

Accessing /data/l4cfd

/data/l4cfd is a large area of disc space set aside on the CIS Linux service to store computational fluid dynamics (CFD) simulations.

In order to make use of this space you need to have been explicitly added to the Unix group normally this is done by sending your ITS username to Grant Ingram but arrangements may vary from year to year.

Using /data/l4cfd

The /data/l4cfd directory is “automounted”, that is it will not appear in the file manager when you are browsing, you must explicitly cd to it.

To do this:

1. ssh into or open a terminal window at an CIS Unix computer
2. type:
cd /data/l4cfd
3. you then need to create a directory to store your files in, i.e. type:
mkdir surname
4. replace surname with your actual surname....
5. then cd to this directory:
cd surname
6. then you can run fluent and icemcfd from the command line

Once the directory is “automounted” it should then be visible from the file manager and the menus in fluent/gambit etc.

Managing files in /data/l4cfd

Although there is a large amount of space on l4cfd it is not infinite, so please carry out the following steps to ensure that it is used efficiently:

- delete old cfd solutions and meshes that you are not using
- use the unix utilities gzip and tar to compress and manage your files
- if you are using the Linux service and you use the file manager to delete files, it doesn't actually delete them it moves them to a directory called /data/l4cfd/.Trash-username – please clear this out regularly
- when you finish your project please clear out your files and give them to your supervisor! It is quite likely that the space will be completely wiped on the 1st of July each year.