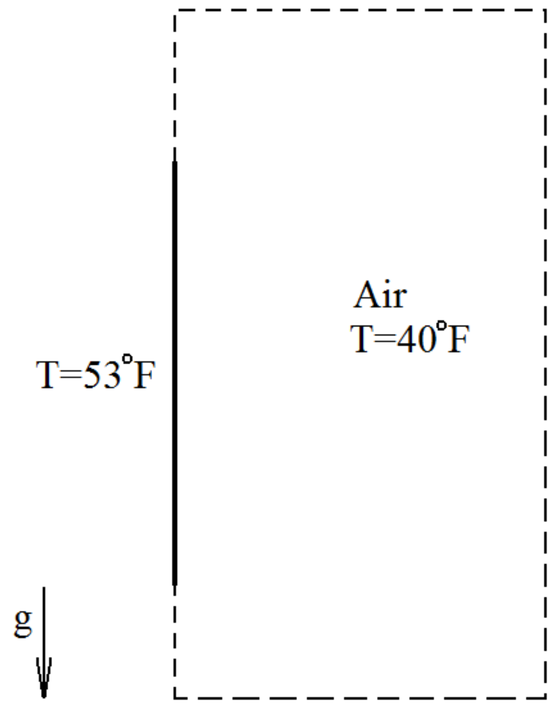
Natural Convection

CFX Tutorial

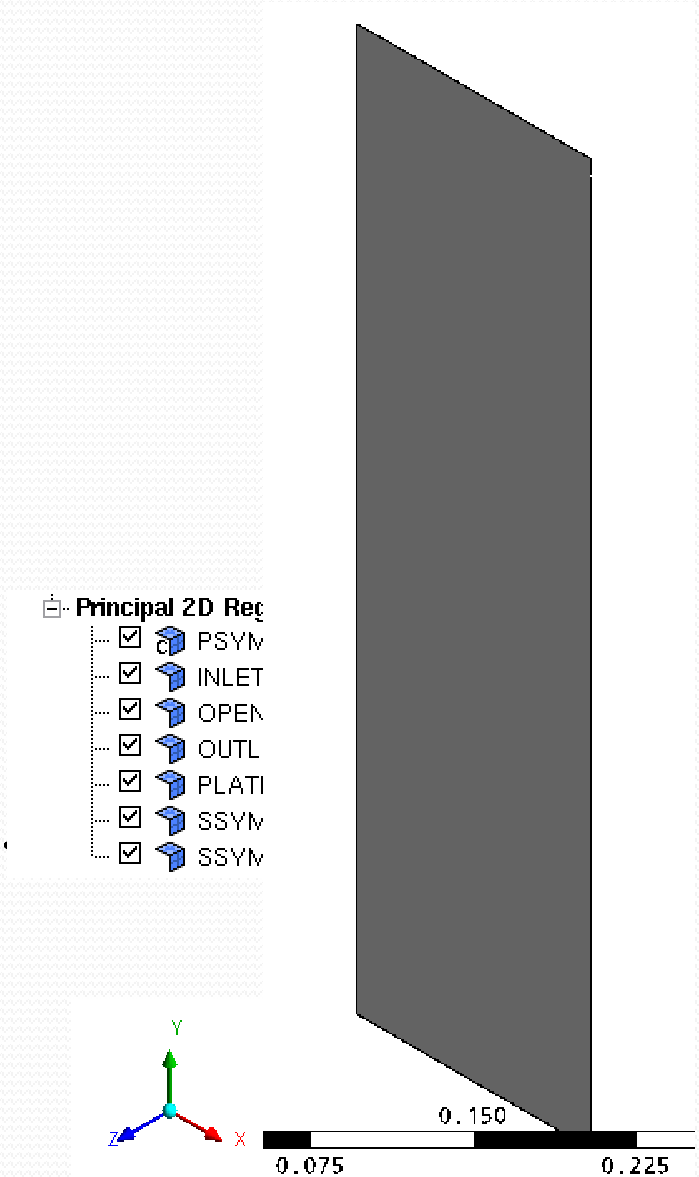
Geometry

- Consider a flat plate of zero thickness standing vertically. The surrounding air is at 40F while the plate itself is at 53F.
- Due to the temperature difference between the plate and the surrounding air, and also because gravity is present, plate will loose heat by natural convection to the surroundings.
- Our goal in this simulation is to find the amount of heat that is going to be lost, and also to find the heat transfer coefficient as a function of distance from the plate's leading edge.

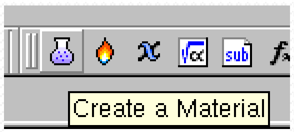


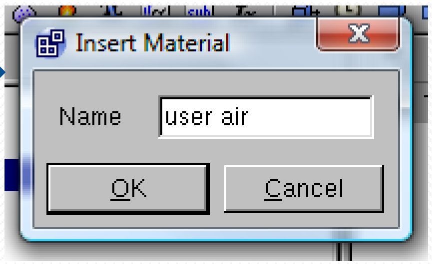
Mesh

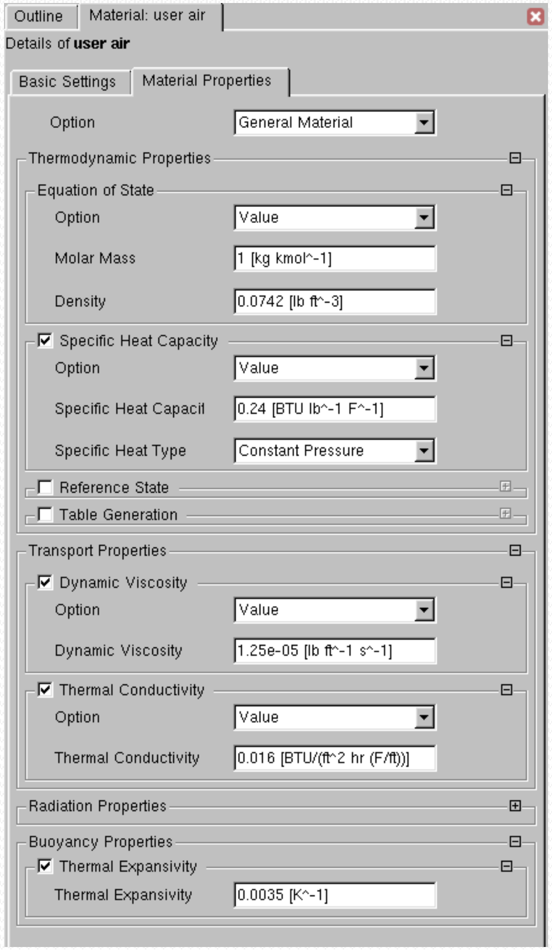
- As the geometry is very similar to flow over a flat plate of zero thickness, and as people have different preferences regarding their meshing program, we will not discuss the meshing method here.
- We will need seven regions shown in the picture to apply the appropriate boundary conditions.
- After creating geometry and mesh, start a new simulation in CFX-Pre and import the mesh.



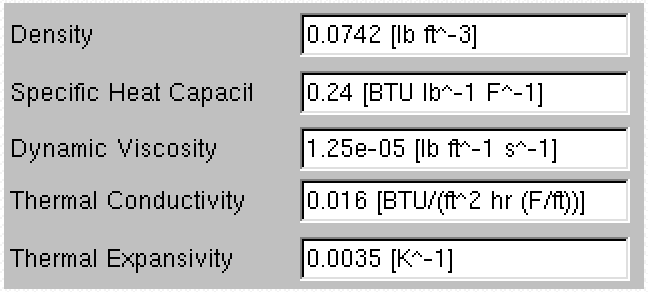
Setting Material Properties



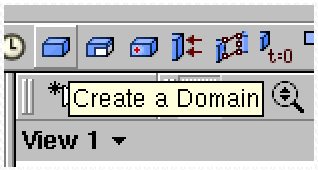


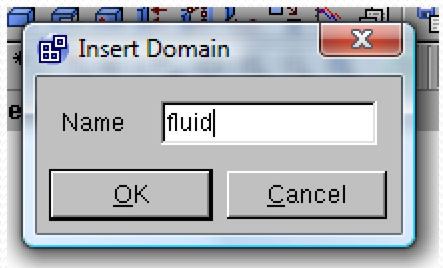


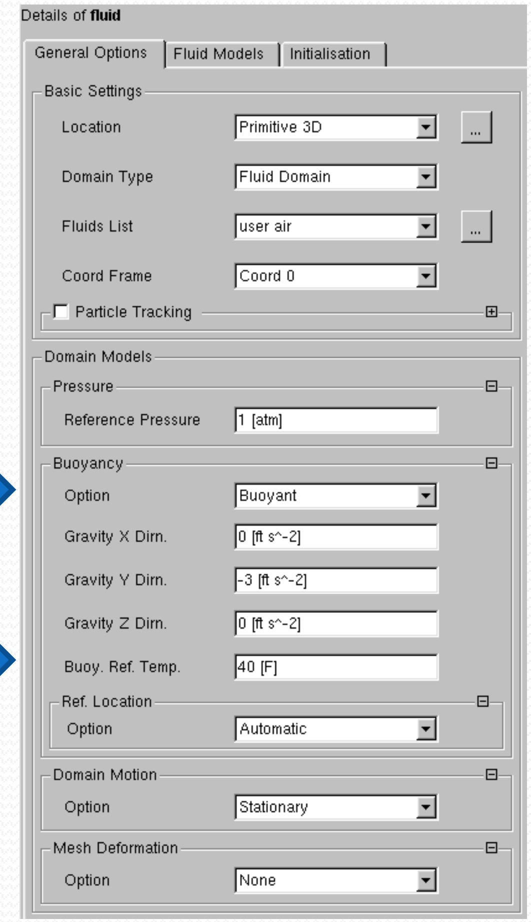
All these properties are necessary for natural convection problems. Note the extra "Thermal Expansivity" which is "Thermal Expansion Coefficient".



Creating the Solution Domain



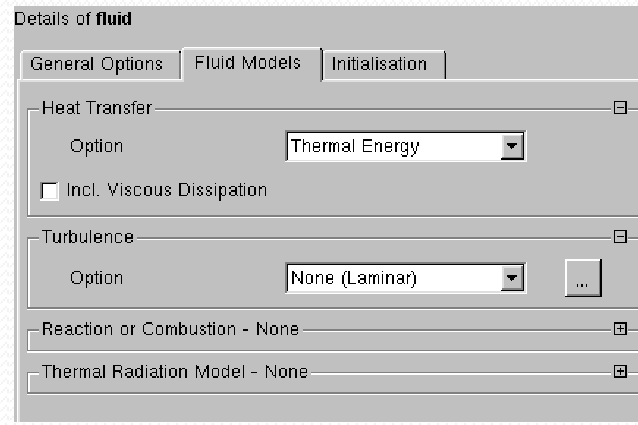




Change this option to “Buoyant” to see the other options. We will start the solution with a acceleration of gravity of ~1/10 of the actual gravity and will increase it as we get converged solutions along the way.

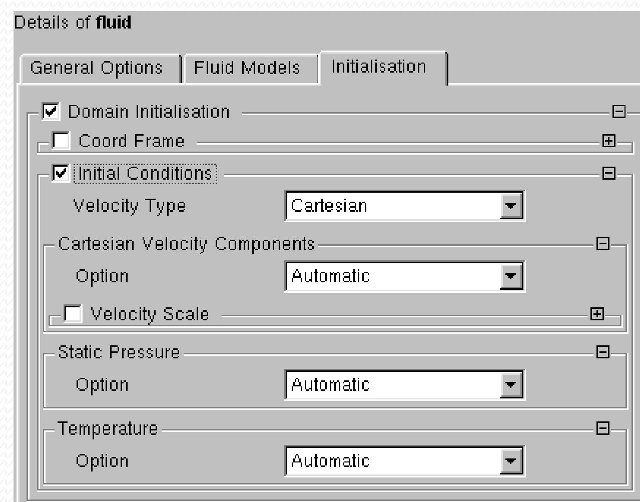
Reference temperature is the temperature at which the fluid has the density provided in the material properties section.

Creating the Soln. Domain (contd.)



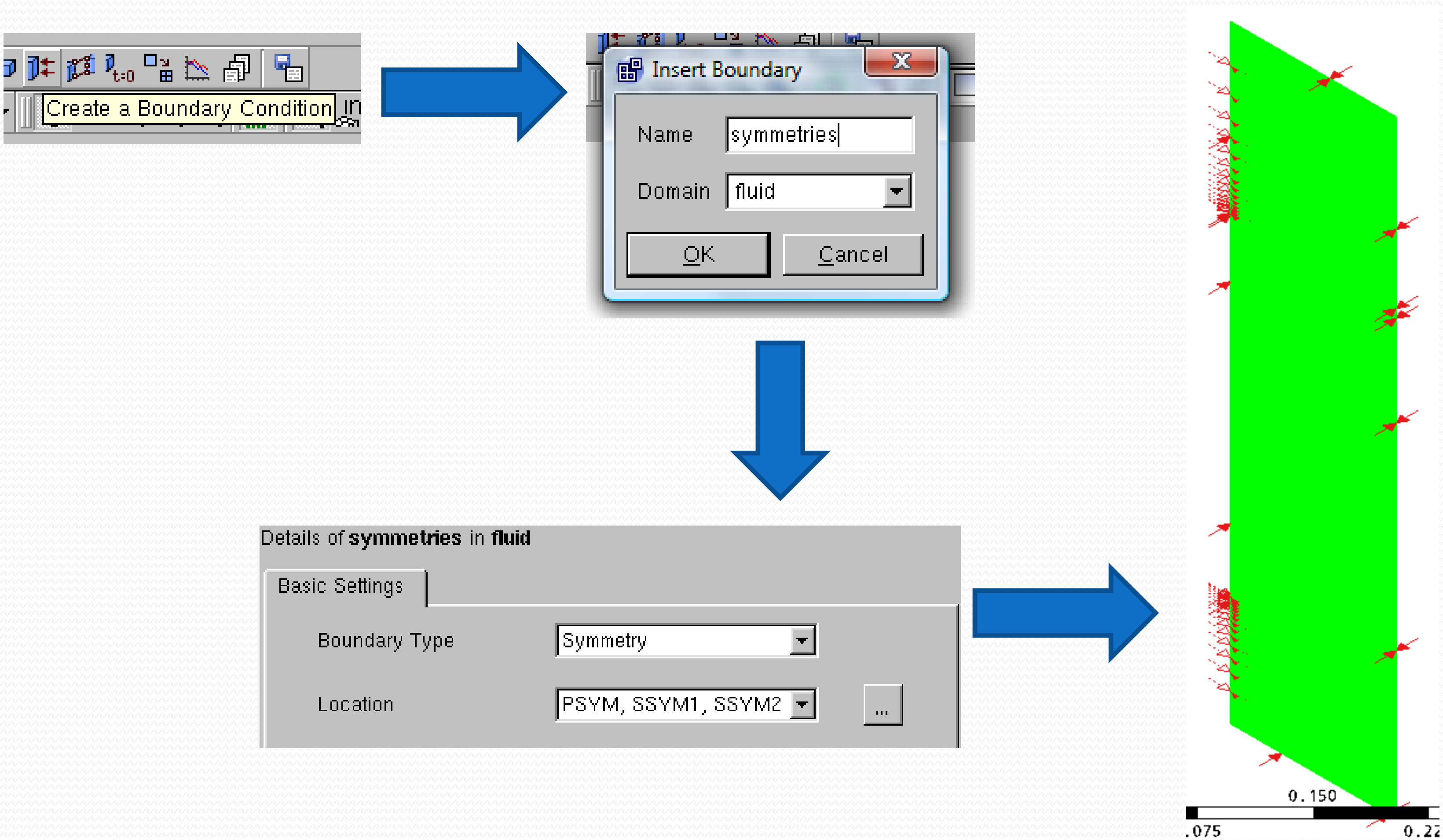
Select “Thermal Energy” for heat transfer

We expect the flow to be laminar. If plate was more than 1.5 ft high, instead of 1 ft, then the flow would have been turbulent.

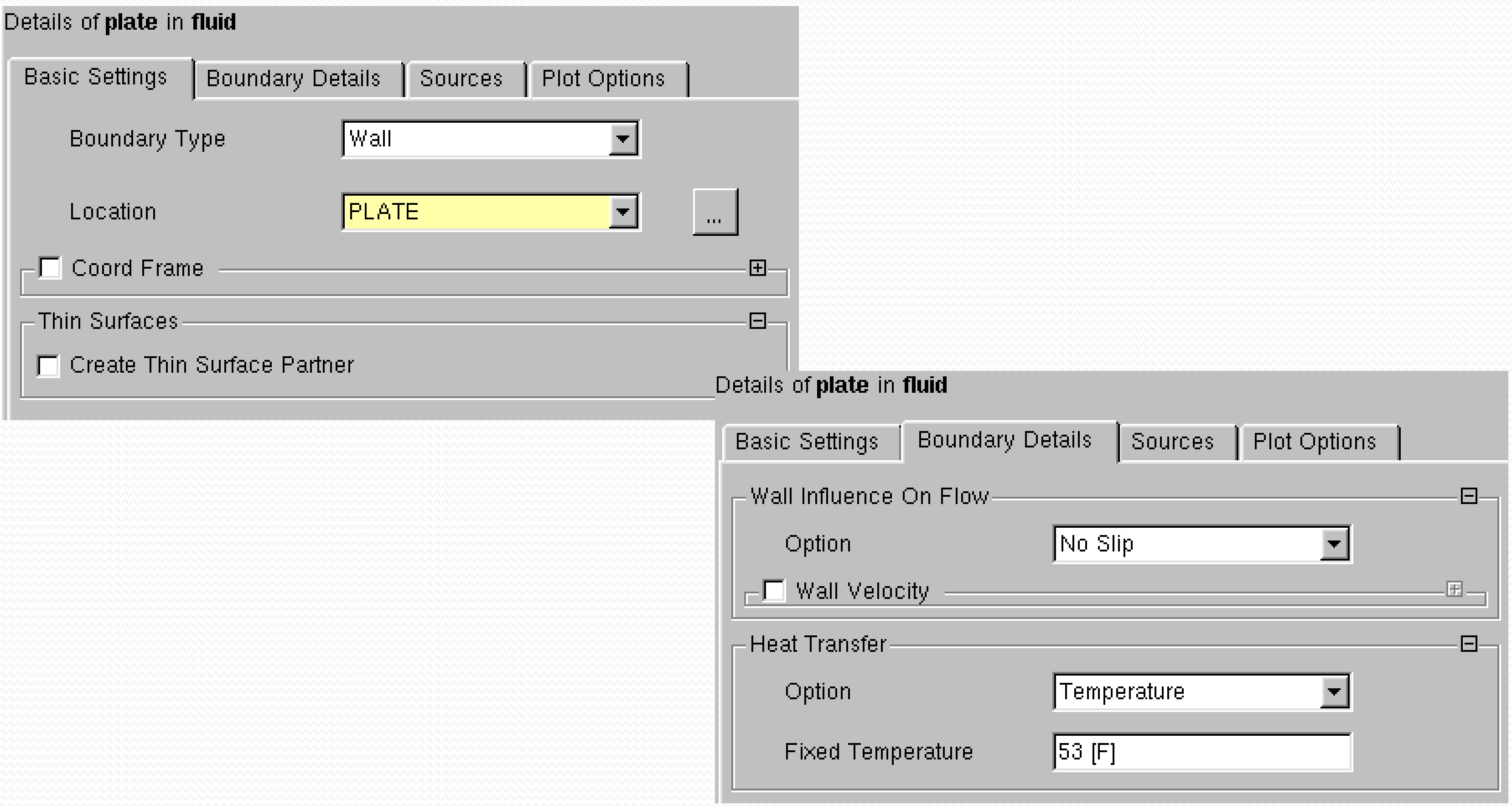


You can either specify the initial conditions using the last tab in the Domain option, or use the following icon on the toolbar to specify them later, since there is only one domain in this problem.

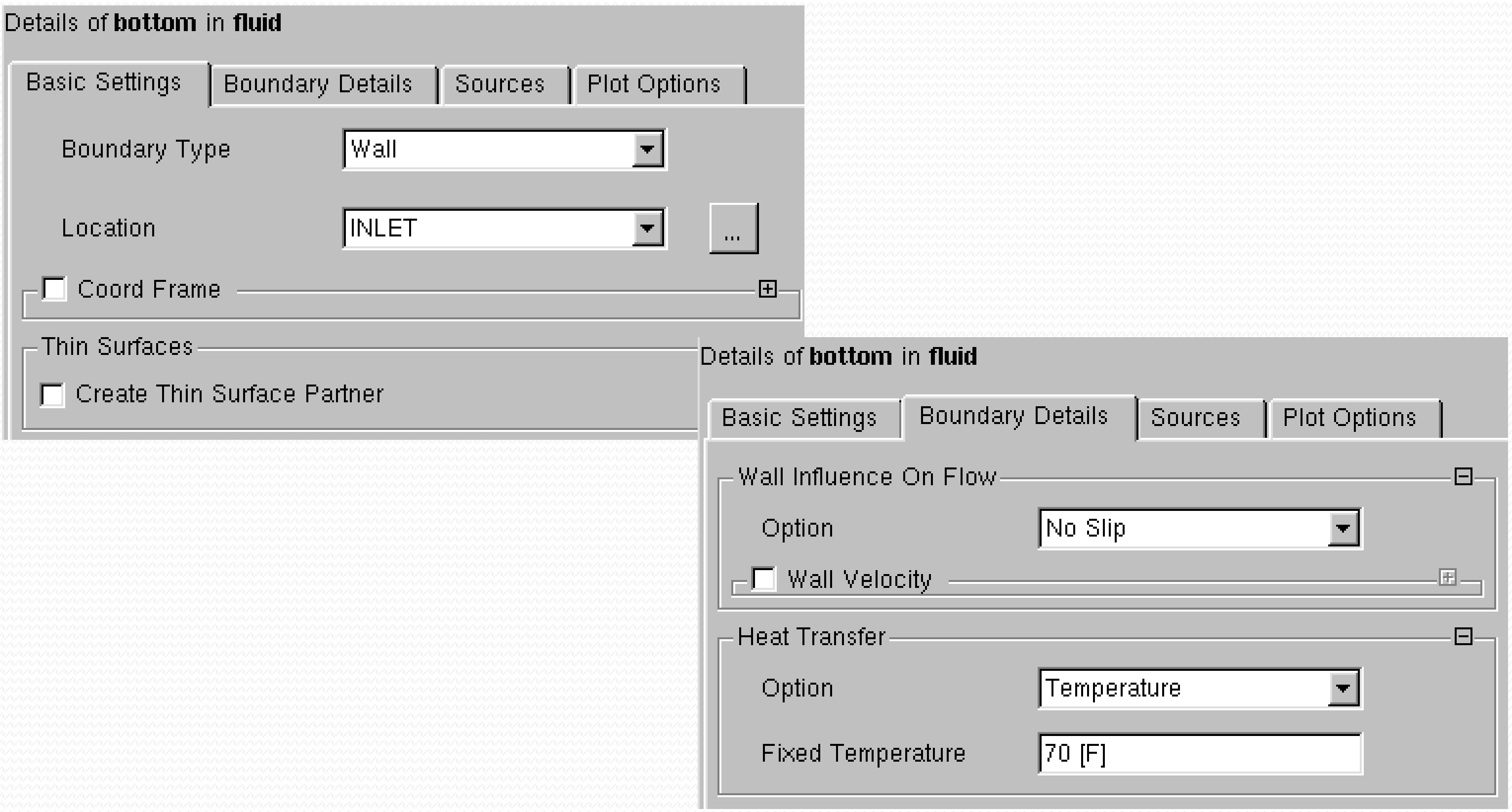
Applying Boundary Conditions



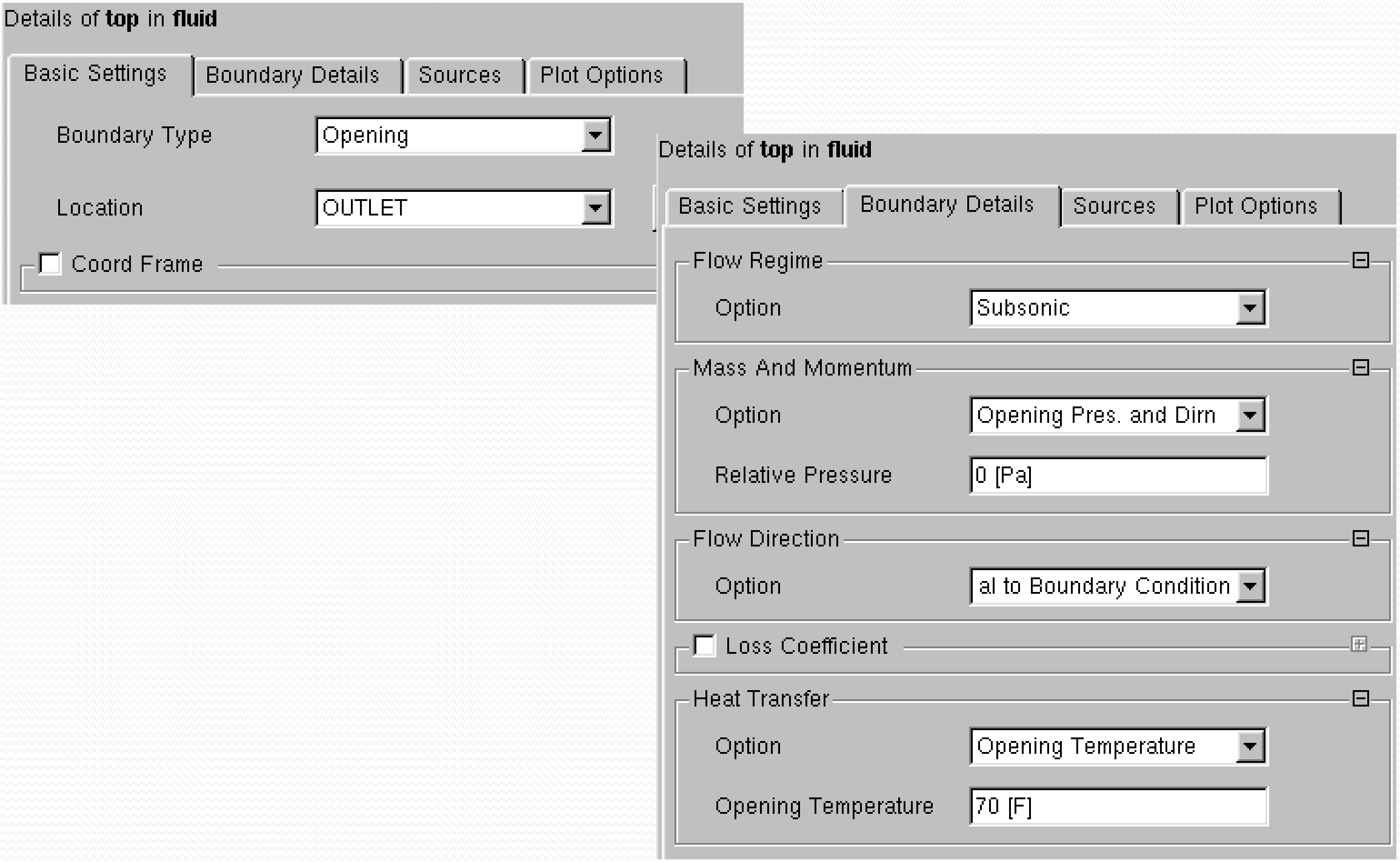
Applying BCs (plate)



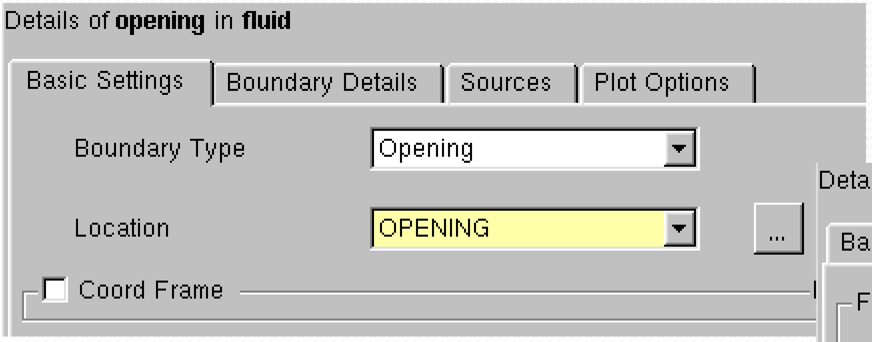
Applying BCs (bottom wall)

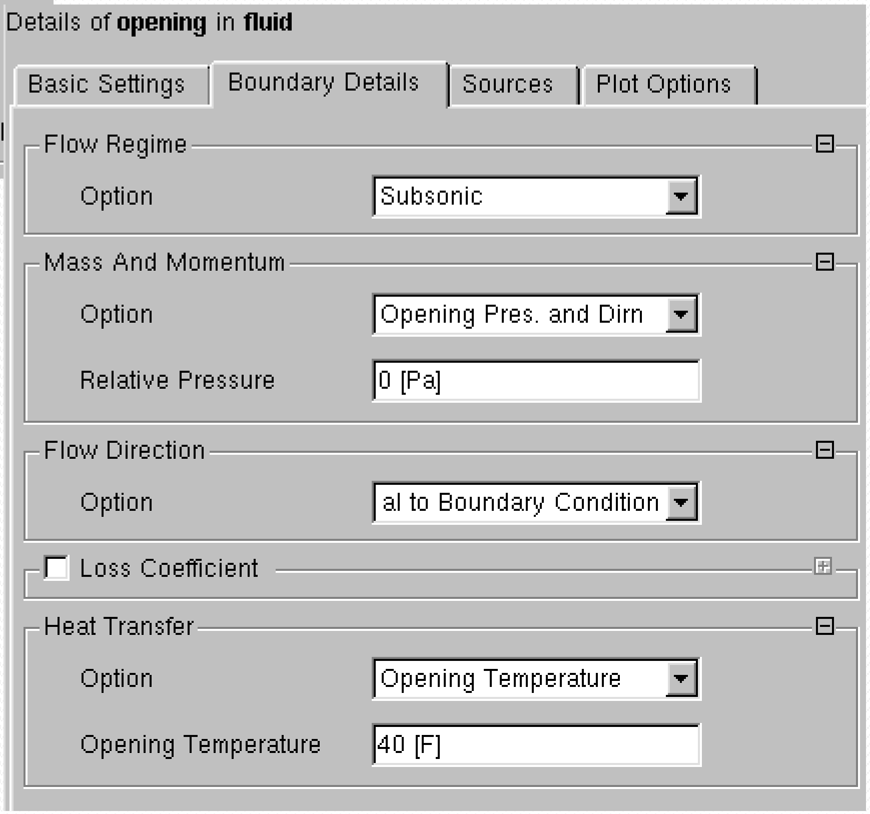


Applying BCs (outlet)

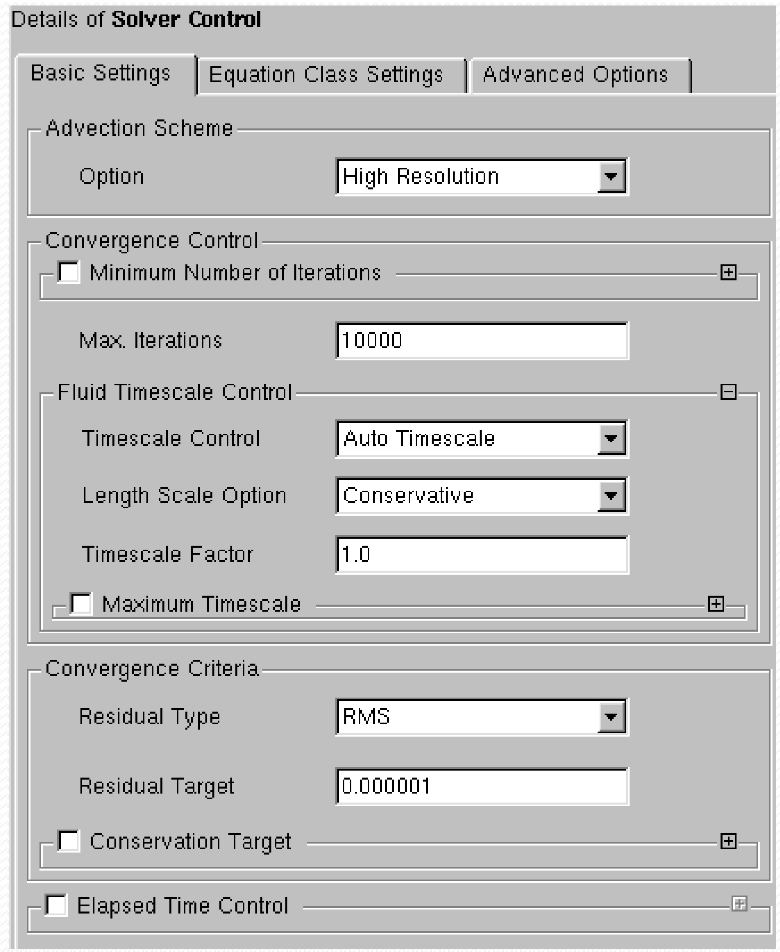


Applying BCs (opening)



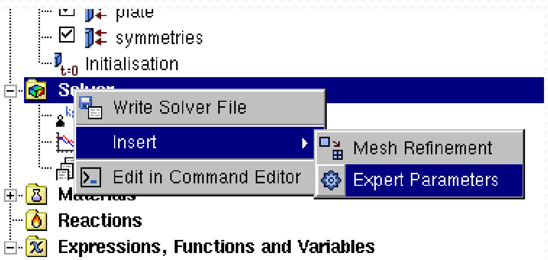


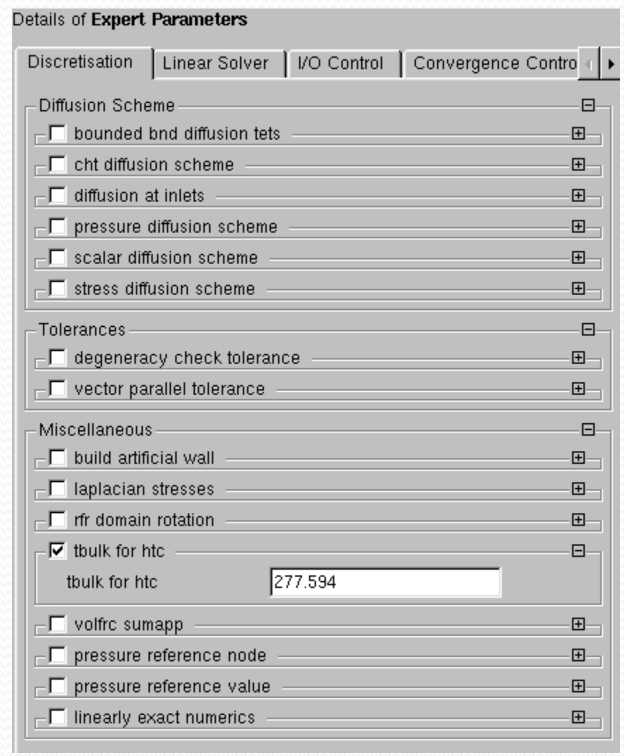
Setting up Solver Parameters



Setting up Solver Params. (contd.)

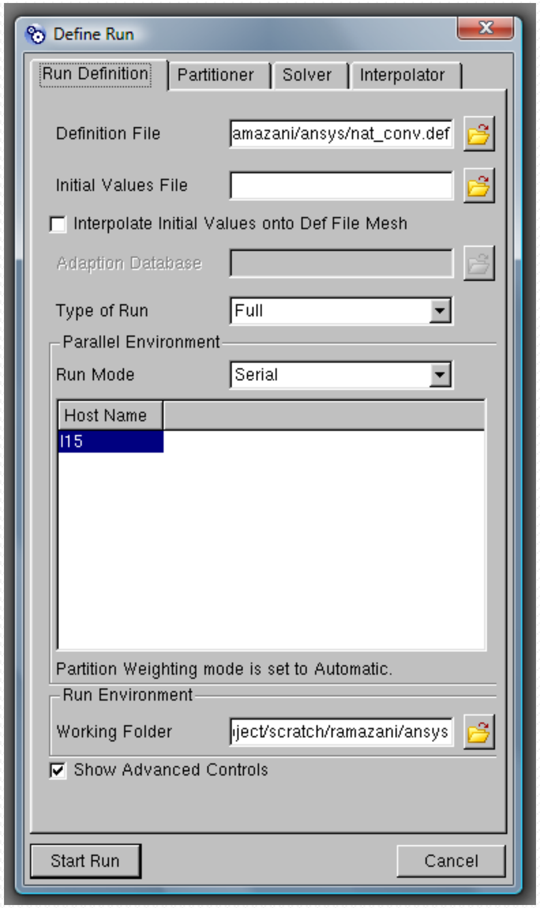
CFX as default calculates heat transfer coefficient by assuming the reference temperature being the temperature of wall-adjacent nodes. We will change this value to the ambient temperature by means of expert parameters.

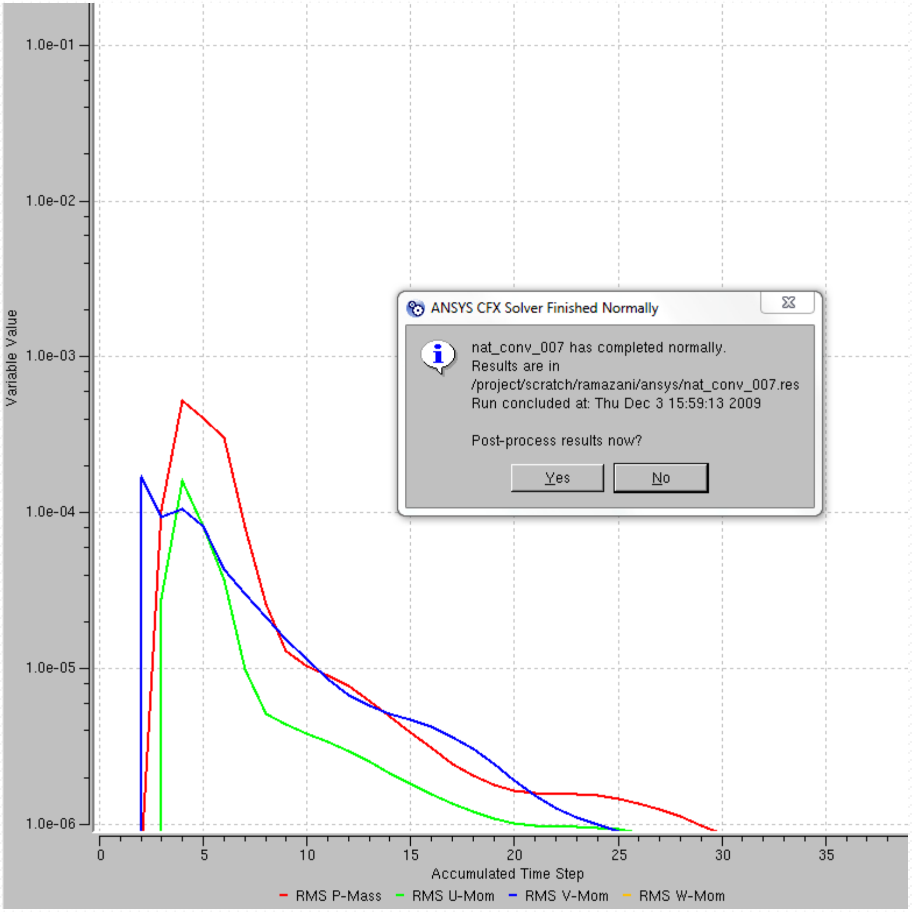




As the solution units for temperature is Kelvin, we need to convert 40F to Kelvin.
We could change the solution units to Rankin if we wanted to, but again we had to convert 40F to Rankin.

Solution

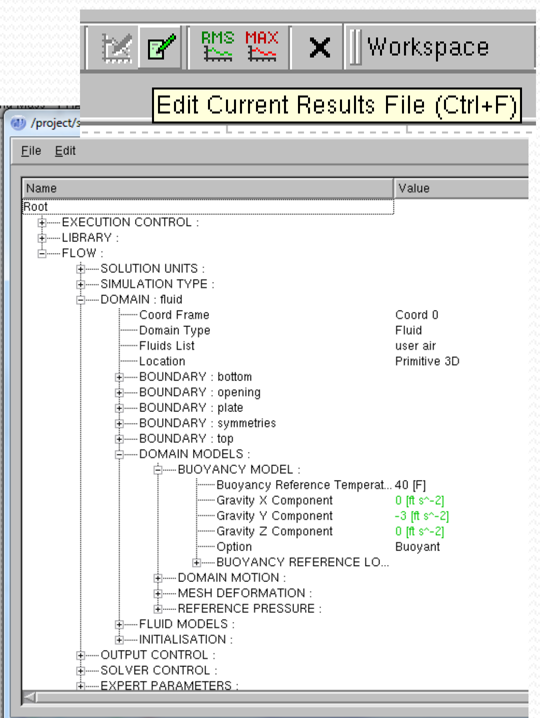

Define Run
Run Definition
Partitioner
Solver
Interpolator
Definition File
amazani/ansys/nat_conv.def
Initial Values File
Interpolate Initial Values onto Def File Mesh
Adaption Database
Type of Run
Full
Parallel Environment
Run Mode
Serial
Host Name
l15
Partition Weighting mode is set to Automatic.
Run Environment
Working Folder
ject/scratch/ramazani/ansys
Show Advanced Controls
Start Run
Cancel


ANSYS CFX Solver Finished Normally
nat_conv_007 has completed normally.
Results are in
/project/scratch/ramazani/ansys/nat_conv_007.res
Run concluded at: Thu Dec 3 15:59:13 2009
Post-process results now?
Yes
No
Variable Value
1.0e-01
1.0e-02
1.0e-03
1.0e-04
1.0e-05
1.0e-06
0
5
10
15
20
25
30
35
Accumulated Time Step
RMS P-Mass
RMS U-Mom
RMS V-Mom
RMS W-Mom

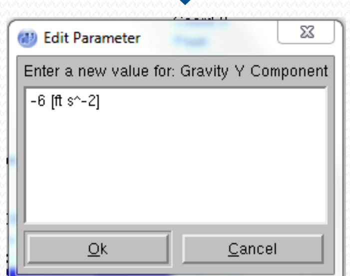
Solution (contd.)

Now that we have a converged solution for diminished gravity, we will increase gravity step by step to get to the actual gravity.

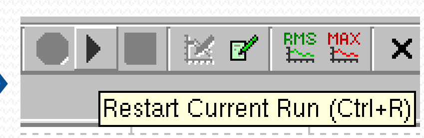
CFX 11.0



"Edit current results file"
>Flow
>Domain
>Domain Models
>Buoyancy Model
>Gravity Y Component



> OK
> File > Save
> File > Exit



CFX 12.0

The same process can be done in CFX12.0 if we launch CFX in standalone mode. Otherwise, we just need to edit the setup cell in the project and update the solution cell.

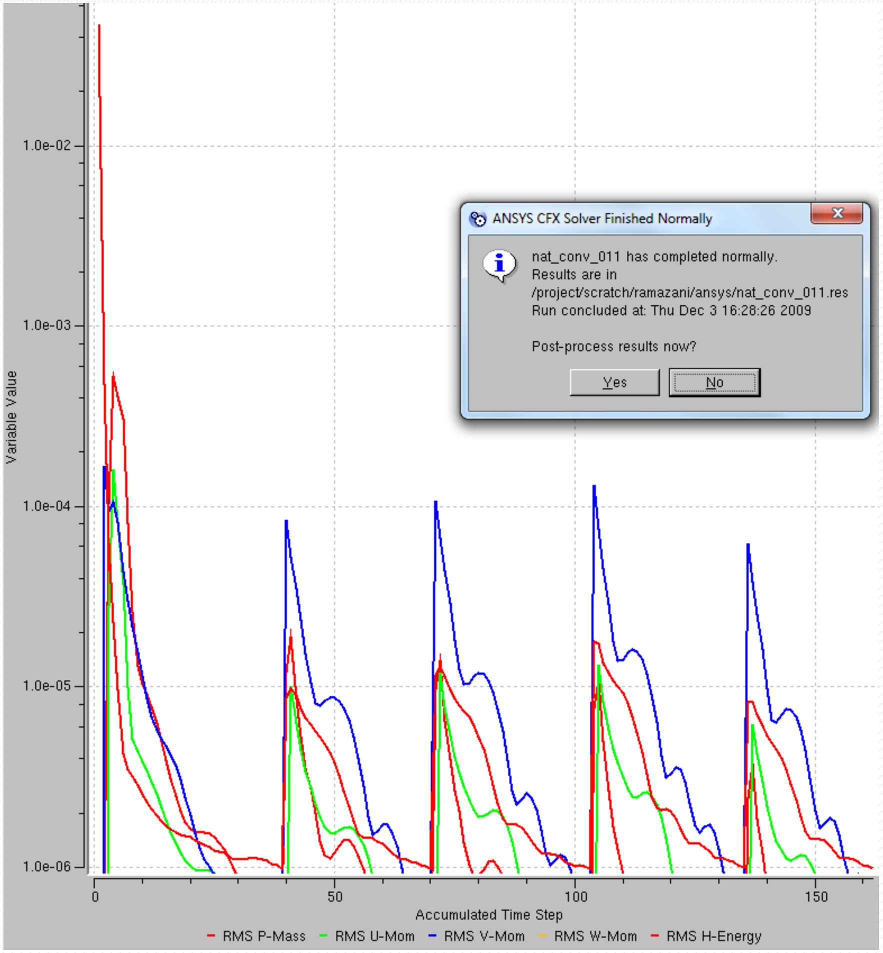
Solution (contd.)

After getting a converged solution at each step, we have to repeat the same process to increase the gravity a little more. We may need to repeat this procedure as little as two to three times, or for more unstable cases more than ten times.

Note:
If you think the convergence rate is very fast, you may can increase the rate at which you increase the gravity.

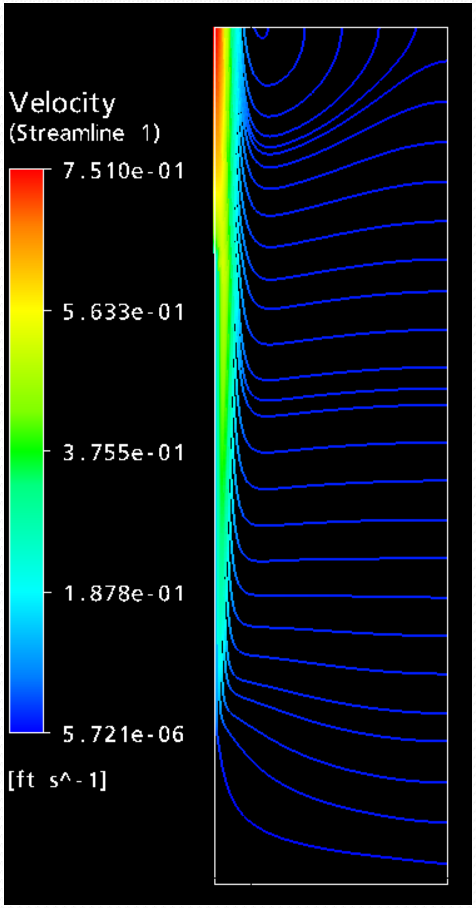
Warning:
If you increase the gravity too much the solution may diverge. If this happens, reload the last converged solution and continue the solution procedure using smaller increment steps.

Solution (contd.)

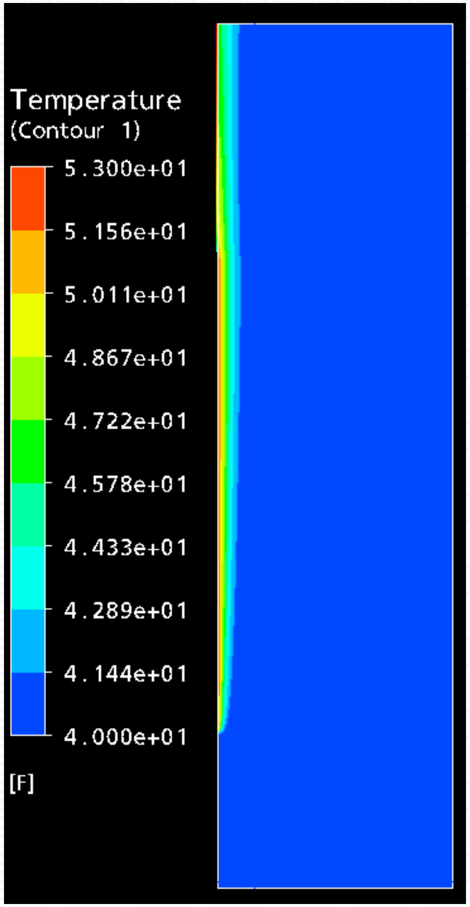


Post-Processing

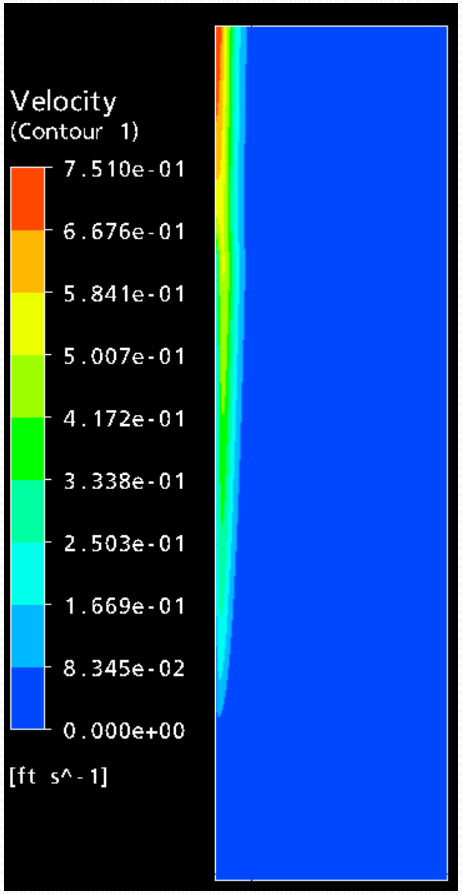
Streamlines



Temperature contours



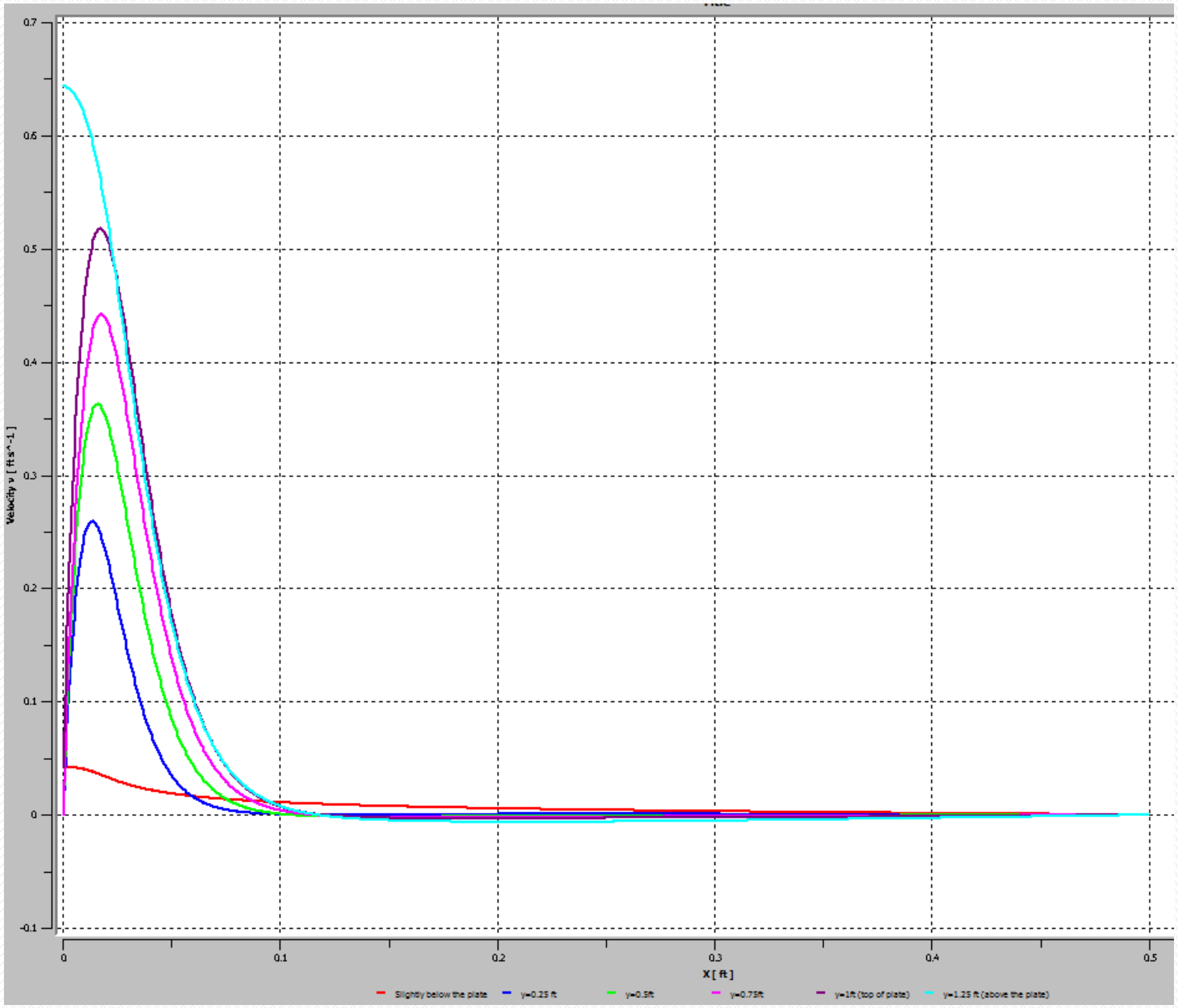
Velocity contours



Post-Processing (contd.)

Notes:

- The non-zero velocity below the plate
- Maximum velocity location shifts to right as y increases
- Above the plate, maximum velocity is at x=0.
- Velocity continues to increase even after the plate ends.



Post-Processing (contd.)

This graph shows the variation of heat transfer coefficient as a function of distance from the upstream edge of the plate.

The average heat transfer coefficient is:
0.55 Btu/hr.ft^2.F