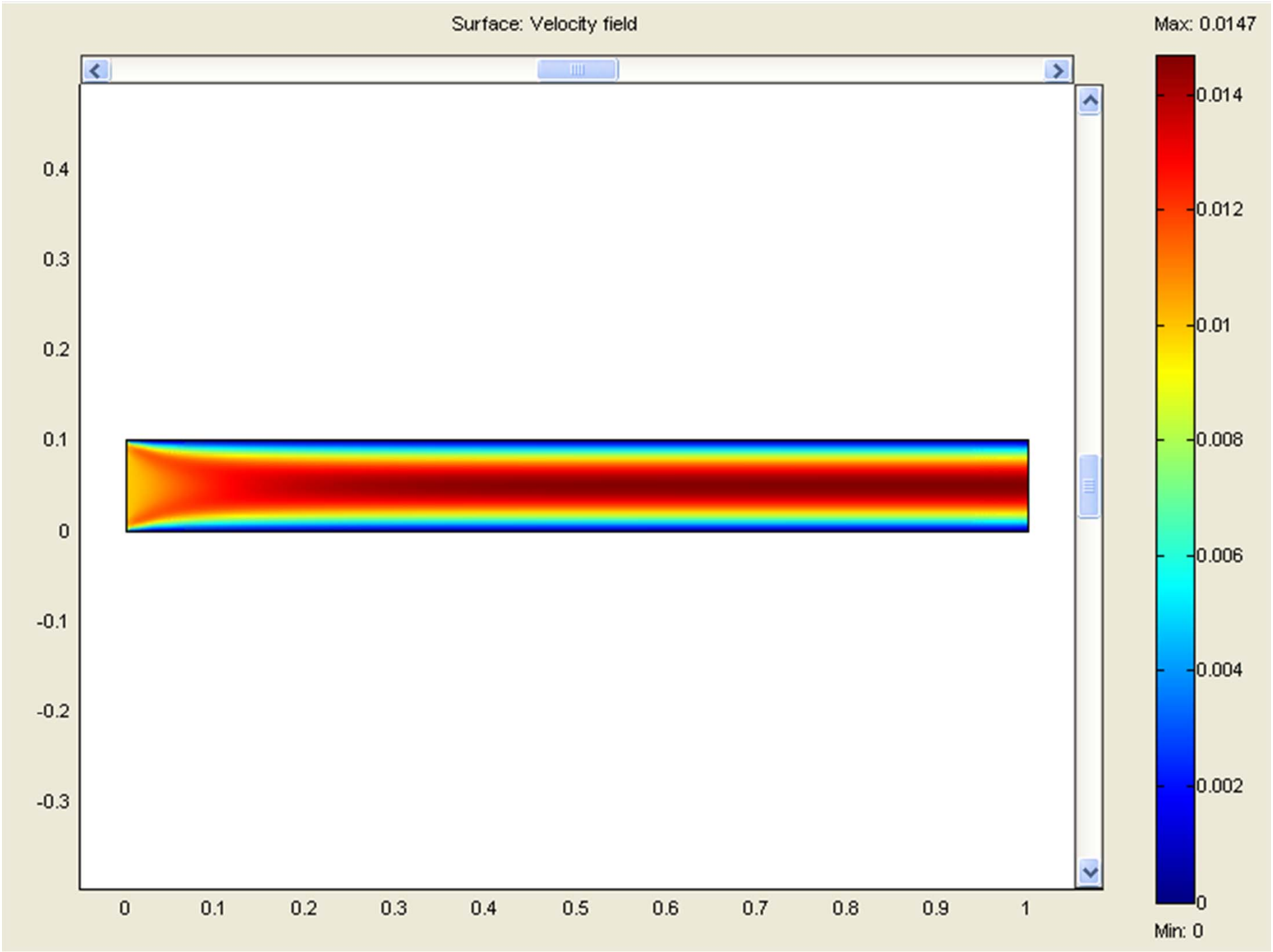
- Drawing a rectangle of size (1x0.1) implies a parallel plate channel of length 1 m and diameter 10 cm.

- For the boundary conditions (Physics → Boundary Settings):

- U0 = inlet velocity = 0.01 m/s is set on boundary 1.
- Boundary 2 and 3 have no slip boundary condition specified
- Boundary 4 is the outflow condition

- For other parameters of the Navier-Stokes equation (Physics → Subdomain Settings):

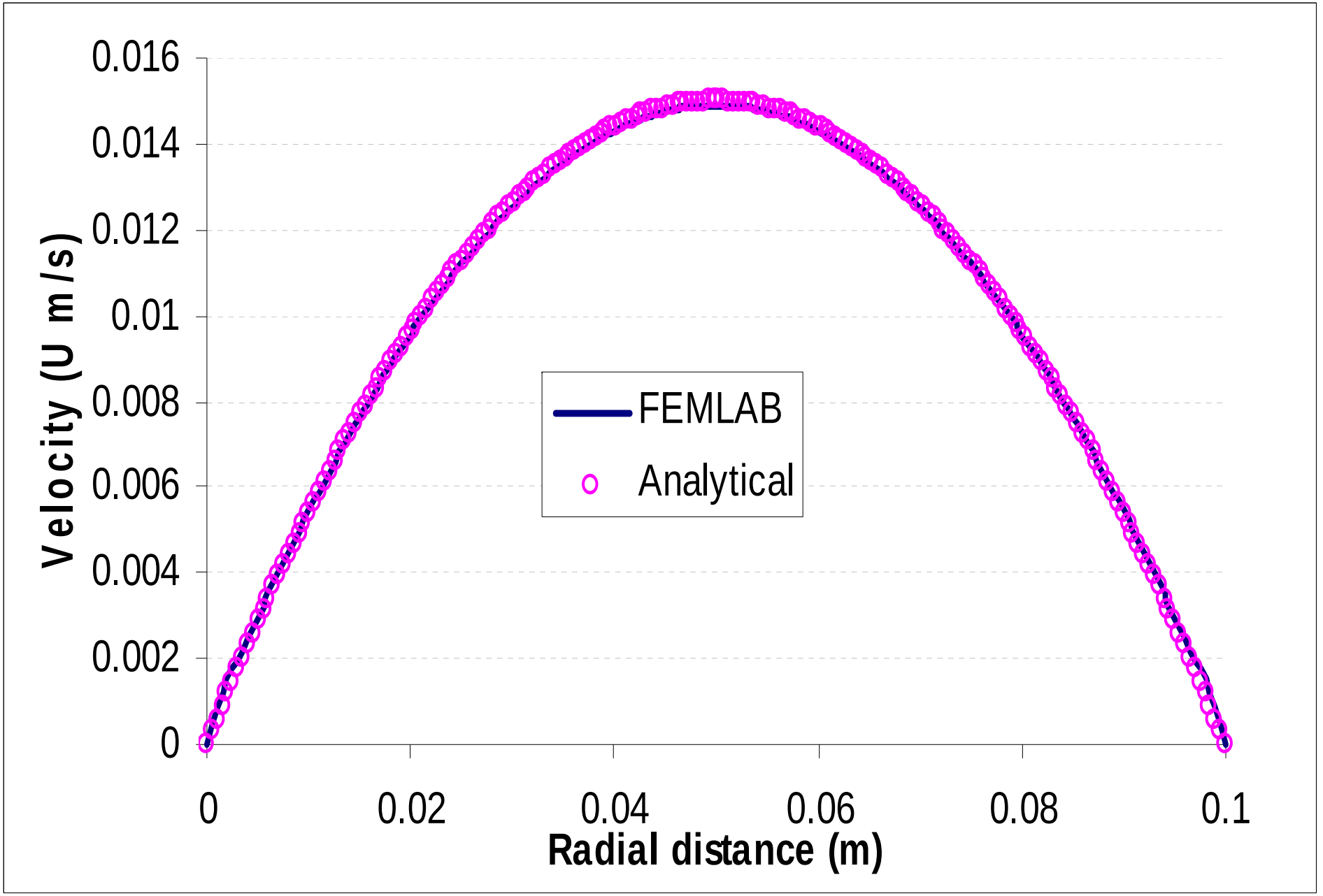
- Density (ρ) = 1.225 kg/m^3.
- Viscosity (μ) = 1.78x10^{-5} kg/m/s
- Dynamic viscosity (η) = 1.45x10^{-5}


Surface: Velocity field
Max: 0.0147
0.014
0.012
0.01
0.008
0.006
0.004
0.002
0
Min: 0
0.4
0.3
0.2
0.1
0
-0.1
-0.2
-0.3
0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
0.9
1

- Analytical Umax is 1.5*U0 = 0.015 m/s for the flow between parallel plates
- This illustrates the relation between the mesh density, solution accuracy and the cpu time.

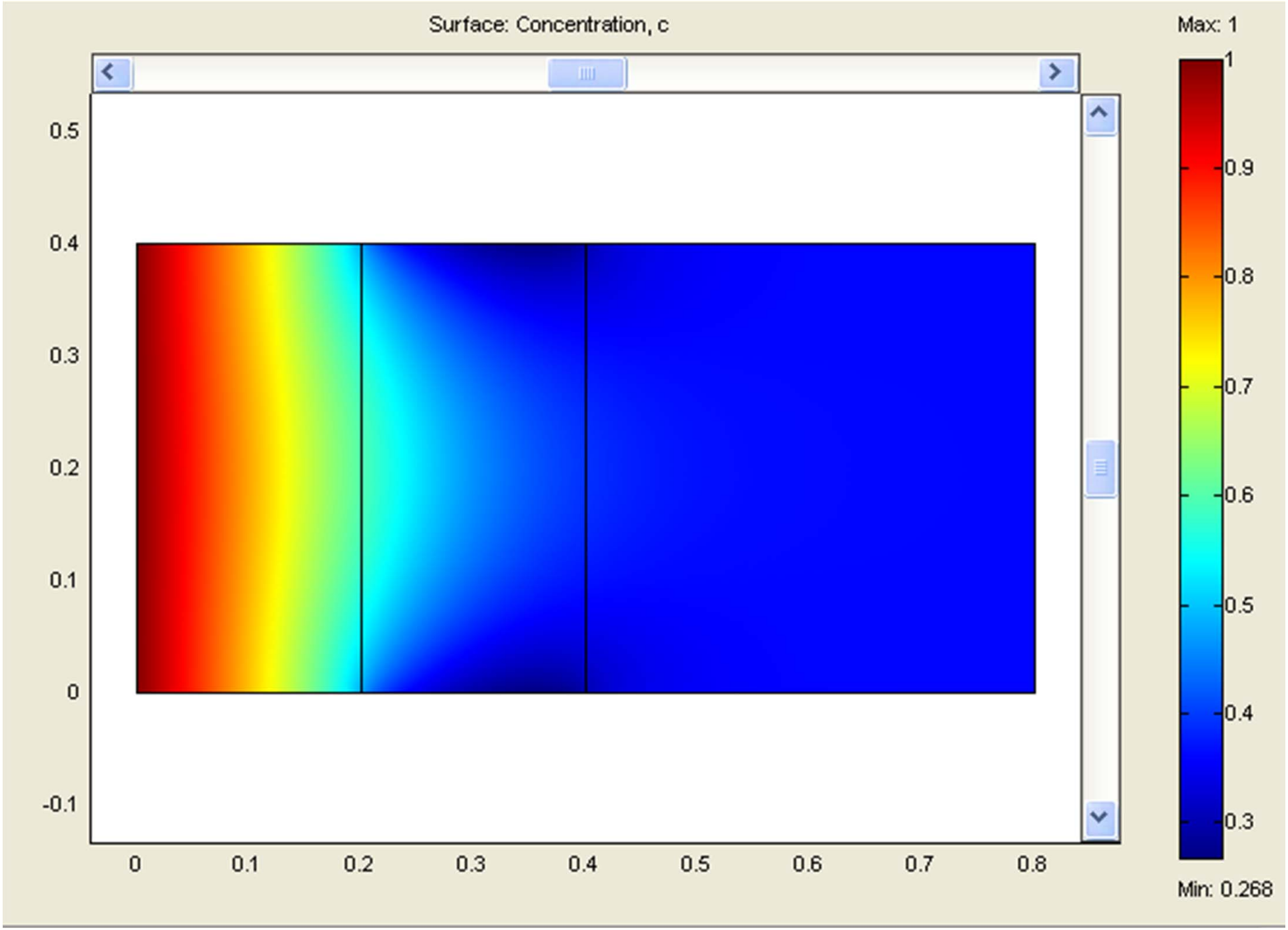
No. of elements	Umax (m/s)	Femlab / Analytical	Solution time (s)
70	0.0138	0.92	0.36
280	0.0144	0.96	0.735
1120	0.0147	0.98	1.671

- The velocity profile, say in the radial direction at the exit, can be plotted using Post Processing → Cross Section Plot Parameters
- Define a line along which the plot is to be made. For the exit here – (x0,y0) = (1,0) and (x1,y1) = (1,0.1). The no. of interpolation points is set to 200 (which is a large enough no.). A figure will be seen once you click "Ok".
- Then to extract the data from the figure to use in matlab or similar applications – click on the ASCII option to write a "data.txt" file.
- Comparing with the analytical solution:


0.016
0.014
0.012
0.01
0.008
0.006
0.004
0.002
0
Velocity (U m/s)
FEMLAB
Analytical
0
0.02
0.04
0.06
0.08
0.1
Radial distance (m)

Diffusion and reaction in a pellet.

- L = 1.6 m (total length of pellet) and D = 0.4 m
- Use symmetry at L = 0.8 so that we account for diffusion from both sides.
- Reaction zone of length 0.2 m at x=0.2
- C = 1 at inlet.
- A first order reaction occurs at the surface in the reaction zone. The reaction rate constant is 2×10^{-4}.
- The diffusivity of gas in the medium is 2.88×10^{-5} m^2/s.
- The contours of concentration are as below:


Surface: Concentration, c
Max: 1
1
0.9
0.8
0.7
0.6
0.5
0.4
0.3
Min: 0.268
0.5
0.4
0.3
0.2
0.1
0
-0.1
0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8

To investigate the effect of mass transfer, lowering the width by a factor of 2.