

arb

code version 0.24

Dalton Harvie

December 12, 2010

1 What's new?

- 12/12/10 The GNU GPL'd sparse linear solver [UMFPACK](#) written by Tim Davis is now interfaced to arb. An easy download and setup script is included.
- 25/11/10 The run procedure has been simplified for version 0.24. Basically you don't have to run `setup_equations` separately anymore - all meta-programming, compilation and running is handled by the single command `./arb`, run from the working directory.

2 What is arb?

arb solves arbitrary partial differential equations on unstructured meshes using the finite volume method. The code is written in fortran95, with some meta-programming done in perl with help from maxima.

The primary strengths of arb are:

- All equations and variables are defined using 'maths-type' expressions written by the user, and hence can be easily tailored to each application;
- All equations are solved simultaneously using a Newton-Raphson method, so implicitly discretised equations can be solved efficiently; and

- The unstructured mesh over which the equations are solved can be composed of all sorts of convex polygons/polyhedrons.

arb requires a UNIX type environment to run, and has been tested on both the Apple OsX and ubuntu linux platforms. Certain third party programs are used by arb:

- A fortran compiler; ifort and gfortran are supported;
- [maxima](#);
- perl;
- A sparse matrix linear solver: [UMFPACK](#), [pardiso](#) and (currently a single) Harwell Subroutine Library routines are supported;
- Some general linear algebra routines: Lapack and the Numerical Recipes in Fortran Library are supported;
- [gms](#); and
- The core unix routines tar, gzip and rsync.

Further details about how to get and install this software are given in section [3.2](#).

arb is copyright Dalton Harvie (2009,2010), but released under the [GNU General Public License \(GPL\)](#). Further details of this license can be found in the license directory once the code has been unpacked.

3 Working with arb

3.1 Managing arb's files

All routine setup, compiling and running of arb is conducted from a working directory which contains a specific structure of files and subdirectories. Two routines, pack and unpack, are provided to automate the process of 'transporting' arb and creating this required file/subdirectory structure.

3.1.1 Unpacking the code

To create a new version of arb, a new working directory should have been obtained containing the four files

- `archive.tar.gz`
- `unpack`
- `README`
- `LICENSE`

Using the command

```
./unpack
```

from within this directory will unpack the archive, creating the necessary file/sub-directory structure.

If you're impatient to get things going (who isn't?), you next need to check that the required third-party software is installed, as detailed in section [3.2](#).

3.1.2 The working directory and file structure

Once the archive is unpacked the working directory will contain the following subdirectories and files/links:

- `src` directory: contains the main fortran source code of arb, and the `makefile` necessary to build everything (except for the contributed libraries). It also contains the meta-programming perl script `setup_equations.pl` and a template file `equations_module.f90` which are used to create the fortran file `equations_module.f90` within `build` which is specific to each problem.
- `src_equations/contributed` directory: This directory may/should/can contain contributed third party code that can be used by arb, along with associated interface modules — for example, linear solver routines. There is a separate subdirectory for each package. Each directory contains error

handling modules that handle runtime cases where the third party routines are not available, and also some brief installation instructions. Most directories work on the drop-box principle — if the required files are available then they will be included in the arb executable.

- `build` directory: all building is done within this directory.
- `tmp` directory: temporary files are stored within this directory, including the `file setup/debug` which is used for debugging the equation setup.
- `output` directory: output files from a simulation are placed in here.
- `misc` directory: contains miscellaneous files — right now a simple script `create_mesh` which builds a `.msh` file from any `.geo` file in the working directory.
- `doc` directory: contains documentation including this manual.
- `gmsh` directory: generally the gmsh structure (`.geo`) and mesh (`.msh`) files are stored in this directory. Examples that are detailed in this manual (section 6) are stored in the `gmsh/manual` subdirectory.
- `examples` directory: example problem-specific files `equations.in` and `constants.in` are stored here. Examples that are detailed in this manual (section 6) are stored in the `examples/manual` subdirectory.
- `packer` directory: contains various files specific to the pack and unpack operations. A history file within this directory records the distribution history of the specific code version.
- `license` directory: contains license details, including the (GNU GPL) license under which arb is released.
- `equations.in` and `constants.in` files: these files and an associated `.msh` or `.geo` file contain all the problem-specific information required for a particular simulation. `equations.in` is read by `setup_equations` when creating the equation-specific fortran code `equations_module.f90` within `build`. `constants.in` is read by both `setup_equations` and by the executable fortran code `arb`. It contains constant and numerical simulation data which can be (mainly) edited without requiring `arb` to be recompiled.
- `arb` script: this shell script is a wrapper script for setting-up, making and running `arb`. You can pass options to this script to control the compilation process — for example choose between the `gnu` and `intel` compilers. It also

performs simulation restart file management (not implemented!). This script works out when the equation meta-programming has to be redone or not, and when recompilation is necessary. So if in doubt, just type `arb.....`

- `pack` file: this shell script is used to pack up a simulation.

3.1.3 Packing the code

To pack a simulation ready to transport or backup, use the command

```
./pack
```

from within the working directory. This will create a subdirectory with a name of the form `arb_[date]_[version]` which contains all files necessary to run `arb`. Following the command with a name, as in

```
./pack a_name
```

will create the archive in a subdirectory named `a_name` instead of the default.

The script `pack` accepts a number of options. By default only files within the `examples` or `gmsh` directories that are specific to this manual are included in the archive. The options `--example`, `--gmsh`, `--misc` or `--all` specify that all files within either the `example`, `gmsh`, `misc` or all three directories are contained within the archive. Including the `gmsh` option may be necessary to transport a particular `.msh` or `.geo` mesh file with the other files. Also, by default only source code within the `src/contributed` directory that is not subject to a non-free third party license is included in the archive. Using the options `--contributed` or `--all` causes all files in these directories to be archived. Using `--distribute` means that no third party software is included in the archive. The option `--build` means that all files in the `build` directory will be included in the archive. This may be useful if you want to transport the simulation to another machine that may not have `maxima` installed (for example).

Other options to `pack` include `--notar` and `--help`.

3.2 Checking for prerequisite software

A number of third-party applications must be installed in order to use `arb`.

3.2.1 Maxima

Equation generation is performed using the Maxima Computer Algebra system. It is released under the [GNU General Public License \(GPL\)](#) and is available for free.

To check whether you have it installed already try typing `maxima`. If the program is installed you will enter a symbolic maths environment. Then check that the command `load(f90);` finds these libraries, and then quit using `quit();`.

If there is something missing then on ubuntu linux the command `apt-get install maxima maxima-share` should sort things out. On OSX maxima can be installed using the [macports](#) software, using `port install maxima`. Alternatively, a precompiled version for the mac is available from [sourceforge](#), and really this is the much easier way of getting things going. If you do use the binary version then you'll need to get the executable command (ie, `Maxima.app/Contents/Resources/bin/maxima`) into your path.

3.2.2 Perl

Most likely you already have a version of perl installed.

3.2.3 A fortran compiler and the lapack libraries

Two different compilers have been tested with arb. The [Intel Fortran compiler](#) `ifort` is probably the faster of the two. It also includes the Intel Maths Kernel library which itself includes the excellent Pardiso routines (see section below) and lapack libraries: however this compiler is not free except on linux and even then, only under specific non-commercial circumstances. The non-commercial download site for `ifort` is [here](#). Make sure you install both the compiler and MKL (Math Kernel Libraries).

The GNU compiler `gfortran` is an easier option to get going, and is freely available on both the OSX and linux platforms. It does not include the Pardiso routines but with the new interface to the UMFPACK routines this isn't a tremendous disadvantage.

On ubuntu linux `gfortran` and the lapack/blas libraries can be installed with the command `apt-get install gfortran liblapack-dev libblas-dev`. On OSX `gfortran` can be installed using the macports software and the command

port install gfortran. Binary versions of gfortran are also be available from the web for OsX, for example from [here](#). Binaries are easier to install, but check that once installed the gfortran executable is in your path - that is, typing gfortran at the command line should find the compiler. I think that the lapack/blas libraries get installed on OsX when you install the Xcode development tools.

Compiler choice is made automatically by the arb script (defaulting to ifort if it exists, otherwise using gfortran). This can be overwritten with the arb options `--compiler-gnu` or `--compiler-intel`.

ifort versions of 11.1.069 and newer (not version 2011 yet) have been tested on both linux and OsX. gfortran version 4.2.4 on ubuntu 8.04 doesn't seem to work on some computers (internal compiler error) whereas version 4.4.3 on ubuntu 10.04 does. Version 4.6 on osX also works fine. On ubuntu 8.04 you could try downloading a newer binary version of gfortran but I haven't tried this.

3.2.4 **Pardiso**

The pardiso sparse linear matrix solver is included as part of the Intel Math Kernel Library which is packaged with the [Intel Fortran compiler](#) (see above).

If using the intel compiler then this solver will automatically become available (check the initial output when running arb to see if this has been found).

3.2.5 **UMFPACK**

The [UMFPACK](#) sparse linear solver is part of the suitesparse collection of sparse matrix routines written by Prof. Tim Davis. It is written in c and released under the GNU GPL (see [conditions here](#)).

UMFPACK depends on '[METIS - Serial Graph Partitioning and Fill-reducing Matrix Ordering](#)'. METIS is freely available but not free distributable. For more details see the [conditions here](#).

The installation process for the UMFPACK/METIS combination has been automated so that it can be easily used with arb. To install these packages and compile them in a form that is suitable for arb:

```
cd src/contributed/suitesparse
make
```

The `make` command will download the latest version of UMFPACK and version 4.0 of METIS (using `wget`), and then compile these libraries using the `gcc` compiler. A wrapper script for using UMFPACK from fortran (included with UMFPACK) will also be compiled. The files will be placed in the `src/contributed/suitesparse` directory, so will not overwrite any alternative suitesparse or metis libraries already on your system. You need to have `gcc` installed on your system.

If all goes well the following files will be placed in `src/contributed/suitesparse` to be used by `arb`:

```
libamd.a
libcamd.a
libccolamd.a
libcholmod.a
libcolamd.a
libmetis.a
libumfpack.a
umf4_f77wrapper.o
```

All of these files are required for UMFPACK to successfully run.

Note that the compilation is dependent on the type of machine architecture and the libraries will need to be remade if transferred from one machine to another - do a `make clean` and then a `make` again.

To remove the compiled libraries and files type

```
make clean
```

This will leave only the downloaded files (ready to be reused). To remove the suitesparse and metis downloads as well, type

```
make clean_all
```

`arb` has been tested with the following combinations of `umfpack` and `metis`:

- UMFPACK.tar.gz 02-Jun-2010 11:46 and metis-4.0.tar.gz (4.0.1)

3.2.6 Harwell Subroutine Library

TODO.

3.2.7 Numerical Recipes in Fortran 77

TODO.

3.2.8 Gmsh

While not integral to the arb code, the mesh and data format which arb uses is that developed for gmsh. Gmsh is a mesh element generator which can be run using scripts or via a graphical interface and can be used for post-processing (visualisation) too. Gmsh uses the [GNU General Public License \(GPL\)](#).

There is some great introductory material available on the [gmsh website](#) on the use of this program, particularly these online [screencasts](#).

3.3 Running a simulation

3.3.1 The super quick guide: A heat conduction simulation

Once the code is unpacked you should be left in the working directory. Copy the three required input files from the examples directory, and create the .msh file using:

```
cp examples/manual/heat_conduction_around_ellipse/*.in .
cp examples/manual/heat_conduction_around_ellipse/*.geo .
misc/create_msh 2 1
```

The last command accepts two arguments; the first is the number of dimensions, while the second controls the relative cell size — the smaller the parameter, the smaller the mesh element size and greater the number. The file `surface.msh` should now be present in the working directory.

Next edit the file `constants.in` to specify your choice of linear solver. If you are using the intel compiler with MKL libraries a sensible choice is:

```
LINEAR_SOLVER "INTEL_PARDISO" # pardiso solver contained  
in intel mkl library
```

If using the gnu compiler then you'll probably need to use the UMFPACK routines:

```
LINEAR_SOLVER "SUITESPARSE_UMF" # suitesparse umf solver  
(UMFPACK) by Timothy A. Davis
```

Now type

```
./arb
```

If all goes well the simulation will run and output will be produced in the directory output. To view the output type

```
gmsht output/output.msh
```

You should see the temperature field around a heated ellipse.

3.3.2 A more detailed guide: Newtonian fluid flow

The information about running a simulation here is presently out of date, but there is some useful info about the mesh and input file structure.

To discuss the working method in more detail we use the example of steady-state flow of a Newtonian fluid around a cylinder that is contained in a 2d channel.

There are 5 basic steps to setting up and running an arb simulation:

1. **Create a mesh:** Geometry definition and mesh creation can be performed in gmsht. The domain geometry details are stored in a .geo file, which can be created by hand (file editing) or with the help of the gmsht GUI. An example .geo file is included in Section 6.1 for the considered case of the 2d channel containing a cylinder. This file is included in the gmsht/manual/2d_channel_with_cylinder/ directory.

Once a geometry (.geo file) has been created, gmsht can mesh this to produce a .msh file which is the file that is read in by arb. For the

considered test case place the generated mesh file `surface.msh` in the `gmsh/manual/2d_channel_with_cylinder/` directory. When creating a 3D mesh be sure to optimise the mesh after creation (greatly improves mesh quality).

`arb` uses the concept of 'regions' to locate various equations (for example boundary conditions, domain equations etc) and these must be defined in the `.geo` file prior to meshing. Regions have names that are delimited by the `<` and `>` signs: for example `<inlet>` and `<outlet>` (generally any user-defined names are delimited this way in `arb`).

In the example geometry file the boundary regions `<inlet>`, `<outlet>` and `<cylinder>` have all been defined as physical entities. The area region of `<flow domain>` which contains all mesh cells within the flow domain has also been created. This region is necessary as by default, `gmsh` does not write out mesh cells to a mesh file unless they are part of a physical entity. Note that certain region names are reserved: see section 4.3 for more details.

2. **Edit** `equations.in` and `constants.in`: Aside from the mesh file, all information that is specific to a simulation is contained in these two `.in` files. `equations.in` is read only by the `setup_equations` perl script when performing the fortran 'meta-programming'. `constants.in` is also read by this perl script, but is also read by the `arb` fortran executable each time a simulation is run. In general physical constants that would change regularly between similar simulations should be defined in `constants.in` as their values can be changed without having to rerun the meta-programming script `setup_equations`. On the other hand all equations, which are processed by `maxima` via `setup_equations`, must be defined in `equations.in`. Hence any change to `equations.in` requires `setup_equations` to be rerun and the fortran executable `arb` remade.

Note that some simulation specific information such as the mesh file name, newton solver convergence tolerance, number of newton iteration steps and dimensions of the physical problem are defined in the `constants.in` file and read directly by the `arb` executable: Hence these can be changed without rerunning `setup_equations` or recompiling `arb`. Similarly any region definitions contained in `constants.in` are only read by the `arb` executable and can be changed without rerunning `setup_equations`.

Section 6.1 gives example `equations.in` and `constants.in` files for the considered test case of steady-state fluid flow around a cylinder. These files are given in `examples/manual/steady_state_channel_flow_with_cylinder` and should be copied to the working directory. The quoted string after the

READ_GMSH keyword in `constants.in` should refer to the location of the mesh file, relative to the working directory.

Further details regarding the syntax of both `equations.in` and `constants.in` can be found in Sections 4 and 5.

3. **Run `setup_equations`:** From the working directory running

```
./setup_equations
```

reads both `equations.in` and `constants.in`, and using maxima, creates the fortran source code file `src/equations_module.f90`. `setup_equations` writes some progress information to the screen - if all goes well this output will end with the statement 'success'.

If the script is not successful then errors in the `equations.in` and `constants.in` files will need to be found and corrected. Aside from the screen output, more debugging information is written by `setup_equations` to the file `tmp/debug`. Other files in the `tmp` directory trace the interaction between the perl script and maxima.

4. **Compile and run `arb`:** The `arb` script in the working directory is a 'wrapper' for handling the running of `arb`. Typing

```
./arb -q -m &
```

from the working directory will remove old output files, compile `arb` (make option `-m`), run `arb` and direct the output to `output/output.scr` (quiet option `-q`), all in the background (`&`).

Other options to the `arb` run script include `-d` to make with debugging options and run in a debugging environment, and `-c` to continue from a previous run (not implemented yet).

5. **View results:** `arb` produces a file `output/output.msh` which can be opened by `gms` for viewing. This file includes the mesh information as well as variable data, so can be passed to `arb` as an input `.msh` file for subsequent simulations (although only mesh read, not data read, is currently implemented).

4 Inside arb

4.1 Code structure

4.2 Mesh structure

arb uses an unstructured mesh composed of cell elements that are separated by face elements. The dimension of the domain cell elements is specified in `constants.in` using the `DIMENSIONS` keyword. The dimension of the face elements is always 1 less than that of the domain cell elements. arb has been coded to be able to handle any poly-sided cells, however in practice it has only been tested to date (v0.23) with tetrahedron in 3D, triangles in 2D and lines in 1D: these are the default element geometries created by gmsh.

Boundary cells are created by arb after a mesh has been imported. They have a dimension that is 1 less than that of the domain cells, so have no volume/area/width in 3D/2D/1D, respectively. Each boundary cell has the same geometry, and is coincident with, a boundary face. Hence, a mesh has the same number of boundary faces as boundary cells.

4.3 Regions

Regions are sets of mesh elements that are used to locate user-defined variables and equations. Each region may contain only mesh elements of the same centring (that is, either cell or face elements, but not both). Regions can be defined by the user directly in gmsh when the mesh is generated, or via statements in the `constants.in` file that are interpreted when arb is run. There are also several generic system generated regions. Region names must be delimited by the `<>` characters, but apart from these two characters their names may contain any non-alphanumeric characters.

4.3.1 Defining regions via gmsh

Regions are specified in gmsh by defining and then naming physical entities. To do this via the gmsh GUI:

- Add a physical entity (under the physical groups tab) by selecting various elemental entities.

- Edit the geometry file (using the edit tab) and change the physical entity's name from the numerical name given by gmsh to the required <> delimited name suitable for arb.
- Save the .geo file.
- Reload the .geo file again (using the reload tab). If you now check under the visibility menu the physical entity will be visible.

4.3.2 Defining regions within the constants.in file

Currently there are two types of region specification statements that can be used in the constants.in file:

Compound region:

```
CELL_REGION <name> "COMPOUND +<region1>+<region2>-<
    region3>" # comments
FACE_REGION <name> "COMPOUND <region1>-<region2>" #
    comments
```

A compound region is defined using other existing regions. All regions that are used in the definition (ie, <region1>, <region2> and <region3> in the above examples) must have the (same) centring that is specified by the REGION keyword. If a + sign precedes a region name in the list of regions, then all the mesh elements that are in the following region are added to the new compound region, if they are not already members. If a - sign precedes a region name in the list of regions, then all the mesh elements that are in the following region are removed from the new compound region, if they are (at that stage) members of the new compound region. If no sign immediately precedes a region name in the defining list then a + sign is assumed. When constructing a compound region arb deals with each region in the defining list sequentially; so whether a mesh element is included in the compound region or not may depend on the order that the regions are listed.

At region:

```
CELL_REGION <name> "AT x1 x2 x3" # comments
FACE_REGION <name> "AT x1 x2 x3" # comments
```

This statement defines a region that contains one cell or one face mesh element. The element chosen lies closest to the point (x_1, x_2, x_3) . The values x_1 , x_2 and x_3 can be real or double precision floats.

4.3.3 System generated regions

The following regions are generated by `arb` at the start of a simulation. The names cannot be used for user-defined regions:

region name	description
<all cells>	all cells
<domain>	internal domain cells
<boundary cells>	cells located on the boundary
<all faces>	all faces
<domain faces>	internal domain faces
<boundaries>	faces located on the boundary

Additionally, there are a number of system regions which may be used in user-written expressions (see section 5) which specify sets of mesh elements relative to the current position. These names cannot be used for user-defined regions either:

region name	rel. to	description
<celljfaces>	cell	faces that surround the current cell
<nobcelljfaces>	cell	faces that surround the current cell, unless the current cell is on a boundary. In that instance move to the neighbouring domain cell and then cycle around the surrounding face cells.
<cellicells>	cell	cells that are local to the current cell (more than just the adjacent cells)
<faceicells>	face	cells that are local to the current face (more than just the adjacent cells)
<adjacentcellicells>	cell	cells that are strictly adjacent to the current cell
<adjacentfaceicells>	face	cells that are strictly adjacent to the current face (always two)

<code><upwindfaceicells></code>	face	the cell that is upwind of the face, used when performing <code>faceave[advection]</code> averaging (see section 5. Not really a user region.
<code><downwindfaceicells></code>	face	the cell that is downwind of the face, used when performing <code>faceave[advection]</code> averaging (see section 5. Not really a user region.
<code><cellkernel[l=0]></code>	cell	surrounding faces used in a cell averaging kernel (see section 5. Not really a user region.
<code><cellkernel[l=1-3]></code>	cell	surrounding cells used in cell derivative kernels (see section 5. Not really a user region.
<code><cellkernel[l=4]></code>	cell	surrounding nodes used in a cell averaging kernels (see section 5. Not really a user region.
<code><facekernel[l=0-6]></code>	face	surrounding cells used in face averaging and derivative kernels (see section 5. Not really a user region.
<code><noloop></code>	face/cell	dummy region which specifies no elements.

4.4 Variable types

There are six types of user defined variables: constant, transient, dependent, independent, equation and output. Each of these are stored in `arb` using the same general data structure (fortran type `var`). Any of these variables can be defined by a user-written expression in `equations.in` which is read by `setup_equations` and interpreted by `maxima`. Additionally, the constant type may be defined in `constants.in` and there given (only) a numerical value. Along with the user defined variables, there are also system defined variables which can be used in user-written expressions.

All variables have an associated compound variable type (scalar, vector or tensor) which is used mainly for output purposes.

Details of both the user and system defined variables are given in this section.

4.4.1 Constant type variable defined in `equations.in`

Synopsis:

Constant variables are evaluated once at the start of a simulation. If defined in `equations.in` they are defined using an expression which may contain only system variables and other constants — in the latter case the constants must

have been defined in either the `constants.in` file or previously (above) in the `equations.in` file.

Defining statements:

```
CELL_CONSTANT <name> [multiplier*units] "expression" ON
    <region> options # comments
FACE_CONSTANT <name> [multiplier*units] "expression" ON
    <region> options # comments
NONE_CONSTANT <name> [multiplier*units] "expression"
    options # comments
CONSTANT <name> [multiplier*units] "expression" options
    # comments
```

Statement components:

- (CELL_|FACE_|NONE_|)CONSTANT (*required*): This keyword specifies the centring of the variable. Constants that have cell or face centring vary over the simulation domain, and have values associated with each cell or face, respectively (subject to the region statement, below). None centred constants have one value that is not linked to any spatial location. If the centring specifier is omitted from the keyword (as in `CONSTANT`) then none centring is assumed (ie., keyword `CONSTANT` is equivalent to keyword `NONE_CONSTANT`).
- <name> (*required*): Each variable must have a unique name, delimited by the < and > characters. Besides these characters, the variable may contain spaces and any other non-alphanumeric characters. If the name ends with a direction index, as in <u[l=1]> or <gradp [l=3]>, then the variable is considered to be a component of a three dimensional vector compound. Similarly, if the name ends with a double direction index, as in <tau[l=1,3]>, the variable is considered to be a component of a three by three tensor compound. Components of compounds that are not explicitly defined are given a zero value (when used in dot and double dot products for example). All defined components that are members of the same compound must be of the same variable type, have the same centring, be defined over the same region and have the same units and multiplier. Certain names are reserved for system variables (see section 4.4.9).
- multiplier (*optional*): When reading in numerical constants, each value is multiplied by this value. At present not in use in `equations.in`.

- *units (optional)*: A string which specifies the units for the variable. At present this string is not interpreted by the code at all and the user must ensure that the units used are consistent.
- *"expression" (required)*: When a constant is defined in `equations.in`, this double-quoted expression is used to specify the value of the constant. As they may contain system variables and also other constants, they may vary throughout the domain. For more details regarding the syntax of these expressions, see section 5.
- *ON <region> (optional)*: This part of the statement determines over what region the variable should be defined. It is only applicable for cell and face centred variables, and must in these cases refer to a region that has the same centring as the variable. If omitted then by default a cell centred constant will be defined on `<all cells>` and a face centred constant on `<all faces>`. Note that referring to a variable value outside of its region of definition will produce an error when running `arb`.
- *options (optional)*: This is a comma separated list of options. Valid options for the constant variable type include:
 - `componentoutput`: This component to be written to `output.msh`. Default is `nocomponentoutput` for constants.
 - `compoundoutput`: The compound variable that is component is a member of to be written to `output.msh`. If the compound is cell centred then the data is output at cell and node centres (which looks better than only cell centred data when rendered). Default is `nocompoundoutput` for constants.
- *comments (optional)*: Anything written beyond the `#` is regarded as a comment.

Examples:

```
CELL_CONSTANT <test constant> "<cellx[1=1]>^2" ON <
  boundaries> # a test
FACE_CONSTANT <test constant 2> [m] "<facex[1=2]>" #
  another test
```

4.4.2 Constant type variable defined in `constants.in`

Synopsis:

Constant variables defined in `constants.in` are set to numerical values read directly by the `arb` executable, rather than expressions interpreted by `maxima`.

Defining statements:

```
CELL_CONSTANT <name> [multiplier*units] value ON <region>
> options # comments
FACE_CONSTANT <name> [multiplier*units] value ON <region>
> options # comments
NONE_CONSTANT <name> [multiplier*units] value options #
comments
CONSTANT <name> [multiplier*units] value options #
comments
```

Statement components:

The components of these statements are the same as in section 4.4.1 with the exception of:

- `value` (*required*): A numerical value of real or double precision type.

Examples:

```
CONSTANT <mu> [Pa.s] 1.0d-3 # fluid viscosity
NONE_CONSTANT <rho> [997*kg/m^3] 1.0 # fluid density
```

4.4.3 Constant type variable defined per region in `constants.in`

Synopsis:

This definition can be used in the `constants.in` file to assign different numerical values to either a cell or face centred constant in specific regions. Two statements are required for this type of constant definition: The first defines the list of regions where the next constant will be set (`REGION_LIST`) and the second defines the constant and sets/lists the corresponding numerical values (`(CELL_|FACE_)REGION_CONSTANT`). The region names in the `REGION_LIST`

statement must have the same centring as the following REGION_CONSTANT statement. Furthermore, the <region> over which the constant is defined must include all of the regions listed within the previous REGION_LIST statement.

Defining statements:

```
REGION_LIST <region1> <region2> ... <regionN> # comments
CELL_REGION_CONSTANT <name> [multiplier*units] value1
    value2 ... valueN ON <region> options # comments
FACE_REGION_CONSTANT <name> [multiplier*units] value1
    value2 ... valueN ON <region> options # comments
```

Statement components:

The components of these statements are the same as in section 4.4.1 with the exception of:

- <region1> <region2> ... <regionN> (*required*): A list of regions that have the same centring as the following REGION_CONSTANT statement.
- value1 value2 ... valueN (*required*): A list of numerical values for the constant, corresponding in a one-to-one fashion with the list of regions given in the previous REGION_LIST statement.

Examples:

```
REGION_LIST <inlet> <outlet> # some face regions
FACE_REGION_CONSTANT <electric field> [V/m] 10 20. ON <
    boundaries>
```

4.4.4 Transient type variable defined in equations.in

Synopsis:

These will be implemented for version 0.3 (transient).

4.4.5 Dependent type variable defined in equations.in

Synopsis:

Dependent variables depend on the independent variables and other previously defined (ie, above in the file) dependent variables.

Defining statements:

```
CELL_DEPENDENT <name> [multiplier*units] "expression" ON
    <region> options # comments
FACE_DEPENDENT <name> [multiplier*units] "expression" ON
    <region> options # comments
NONE_DEPENDENT <name> [multiplier*units] "expression"
    options # comments
DEPENDENT <name> [multiplier*units] "expression" options
    # comments
```

Statement components:

Along with the information presented in section 4.4.1, the following applies to dependent variables:

- (CELL_|FACE_|NONE_|)DEPENDENT (*required*): If no centring is specified then none centring is assumed.
- "expression" (*required*): This is an expression for the dependent variable in terms of constant, transient, independent, previously defined dependent (appearing above in equations.in) and system variables.
- ON <region> (*optional*): If omitted then by default a cell centred dependent will be defined on <all cells> and a face centred dependent on <all faces>.
- options (*optional*): This is a comma separated list of options. Valid options for dependent variables (as well as those given in section 4.4.1) include:
 - noderivative: Normally the derivative of this variable's expression is calculated with respect to each independent variable (the Jacobian) when performing the Newton-Raphson solution procedure. Including this option sets this derivative to zero. This may be required for functions for which the derivative cannot be calculated or for functions that undergo step changes (not continuous) which are not amenable to solution via the Newton-Raphson procedure. Using this option will usually slow convergence.

- positive/negative: Including one of these options causes the code to check the sign of the dependent variable. This is particularly useful for quantities like concentrations that are only physically meaningful when being positive. By using an expression such as "1-<con>" and including the option positive an upper limit for a variable can also be enforced.

Examples:

```
FACE_DEPENDENT <tau[l=1,1]> "<p> - <mu>*2.d0*facegrad[l
=1](<u[l=1]>)" compoundoutput
CELL_DEPENDENT <graddivp[l=1]> "celldivgrad[l=1](<p>)" #
divergence based pressure gradient
```

4.4.6 Independent type variable defined in equations.in

Synopsis:

Independent variables are those upon which the equations and dependent variables ultimately depend.

Defining statements:

```
CELL_INDEPENDENT <name> [multiplier*units] magnitude "
expression" ON <region> options # comments
FACE_INDEPENDENT <name> [multiplier*units] magnitude "
expression" ON <region> options # comments
NONE_INDEPENDENT <name> [multiplier*units] magnitude "
expression" options # comments
INDEPENDENT <name> [multiplier*units] magnitude "
expression" ON <region> options # comments
```

Statement components:

Along with the information presented in section 4.4.5, the following applies to independent variables:

- (CELL_|FACE_|NONE_|)INDEPENDENT (*required*): If no centring is specified then cell centring is assumed.

- *magnitude (required)*: An order of magnitude estimate (positive and greater than zero real or double precision value) must be specified for all independent variables. This magnitude is used when checking on the convergence of the solution.
- *"expression" (required)*: For an independent variable the expression specifies the variable's initial value. The expression may contain constants, previously defined (initial) dependent values (those appearing above in `equations.in`) and system variables.
- *ON <region> (optional)*: If omitted then by default a cell centred independent will be defined on `<all cells>` and a face centred independent on `<all faces>`.
- *options (optional)*: The `noderivative` option is not applicable for independent variables.

Examples:

```
CELL_INDEPENDENT <u[l=1]> 1.d0 "<u_av>" # a velocity
                component
CELL_INDEPENDENT <p> [] 1.d0 "1.d0-<cellx[l=1]>" #
                pressure
NONE_INDEPENDENT <p_in> [Pa] 1.d0 "1.d0" # the pressure
                at the inlet
```

4.4.7 Equation type variable defined in `equations.in`

Synopsis:

Equation variables represent the equations to be satisfied. The equation expressions should be formulated so that when the equation is satisfied, the expression equals zero. The number of equations must equal the number of independent variables. Furthermore, for the system to be well posed the equations must be independent (no single equation can be made from a combination of the other equations).

Defining statements:

```
CELL_EQUATION <name> [multiplier*units] "expression" ON
               <region> options # comments
```

```

FACE_EQUATION <name> [multiplier*units] "expression" ON
    <region> options # comments
NONE_EQUATION <name> [multiplier*units] "expression"
    options # comments
EQUATION <name> [multiplier*units] "expression" options
    # comments

```

Statement components:

Along with the information presented in section 4.4.5, the following applies to equation variables:

- (CELL_|FACE_|NONE_|)EQUATION (*required*): If no centring is specified then none centring is assumed.
- "expression" (*required*): For an equation variable the expression should equal zero when the equation is satisfied. The expression may contain constant, transient, dependent, independent and system variables.
- ON <region> (*optional*): If omitted then by default a cell centred equation will be defined on <domain> and a face centred equation on <boundaries>.

Examples:

```

CELL_EQUATION <continuity> "celldiv(<u_f>)" ON <domain>
    # continuity
FACE_EQUATION <outlet noslip> "dot(<u[l=:]>,<facetang1[l
    =:]>)" ON <outlet> # no component tangential to
    outlet
NONE_EQUATION <p_in for flowrate> "<u_av_calc>-<u_av>" #
    set flowrate through inlet to give required average
    velocity

```

4.4.8 Output type variable defined in equations.in

Synopsis:

Output variables are evaluated once convergence of the solution has been reached: They are only for output purposes.

Defining statements:


```

CELL_OUTPUT <name> [multiplier*units] "expression" ON <
    region> options # comments
FACE_OUTPUT <name> [multiplier*units] "expression" ON <
    region> options # comments
NONE_OUTPUT <name> [multiplier*units] "expression"
    options # comments
OUTPUT <name> [multiplier*units] "expression" options #
    comments

```

Statement components:

Along with the information presented in section 4.4.5, the following applies to output variables:

- (CELL_|FACE_|NONE_|)OUTPUT (*required*): If no centring is specified then none centring is assumed.
- "expression" (*required*): For an output variable the expression may contain constant, transient, dependent, independent, equation and system variables.
- ON <region> (*optional*): If omitted then by default a cell centred output variable will be defined on <all cells> and a face centred output variable on <all faces>.
- options (*optional*): The noderivative option is not applicable for output variables (this option is implicitly set anyway for these variables).

Examples:

```

NONE_OUTPUT <F_drag> [N] "facesum(<facearea>*dot(<
    facenorm[l=:]>,<tau[l=:,1]>),<cylinder>)" # force on
    object in axial direction

```

4.4.9 System variables

4.5 Simulation options

4.6 Data visualisation

5 Expression language reference

Sorry - there's lots missing in this section!

5.1 Operators

5.1.1 `celldiv`: Divergence calculated around a cell

Summary: Uses Gauss' theorem to calculate the divergence of a face centred component around a cell.

Statement:

`celldiv[options](face_centred_expression)`

Centring:

Operator is cell centred, contents of operator is face centred.

Details:

Using Gauss' theorem to evaluate divergences around cells is probably the defining characteristic of Finite Volume methods. `celldiv` performs this operation.

To discretise the divergence of a vector \mathbf{u} over a cell within the domain, Gauss' theorem gives

$$\frac{\int_{V_{\text{cell}}} \nabla \cdot \mathbf{u} dV}{\int_{V_{\text{cell}}} dV} = \frac{1}{V_{\text{cell}}} \int_{S_{\text{cell}}} \mathbf{n}_{\text{cell}} \cdot \mathbf{u} dS = \frac{1}{V_{\text{cell}}} \sum_j (\mathbf{n}_{\text{cell}} \cdot \mathbf{n}_j) (\mathbf{u} \cdot \mathbf{n}_j) S_j$$

where V_{cell} and S_{cell} are the volume and total surface area of the cell, respectively, \mathbf{n}_{cell} is a unit normal pointing outward from the cell, \mathbf{n}_j is a normal associated with surrounding face j , and the sum is conducted over all faces (index j) which surround the cell. Taking the divergence of a vector results in a scalar. The above vector divergence is represented by `celldiv` as

`celldiv(dot(<u[l=:]>, <facenorm[l=:]>))`

where in this case the vector \mathbf{u} is represented by the three component variables `<u[l=1]>`, `<u[l=2]>` and `<u[l=3]>`, and the unit normal associated with

the face j , \mathbf{n}_j , is given by the system component variables `<facenorm[l=1]>`, `<facenorm[l=2]>` and `<facenorm[l=3]>`.

Note that the region used by `arb` in performing the sum \sum_j is `<nobcelljfaces>`. This relative region specifies all faces that surround a given cell, unless that cell is a boundary cell. As boundary cells are not fully surrounded by faces Gauss' theorem can not be applied. Hence, if the operator `celldiv` is used at a boundary cell then the region `<nobcelljfaces>` is taken relative (moved) to the closest domain cell that is adjacent the boundary cell, so this is where `celldiv` becomes evaluated. Physically it is inadvisable to use an equation that involves a divergence at a boundary cell anyway.

Options:

- `noderivative`: No derivatives with respect to the independent variables for the Newton-Raphson Jacobian are calculated for this operator (and its contents).

Examples:

```
CELL_EQUATION <continuity> "celldiv(<u_f>)" ON <domain>
# continuity equation
CELL_EQUATION <momentum[l=1]> "celldiv(<J_f[l=1]>)" ON <
domain> # momentum conservation in direction l=1
CELL_EQUATION <momentum[l=2]> "celldiv(<J_f[l=2]>)" ON <
domain> # momentum conservation in direction l=2
```

- 5.1.2 `cellgrad` or `facegrad`: **Gradient evaluated at a cell or face**
- 5.1.3 `celldivgrad`: **Gradient evaluated at a cell calculated via a divergence**
- 5.1.4 `cellave` or `faceave`: **Interpolation of a quantity from one centering to another**
- 5.1.5 `celllimiter`: **Gradient limiter for ensuring advection stability**
- 5.1.6 `cellif` or `faceif` or `noneif`: **If conditional statement**
- 5.1.7 `cellsum` or `facesum`: **Sum performed over a region of elements**
- 5.1.8 `cellmax` or `facemax` or `nonemax`: **Picks the maximum from a region of elements**
- 5.1.9 `cellmin` or `facemin` or `nonemin`: **Picks the minimum from a region of elements**
- 5.1.10 `celldelta` or `facedelta`: **A delta function to identify specific regions**

6 Example applications

6.1 Incompressible steady-state Newtonian flow through a 2D channel containing a cylinder with set inlet velocities

Listing 1: ../../gmsh/manual/channel_with_cylinder/surface.geo

```
// 2d_channel_with_cylinder

lc = 0.05; // characteristic mesh length variable

// setup domain boundaries
Point(1) = {0, 0, 0, lc/2};
Point(15) = {0.2, 0, 0, lc/4};
Point(2) = {2.2, 0, 0, lc} ;
Point(3) = {2.2, 0.41, 0, lc} ;
Point(16) = {0.2, 0.41, 0, lc/4};
Point(4) = {0, 0.41, 0, lc/2} ;

Line(1) = {1,15} ;
Line(15) = {15,2} ;
Line(2) = {2,3} ;
Line(3) = {3,16} ;
Line(16) = {16,4} ;
Line(4) = {4,1} ;

// create an elementary entity that is the domain
boundary
Line Loop(5) = {1,15,2,3,16,4} ;

// create the physical entities for the inlet and output
which become the arb regions
Physical Line("<inlet>") = {4};
Physical Line("<outlet>") = {2};

// create the cylinder
Point(5) = {0.2, 0.15, 0, lc/4};
Point(6) = {0.25, 0.2, 0, lc/4};
Point(7) = {0.2, 0.25, 0, lc/4};
Point(8) = {0.15, 0.2, 0, lc/4};
Point(9) = {0.2, 0.2, 0, lc/4};
```

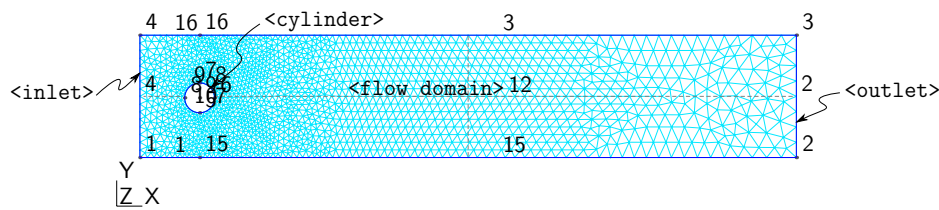


Figure 1: Geometry and resulting mesh from the `surface.geo` file for the flow through a 2d channel containing a cylinder problem.

```

Ellipse(7) = {5, 9, 9, 6};
Ellipse(8) = {6, 9, 9, 7};
Ellipse(9) = {7, 9, 9, 8};
Ellipse(10) = {8, 9, 9, 5};

// create an elementary entity that is the cylinder
boundary
Line Loop(11) = {7, 8, 9, 10};

// create the physical entity for the cylinder boundary
which becomes the arb region
Physical Line("<cylinder>") = {7, 8, 9, 10};

// all of the flow domain must be included as a physical
entity to be output under gmsh
Plane Surface(12) = {5, 11};
Physical Surface("<flow domain>") = {12};

```

Listing 2: `../examples/manual/steady_state_channel_flow_with_cylinder/constants.in`

```

# arb finite volume solver
# Copyright 2009,2010 Dalton Harvie (daltonh@unimelb.edu
  .au)
#
# arb is released under the GNU GPL. For full details
  see the license directory.
#
#
-----

# file constants.in
VERSION 0.24

```

```

#
-----

# user-defined constants

# REGION_LIST line has the ordered names of any regions
  (quoted)
# (CELL_|FACE_)REGION_CONSTANT sets a constant that
  varies with region and has the form: <name> [
  multiplier*units] value_for_region_1 ..
  value_for_region_n options # comments
# (CELL_|FACE_|NONE_|)CONSTANT sets a constant specific
  to one region or no regions and has the form: <name>
  [multiplier*units] value ON <region> options #
  comments

# physical data
CONSTANT <mu> [Pa.s] 1.d-3 # viscosity of liquid
CONSTANT <rho> [kg/m^3] 1.d0 # density
CONSTANT <u_av> [m/s] 0.2d0 # average inlet velocity

# numerical data
CONSTANT <C_{Rhie-Chow}> [] 1.0d+0 # multiplier for Rhie
  -Chow-type pressure oscillation control
CONSTANT <adv_limiter> [] 1.d0 # multiplier used to
  limit gradients when calculating advection fluxes

#
-----

# system constants

NEWTRESTOL 1.d-12 # convergence criterion for newton
  solver
NEWTSTEPMAX 20 # maximum number of steps for newton
  solver

#
-----

# geometry

# CELL_REGION/FACE_REGION specified by: <name> "location

```

```

    string" # comments
# where location string could be: "AT x1 x2 x3" for a
  single point closest to these coordinates
# where location string could be: "COMPOUND +<a region
  >-<another region>" for a + and - compound region
  list

FACE_REGION <walls> "COMPOUND <boundaries>-<inlet>-<
  outlet>"

# DIMENSIONS is the number of dimensions used in the
  problem
DIMENSIONS 2

# READ_GMSH instructs arb to read a gmsh file
#READ_GMSH "gmsh/manual/channel_with_cylinder/surface.
  msh"
READ_GMSH "surface.msh"

# linear solver that is used to invert jacobian
#LINEAR_SOLVER "HSL_MA28" # hsl archive direct solver
#LINEAR_SOLVER "INTEL_PARDISO" # pardiso solver
  contained in intel mkl library
#LINEAR_SOLVER "INTEL_PARDISO_OOC" # pardiso solver
  contained in intel mkl library
LINEAR_SOLVER "SUITESPARSE_UMF" # suitesparse umf solver
  (UMFPACK) by Timothy A. Davis

#
  -----

```

Listing 3: ../../examples/manual/steady_state_channel_flow_with_cylinder/equations.in

```

# arb finite volume solver
# Copyright 2009,2010 Dalton Harvie (daltonh@unimelb.edu
  .au)
#
# arb is released under the GNU GPL. For full details
  see the license directory.
#
#
  -----

```



```

# file equations.in
VERSION 0.24
#
-----

# statement reference
# (CELL_|FACE_|NONE|)CONSTANT <name> [units] "expression
    (involving only constants)" ON <region> options #
    comments
# (CELL_|FACE_|NONE|)TRANSIENT <name> [units] magnitude
    "expression" ON <region> options # comments
# (CELL_|FACE_|NONE|)DEPENDENT <name> [units] magnitude
    "expression" ON <region> options # comments
# (CELL_|FACE_|NONE|)INDEPENDENT <name> [units] "
    expression (initial value)" ON <region> options #
    comments
# (CELL_|FACE_|NONE|)EQUATION <name> [units] "expression
    (equation equaling zero)" ON <region> options #
    comments
# (CELL_|FACE_|NONE|)OUTPUT <name> [units] "expression"
    ON <region> options # comments

# options include:
# noderivative - for DEPENDENT, EQUATION
# positive,negative - for DEPENDENT, INDEPENDENT,
    EQUATION
# harmonic - for CONSTANT, TRANSIENT, DEPENDENT,
    INDEPENDENT
# compoundoutput/nocompoundoutput - for ALL
# componentoutput/nocomponentoutput - for ALL

# independent variables used for flow problems
CELL_INDEPENDENT <u[l=1]> [] 1.d0 "<u_av>" # velocity
    component
CELL_INDEPENDENT <u[l=2]> [] 1.d0 "0.d0" # velocity
    component
CELL_INDEPENDENT <p> [] 1.d0 "1.d0-<cellx[l=1]>" #
    pressure

# total stress tensor
FACE_DEPENDENT <tau[l=1,1]> "<p> - <mu>*2.d0*facegrad[l

```

```

=1](<u[l=1]>)" compoundoutput
FACE_DEPENDENT <tau[l=1,2]> "- <mu>*(facegrad[l=2](<u[l=1]>)+facegrad[l=1](<u[l=2]>))"
FACE_DEPENDENT <tau[l=2,2]> "<p> - <mu>*2.d0*facegrad[l=2](<u[l=2]>)"
FACE_DEPENDENT <tau[l=2,1]> "<tau[l=1,2]>"

# a Rhie-Chow-type correction is applied to the face
# velocities
CELL_DEPENDENT <graddivp[l=1]> "celldivgrad[l=1](<p>)" #
# pressure gradient calculated via a divergence (
# consistent with momentum conservation)
CELL_DEPENDENT <graddivp[l=2]> "celldivgrad[l=2](<p>)" #
# pressure gradient calculated via a divergence (
# consistent with momentum conservation)
FACE_DEPENDENT <p_error> "facegrad(<p>) - dot(<graddivp[
l=:]>,<facenorm[l=:]>)" # difference between face
# centred and cell divergence type gradient at face and
# normal to the face
FACE_DEPENDENT <u_f_{correction}> "-<C_{Rhie-Chow}>*
# facedelta(<domain faces>)*facemin(<facedx>^2/<mu>,
# sqrt(<facedx>/(<rho>*facemax(abs(<p_error>),1.d-6)))
# )*<p_error>" compoundoutput # the Rhie-Chow type
# velocity correction, only applied (nonzero) on the
# domain faces

# flux of mass (volume) and momentum components over
# each face
FACE_DEPENDENT <u_f> "dot(<u[l=:]>,<facenorm[l=:]>) + <
# u_f_{correction}>" # volume (velocity) transport
FACE_DEPENDENT <J_f[l=1]> "dot(<facenorm[l=:]>,<tau[l
# =:,1]>)+<rho>*faceave[advection](<u[l=1]>,<u_f>,<
# adv_limiter>)*<u_f>" compoundoutput # component of
# momentum transport from stress and advection
FACE_DEPENDENT <J_f[l=2]> "dot(<facenorm[l=:]>,<tau[l
# =:,2]>)+<rho>*faceave[advection](<u[l=2]>,<u_f>,<
# adv_limiter>)*<u_f>" # component of momentum
# transport from stress and advection

# conservation equations solved over each domain cell (
# finite volume method)
CELL_EQUATION <continuity> "celldiv(<u_f>)" ON <domain>
# continuity

```

```

CELL_EQUATION <momentum[l=1]> "celldiv(<J_f[l=1]>)" ON <
    domain> # momentum component
CELL_EQUATION <momentum[l=2]> "celldiv(<J_f[l=2]>)" ON <
    domain> # momentum component

# boundary conditions on nonslip walls
FACE_EQUATION <wall noflux> "<u_f>" ON <walls> # no flux
    normal to walls
FACE_EQUATION <wall noslip> "dot(<u[l=:]>,<facetang1[l
    =:]>)" ON <walls> # nonslip
FACE_EQUATION <wall p extrapolation> "dot(<graddivp[l
    =:]>,<facenorm[l=:]>)" ON <walls> # extrapolate
    pressure to the wall using zero gradient normal to
    wall

# boundary conditions on outlet - fully developed flow
FACE_EQUATION <outlet fully developed> "facegrad(dot(<u[l
    =:]>,cellave[lastface](<facenorm[l=:]>)))" ON <
    outlet> # normal velocity component is fully
    developed
FACE_EQUATION <outlet noslip> "dot(<u[l=:]>,<facetang1[l
    =:]>)" ON <outlet> # no component tangential to
    outlet
FACE_EQUATION <outlet p> "<p>" ON <outlet> # specified
    uniform (zero) pressure

# boundary conditions on inlet - fully developed flow
FACE_EQUATION <inlet fully developed> "facegrad(dot(<u[l
    =:]>,cellave[lastface](<facenorm[l=:]>)))" ON <inlet>
    # normal velocity component is fully developed
FACE_EQUATION <inlet noslip> "dot(<u[l=:]>,<facetang1[l
    =:]>)" ON <inlet> # no component tangential to inlet

# set velocity distribution corresponding to fully
    developed Cartesian flow
FACE_EQUATION <inlet flowrate> "<u_f>+6.d0*<u_av>*<cellx
    [l=2]>*(0.41d0-<cellx[l=2]>)/(0.41d0^2)" ON <inlet> #
    specified velocity distribution

# alternatively, set uniform inlet pressure giving
    required average velocity
#NONE_DEPENDENT <u_av_calc> "facesum(-<u_f>*<facearea>,<
    inlet>)/facesum(<facearea>,<inlet>)" # calculate

```

```

    average velocity directed into the domain
#NONE_INDEPENDENT <p_in> [Pa] 1.d0 "1.d0" # define the
    pressure at inlet
#NONE_EQUATION <p_in for flowrate> "<u_av_calc>-<u_av>"
    # set flowrate through inlet to give required average
    velocity
#FACE_EQUATION <inlet flowrate> "<p>-<p_in>" ON <inlet>
    # apply specified pressure over inlet

# calculate drag and lift on object
NONE_OUTPUT <F_drag> [N] "facesum(<facearea>*dot(<
    facenorm[l=:]>,<tau[l=:,1]>),<cylinder>)" # force on
    object in axial direction
NONE_OUTPUT <F_lift> [N] "facesum(<facearea>*dot(<
    facenorm[l=:]>,<tau[l=:,2]>),<cylinder>)" # force on
    object in vertical direction
NONE_OUTPUT <C_drag> "2.d0*<F_drag>/(<rho>*<u_av>^2*0.1
    d0)" # drag coefficient
NONE_OUTPUT <C_lift> "2.d0*<F_lift>/(<rho>*<u_av>^2*0.1
    d0)" # lift coefficient

#
    -----

```