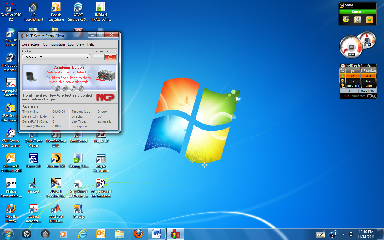
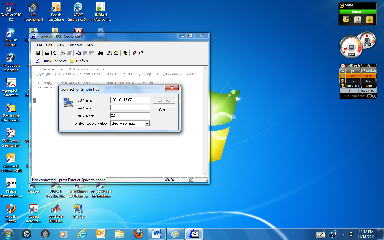
Procedure to start OpenFOAM: by Mohamed Alqadhi

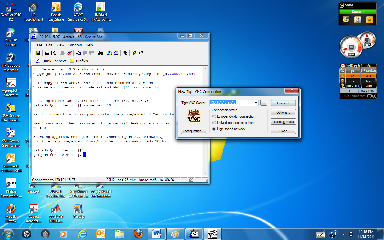
15 November 2011

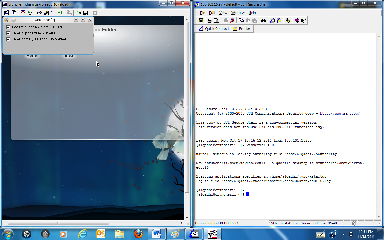
1. Log on to **VPN**

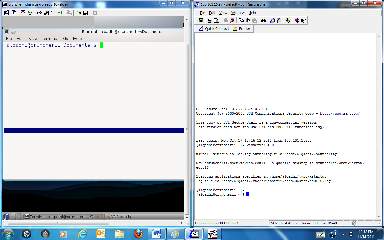


1. Log on to **SSH Secure Shell** using one of the below addresses.
2. 130.101.15.98 for **cruncher I**
3. 130.101.15.97 for **cruncher II**
4. Enter user name and password when prompted.



1. Once logged on to shell, type in **vncserver :10** on the command line.
2. Go to tight VNC and logon to view screen with same password as shell.
3. A screen will pop up to show what it would look like when logged on in front of cruncher I or II.
4. You will have two screens.
   1. One the shell terminal.
   2. And second the cruncher window screen view.



1. Once on the cruncher go to and open up a second terminal called the **terminal emulator** from the start menu on cruncher.
2. Go to shell and open up from the command line the file you want, e.g. **cd OpenFOAM**
3. Once the final file destination is opened on shell, then you can go to the terminal emulator on cruncher and do the same as you did on shell.
4. For example opening **cavity** file for both terminals, shell first, then cruncher second.
5. Next type **paraFoam** in the cavity directory on the terminal in cruncher and paraview will open.
6. There post processing of the model will take place. Preprocessing can be described by the **user guide on OpenFOAM.com website.**