Development of a GUI Based Front End for Open Source CFD Program, OpenFOAM

Sam Hee Han, Young Jin Lee, Hyong Chol Kim, Sun Byung Park, Hyun Jik Kim
Nuclear Safety Evaluation (NSE)
Corresponding author : yjlee@nse.re.kr

1. Introduction

A GUI (Graphic User Interface) based front end, Run_FOAM, was developed to run OpenFOAM CFD program[1] under Windows operating system. OpenFOAM is a free, open source CFD software package developed by OpenCFD Ltd. The package provides many ready-made numerical solvers, but users can also develop customized numerical solvers from the program sources provided in the package. However, OpenFOAM is sorely lacking in user friendliness as it runs in console mode under Linux. Run_FOAM was developed to greatly simplify the task of running an OpenFOAM calculation under Windows OS. Run_FOAM was written using Delphi object pascal language, and GLScene package was used for the 3D graphics. Verification of Run_FOAM was carried out by performing some OpenFOAM CFD calculations provided in OpenFOAM package, and these showed that the use of Run_FOAM is simple whilst providing sufficient allowances in user modifications.

2. Development of GUI Front End, Run_FOAM

2.1 Modification of OpenFOAM Windows Version

In order to run OpenFOAM under Windows, a version of OpenFOAM suitable for windows environment is needed. However, developing a windows version from the source is not a trivial matter because the program sources (written in C++ language) are designed for the Linux OS where the lower and the upper case letters are differentiated. Fortunately, a Windows version of the OpenFOAM can be downloaded from the hompage of blueCAPE[2] company. The OpenFOAM package by blueCAPE is released under GNU license[3] and blueCAPE provides a cross-compiled version of OpenFOAM package that contains OpenFoam in binary forms. The source forms of OpenFOAM can be downloaded from SourceForge website[4]. Installation of the package was found to be straight forward. After installation, the installed programs and scripts were modified in such a way that the entire package can be relocated to any given directory. In addition to OpenFOAM package, the paraView package[5], which is also released under GNU license, was installed and then it was also modified to be relocatable.

2.2 Development of GUI

Run_FOAM was developed using Delphi[6] program language and GLScene[7] 3D graphics package. Figure 1 shows the screen capture of an instance of Run_FOAM running under Windows OS.

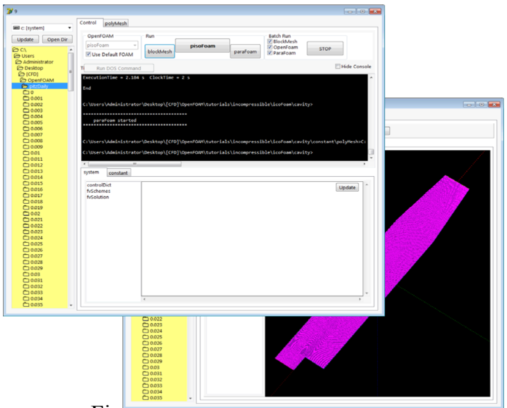

Figure 1. Screen Capture of Run_FOAM

The major functions of the program can be divided into 3 sections: 1) section for selecting a work directory, 2) section for adjusting controls for running OpenFOAM program, and 3) section for viewing the generated mesh.

The first section, selecting a work directory, was developed using the Delphi's DirectoryListBox object, by which a user can select the desired directory

The second section consists of a sub-section where problem configuration can be modified, and a sub-section to optionally select and run the meshing program 'blockMesh', the OpenFOAM solver, or the paraFoam post-pocessor. There is also a sub-section to show the console output. OpenFOAM configuration files such as 'controlDict', 'fvSchemes' and 'fvSolution' are shown and these can be modified on screen. Appropriate OpenFOAM solver is pre-selected by the GUI front end by examining the command in 'controlDict' file, but user can override and "type in" names of other solvers or select from a list of solvers.

The last section is used for viewing and examining the meshing scheme and user can view the mesh in text form or in 3D graphics. User can rotate and zoom to examine the mesh.

3. Exercise Calculations

Many sample CFD calculations are provided in the OpenFOAM package. In the package, there is the tutorial directory where about 100 cases of OpenFOAM sample calculation in 14 categories are provided. In the current study, 2 OpenFOAM cases from the package and 1 exercise case developed from scratch were used for verification calculation.

3.1 Calculation of a 2D airfoil

This exercise is given in the tutorial section of OpenFOAM package. It is an incompressible case solved with simple algorithm. Run_FOAM program was used to run the exercise. Running the problem with Run_FOAM was very simple in that all user had to do was to locate the directory using the GUI component and then press the button with the caption 'simpleFoam'. Run_Foam program automatically selects the solver, in this case simpleFoam, by analyzing the entry in 'controlDict' file. User, if he wishes, can disable the automatic solver selection and choose a different solver. After the solver finishes, user can press the button with caption 'paraFoam' to post-process the result with paraView. Figure 2 shows the post-processed results of the velocity distribution around the airfoil.

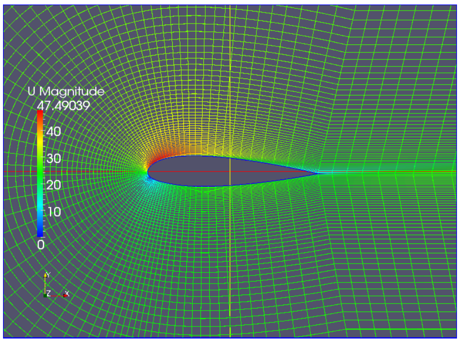

Figure 2. Velocity Distribution Around the Airfoil
(Post-Processing by ParaFoam)

3.2 Multiphase Bubble Column Calculation

A bubble column calculation using multi-phase solver, bubbleFoam, was carried out using Run_Foam similar to the previous exercise. Figure 3 shows the post-processed results for the void fraction.

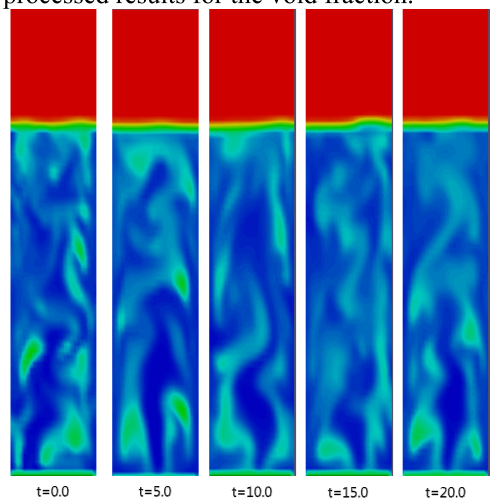

Figure 3. Void Fraction Visualization
(Post-Processing by ParaFoam)

3.3 A simpleFoam calculation of a complex pipe flow

As an exercise, an arbitrarily shaped duct was designed using a CAD program which is then exported as an IGES file. The IGES file was fed to Netgen[8] mesh generator program to generate a mesh. The mesh generated by Netgen was exported in OpenFoam format. The boundary conditions were set manually for t=0. Run_Foam was then run for this case, and Figure 4 shows the post-processed results.

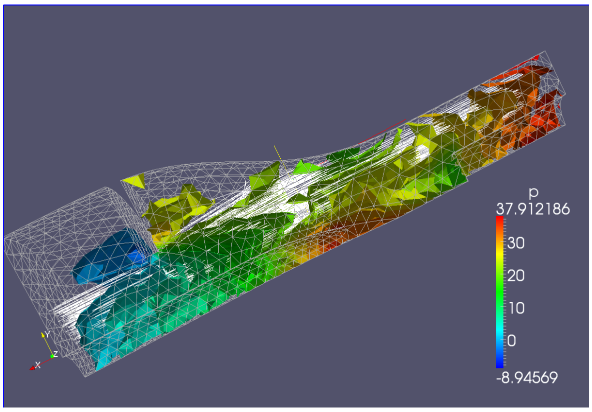

Figure 4. Pressure Distribution Visualization
(Post-Processing by ParaFoam)

4. Conclusions

Run_Foam, a GUI based front end program to simplify running OpenFoam CFD cases, has been developed. By incorporating numerous GUI in the program, Run_Foam has demonstrated that running an OpenFoam case can be easily accomplished. There is a potential for further development as the OpenFoam has the great advantage of being free to develop and to use. There is also a potential to couple or interface the OpenFoam with the systems analysis code such as Relap5.

REFERENCES

[1] "The OpenFOAM® Foundation", http://www.openfoam.org
[2] "Welcome to blueCAPE's webpage!", http://www.bluecape.com.pt
[3] "GNU General Public License", http://en.wikipedia.org/wiki/GNU_license
[4] "OpenFOAM - The Open Source CFD Toolbox", http://sourceforge.net/projects/foam/?source=directory
[5] "ParaView", http://www.paraview.org/
[6] "Borland Software Corporation, Delphi 6 for Windows Developer's Guide", 1998.
[7] "GLScene : OpenGL Solution for Delphi", http://glscene.sourceforge.net/wikka/HomePage
[8] "NETGEN - automatic mesh generator", http://www.hpfem.jku.at/netgen/