HIGH PERFORMANCE COMPUTING WITH PLYMOUTH UNIVERSITY

Using Ansys CFX on the HPC

Author:
Peter Mills
HPC Administrator
University of Plymouth

Using Ansys CFX on the HPC

Getting Started

The following is a guide to using Ansys CFX on the High Performance Computing Clusters.

Before we can consider running any code we need to get our files on to the cluster.
It is recommended that you use an ftp client such as Tunnelier.

The software and the user guide for Tunnelier is available from the HPC website - http://www.plymouth.ac.uk/pages/view.asp?page=33934

Understanding the file format in Ansys

To start a run in Ansys you need to have three files, these are

- The <filename>.sh is the bash script for running the job and contains run commands and other debugging information

- The timelimit.ccl defines the run time for the job if you open and edit this in note / wordpad you can change the time that the job will run for before it stops.

- The <filename>.def file is a definition file that is exported from Ansys CFX this contains all the run and settings data for your CFX model.

When your Ansys job is run, it will generate additional files, these are:-

- The <filename>.out file is the results from Ansys in text form

- The <filename>.res is the graphical results file that the user can view once the run is complete.

Note: The .out and .res are the output files generated by the job run.

These are the files that need to be copied back from the cluster to the local desktop to view their results.

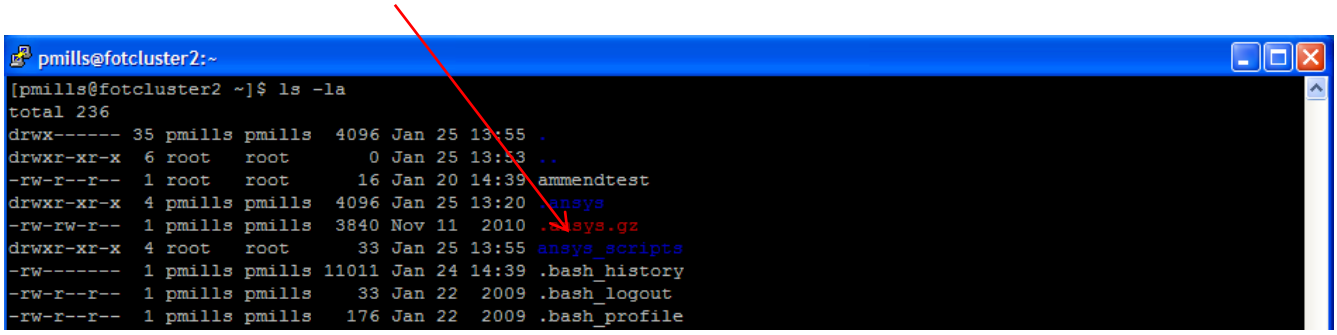
- .res files will only be generated after the analysis has been successfully completed, if no .res appears then the run failed and by reading the .out you will be able to see why the run has failed.

- Once the user has download the .res and .out to the desktop they should be deleted from the cluster as they take up a lot of file space.

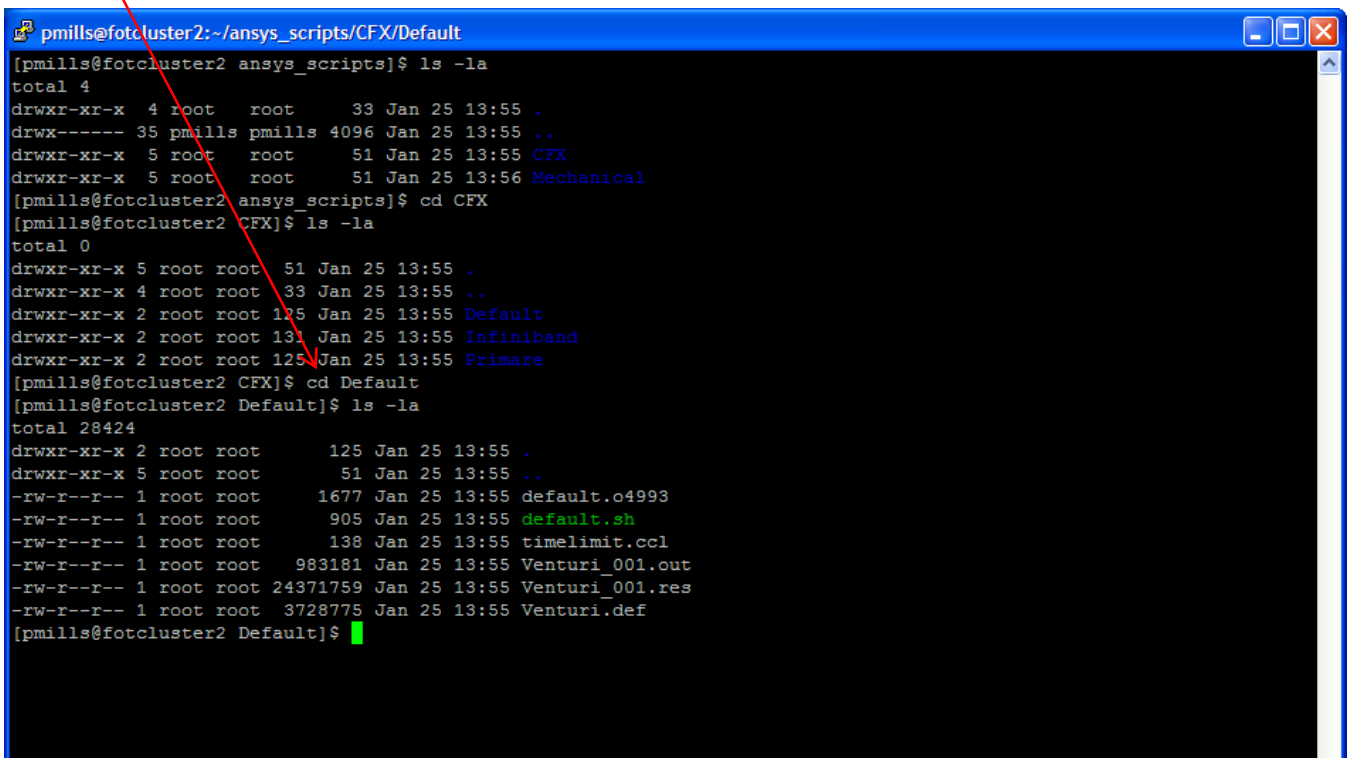
Example Files

Firstly, login to the cluster you will be taken to your home folder.

To start you off, you will find in your home directory a folder ansys_scripts



Now cd into the CFX folder, followed by the Default folder and you will see the files as shown below.



You should now see the files mentioned earlier in understanding the file formats in Ansys.

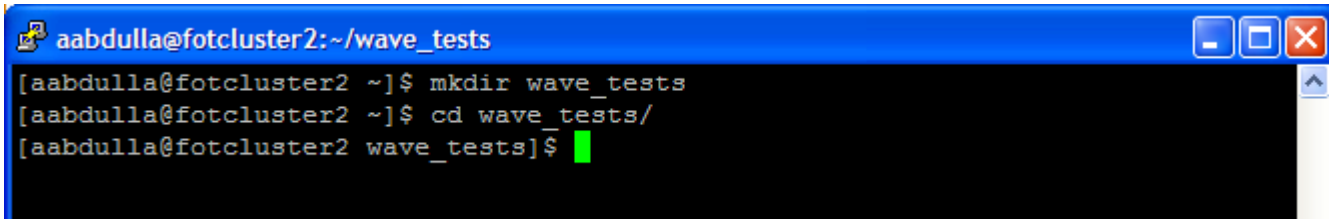
A copy of these files should be placed in what would be the working folder for the users Ansys runs. **Note -** The Venturi_001.out and the Venturi_001.res files are not required.

To copy the required files, firstly go back to the root of your home folder by using the
command cd ~

Now create a new folder from which you will do your Ansys job runs.

Let's assume you wish to name the folder "wave_tests" the user should enter:

mkdir wave_tests > now type cd wave_tests

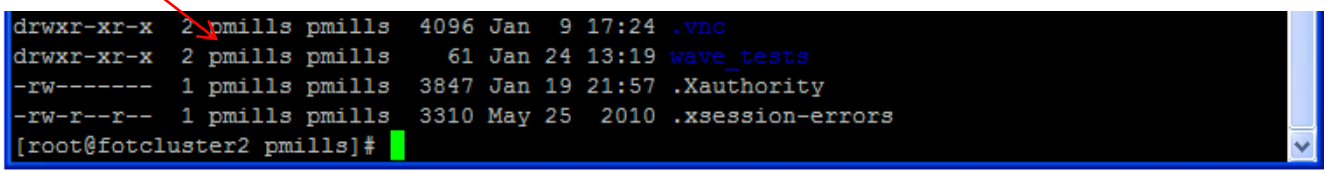


Finally to copy over the required sample files type the following –

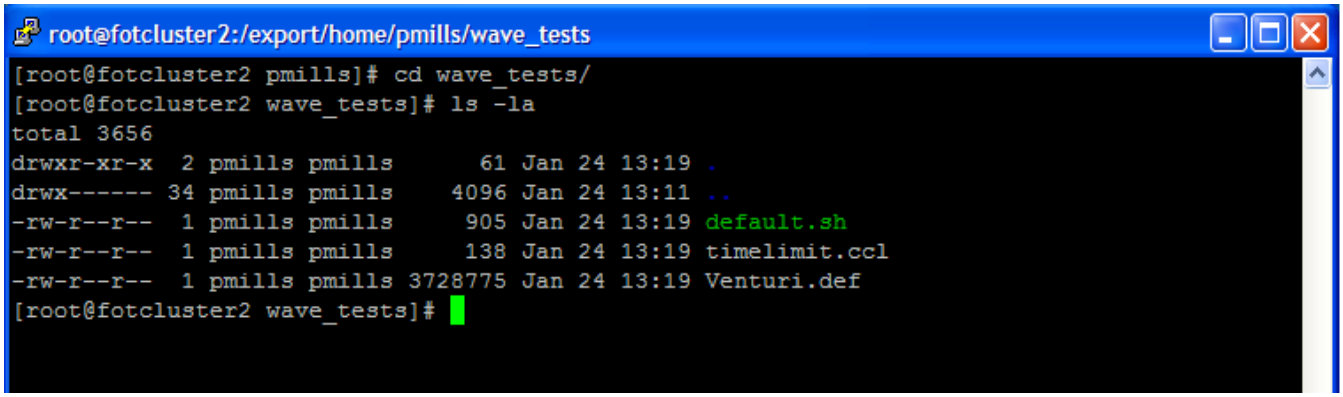
cp /export/home/<username>/ansys_scripts/CFX/Default/default.sh timelimit.ccl Venturi.def /export/home/<username>/wave_tests/

Before we can use these files we need to check the ownership of them.

Screenshot with ownership showing the correct ownership of the folder.



Also the files within the folder are also correct



The files are now ready for use.

Setting up the files ready for job Submission

Once we have the three files in our home folder, we need to edit the script file (default.sh)
So that the file will run correctly and all the results will be outputted to the users home folder.

From within your home folder type - nano default.sh

When the file is displayed browse to the bottom of the file and you will see the following

```
#
# run the cfx job
#
parallel="-par -part $nodes -par-host-list $par_host_list"
cd /home/pmills/wave_tests
/opt/ansys_inc/v120/CFX/bin/cfx5solve -def Venturi.def -ccl timelimit.ccl
$parallel
```

Change the line

- cd /home/pmills/wave_tests

To the users name and the folder from which the script will be run.

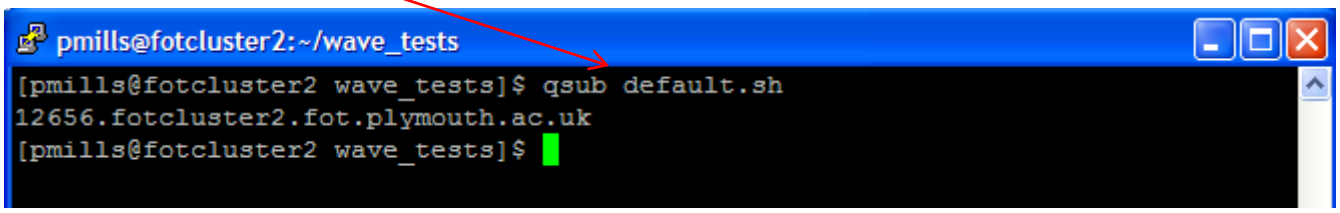
- cd /home/<username>/<folder name>
- Save the file, we are now ready to go.

Submitting an Ansys CFX job

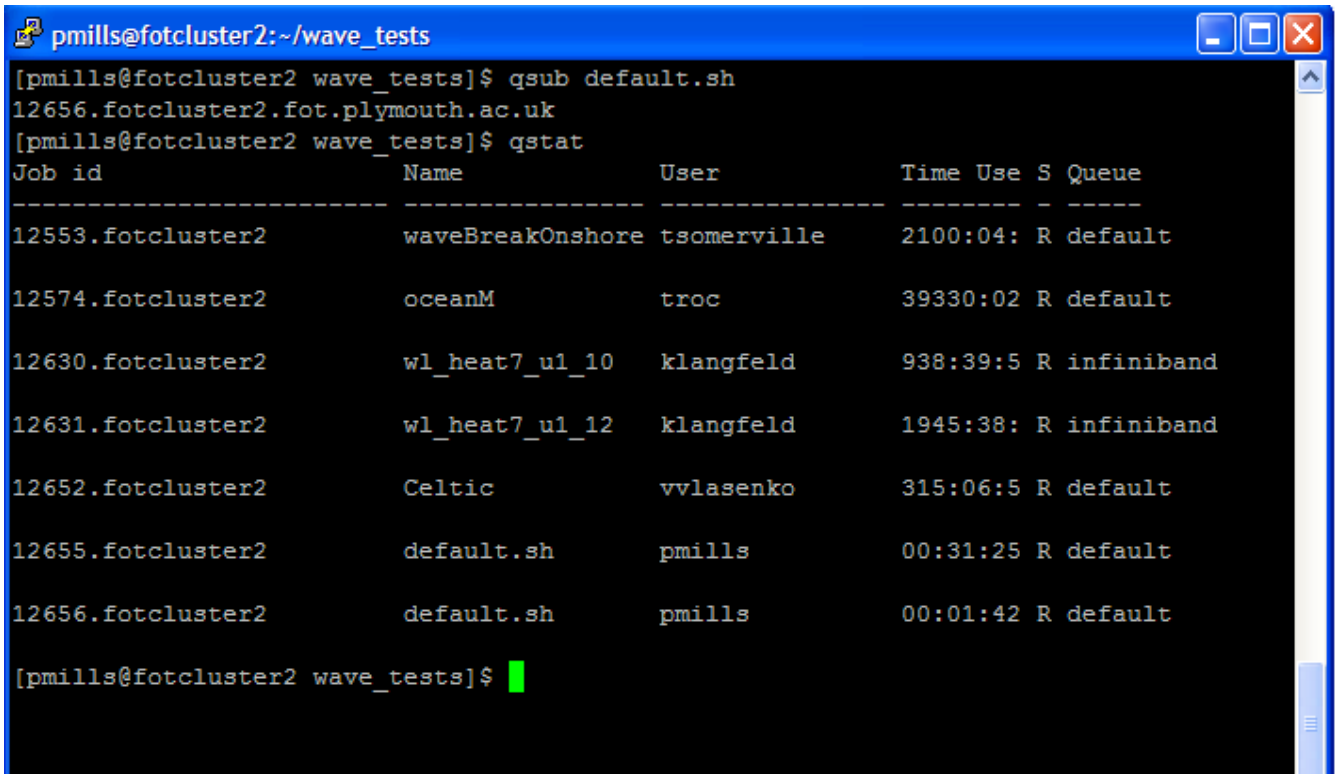
When we submit the job, the scheduler on the cluster will decide which node/s the job will be run on.

To submit a job to the cluster we need to submit the script file using the following format

- qsub <filename>.
  .eg. qsub default.sh

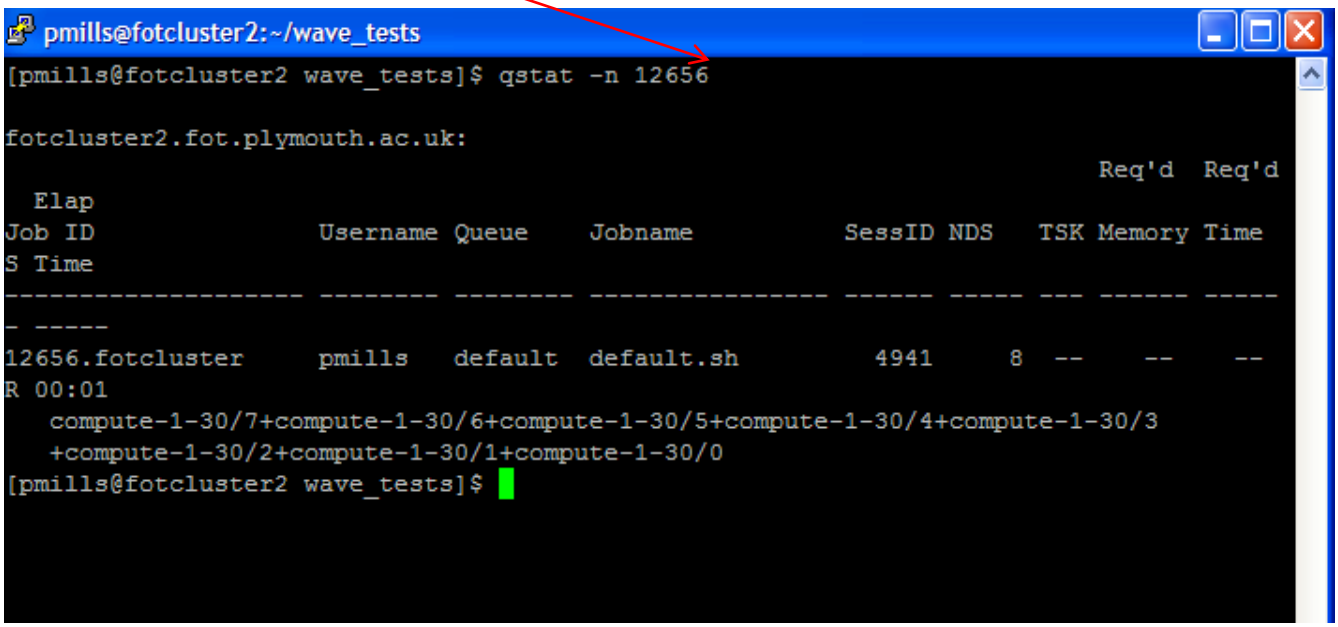


[
To check the progress of our job we use qstat



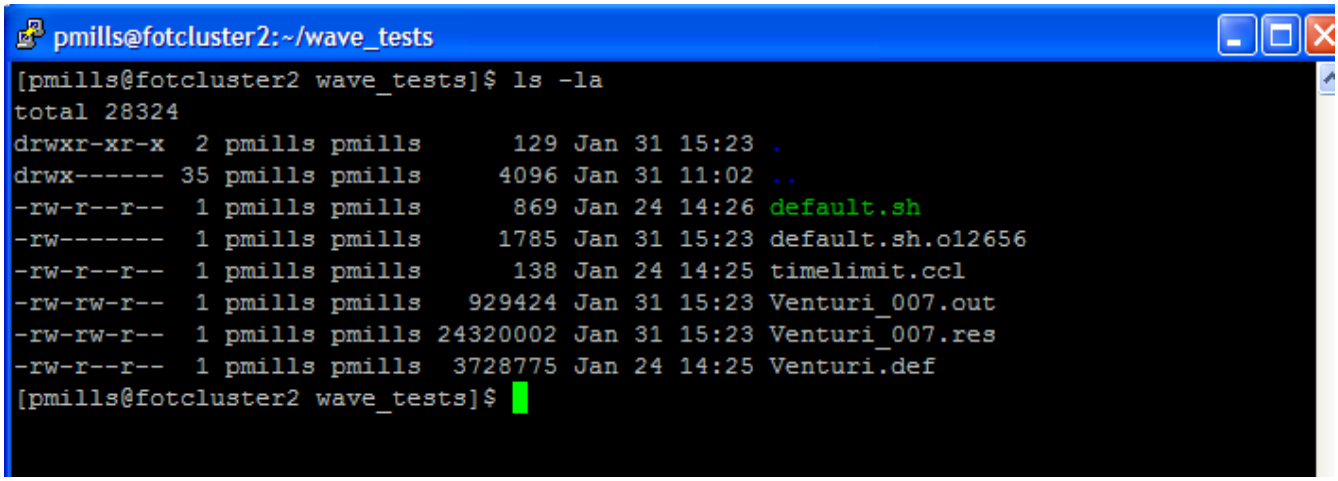
To check to see which nodes the job is running on we use the command

- qstat –n <job number>



Once the job has finished, we can go back to our home folder and we should now have the additional files

1) .res (results) file. This is the file that you would download to your local machine.
2) .out (output)file. Output summary of the job submission.
3) .sh.o12656 file. Copy of script file, with job number appended to the name.



The output and results file can now be copied back to your local machine so that you are able to see the output details.