

# Debugging OpenFOAM implementations with GDB

(Acknowledgements to Dr. Fabian Peng-Kärrholm)

- It is impossible to do bug-free programming (trust me!), so you should always verify your implementations.
- When you run into problems, such as code crash, or mysterious behaviour, you also need some debugging approach.
- There are many debugging approaches, and we will discuss three alternatives here:
  - Info statements in the code
  - Built-in DebugSwitch option in OpenFOAM (similar to the above - you will see)
  - Debugging using the Gnu debugger, GDB (<http://www.gnu.org/software/gdb/>)
- We will now go through these alternatives...

## Debugging using Info statements

- The simplest way to do debugging is to write out intermediate results to the screen, and check that those results are correct.
- In OpenFOAM this is done using `Info` statements.
- This kind of debugging does not allow any control of the running of the code while debugging. It will just print out additional information.
- `Info` debugging requires that new lines are inserted in the source code, and that the source code must be re-compiled whenever a new `Info` statement has been added.
- When the code has been debugged, all the `Info` statements must be deleted, or commented, so that you don't get all that information when you are no longer debugging.
- OpenFOAM provides an alternative to removing all the `Info` statements, so that these `Info` statements can be activated again later.
- This brings us to the next level of debugging in OpenFOAM...

## Debugging using OpenFOAM DebugSwitches

- In `$WM_PROJECT_DIR/etc/controlDict` (global `controlDict` dictionary), you can find a list of `DebugSwitches`:

```
DebugSwitches
{
    Analytical          0;
    APIdiffCoefFunc    0;
    Ar                  0;
    ...
}
```

- Each class thus has its own `DebugSwitch`.
- `DebugSwitches` set to zero will produce no debug information.
- Different levels of debugging can be chosen by setting a `DebugSwitch` to 1, 2, 3 ...
- It is recommended to make a copy of that file, to `$HOME/.OpenFOAM/1.7.x/controlDict` and make any modifications there. This file will override the original file.

```
mkdir -p $HOME/.OpenFOAM/1.7.x
cp $WM_PROJECT_DIR/etc/controlDict $HOME/.OpenFOAM/1.7.x/controlDict
```

## What is a DebugSwitch?

- Let's have a look at the `lduMatrix DebugSwitch`, which is set to 1.
- The `lduMatrix` class is implemented in  
`$FOAM_SRC/OpenFOAM/matrices/lduMatrix/lduMatrix`
- Looking inside `lduMatrix.C`, we see a line:  

```
defineTypeNameAndDebug(lduMatrix, 1);
```

This line defines the `DebugSwitch` name `lduMatrix`, and sets its default value to 1.
- In `lduMatrixTests.C`, in the same directory, you can find a member function `print()`, where all its contents are within an if-statement:  

```
if (debug)
{
...
}
```
- Boolean `debug` corresponds to the `lduMatrix DebugSwitch`, and it is `true` if the `DebugSwitch` is *greater than* 0. The default value, both in the class definition, and in the global `controlDict` is 1, so the contents of the if-statement will be executed.

## Result of DebugSwitch lduMatrix 1; (or 2;)

- We see that the class will print out the member data `solverName_`, `fieldName_`, `initialResidual_`, `finalResidual_`, and `noIterations_` (unless the solution is singular). In other words, this is most of the information we usually get in the log-file:

```
DILUPBiCG: Solving for Ux, Initial residual = 1, Final residual = 2.96338e-06, No Iterations 8
DILUPBiCG: Solving for Uy, Initial residual = 0, Final residual = 0, No Iterations 0
DICPCG: Solving for p, Initial residual = 1, Final residual = 7.55402e-07, No Iterations 35
```

- In the same file we can see that a `lduMatrix DebugSwitch` value  $\geq 2$  will give some extra information. In this case:

```
Normalisation factor = 0.004
DILUPBiCG: Iteration 0 residual = 1
DILUPBiCG: Iteration 1 residual = 0.153298
DILUPBiCG: Iteration 2 residual = 0.0375508
DILUPBiCG: Iteration 3 residual = 0.00820823
DILUPBiCG: Iteration 4 residual = 0.00174405
DILUPBiCG: Iteration 5 residual = 0.000357418
DILUPBiCG: Iteration 6 residual = 7.4212e-05
DILUPBiCG: Iteration 7 residual = 1.51806e-05
DILUPBiCG: Iteration 8 residual = 2.96338e-06
```

- In summary, the DebugSwitches only control different levels of Info-statements. No re-compilation is needed when switching the level, but if new Info-statements are included, re-compilation is needed. You can use this feature in your own development.
- This still offers no control of the running of the code while debugging...

## Debugging OpenFOAM implementations with GDB

- Now it is time for some real debugging with the Gnu debugger...
- GDB can be used for
  - Programs written in C, C++, Ada, Pascal etc
  - Running and stopping at specific positions in the code
  - Examining variables at run-time
  - Changing your program at run-time
- Bad news: The complete code needs to be re-compiled with a debug flag. This will produce ~1Gb extra of OpenFOAM binaries.
- Good news: I have done it for you here at Chalmers.
- More bad news: The code will run much slower in debug mode, in particular when running under GDB. The reason is that nothing can be optimized, and that there is much more information to keep track of.
- Best news: GDB can help you implement a bug-free code, which can be run in an optimized version.

## Compiling OpenFOAM in Debug mode

- In `$WM_PROJECT_DIR/etc/bashrc` you find an environment variable `WM_COMPILE_OPTION` that can be set to `Debug`. That is what you need to do if you want to compile using the debug flag, or use the Debug version.
- What you need to do right now is to do a new ssh to the student computers and type the following lines (or similar on your laptop if you have compiled the Debug version):

```
export WM_COMPILE_OPTION=Debug;  
export FOAM_INST_DIR=/chalmers/sw/unsup64/OpenFOAM;  
. $FOAM_INST_DIR/OpenFOAM-1.7.x/etc/bashrc
```

Make sure that you use the Debug mode by typing:

```
which icoFoam
```

which should point at a path containing `linux64GccDPDebug`.

Now you can compile or run all or parts of OpenFOAM in Debug mode.

- Now it is time to learn the basics of GDB...

## Debugging the icoFoam solver

- Let's practice GDB on the icoFoam solver.
- The objective is to find out what a part of the code does.
- In `$FOAM_SOLVERS/incompressible/icoFoam/icoFoam.C` there is a line saying:  
`adjustPhi(phi, U, p);`  
What does this line do?
- Make sure that you have an icoFoam/cavity case, that you have run blockMesh on it, and you are inside that case directory while running GDB.
- Start icoFoam under GDB in the cavity case directory:  
`gdb icoFoam`
- Set a break point at the line in icoFoam.C where `adjustPhi(phi, U, p);` is used:  
`b icoFoam.C:75`
- Start the execution by typing:  
`run` (this will take some time...)
- The execution will stop at the breakpoint saying:  
Breakpoint 1, main (argc=1448, argv=0x3d1) at icoFoam.C:76  
76                    `adjustPhi(phi, U, p);`

## Line-by-line checking of `adjustPhi`

- There are two ways of stepping in the file (**don't do these now!**):
  - `n` (next) will step to the next line in the current file, but will not go into functions and included files.
  - `s` (step) will go to the next line, also in functions and included files.Both can be followed by a number specifying how many times to repeat the command.
- Step to the next line by typing `'s'` (allowing GDB to go inside the `adjustPhi` function). We are now at line 41 in `cfTools/general/adjustPhi/adjustPhi.C`
- Type `'where'` to see which file you are debugging, and which file called it.
- Type `'list cfTools/general/adjustPhi/adjustPhi.C:30,50'` to see source lines 30-50 of `adjustPhi.C`. Line 41 is the first line of the `adjustPhi` function.
- Type `'n 3'` to avoid going into the evaluation of the boolean, instead we will see that we are at line 46 in `adjustPhi.C`, so the if-statement is evaluated.
- Type `'p massIn'` and then `'p fixedMassOut'`. Note that line 46 has not yet been evaluated, so `fixedMassOut` can have any value!!
- Type `'run'` to restart the debugging from the beginning if needed. Type `'quit'` to quit.
- Stepping and listing the code will show every single step that will be taken. You will understand that `adjustPhi` ensures global continuity.

## Learn more on GDB

- See `gdbfoam.pdf`, by Dr. Fabian Peng-Kärrholm, on the 2007 course homepage.
- See <http://www.gnu.org/software/gdb>
- There are some interfaces to GDB:
  - See <http://www.gnu.org/software/gdb/links/>
  - emacs ... next slide

## GDB in emacs

- In the cavity case, run emacs
- Click `Tools/Debugger (GDB)`
- Type `icoFoam` (appears at the bottom of emacs, as part of a GDB command)
- Type `b icoFoam.C:75`, as before.
- Click on `GO`  
The execution stops at line 75 in `icoFoam.C`, and emacs shows the output of `icoFoam` in the upper window, and the surrounding code in the lower window.
- Click on `Next Line` to go to the next line in `icoFoam.C`, without entering any functions.
- At the line with `if (nonOrth == nNonOrthCorr)`, type in the upper window:

```
p nonOrth  
p nNonOrthCorr
```

They are the same, to the boolean is true, which means that the line inside the if-statement will be executed (check by clicking on `Next Line`). Let's examine that line closer by clicking `Step Line`. Emacs will ask if you want to follow a symbolic link - just type `yes`. We now see that we are in `fvMatrix.C`, line 785.

## GDB in emacs

- Will we enter the if-statement? Type in the upper window:  

```
p !psi_.mesh().fluxRequired(psi_.name())
```

which returns false, so we will not enter (check with `Next Line`)
- Click `Next Line` until you are at the line with `forall`, then click on `Step Line` and type `yes` on the question on the symbolic link.
- The `size()` function returns the size of member data `ptrs_` in the templated `PtrList` class. Click `Step Line` and type `yes`.
- The `PtrList` seems to be pointing at the `List` class to evaluate the `size()` function. Click on `Step Line` twice and we are back to the `PtrList`. Click on `Step Line` again, and we are in a `forall` loop in the `fvMatrix.C` file.
- Let's not dig any further, so click on `Finish Function` to get back to `icoFoam.C`, and we can continue the execution on the highest level.

## Debugging a simple C++ code

```
#include <iostream>
using namespace std;

main()
{
const int constantInteger=5.1;
const float constantFloat=5.1;
cout << "constantInteger = " << constantInteger << endl;
cout << "constantFloat = " << constantFloat << endl;
}
```

Compile and debug with:

```
g++ -g basic.C -o basic
emacs
```