Remote Access and Use Garfield and ANSYS

UTA-HEP/IF-0020
Jul. 17, 2015

Joshua R. Medford
Department of HEP
The University of Texas at Arlington

1. Gaining access to Garfield

Garfield is a computer program for detailed simulations of two- and three-dimensional drift chambers. Garfield offers various types of simulations in order to gain a better understanding on how a particular GEM detector will respond in certain conditions. Along with HEED and Magboltz, this program offers the production of simulations that are able to portray any situation with any parameter that will give the most predictable outcome.

1. First you need to have some form of SCL6 (red hat or Ubuntu – latest version)
2. Then you must request access to tier3-atlas
   a. Email Dr. Mark Sosebee at sosebee@uta.edu to request account creation for the tier-3 system
3. Once you have an account, you will be able to access and use Garfield and ROOT
   a. If you wish to use this account off campus you must download Cisco Anyconnect
   b. Go to http://www.uta.edu/oit/cs/software/vpn/
   c. Download for Linux
   d. Open terminal
   e. 'cd' to the directory of the setup script for the vpn setup
   f. enter the command 'sudo sh vpnsetup.sh'
      i. this will prompt you for your password
   g. search for Cisco and run
   h. connect to 'vpn.uta.edu'
   i. enter your mymav account username and password
4. Now you may connect to the tier-3 system by entering the following command in your terminal 'ssh -l **yourusername** -X master.tier3-atlas.uta.edu' followed by your password
   a. If you are prompted something that ask you if you are sure you want to connect, enter the command 'yes'
5. Now you must setup the environment with the following commands in order to use Garfield
   a. source setup-atlas-sw-cvmfs
   b. source setup-garfield
   c. localSetupROOT
6. To use Garfield you must:
   a. 'cd garfield'
   b. 'cd Garfroot'
      i. You can only run simulations and produce histograms through Garfroot

Note: There will be example files already loaded into directories that are titled to what the simulations correspond to (e.g. 'triplegemgain' will simulate gain in a triple GEM setup with specific ANSYS files). If you wish to obtain more examples or view them, go to http://garfieldpp.web.cern.ch/garfieldpp/ and choose the directory labeled 'Examples.' If you wish to download files, you must first create a directory with the proper name and then use the command 'wget' to obtain the makefile and simulation files/scripts. If you ever have any questions about the simulations, you may contact me (joshua.medford@mavs.uta.edu) or the creator of the simulation(s) that are listed on the respective example page.

2. Editing and Running Simulations

If you wish to edit or run the current example files, you must do the following:

1. Go the directory that is titled with the simulation that you wish to edit/run
2. 'rm x' where rm=remove and x=the name of the file
   a. The file you want to delete will be highlighted in green if you type the command 'ls' in the directory of the simulation you want to run
   b. The file that is the simulation file will be similar in name to the file/script that you edit (e.g. the simulation 'edep' is the file that will need to be removed every time you edit the file titled 'edep.C.'
3. Enter the command 'gedit x' where is x is the name of the script for the simulation file (e.g. you must enter 'gedit edep.C' in order to edit the simulation for primary ionization)
   a. This will open a script that has several values that can be edited
   b. You can find a list of proper values here: http://garfieldpp.web.cern.ch/garfieldpp/documentation/UserGuide.pdf
4. After you make your editions, you must save the file and then close it
5. Now you must create the program again; simply enter the command 'make'
   a. The program will now be recreated with all of the editions you have made
6. To now run this program enter './edep' and the simulation to begin (edep is used as an example, the name of the file depends on the name of the program/simulation you are running)
7. After a while, plots and calculations will be available for you to gather, if the simulation is successful

Note: In order to obtain those plots/graphs there are two things that you can do; take a screenshot or save the file as a jpeg and use a program called WinSCP. This program can be downloaded at http://winscp.net/eng/download.php. This program allows you to access the files on your computer and files from another server (tier atlas) and move and copy them as you please across servers. The host name will be 'master.tier3-atlas.uta.edu.' (You must be connected through VPN!
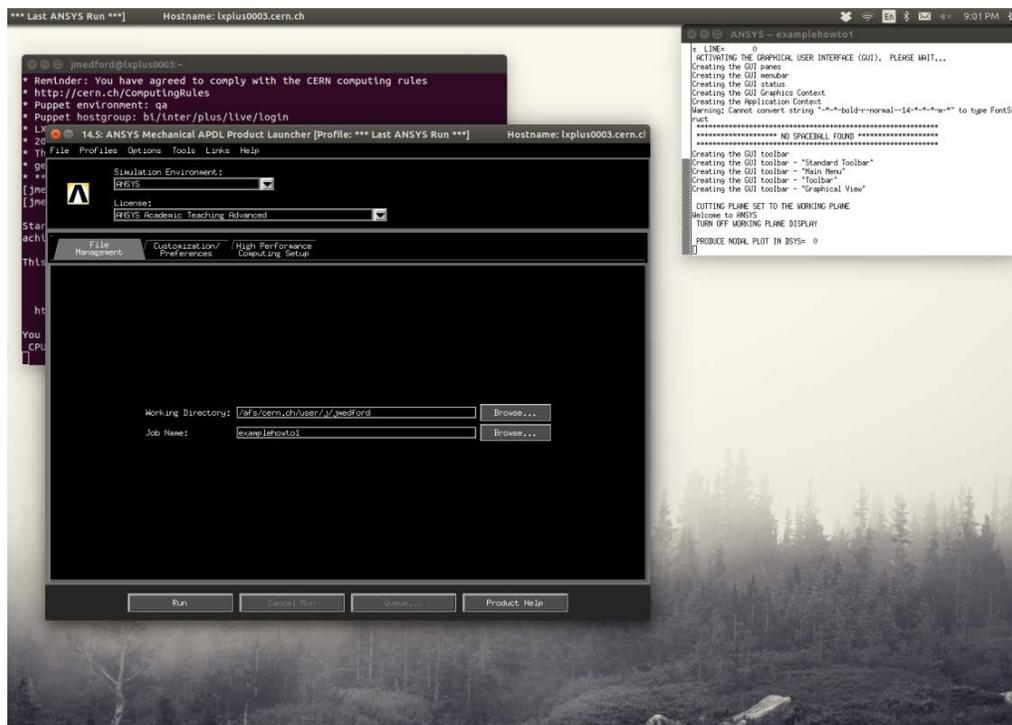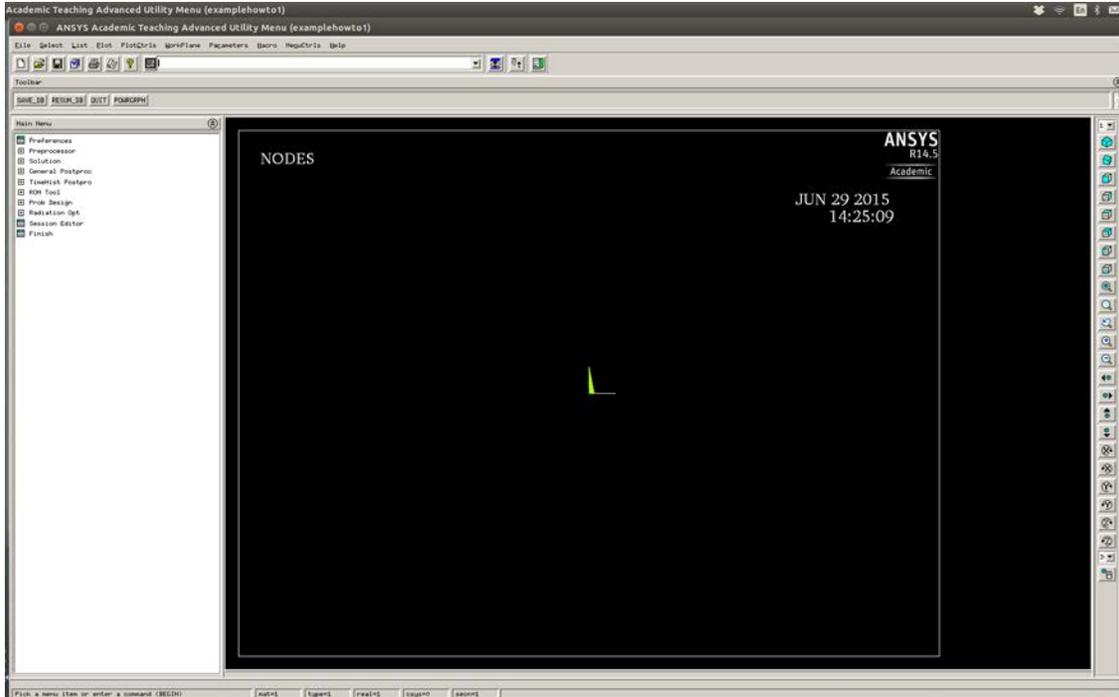
3. ANSYS

There are some files that require the use of geometry files produced from an outside program called ANSYS. ANSYS is engineering simulation software that helps predict how a product will operate in the

real world. Products like GEM detectors can be built with particular layouts to give a better understanding of how they operate before you actually begin to build your prototype.

In order to gain remote access you must do the following:

1. Request to be added to the RD51 email group via Dr. Yu.
2. Gain computer access to CERN
   a. Veronique Wedlake will walk you through
   b. You will need a copy of a passport in order to do this
3. Once you have an account with CERN that has computer access you can now access ANSYS
4. Open a terminal in Linux
5. Enter the command 'ssh –X **yourusernameforCERN**@lxplus.cern.ch' followed by your CERN account password
6. Enter the command '. /afs/cern.ch/project/parc/env/parc_apps.sh'
7. Enter the command 'ansys14'
   a. An older version must be used due to connectivity issues; using an older version is the fastest way to produce files
   b. If you are prompted by something that tells you the version you requested is obsolete, use the oldest version they list (e.g. ansys13 is obsolete, you can only use ansys14, ansys15 or ansys16; you will use ansys14)
8. ANSYS will then begin to initialize
9. After initialization is complete, a product launcher will appear
   a. Keep the default working directory
   b. You can change the job name file to whatever you please
   c. Hit 'Run'
   d. An ANSYS file manager will appear that will track the progress of the program startup
      i. This will take several minutes

10. After loading is complete a new GUI will appear labeled 'ANSYS Academic Teaching Advanced Utility Menu'
11. The best way to use this GUI is to copy and paste an already made or edited script from the Garfield example page into the script header
    a. Once you paste it in (it won't look like it all fits, but it does) press enter and it will begin its simulation/creation process
12. After you enter the script, four files will be created along with an image of the geometry the script calls for
    a. These four files must be saved as '.lis' files (e.g. PRNSOL.lis) so that Garfield can read them
    b. You can transfer these files with the WinSCP program mentioned earlier to the appropriate Garfield directory on the atlas server
        i. The host name will be 'lxplus.cern.ch'

4