ANSYS FLUENT

ANSYS FLUENT is available on the cluster nodes but because of space constraints it's not available on the submit host. It can be invoked as fluent after using the module load ansys command.

Documentation

Documentation for ANSYS FLUENT is available via the ANSYS customer support portal (you must register to get access to the portal, and this registration requires FEUP's ANSYS account number which we can't freely distribute - contact us to get this).

Running FLUENT

If you need to use FLUENT interactively, you can use interactive jobs as documented at this page (basically use the command qsub -I).

In both interactive and batch mode, FLUENT should always be invoked with the fluent -g command (the -g disables the graphical display which isn't normally available on the cluster nodes). This will start the FLUENT program which will then prompt for the solver version to use:

```
$ fluent -g
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/bin/fluent -r18.2.0 -g
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/cortex/lnamd64/cortex.18.2.0 -f
fluent -g (fluent "  -alnamd64 -r18.2.0 -t0 -
path/soft/ANSYS/18.2/v182/fluent -ssh")
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/bin/fluent -r18.2.0 -alnamd64 -t0
-path/soft/ANSYS/18.2/v182/fluent -ssh -cx
cfpsmall05.grid.fe.up.pt:37347:34395

The versions available in /soft/ANSYS/18.2/v182/fluent/fluent18.2.0/lnamd64
are:
2d       2d_node  2ddp_host  3d       3d_node  3ddp_host
2d_host  2ddp     2ddp_node  3d_host  3ddp     3ddp_node
 The fluent process could not be started.

version>
```

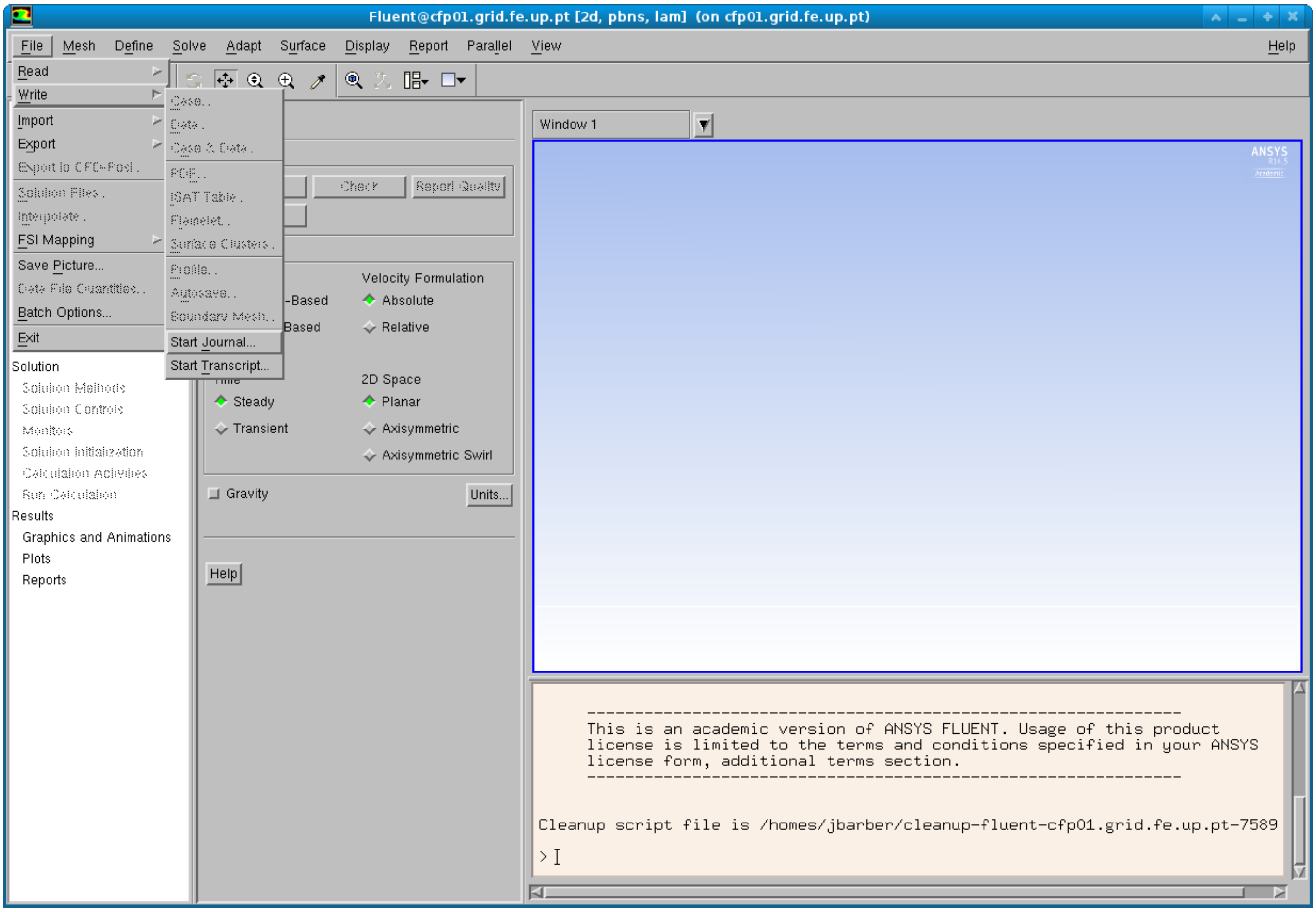
You can skip this prompt by providing the solver on the command line:

```
$ fluent -g 2d
```

which will place you directly in the command line prompt for FLUENT. You can now type your commands to drive FLUENT, or you can use a pre-created journal file.

Journal Files for batch processing

ANSYS FLUENT can be controlled non-interactively through a simple file called the journal. The journal contains a series of instructions in the ANSYS FLUENT TUI language which is described in more detail in Chapter 3 of the ANSYS FLUENT User's Guide. You can either manually create a journal file (with a text editor), or use the FLUENT GUI on your workstation to record the commands to run your analysis on a smaller data set. You do this by first starting FLUENT and then opening the menu File→Write→Start Journal (as shown in the screenshot below) and selecting the file to save the commands in. All of the operations that you do after this will be saved to this file. Once you have finished recording the commands you go File→Write→Stop Journal to make sure they are all recorded in the journal file. You can now modify the saved journal file to use the final data files and copy it to the cluster to run your analysis.



The simplest journal (in the file *simple.journal*) is just:

```
exit
```

you can then reference the journal from the command line using the -i argument to FLUENT:

```
$ fluent -g 2d -i simple.journal
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/bin/fluent -r18.2.0 -g 2d -i
simple.journal
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/cortex/lnamd64/cortex.18.2.0 -f
fluent -g -i simple.journal (fluent "2d -pshmem  -host -alnamd64 -r18.2.0 -
t1 -mpi=ibmmpi -path/soft/ANSYS/18.2/v182/fluent -ssh")
```

```
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/bin/fluent -r18.2.0 2d -pshmem -
host -alnamd64 -t1 -mpi=ibmmpi -path/soft/ANSYS/18.2/v182/fluent -ssh -cx
cfpsmall05.grid.fe.up.pt:39633:44138
Starting
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/lnamd64/2d_host/fluent.18.2.0 host
-cx cfpsmall05.grid.fe.up.pt:39633:44138 "(list (rpsetvar (QUOTE
parallel/function) "fluent 2d -flux -node -alnamd64 -r18.2.0 -t1 -pshmem -
mpi=ibmmpi -ssh") (rpsetvar (QUOTE parallel/rhost) "") (rpsetvar (QUOTE
parallel/ruser) "") (rpsetvar (QUOTE parallel/nprocs_string) "1") (rpsetvar
(QUOTE parallel/auto-spawn?) #t) (rpsetvar (QUOTE parallel/trace-level) 0)
(rpsetvar (QUOTE parallel/remote-shell) 1) (rpsetvar (QUOTE parallel/path)
"/soft/ANSYS/18.2/v182/fluent") (rpsetvar (QUOTE parallel/hostsfile) "") )"

              Welcome to ANSYS Fluent Release 18.2

              Copyright 2017 SAS IP, Inc. All Rights Reserved.
              Unauthorized use, distribution or duplication is prohibited.
              This product is subject to U.S. laws governing export and re-
export.
              For full Legal Notice, see documentation.

Build Time: Jul 25 2017 20:10:41 EDT  Build Id: 10098


     --------------------------------------------------------------
     This is an academic version of ANSYS FLUENT. Usage of this product
     license is limited to the terms and conditions specified in your ANSYS
     license form, additional terms section.
     --------------------------------------------------------------
Host spawning Node 0 on machine "cfpsmall05" (unix).
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/bin/fluent -r18.2.0 2d -flux -node
-alnamd64 -t1 -pshmem -mpi=ibmmpi -ssh -mport
192.168.147.149:192.168.147.149:34345:0
Starting
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/multiport/mpi/lnamd64/ibmmpi/bin/m
pirun -e MPI_IBV_NO_FORK_SAFE=1 -e MPI_USE_MALLOPT_MMAP_MAX=0 -np 1
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/lnamd64/2d_node/fluent_mpi.18.2.0
node -mpiw ibmmpi -pic shmem -mport 192.168.147.149:192.168.147.149:34345:0

----------------------------------------------------------------------------
---
ID    Hostname    Core  O.S.      PID    Vendor
----------------------------------------------------------------------------
---
n0    cfpsmall05  1/8  Linux-64  24620  Intel(R) Xeon(R) E5430
host  cfpsmall05       Linux-64  24438  Intel(R) Xeon(R) E5430

MPI Option Selected: ibmmpi
----------------------------------------------------------------------------
---
```

```
Cleanup script file is /homes/jbarber/cleanup-fluent-cfpsmall05-24438.sh

 +--------------------------------------------------------------------+
 |                    ANSYS Product Improvement                       |
 |                                                                    |
 |   ANSYS Product Improvement Program helps improve ANSYS            |
 |   products. Participating in this program is like filling out a    |
 |   survey. Without interrupting your work, the software reports     |
 |   anonymous usage information such as errors, machine and          |
 |   solver statistics, features used, etc. to ANSYS. We never        |
 |   use the data to identify or contact you.                         |
 |   The data does NOT contain:                                       |
 |   - Any personally identifiable information including names,       |
 |     IP addresses, file names, part names, etc.                     |
 |   - Any information about your geometry or design specific         |
 |     inputs.                                                        |
 |   You can stop participation at any time. To change your           |
 |   selection go to Help >> ANSYS Product Improvement Program        |
 |   in the GUI.                                                      |
 |   For more information about the ANSYS Privacy Policy, please      |
 |   check: http://www.ansys.com/privacy                              |
 |                                                                    |
 +--------------------------------------------------------------------+



>
> exit
$
```

Parallel Jobs

The above examples show FLUENT being used with as single process, it is also possible to use more processes to speed up the analysis.

Adding more processes to the analysis will not necessarily make the analysis run faster. You may need to test various combinations of nodes and processes per node to get the best performance. If your data or analysis changes, then you may have to repeat this testing to find the optimum performance.

You can use more processes by changing the command line to use the -t, -ssh, and (optional) -cnf argument to FLUENT, and by requesting more nodes and processes-per-node from the cluster scheduler (described in more detail here).

You must always use the *-ssh* argument when running a parallel job, even if you're using the GUI.

Let's first cover the FLUENT command. The -ssh argument is required whenever you also use the -t argument to run a parallel job. The -t argument is used to specify how many processes in total you want to use, so if you wanted to use 16 processes you would use -t16. Finally, the -cnf argument is used to specify a text file that contains the hosts to use. The values for -t and -cnf are supplied by the queuing system as environment variables, so you can use the command:

```
$ fluent -g 2d -ssh -t$PBS_NP -i test.journal
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/bin/fluent -r18.2.0 -g 2d -ssh -t1
-i test.journal
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/cortex/lnamd64/cortex.18.2.0 -f
fluent -g -i test.journal (fluent "2d -pshmem  -host -alnamd64 -r18.2.0 -t1
-mpi=ibmmpi -path/soft/ANSYS/18.2/v182/fluent -ssh")
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/bin/fluent -r18.2.0 2d -pshmem -
host -alnamd64 -t1 -mpi=ibmmpi -path/soft/ANSYS/18.2/v182/fluent -ssh -cx
cfpsmall05.grid.fe.up.pt:45127:41088
Starting
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/lnamd64/2d_host/fluent.18.2.0 host
-cx cfpsmall05.grid.fe.up.pt:45127:41088 "(list (rpsetvar (QUOTE
parallel/function) "fluent 2d -flux -node -alnamd64 -r18.2.0 -t1 -pshmem -
mpi=ibmmpi -ssh") (rpsetvar (QUOTE parallel/rhost) "") (rpsetvar (QUOTE
parallel/ruser) "") (rpsetvar (QUOTE parallel/nprocs_string) "1") (rpsetvar
(QUOTE parallel/auto-spawn?) #t) (rpsetvar (QUOTE parallel/trace-level) 0)
(rpsetvar (QUOTE parallel/remote-shell) 1) (rpsetvar (QUOTE parallel/path)
"/soft/ANSYS/18.2/v182/fluent") (rpsetvar (QUOTE parallel/hostsfile) "") )"

              Welcome to ANSYS Fluent Release 18.2

              Copyright 2017 SAS IP, Inc. All Rights Reserved.
              Unauthorized use, distribution or duplication is prohibited.
              This product is subject to U.S. laws governing export and re-
export.
              For full Legal Notice, see documentation.

Build Time: Jul 25 2017 20:10:41 EDT  Build Id: 10098



     --------------------------------------------------------------
     This is an academic version of ANSYS FLUENT. Usage of this product
     license is limited to the terms and conditions specified in your ANSYS
     license form, additional terms section.
     --------------------------------------------------------------
Host spawning Node 0 on machine "cfpsmall05" (unix).
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/bin/fluent -r18.2.0 2d -flux -node
-alnamd64 -t1 -pshmem -mpi=ibmmpi -ssh -mport
192.168.147.149:192.168.147.149:36981:0
Starting
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/multiport/mpi/lnamd64/ibmmpi/bin/m
pirun -e MPI_IBV_NO_FORK_SAFE=1 -e MPI_USE_MALLOPT_MMAP_MAX=0 -np 1
/soft/ANSYS/18.2/v182/fluent/fluent18.2.0/lnamd64/2d_node/fluent_mpi.18.2.0
node -mpiw ibmmpi -pic shmem -mport 192.168.147.149:192.168.147.149:36981:0

-----------------------------------------------------------------------------
---
ID    Hostname    Core  O.S.      PID    Vendor
-----------------------------------------------------------------------------
```

```
---
n0    cfpsmall05  1/8  Linux-64  25502  Intel(R) Xeon(R) E5430
host  cfpsmall05       Linux-64  25320  Intel(R) Xeon(R) E5430

MPI Option Selected: ibmmpi
----------------------------------------------------------------------------
---

Cleanup script file is /homes/jbarber/cleanup-fluent-cfpsmall05-25320.sh

 +--------------------------------------------------------------------+
 |                    ANSYS Product Improvement                       |
 |                                                                    |
 |   ANSYS Product Improvement Program helps improve ANSYS            |
 |   products. Participating in this program is like filling out a   |
 |   survey. Without interrupting your work, the software reports     |
 |   anonymous usage information such as errors, machine and         |
 |   solver statistics, features used, etc. to ANSYS. We never        |
 |   use the data to identify or contact you.                         |
 |   The data does NOT contain:                                       |
 |   - Any personally identifiable information including names,       |
 |     IP addresses, file names, part names, etc.                     |
 |   - Any information about your geometry or design specific         |
 |     inputs.                                                        |
 |   You can stop participation at any time. To change your           |
 |   selection go to Help >> ANSYS Product Improvement Program        |
 |   in the GUI.                                                      |
 |   For more information about the ANSYS Privacy Policy, please      |
 |   check: http://www.ansys.com/privacy                              |
 |                                                                    |
 +--------------------------------------------------------------------+


> exit
$
```

Using more that 4 processes in parallel requires free FLUENT_PARALLEL licenses. If there are no free licenses then the command will fail. The submit host has a command *show_licenses* **which reports the number of free licenses. The** *aa_r_hpc* **license shown by this tool is the same as the FLUENT_PARALLEL license.**

To do the same thing with a batch job and the scheduler, create a script as follows (called *test.sh* in this example):

```
#!/bin/bash
module load ansys
fluent -g 2d -ssh -t$PBS_NP -cnf=$PBS_NODEFILE -i test.journal
```

and on the submit host, run it on the cluster with the *qsub* command:

```
$ qsub -l nodes=1:ppn=2 test.sh
```

As you need more nodes and processes, you can modify the nodes=1 and ppn=2 arguments.

$PBS_NP is the product of the nodes and ppn argument.

A more complicated Journal example

The following journal illustrates processing a “real” data set from the ANSYS examples:

```
/file/read-case-data
/usr/local/ansys/v145/icemcfd/Samples/Visual3_Files/Visual3_Examples/Pipe_Ne
twork/pp
/file/auto-save/root-name pp
/file/auto-save/data-frequency 10
/solve/iterate 100
exit
```

The first line reads the case and data files, the second sets the output to be saved to files in the current directory starting with pp, the third sets the data to be saved every 10 iterations, the forth line starts the solver running for 100 iterations, and the final line stops FLUENT.

The journal file can be executed with the FLUENT 3D solver as follows:

```
$ fluent -g 3d -ssh -t$PBS_NP -cnf=$PBS_NODEFILE -i test.journal
```

Parallel Interactive Jobs

It is possible to run interactive jobs over multiple nodes, with a command such as follows:

```
$ fluent 3d -ssh -t$PBS_NP -cnf=$PBS_NODEFILE
```

Note that you have to know which FLUENT version you want to run - it's probably one of:

- 2d
- 2ddp
- 3d
- 3ddp