



## ***IMPACT* Module for *IcoFoam***

Illinois Rocstar LLC

November 18, 2014

## Table of Contents

1	OpenFoam	1
2	<i>IMPACT</i>	2
3	Obtaining, placing, and editing the module files	2
4	Building the module library and driver	3
5	Runing and testing the <i>IcoFoam</i> module	4

## List of Figures

## List of Tables

# 1 OpenFoam

In order to build a module for *IcoFoam* it is necessary to first download and build *OpenFoam* and specifically for this case *OpenFoam Extend* (*OpenFoam* and *OpenFoam Extend* will be used interchangeably within this document). Information for downloading this software can be found at <http://www.extend-project.de/>.

**Note:** The same *mpicc*, *mpicxx*, and/or *mpif90* must be used when compiling *IMPACT*, *OpenFoam*, and the module driver.

Is the above correct, Mike C.? I couldn't remember if I had set those variables before compiling *OpenFoam* but I assume I did

For builds outside of Illinois Rocstar make note of the compiler used to build *OpenFoam* and then follow the build instructions for *OpenFoam* found online or in the documentation. If you are working within Illinois Rocstar then load the `openmpi-x86_64` module and set the environment variables `CC`, `CXX`, and `FC` to `mpicc`, `mpicxx`, and `mpif90` respectively. Then, source the appropriate file (`cshrc` when using *C Shell* or `bashrc` when using *Bash*) found in the `etc` directory under the main *OpenFoam* source directory by entering one of the following commands

```
> source etc/cshrc
```

or

```
> source etc/bashrc
```

from the main *OpenFoam* source directory. Finally, run

```
> wmake/wmake all
```

from the main *OpenFoam* source directory. There may be additional changes that must be made to the `wmake` files in order to get *OpenFoam* to properly compile within your specific environment setup.

After *OpenFoam* is built, ensure that the `icofoam` executable has been created. If you have sourced one of the files for *OpenFoam* as mentioned above you may check that the `icofoam` executable exists by running

```
> which icofoam
```

and ensuring that the `icofoam` command returned is within the path of the *OpenFoam* source files you have been building (*OpenFoam* builds in-source).

Within Illinois Rocstar, a test case exists for *IcoFoam* from *OpenFoam Extend*. The documentation for this test case can be found in Section 9.2.2 of the *Uniphysics Validation for Development of Multiphysics Coupling in MP-Infra* document located internally in the Illinois Rocstar source repository under `/MPInfra/data/documentation/validation_uniphysics/Tex`. After ensuring that the test run works, the next steps toward building a module can be taken.

## 2 *IMPACT*

The Illinois Rocstar software *IMPACT* is required for integrating the *IcoFoam* module. For an external user the *IMPACT* software may be downloaded from <http://sourceforge.net/p/openmultiphys/wiki/Consortium%20for%20Open%20Multiphysics/>. For an internal user, *IMPACT* is located in the repository under *IMPACT/trunk*. Follow the build instructions from the User's Guide, keeping in mind that the same *mpicc*, *mpicxx*, and/or *mpif90* must be used when compiling *IMPACT*, *OpenFoam*, and the module driver. Note that internal users may not need to build *IMPACT* but instead simply load the *IMPACT* module provided to their machine.

Ensure that the locations of the *IMPACT* `include` and `lib` directories are known since they will be needed in [Section 3](#). If you use `make install` when installing *IMPACT* these directories should both be located under the install directory. Otherwise, they should be located in the main source directory and the main build directory for *IMPACT* respectively.

## 3 Obtaining, placing, and editing the module files

For a user internal to Illinois Rocstar the files necessary for building a module for *IcoFoam* within *OpenFoam Extend* can be found in the source repository under `/MPIinfra/Third_Party_Modules/OFTest/icoFoam/trunk/`. For an external user these files may be found on the open multiphysics website (to come). The files will contain the appropriate directory structure for building the driver. However, the files used with *OpenFoam* to create a module library must be placed in the appropriate locations within the *OpenFoam* source files in order for them to build. These files are located under the `native` directory of the module main source directory. The locations for placing these files will be given, but it should be noted that these are specific for *OpenFoam Extend 3.1*. In the case of another version of *OpenFoam Extend* it may be necessary to place these files in different locations. If you are using a different version of *OpenFoam Extend*, it is suggested that you locate the source file `icoFoam.C` and then place the module files in a manner with a similar structure to that shown here.

All paths below are shown from within the *OpenFoam Extend* main source directory.

- The main source file `icoFoam.C` should be located under `applications/solvers/incompressible/icoFoam/`.
- Place `icoFoamHeader.H` and `icoFoamModule.C` in `applications/solvers/incompressible/icoFoam/`.
- Replace the files and options files located under `applications/solvers/incompressible/icoFoam/Make` with the files and options files provided with the module. (The options file will require editing.)
- Place the `Allwmake` file in `applications/solvers/incompressible/`.

It is necessary to edit the options file in the `Make` directory as mentioned above. The options file should appear as shown below



```
EXE_INC = \  
    -I$(LIB_SRC)/finiteVolume/lnInclude \  
    -I/path/to/IMPACT/install/include  
  
LIB_LIBS = \  
    -lfiniteVolume \  
    -llduSolvers \  
    -L/path/to/IMPACT/install/lib \  
    -lSITCOM
```

Edit the locations that say “`path/to/IMPACT/install/`” to be the actual path to your installation of *IMPACT*. Again, note that if you did not do `make install` when compiling and building *IMPACT* the path to the include directory should be the source directory for *IMPACT*, and the path to the lib directory should be the build directory for *IMPACT*.

The module files are now in place and have been edited, so the module library can be built.

## 4 Building the module library and driver

In order to build the *IcoFoam* module library, simply repeat the build steps taken when building *OpenFoam Extend* for the first time (see [Section 1](#)). When the building completes, ensure that the *IcoFoam* library built. It should be located under the `lib` directory of the *OpenFoam Extend* source. There may be another subdirectory under the `lib` directory like `linux64GccDPOpt`. If this is the case, the library should be in this subdirectory. It should be titled `libIcoFoamModule.so`. If the library is in a subdirectory or any location other than the `lib` directory under the main source directory, then create a link to `libIcoFoamModule.so` and the libraries located with it in the `lib` directory. An example command for creating this link is shown below

```
> ln -s /home/user/foam/foam-extend-3.1/lib/linux64GccDPOpt/*  
/home/user/foam/foam-extend-3.1/lib/.
```

This link will help when building the module driver using *CMake*.

If the library built successfully, the driver for the module may now be built and linked to both *IMPACT* and the module library. It is recommended that the driver source and build files be kept in separate directories from one another and from *OpenFoam Extend*. The module driver is built using *CMake* so this must be installed on the system. **Remember** that the same *mpicc*, *mpicxx*, and/or *mpi90* must be used for building *OpenFoam*, *IMPACT*, and the module driver. The build steps are as follows:

- Create a build directory for the driver.  
Example:  

```
> mkdir /home/user/icoFoamModuleBuild
```
- Change directories to the build directory.  
Example:  

```
> cd /home/user/icoFoamModuleBuild
```



- Run *CMake* on the module driver source directory with the `CMAKE_PREFIX_PATH` set to both the *IMPACT* install location (or locations of the *IMPACT* bin and lib directories) and the *OpenFoam Extend* source directory

Example:

```
> cmake -DCMAKE_PREFIX_PATH=/home/user/IMPACT-install\;  
/home/user/foam/foam-extend-3.1 /home/user/icoFoamModule  
(Note that there is no space between \; and /home/user/foam/foam-extend-3.1.  
The new line shown above is used only for visual clarity.)
```

- Run `make` and, if desired, `make install`.

Once the build process has finished ensure that the module driver built by checking the `bin` directory within the module driver build directory for `lcoFoamModuleDriver`.

## 5 Running and testing the *lcoFoam* module

Now that the library and driver for the module have been built, run the driver executable in the same manner in which *lcoFoam* was run in the example problem discussed in Section 9.2.2 of the *Uniphysics Validation for Development of Multiphysics Coupling in MP-Infra* document (mentioned in [Section 1](#)). Ensure that the module driver achieves the same results as using the `icofoam` executable.