

## What is Gerris

Gerris is a Free Software program for solving partial differential equations describing fluid flow. It was created by Stéphane Popinet and is supported by NIWA (National Institute of Water and Atmospheric research) and Institut Jean le Rond d'Alembert.

## Main features

- Solves the time-dependent incompressible variable-density Euler, Stokes or Navier-Stokes equations
- Solves the linear and non-linear shallow-water equations
- Adaptive mesh refinement: the resolution is adapted dynamically to the features of the flow
- Entirely automatic mesh generation in complex geometries
- Second-order in space and time
- Unlimited number of advected/diffused passive tracers
- Flexible specification of additional source terms
- Portable parallel support using the MPI library, dynamic load-balancing, parallel offline visualisation
- Volume of Fluid advection scheme for interfacial flows
- Accurate surface tension model
- Multiphase electrohydrodynamics

## Step 1 - Log in

The example used in this guide is configured to run on the Swansea Sandy Bridge cluster. Connect to `login.hpcwales.co.uk` with your HPC Wales user credentials using your preferred method (e.g. PuTTY from a Windows machine or ssh from any Linux terminal), then `ssh sw-sb-log-001` to connect to the Swansea system.

The steps below involve typing commands (**in bold font**) in the terminal window.

## Step 2 - Load a Gerris module

HPC Wales' modular environment provides access to the Gerris software.

- Load the module

```
module load Gerris/221013
```

- Confirm the loaded module. Loading the Gerris module causes required other modules to automatically be loaded.

```
module list
```

### Step 3 - Create a directory

Gerris can produce large datasets and, when running in parallel, have high IO requirements. For this reason, we will run code using the high-performance Lustre file system mounted at /scratch:

```
cd /scratch/$USER
mkdir -p geris
```

### Step 4 - Obtain a test case

We'll use the supplied Gerris3D example available at:

```
/app/environment/Gerris/221013/example/3.1_rv_tangaroo
```

Let's copy files to our run directory using:

```
cd /scratch/$USER/geris
cp /app/environment/Gerris/221013/example/3.1_rv_tangaroo/* .
```

### Step 5 - Submit a job

Now the current directory should contain all required files to run the Gerris3D example. Inspect the job script *hpcw\_rv\_tangaroo.q*. Gerris has a number of useful features for parallel jobs that are beyond the scope of this document, but here we use the partitioning capabilities to configure and run a parallel task. Please see the Gerris documentation for more on these capabilities.

- Submit the job using: `bsub < hpcw_rv_tangaroo.q`
- Check the job queue using: `bjobs`

This script reserves and runs on 2 processors.

- When completed, script output can be found in a file called *Gerris\_Tangaroo.o.<JobID>* (where <JobID> is the unique ID generated by the queuing software).
- The script performs a number of steps. As output there will be a large (16GB) *paralleltangaroo.output* from the simulation itself, which has subsequently been visualised into the form of an MPEG movie through the use of *gfsview-batch3D* and *ppm2mpeg*. The MPEG movie can be downloaded and viewed.
- The Gerris documentation should also be referred to in order to understand the visualisation configuration in the *sections.gfs* file as used by *gfsview-batch3D*.

## References

- The Gerris web site, available at [http://gfs.sourceforge.net/wiki/index.php/Main\\_Page](http://gfs.sourceforge.net/wiki/index.php/Main_Page)
- Focussed guide to parallel use of Gerris, at:  
[http://gfs.sourceforge.net/wiki/index.php/Gerris\\_in\\_parallel](http://gfs.sourceforge.net/wiki/index.php/Gerris_in_parallel)
- The Gerris tutorial, at <http://gfs.sourceforge.net/tutorial/tutorial/tutorial1.html>