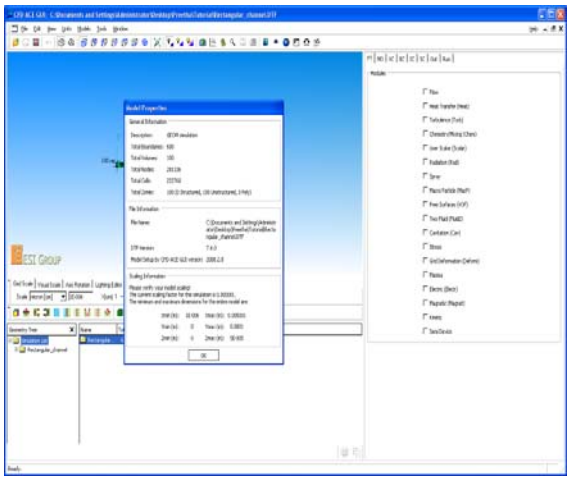
CFD-ACE+ & CFD-VIEW TUTORIALS

A Simple Rectangular Microchannel

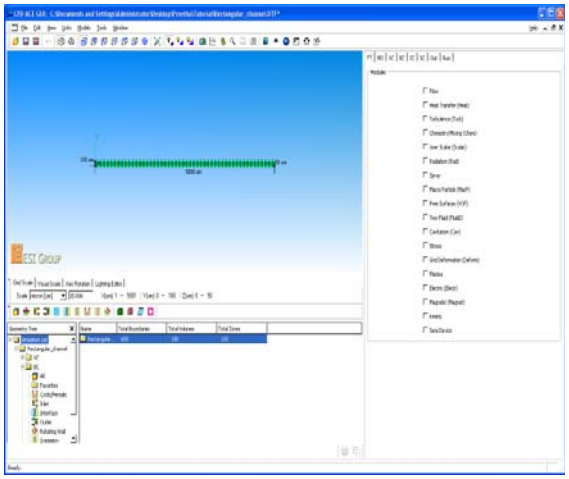
Click File ->Open -> Rectangular_Channel.DTF

Check if Scaling Factor is set to micrometer (1E-006)

CFD-ACE-SOLVER expects all dimensions to be in meters. Click **OK** on Model Properties dialog box after checking scaling information.

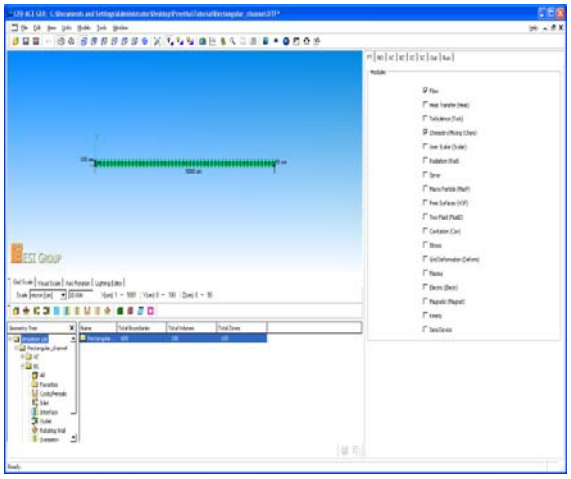


If the scaling factor was not set in CFD-GEOM, it can be applied in the CFD-ACE GUI



Problem Type (PT):

1. Flow Module
2. Chemistry/Mixing Module



Model Options (MO):

Shared

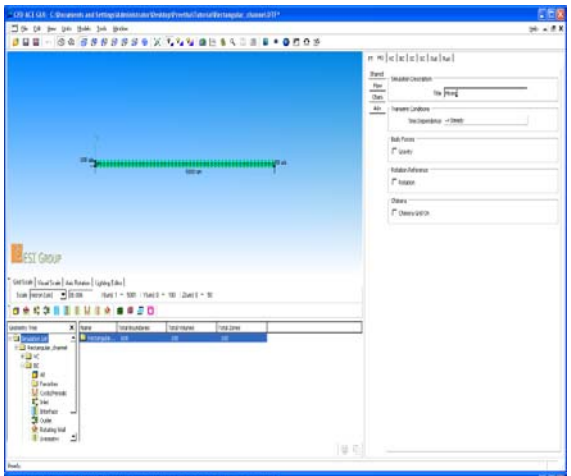
Set Title as **Mixing**

Set Time Dependence = **Steady**

Flow

Reference Pressure = **100000 Pa**

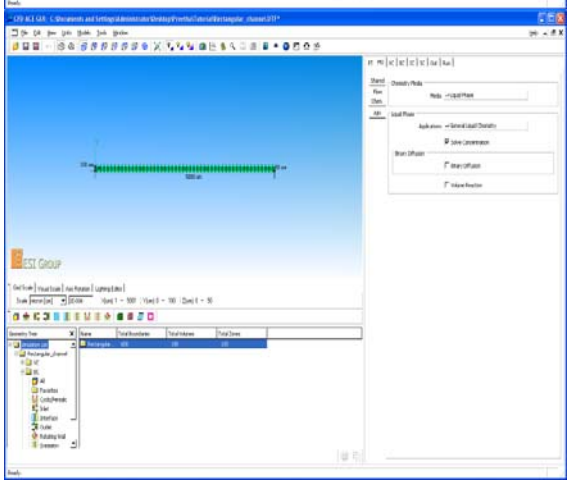
Setting Reference pressure to 0 makes the solver calculate the pressure in absolute pressure units.



Chem

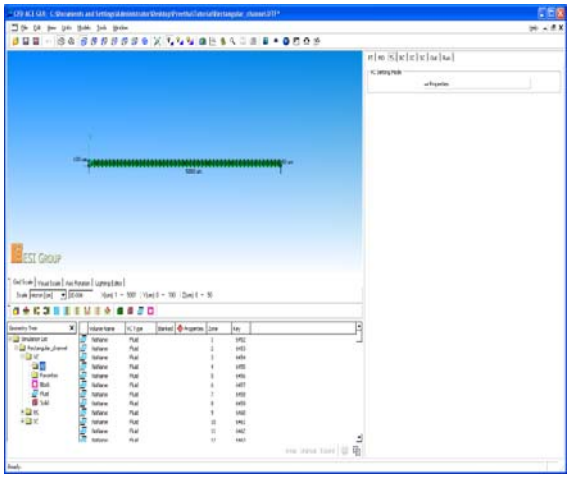
Media : **Liquid Phase**

Check **Solve Concentration**



Volume Conditions (VC):

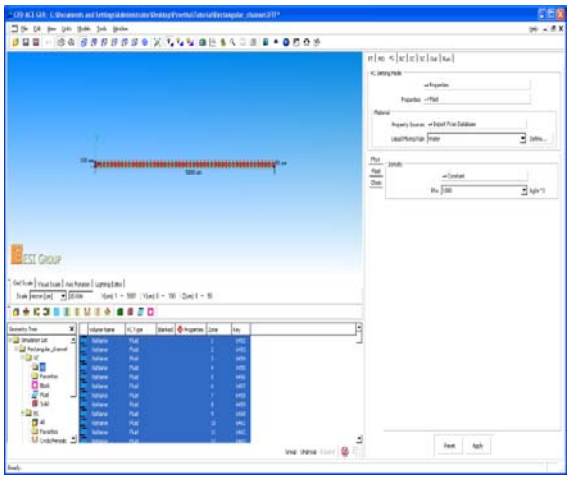
Select all the volumes using the **Select All** button at the bottom of your screen as shown.



Properties
"Fluid"
Material
Property Sources set to **Import from Database**
Liquid Mixing Rule : **Water**

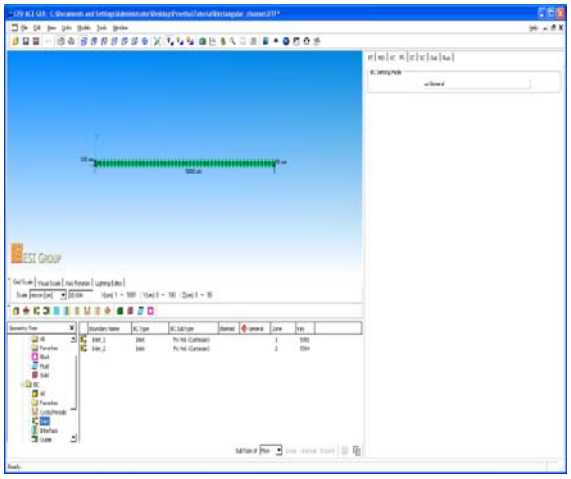
Phys (Physical Properties)
Density = **1,000 kg/m^3** (for water)
Fluid (Fluid Properties)
Viscosity (dynamic) = **1E-03 kg/m-s**

Click **Apply**

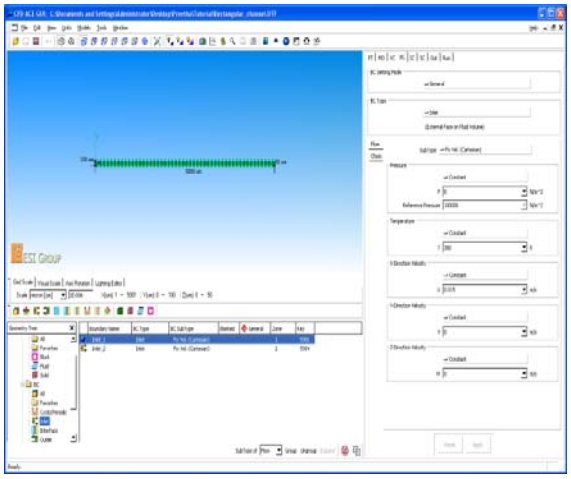


Boundary Conditions (BC):

Model Explorer
Select "**Inlet**" under the BC tree to see just inlet BC's in the Model Explorer

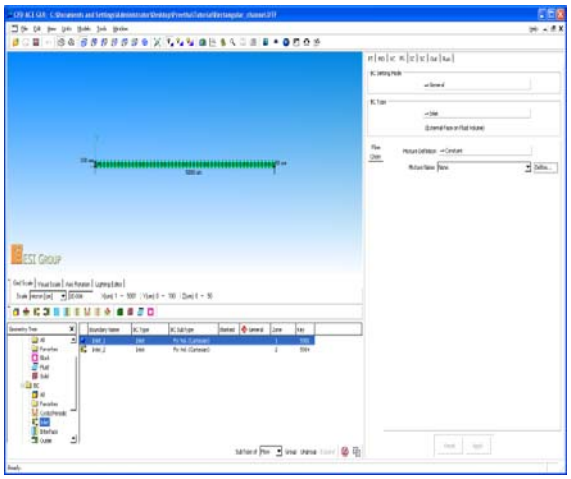


Select the BC named **Inlet_1** (was so named in CFD-GEOM)
→ boundary patch corresponding to **Inlet_1** gets highlighted in red on the model



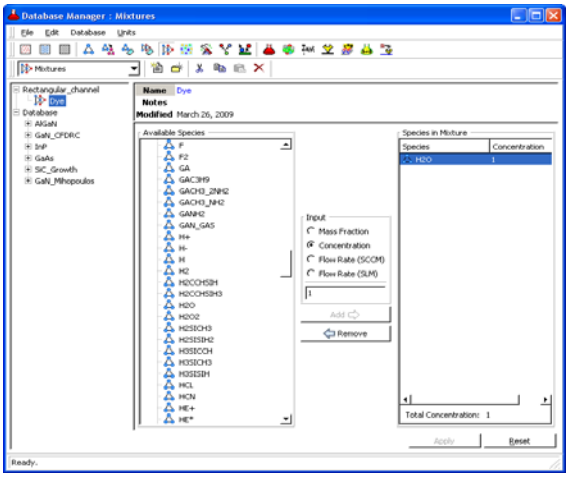
Flow
X-Direction Velocity = **0.015 m/s**

Chem
Click **Define** to create the two mixing fluids (dye in water and just water)



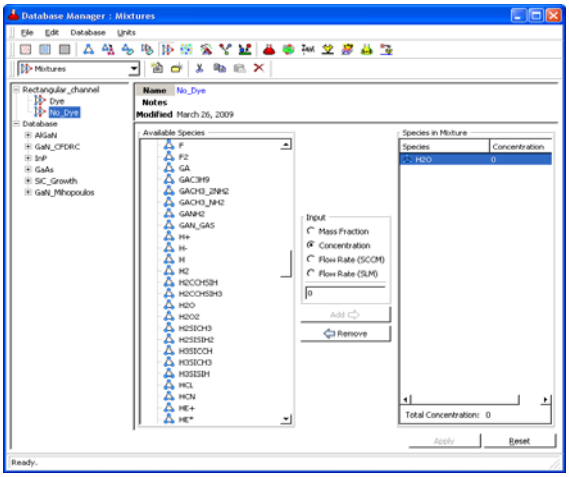
Click on the **LOCAL** collapse bar to see the available mixtures. To create new mixture click on the **New Mixture** icon at the top of the screen as shown.

Mixture Name : **Dye**
User Input : **Concentration**
Available Species: **H_2O**
Click **Add**
Molar Concentration = **1**
Click **Apply**

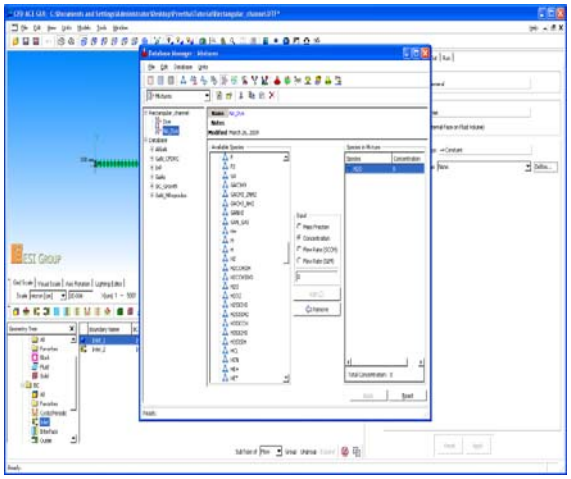


Similarly create the mixture named **No-Dye**

Mixture Name : **No_Dye**
User Input : **Concentration**
Available Species: **H_2O**
Click **Add**
Molar Concentration = **0**
Click **Apply**

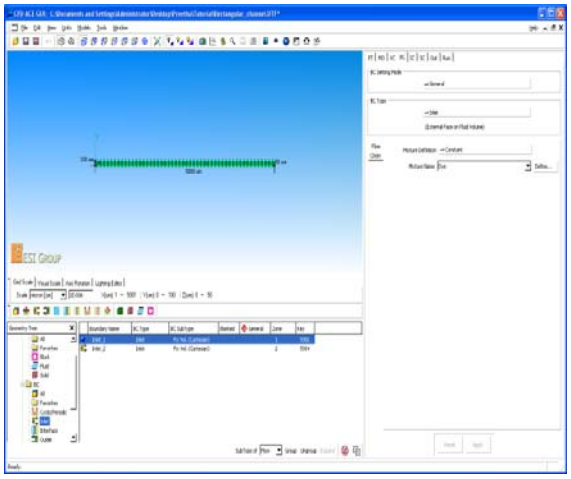


Close the species window to go back to the solver.

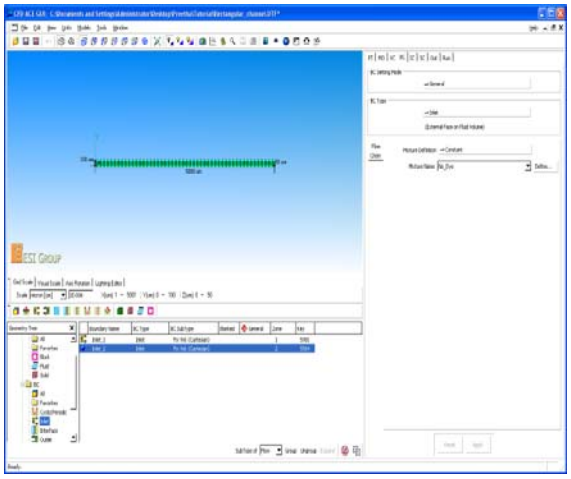


Chem
Mixture Name = **Dye**

Click **Apply**



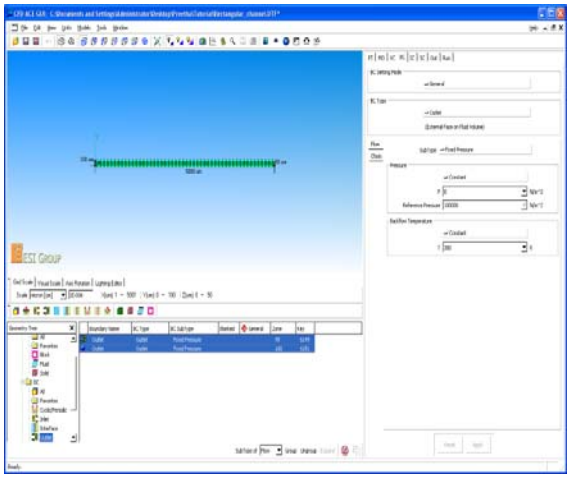
Similarly select the BC named **Inlet_2** (was so named in CFD-GEOM)
→ boundary patch corresponding to **Inlet_2** gets highlighted in red on the model



Flow
X-Direction Velocity = **0.015 m/s**
Chem
Mixture Name = **No_Dye**

Click **Apply**

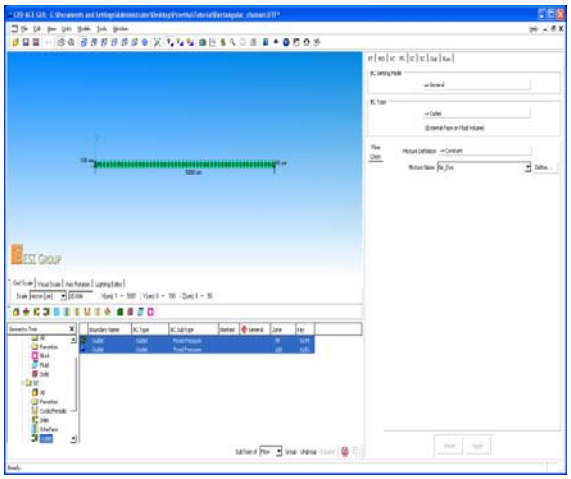
Model Explorer
Select "**Outlet**" under the BC tree to see just outlet BC's in the Model Explorer
Select the BC's named **Outlet** (was so named in CFD-GEOM)
→ boundary patch corresponding to the two **Outlet** boundaries gets highlighted in red on the model



Flow
SubType = **Fixed Pressure**
P = **0**

Click **Apply**

Chem
Mixture Name = **No_Dye**



Click **Apply**

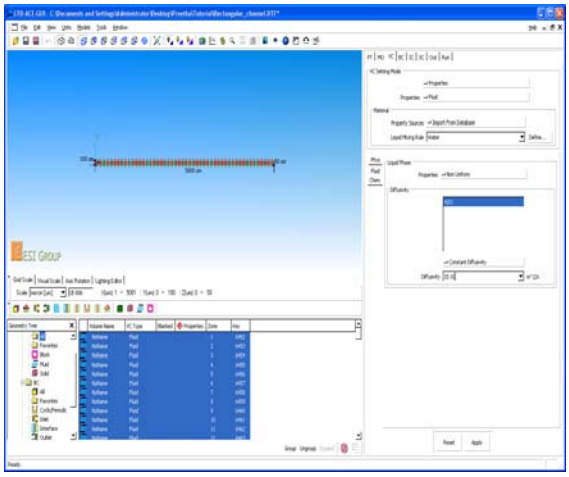
Now go back to the **Volume Conditions (VC)** properties.

Chem

Properties = **Non Uniform**

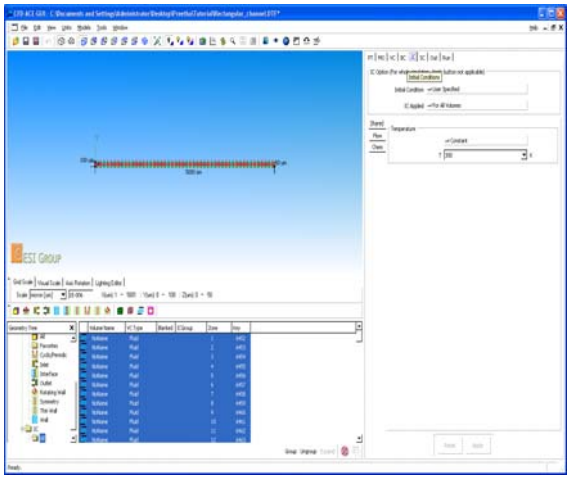
Diffusivity = **1E-10 m^2/s**

Click **Apply**



Initial Conditions (IC):

The Initial Conditions are the "Starting Point" for the Solution.
Although the initial conditions should not affect the final solution, they can affect the path to convergence.
→Bad Initial Conditions Could Cause Divergence!
→ Choosing Realistic Initial Conditions Will Allow an Easier "Start" for the Solver.



Shared

T = **300** (default)

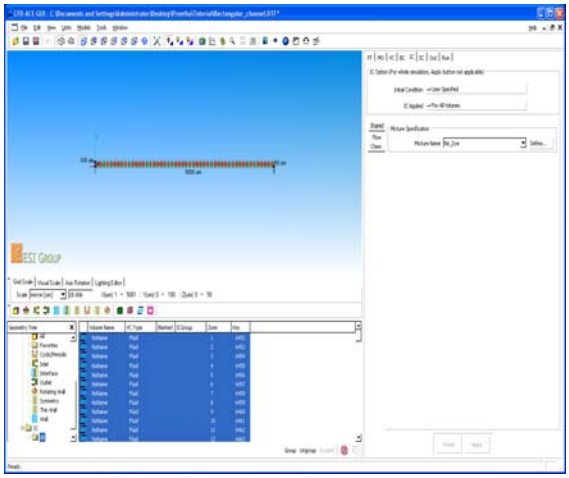
Flow

As the fluid velocities are very small, we do not have to change anything here. (all default values)

Chem

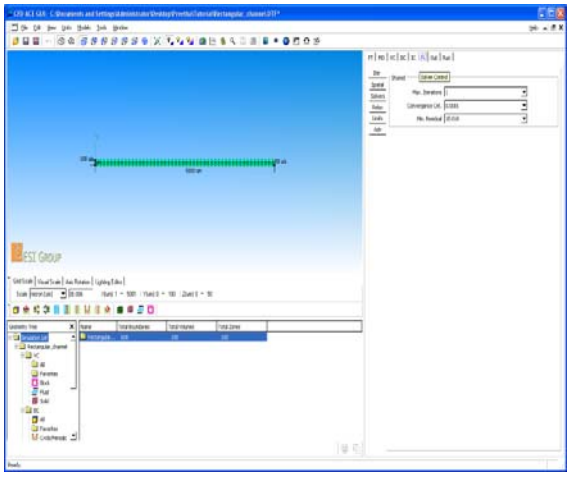
Mixture Name = **No-Dye**

Click **Apply**



Solver Control Parameters (SC):

Solver Control Settings Define the Computational Numerics of the Problem



Iteration

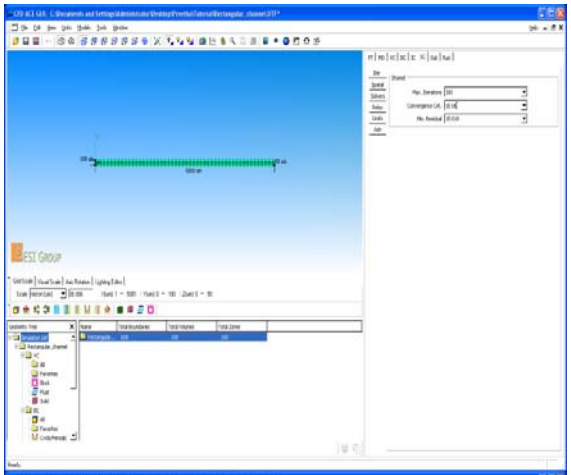
→Defines how many solver iterations to loop through

Max Iterations = 300

→the solver will run 300 iterations or until the convergence criteria is met, whichever comes first

Convergence Crit. = 1E-06

Min. Residual = 1E-018



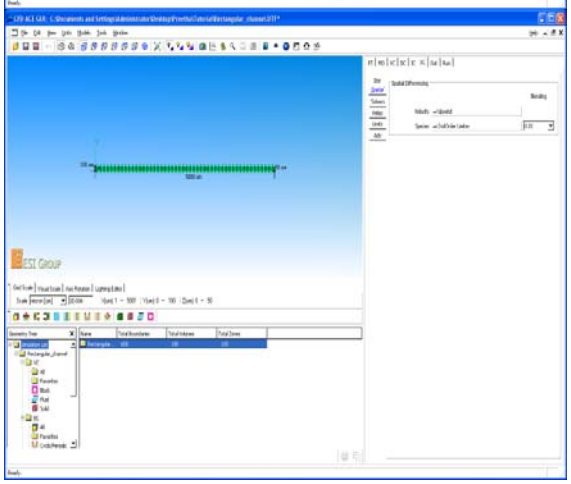
Spatial Differencing

→Defines the differencing scheme used for convective terms

→Leave differencing for Velocity to default values (**Upwind**)

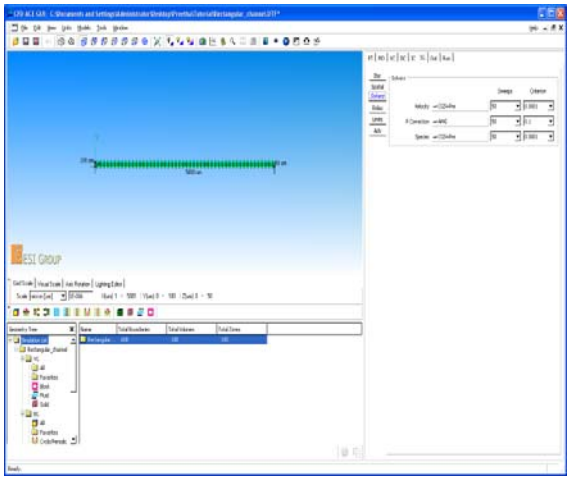
→Change differencing to **2nd Order Limiter** for Species.

→**Blending** of 0.01 implies 1% of upwind mixed with central differencing for the sake of stability



Solvers
→Use the Default Solver (CGS+Pre) for the Velocity and Species Variable
→ Use AMG Solver for Pressure Correction Equation

- the AMG solver sometimes performs better for pure diffusion equations

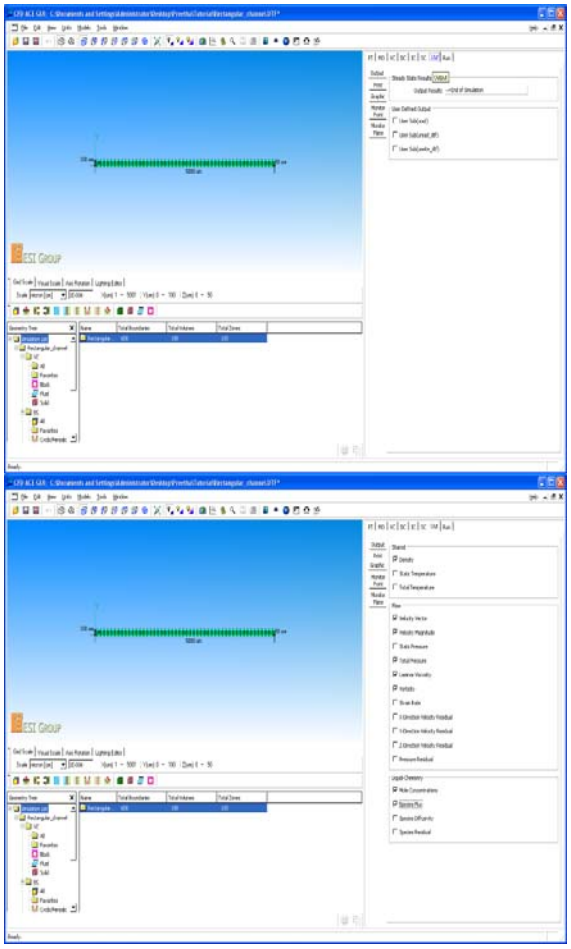


Use Default Values for **Relax**, **Limits** and **Advanced** parameters

Output Options (Out):

Output
→Select Frequency to Write Output File. Write output results at the **end of simulation**

Printed Output
→Select Information to be Written to Text Based Output File (model.out)



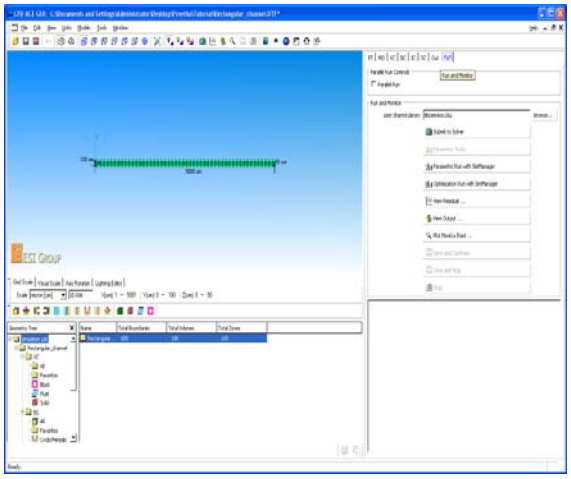
Graphical Output
→Select Information to be Written to DTF File for Graphical Post Processing in **CFD-VIEW**
→Variables Selected are Entirely Optional,

Some of Interest:

- **Velocity Vector**
- **Velocity Magnitude**
- **Total Pressure**
- **Vorticity**
- **Molar Concentration**
- **Species Flux**

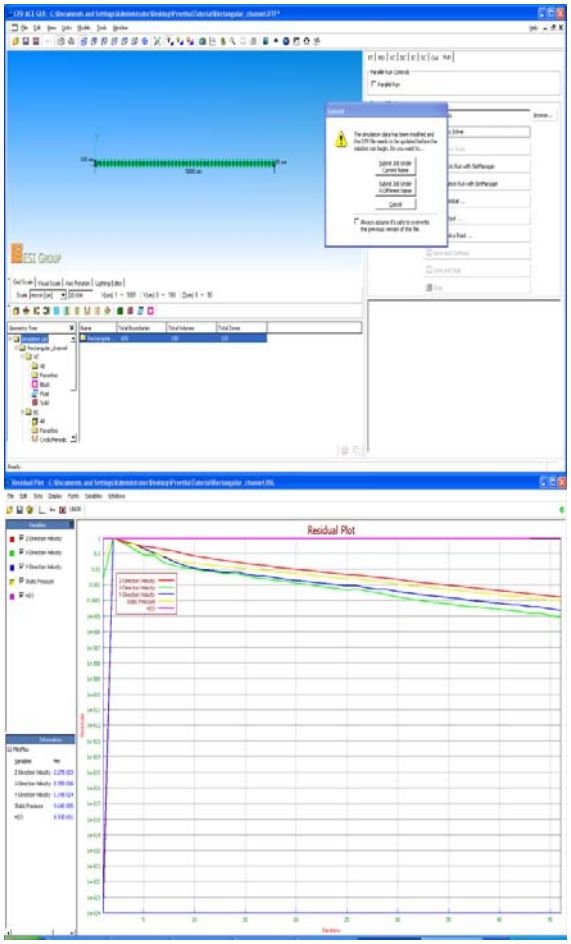
Run and Monitor (Run):

Click "**Submit to Solver**"



Click " **Submit Job Under Current Name**"

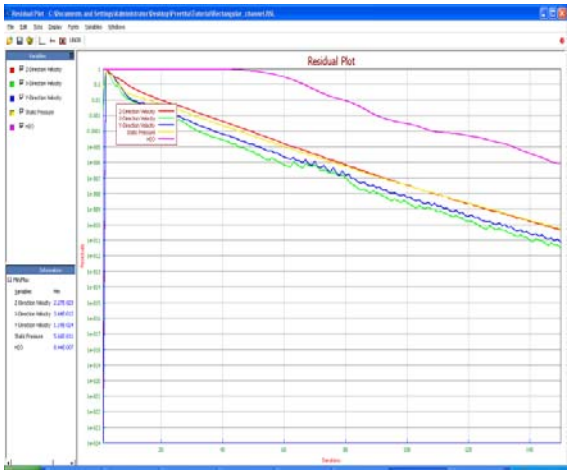
Click "**View Residual**" to see the results converge



Simulation Running.
In the residual window we can see the various input parameters converging. The green button on the upper right hand corner indicates that the simulation is still running.

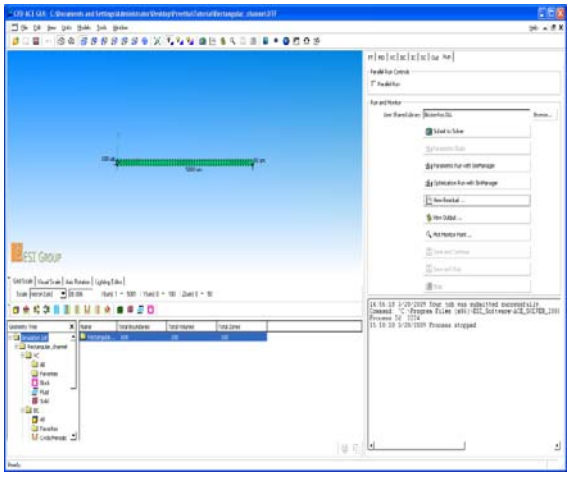
Simulation Done.
The red button on the upper right hand corner is the indication for end of simulation.

We can see that for this case, the output converged in just around 85 iterations although we set total iterations to 300 since the convergence criterion of 1E-06 is met earlier.



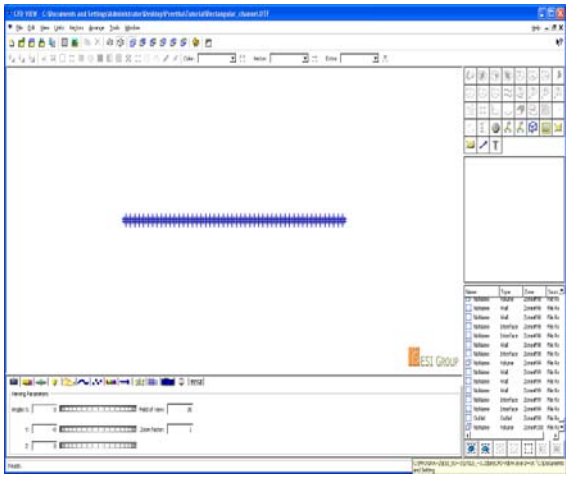
Post-Processing in CFD-VIEW

Click on the CFD-VIEW icon to go directly to CFD-VIEW from CFD-ACE+ solver.

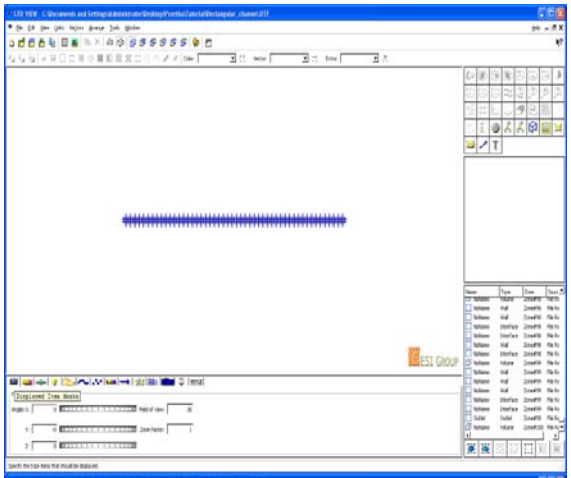


If the file does not load using the CFD-VIEW icon in CFD-ACE+ then,

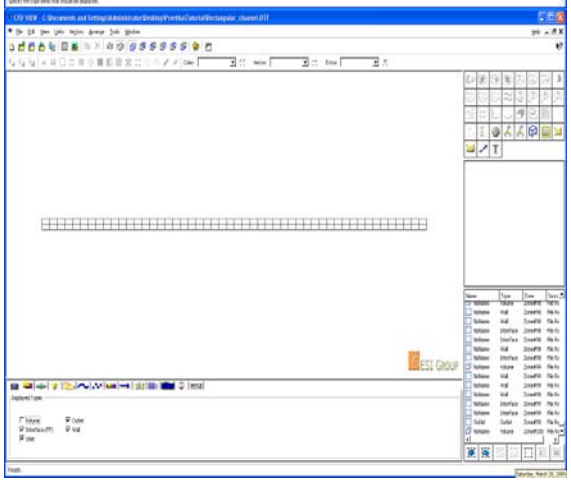
Click *File/Import Additional Data File* from the Menu Bar
Select *DTF/Zone Based* from *Source*
Select *Rectangular_channel.DTF* from file selection box
Click *OK*



Click on the *Displayed Item Masks* icon and check *Volume*



Use the middle scroll wheel to zoom into the model

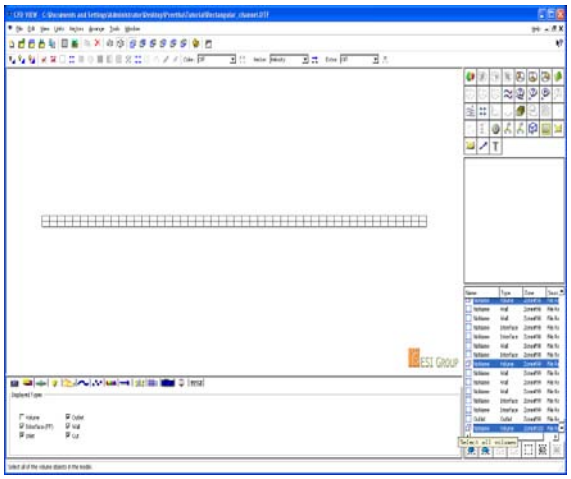


Create a Z-cut:

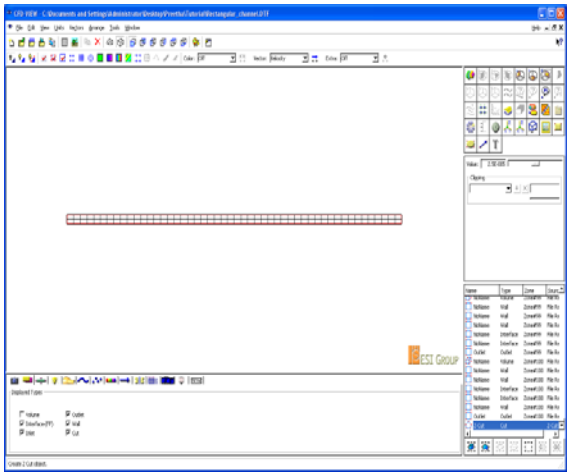
Click *Select All Volumes*

The two volumes in the model get selected.

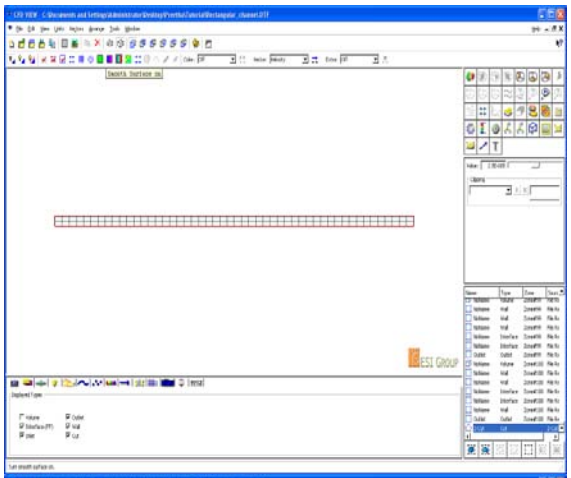
From the *Objects* palette, click the *Z-cut* button.



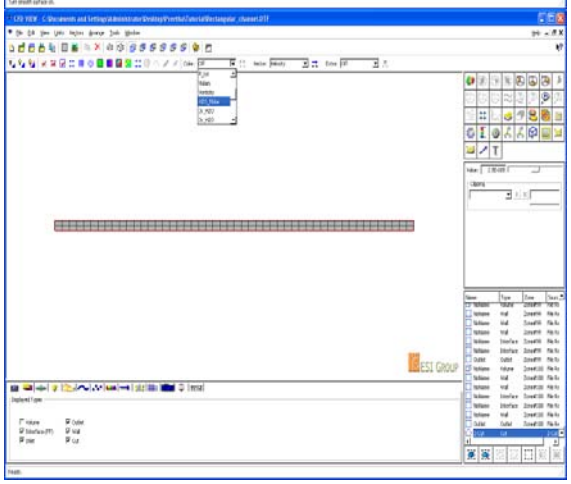
A red outline will appear around the channel. We use the z-cut to visualize the mixing of the two fluids inside the channel.



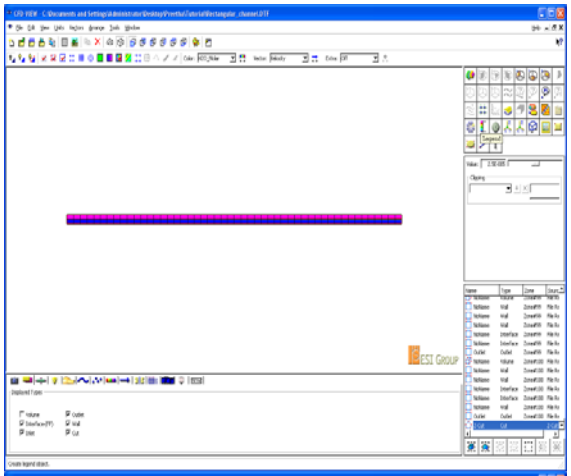
Select the *Smooth Surface On* button from the Visualization panel to apply color to the two fluids flowing in the channel



From the *Visualization Panel*, select '**H_2O_Molar**' from the *Color* pull-down menu

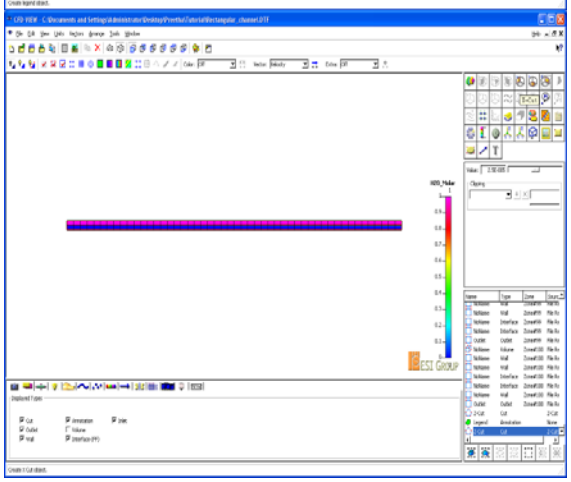


From the *Objects* palette, click the *Legend* button.
Title of legend is determined by *Color* variable



With the Z-cut selected, from the *Objects* palette click on the *X-cut* button.

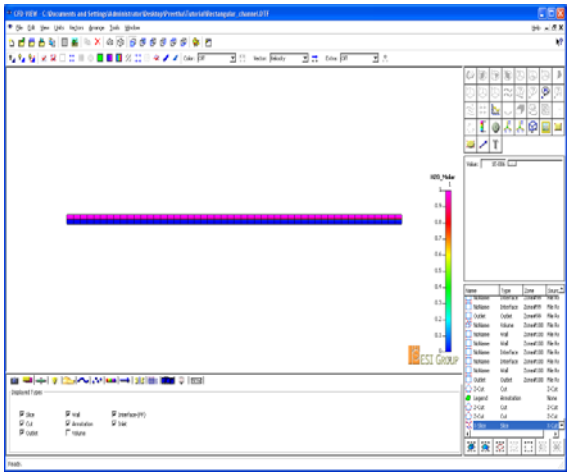
This creates an x-slice across the channel geometry. The chord at the intersection of the z-cut and x-slice is used to plot the molar concentration plot across the channel width at varying distances from the channel entrance.



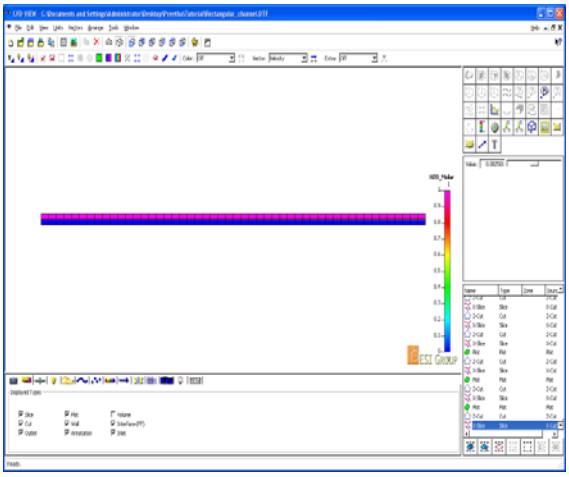
In the *Value* field, change the location of the x-slice to 0.
This corresponds to the channel entrance position.

Note: The units of the *Value* field are in meters.

Next from the *Objects* palette, click on the *Plot* button.



Choose x-slice created change the value now to 0.0025 (half of the channel length)



A plot indicating the variation of the density (***RHO***) appears on the screen.

Change the *Plot Y-axis* field to ***H2O_Molar*** to get the variation of the molar concentration of the two liquids across the width of the channel
i.e,
X-axis = channel width (meters)
Y-axis = dye concentration (0-1)

In the Plot panel click on ***File -->Save Plot***

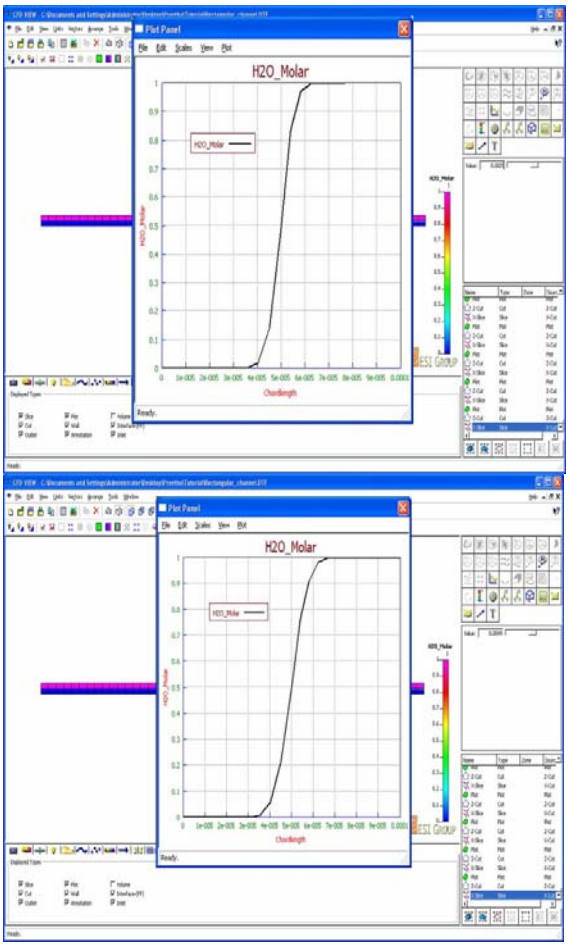
The default file type is **.plt**. Choose the directory you want to save the plot in and save as ***entrance.plt***

Close the plot panel.

Similarly create and save the plot as ***half_channel.plt***

We can see that some mixing has taken place and the ends of the curve are beginning to flatten out.

100% mixing is said to be achieved when the molar concentration curve is a straight line at 0.5 molar across the entire channel width



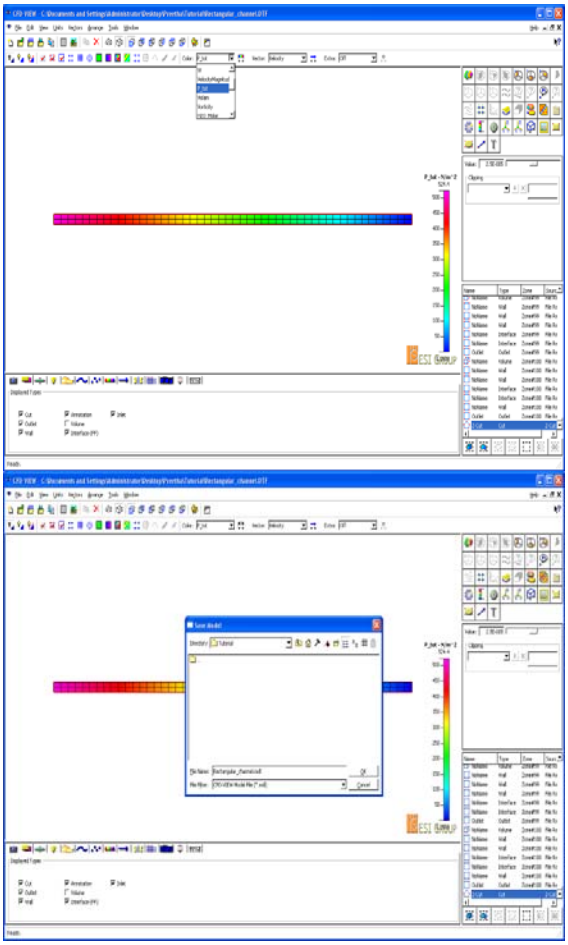
Similarly create a plot at x-value = 0.0049 and save it as ***outlet.plt***

This plot corresponds to the end of the channel.

We can also find the pressure drop across the channel length by selecting the z-cut we created earlier and choosing '**P-tot**' from the *Color* pull-down menu in the *Visualization Panel*

Then click on the legend button in the *Objects* panel. The difference between the upper and lower limit of the legend gives the pressure drop across the channel

Save the file as Rectangular_channel.mdl



The End