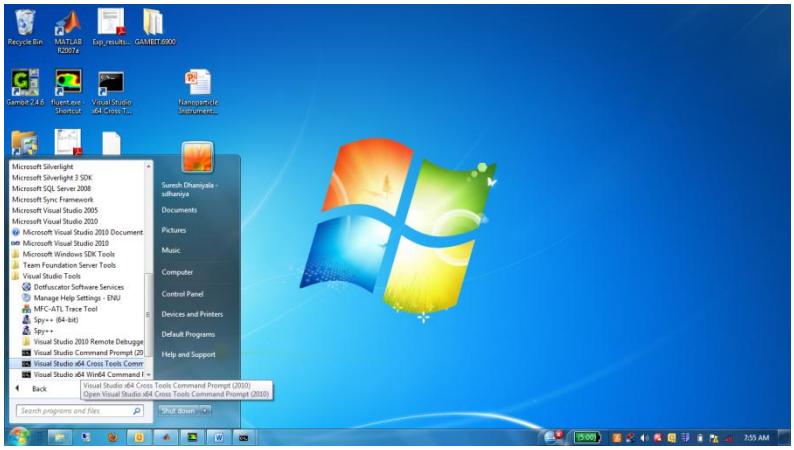
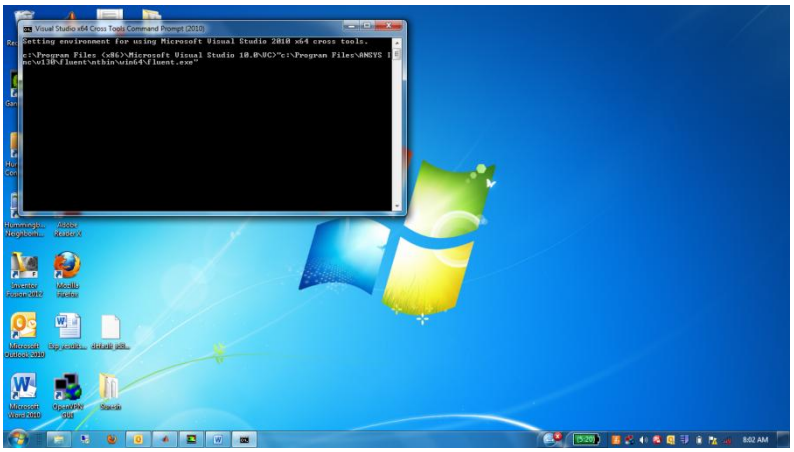
**Procedure to calculate charged particle trajectories in an electric field using FLUENT:**

1) Load FLUENT from “Visual Studio x64 Cross Tools Command Window” (which can be reached by clicking on: START -> Microsoft Visual Studio 2010 -> Visual Studio Tools-> Visual Studio x64 Cross Tools Command Window

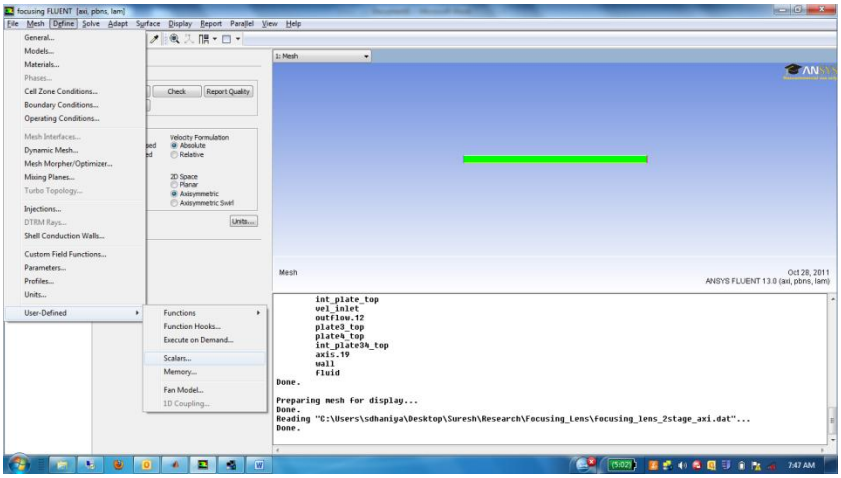


2) In the command window that comes up, run FLUENT. You can type fluent.exe (if the path is set correctly) or the entire path of the FLUENT executable (Most likely -> "c:\Program Files\ANSYS.Inc\ v130 \fluent\ntbin\win64\fluent.exe

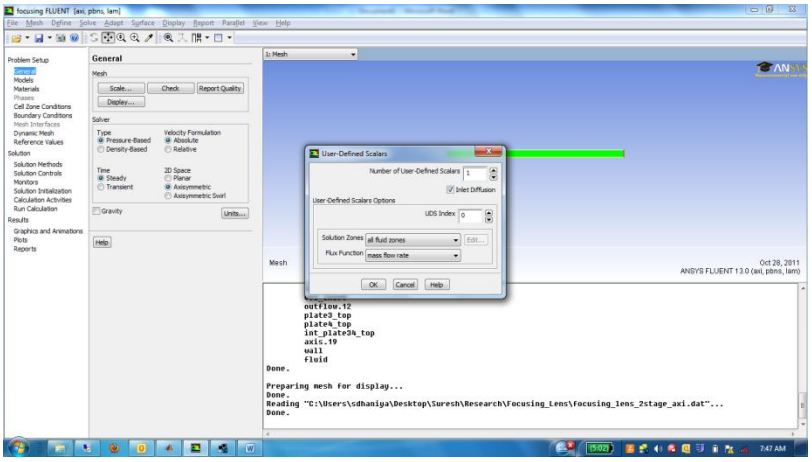
This should load FLUENT. Select the desired options and load your case and data files.



3) For electric field simulations, you need to introduce a scalar. Click on: Define -> User-Defined -> Scalars



4) Set the **"Number of user-defined scalars" to 1**. Click "OK". The variable UDS-0 now represents the potential field in your geometry. For the new variable, FLUENT solves the transport equation,



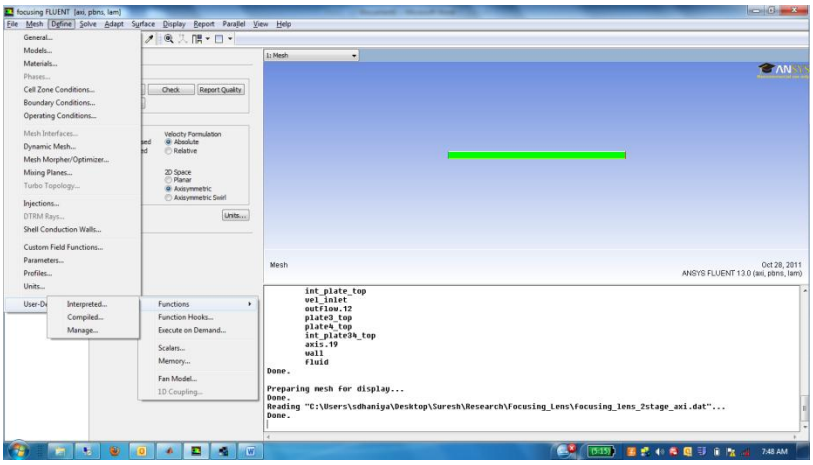
$$\underbrace{\frac{\partial \rho \phi_k}{\partial t}}_{\text{unsteady}} + \underbrace{\frac{\partial}{\partial x_i}\left(F_i \phi_k}_{\text{convection}} \underbrace{- \Gamma_k \frac{\partial \phi_k}{\partial x_i}\right)}_{\text{diffusion}} = \underbrace{S_{\phi_k}}_{\text{sources}}$$

If the problem is steady, the first term ("unsteady") drops out. As we only want to solve the diffusion equation, **select "None" for "Flux Function"** in the User-defined scalar window. The diffusivity coeff ($\Gamma$) is set by default to 1 (this value can be changed under "Define->Materials ->air->UDS-diffusivity). For electric field calculation, the default value of 1 is fine. The source term has a 0 value by default. If need be, this value can be changed under "Define-> Cell Zone Conditions->fluid->edit-> Source terms. For electric field calculation, the default 0 value for the source term is fine.

Thus the equation being solved is: $\nabla^2 \emptyset = 0$.

5) To calculate particle trajectories considering the influence of the electric field, click on:

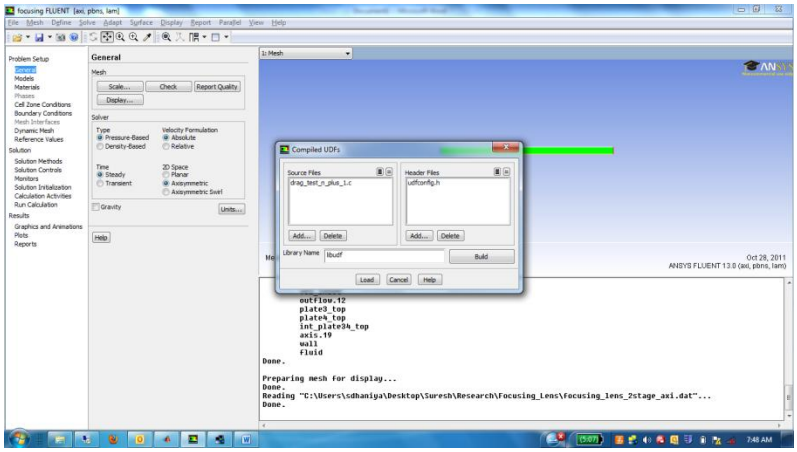
Define->User-Defined -> Functions->Compiled



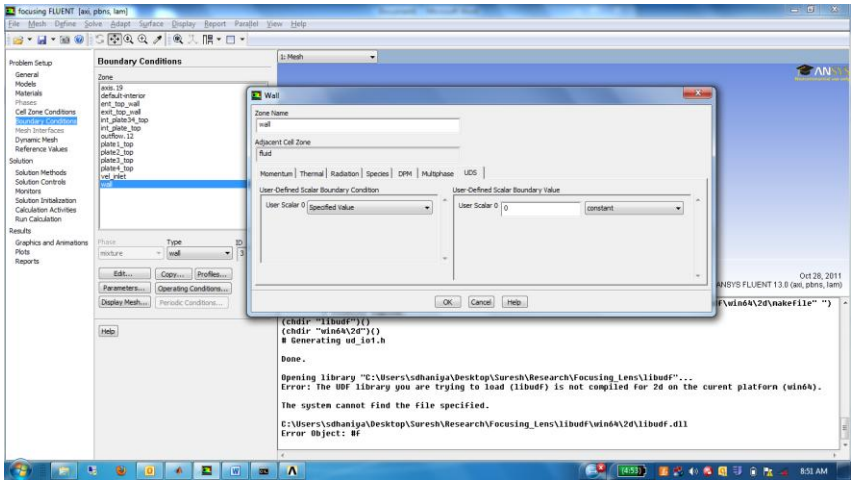
6) In the User-defined functions window, under "Source Files", select the UDF file that has the code for particle trajectory calculation (e.g., "drag_test_n_plus_1.c"). Under the "Header files" option, select udfconfig.h

Click on "Build" and "OK" on the next window that pops up.

If there are no errors, click "Load".



7) Electric field calculation:



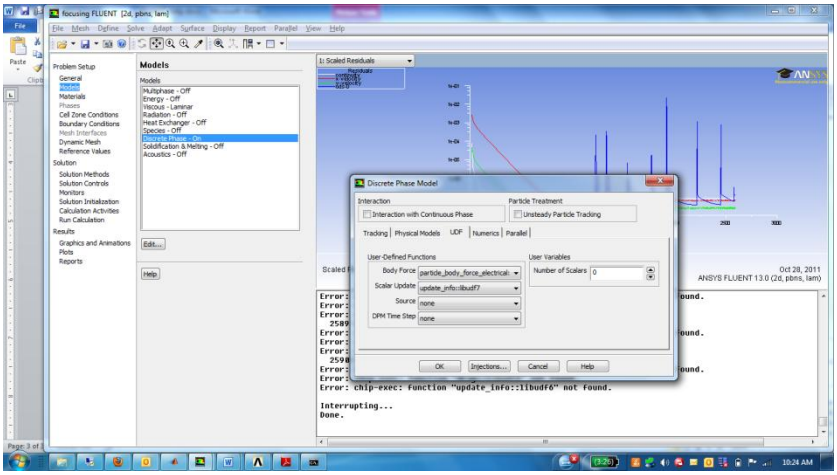
To calculate electric fields, boundary conditions must be selected for the potential function (UDS-0). Under "Boundary Conditions", for the different boundaries, select appropriate boundary conditions for UDS-0.

**For the walls with a known potential (voltage) (conductive surfaces),** for the drop-down menu under "User-defined scalar boundary condition", select "Specified value". Then set the desired value under "User-defined scalar boundary value" (typically the value of the voltage on the different boundaries).

**For insulating walls (and flow inlets and outlets)**, under "User-defined scalar boundary condition", select "Specified flux". Then set the desired value under "User-defined scalar boundary value" (typically 0).

When you click on "Run Calculations", in addition to flows and temperatures, the Laplace equation will be solved with the selected boundary conditions to determine the potential field in the region of interest.

8) Particle trajectory calculations:



Create injections as desired.

To link the UDF for particle trajectory calculation, click on Models-> Discrete-phase->UDF. In the drop-down menu for "Body-Force" and "Scalar-Update", select the udfs that were compiled and loaded under step 6.

9) Run calculations until the residual for UDS-0 is acceptably low.

The particle trajectory calculations should now reflect the net contribution of the drag force and electrical force acting on the particle. Note that the charge on the particle should be set appropriately in the udf code ("drag_test_n_plus_1.c") that was compiled in Step 6.