Open Source Computational Fluid Dynamic: Challenges and its Future

Nugroho Adi Sasongko[1)] Muhamad Fatikul Arif [2)]
[1)] Department of Mechanical Structural Engineering and Material Science - Offshore Technology, Universitetet i Stavanger, Norway
[2)] Department of Polymer Engineering, University of Minho, Portugal

Abstract

Computational fluid dynamics (CFD) is proved to be one of the vital part of the computer aided engineering (CAE) tools and it has been extensively used for various purposes. It approach to model fluid flow phenomena allows engineers and technical analysts to have a powerful fluid simulation on their desktop. However, almost all the available CFD softwares are licensed with very expensive price and it is impossible for the users to fully access and modify the numerical codes. The open source CFD is one of the solutions to solve these limitations. It allows the users to access and develop their own numerical modeling without pay any license. In this work, the milestones and development of CFD are discussed. The development of open source CFD and its comparison to the commercial CFD softwares are then overviewed. The goal of this work is to provide important information about the open source CFD software that becomes extensively used in both industry and academia. Among many open source CFD softwares, OpenFOAM is intensely discussed due to its popularity and performance.

Keywords: CFD, open source, OpenFOAM

I. Introduction

Milestones of computational fluid dynamics

CFD software has developed far beyond what Navier, Stokes or Da Vinci could ever imagine. It is a powerful tool that possible to simulate numerically fluid behavior formulated by them. The development of CFD is influenced by the milestones of many people. Here are some milestones in the early history of CFD:

Numerical methods were known since the Newton's time in 1700s. The solution methods of ordinary differential equations (ODE)s or partial differential equations (PDE)s were conceptually conceived, but only on paper. It is debatable, who actually performed the earliest CFD calculations, although Lewis Fry Richardson in England (1881-1953) developed the first numerical weather prediction system by dividing physical space into grid cells and using the finite difference approximations of Bjerknes's "primitive differential equations"[1]. In 1910, Richardson published 50 pages paper to Royal Society; he performed hand calculations by using human computers, which were iterated 2000 operations per week. His own attempt to calculate weather for a single eight-hour period took six weeks of real time and ended in failure.

- Since the 1940s, analytical solutions to most fluid dynamic problems, especially those arising in aerodynamics, were readily available for simplified or idealized conditions.
- 1943 – Finite element analysis (FEA) was first developed by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variation calculus to obtain approximate solutions to vibration systems
- Around 1960 - First Scientific American articles on CFD was published
- 1965 - Marker and Cell methods - Harlow & Welch
- 1965 - Use in research and "grand challenges" (NASA, Los Alamos...)
- 1970 - Finite difference methods for Navier-Stokes
- 1970 - Finite element methods for stress analysis
- 1980 - Finite volume methods (Imperial College). At the moment, the Finite Volume Method (FVM) method is the most common standard method in numerical fluid dynamic simulation.

Another key event in CFD industry was in 1980 when Suhas V. Patankar published "*Numerical Heat Transfer and Fluid Flow*". It is possibly the most influential book on CFD to date, and the one that spawned a thousand CFD codes[2].

- 1983 - Fluent was launched as commercial CFD software, afterwards a number of commercial CFD software growth and fulfilled the competition of CFD market.
- 1985 - Use in "aero" industries (Boeing, General Electric, ...)
- 1995 - Use in "non-aero" industries (General Motor, Ford, Astra, Ericsson...)
- 2004 - An open source CFD software (FOAM) was released.
- 2006 - Fluent was acquired by ANSYS, Inc. This acquisition makes ANSYS as a strongest computer aided engineering player in numerical FEA and CFD simulation.

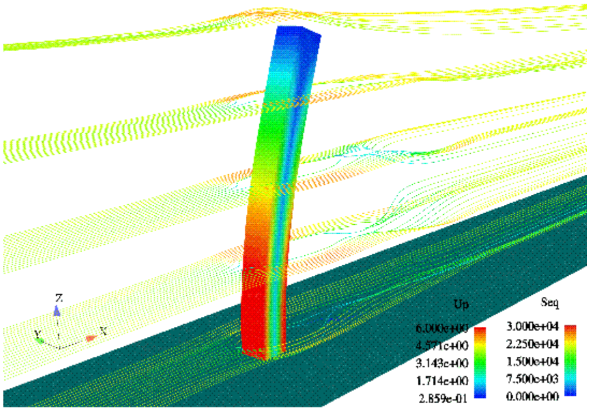

Figure 1. Fluid structure interaction simulation in OpenFOAM
[www.gompute.com]

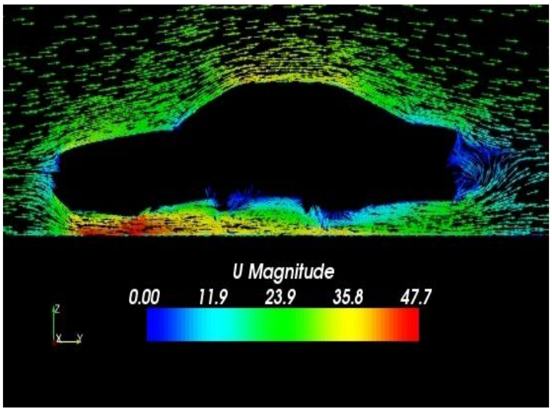

Figure 2. Velocity vector flow near the surface of the car for OpenFOAM on the central symmetry plane.
[www.afs.enea.it]

II. Open source computational fluid dynamic

Open source computational fluid dynamic is a tool that gives free angles of computation by freely available (without pay) and has free license under the GNU general public license. In simple think, we don't have to pay any cents even for massive parallel computers e.g. free CFD license for using in 1000 CPU.

Open source CFD Software is under active development, its capabilities mirror those of commercial CFD software. Open source CFD is a toolbox and it is not a black box that doesn't give any opportunities to the user to know what is the numerical modeling (represented by source code) behind it. Unlike in the commercial CFD packages, the user defined models in the open source CFDs is an integrated part of the main solver, which makes the models efficient.

Moreover, the open source CFD could deal with a specific problem by allowing the user to access the source and transforming a new function through the generic modules, for example:

The partial differential equation for momentum in a compressible flow:

$$\frac{\partial \rho U}{\partial t} + \nabla \bullet \rho U U - \nabla \bullet \mu \nabla U = -\nabla p$$

which is represented in C++ codes as[3]:

```
solve
(
   fvm::ddt(rho, U)
   + fvm::div(phi, U)
    - fvm::laplacian(mu, U)
    ==
    - fvc::grad(p)
);
```

A list of open source CFD softwares that available through the internet is presented below. Possibly there are still a lot of other open source CFD packages that are not listed in here.

- ADFC
- Applied Computational Fluid Dynamics
- CFD2k
- Channelflow
- Code_Saturne
- COOLFluiD
- Diagonalized Upwind Navier Stokes
- Dolfyn
- Edge
- ELMER
- FDS
- Featflow
- Femwater
- FreeFEM
- Gerris Flow Solver
- IMTEK Mathematica Supplement (IMS)
- iNavier
- ISAAC
- Kicksey-Winsey
- MFIX
- NaSt2D-2.0
- NEK5000
- NSC2KE

- NUWTUN
- OpenFlower
- OpenFOAM
- OpenFVM
- PETSc-FEM
- PP3D
- SLFCFD
- SSIIM
- Tochnog
- Typhon solver
- PRIN-3D

III. Comparison between commercial and open source software

In this part, the capability and performance of commercial and open source CFD is compared according to its user traffic and solver capability.

Number of user

So far, it could not be found how many users exactly for each commercial and/or open source CFD software groups in the world. It needs comprehensive research to have any statistical information about the user's number. To simplify the problem, the information available from CFD online forum is used. This forum can be reached from http://www.cfd-online.com/. How big the user's interest on the CFD can be seen indirectly from how much posts in every specific time in CFD online forum.

It is mentioned in the website that 2008 was a successful year for CFD Online. Both the number of read web pages and unique visitors (IP addresses) reached new record levels. The number of read web-pages and unique visitors increased by 12% and 23%, respectively compared to that of the previous years. The bar chart below shows the number of web pages delivered from CFD Online per month over the last 11 years.

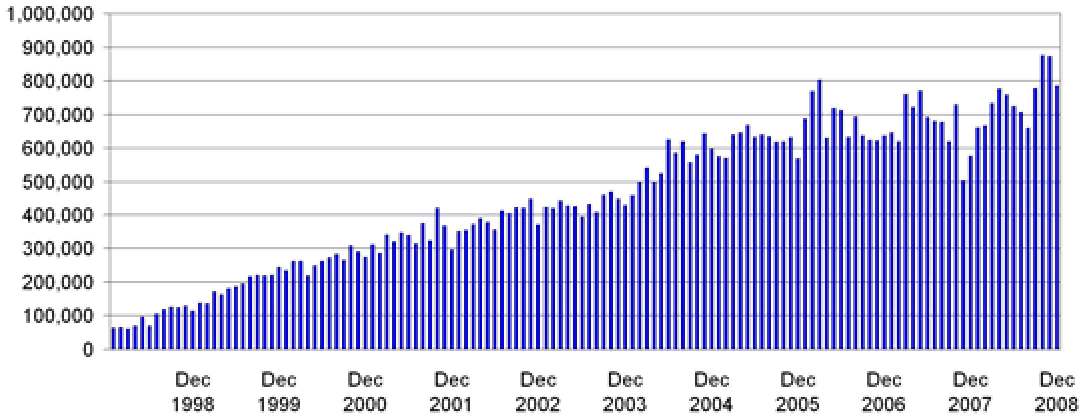

Figure 3. Number of web pages delivered from CFD online per month over the last 11 years [http://www.cfd-online.com]

Where they are from?

Visitors to CFD Online come from all over the world. The pie chart below shows the traffic distribution across different top-level domains.

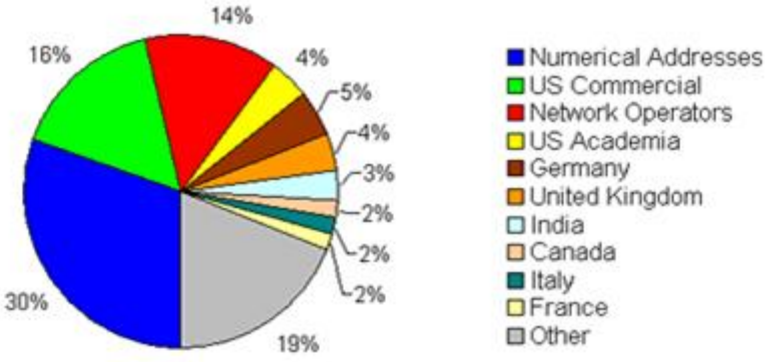

Figure 4. Traffic from different domains recorded in CFD online forum [http://www.cfd-online.com]

CFD online discussion forums

To see how big the activities and competitions between the CFD softwares, it can be traced also from the activities in the online forum. Based on the last activities on 11th October 2009, total threads in this forum is 68,559, posts 231,649, and total registered members is 8,095[4].

Table 1. Software User Forums that available in CFD Online Forum (11th October 2009)

Software User Forums	Type	Threads	Posts
ANSYS (CFX and Fluent)	Commercial	35,102	103,988
CD-adapco	Commercial	5,000	16,694
FloEFD, FloWorks & FloTHERM	Commercial	118	434
FLOW-3D	Commercial	240	1,093
NUMECA	Commercial	223	586
OpenFOAM	OpenSource	7,039	39,913
Phoenics	Shareware	1,288	3,778
Total		**49,010**	**166,486**

This data further can be represented in a simple bar chart as shown in Figure 5.

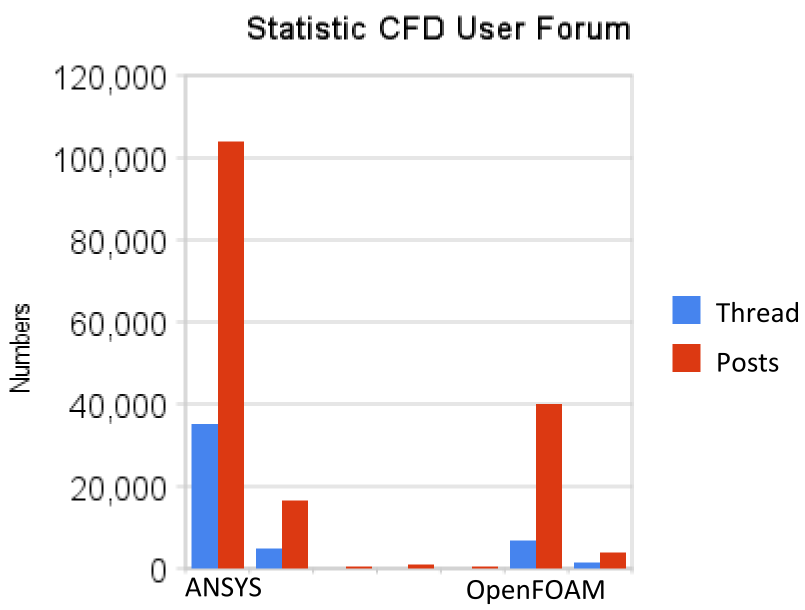

Figure 5. Statistic CFD User Forum by Chart (11th October 2009)

It can be seen from Table 1 and Figure 5 that ANSYS products are ranked in the first position, leave far behind other CFD softwares. ANSYS itself has 2 CFD commercial products which are CFX and Fluent.

OpenFOAM is ranked in the second position after ANSYS, based on the user activities communication or discussion. The rest CFD softwares (CD Adapco, FloEFD FloWorks & FloTHERM, FLOW-3D, NUMECA and Phoenics) are far away behind based on this statistical data.

Fluent itself has started its public forum since 1998 (already 11 years). Meanwhile OpenFOAM public forum has been started since 2002 or 7 years ago. It might be difficult to formulate the exact progress of each CFD softwares from the number of user interest. However, it could be inferred from these data, that the growth of user interest in open source CFD is significant.

Solver capability comparison

In this part, the capability of the open source CFDs will be compared to that of the commercial CFDs. To simplify the research, one candidate software from each group is chosen. For commercial CFD software, ANSYS Fluent is chosen due to its popularity and performance. In addition, ANSYS has almost all the capabilities that commercial CFD softwares have.

From the open source side, OpenFOAM is chosen since it is the most progressive open source software at the moment. OpenFOAM 1.6.x, the latest release of OpenFOAM will be used in this benchmarking. Based on the author experience on both softwares, these comparisons could be assured as the best setup. Thus, it is expected that the comparison is fairly rated.

In this study, the basic information about OpenFOAM from Christian Andersen and Niels E. L. Nielsen Master Thesis report February 2008 (Aalborg Universitet, Denmark) is used[5].

Since the thesis work was performed in 2007 and the OpenFOAM 1.4.1 version (release 2007) was used, it is worthy to employ the capability of the latest OpenFOAM 1.6.x (release July 2009) for benchmarking in this work.

A. Installation

Table 2. Installation of OpenFOAM and Fluent

	OpenFOAM	**ANSYS Fluent**
Operating system compatability	Unix Linux MS Windows (built with MinGW C++ as a set of native windows applications, which improves performance and eliminates the need for Unix emulations). MacOSX	Unix Linux MS Windows MacOS
Installation	Not Easy	Easy
License	Free under GNU license	Expensive
Software upgrading	No Cost	Need cost

B. General setting

In this part, the software capabilities in the case of running-time are investigated.

Table 3. Runtime options and general considerations

	OpenFOAM	**ANSYS Fluent**
Parallelisation	Yes	Yes
Scripting	Yes (C++)	Yes (but not good)
Solution monitoring	Yes	Yes
On-the-fly post-processing	Yes (PyFoam)	Yes
Adjustment of parameters during solving	Yes (easy)	Yes (could be better)
Discretization	FVM	FVM & FEM (CFX)

C. Physical models

In this part, the possibilities of real physical phenomena that could be approached by numerical simulation in both softwares are presented.

Table 4. Physical models available in OpenFOAM and Fluent

	OpenFOAM	ANSYS Fluent
Change in Density		
Incompressible flow	Yes	Yes
compressible flow	Yes	Yes
Chemistry		
Transient global/elementary reactions	Yes	Yes
Steady global/elementary reactions	No	Yes
Thermodynamics		
Conduction	Yes	Yes
Convection	Yes	Yes
Radiation	Yes	Yes
Particle Flow		
Particle tracking	Yes	Yes
Evaporation	Yes	Yes
Particle Combustion	Yes	Yes
Multiphase Flow		
VOF	Yes	Yes
Euler - euler	Yes	Yes
Euler - lagrange	Yes	Yes
Phase Change		
Due to cavitation (pressure)	Yes	Yes
Due to heat transfer (temperature)	No	Yes
Movement by Torque	No	Yes
Direct simulation Monte-Carlo (DSMC)	Yes	Yes

D. Mesh and boundary conditions

Table 5. Mesh types and boundary conditions available in OpenFOAM and Fluent

	OpenFOAM	ANSYS Fluent
Mesh		
Polyhedral (tets, hex etc.)	Yes	Yes
2D and axi-symmetric	Yes (manipulated by 3D with 1 cell thick)	Yes
Dynamic Mesh	Yes	Yes
MRF (multi reference frame)	Yes (has a new approach General Grid Interface (GGI))	Yes
BC type		

Exterior/interior wall	Yes	Yes
Interior face	No	Yes
Porous jump	No	Yes
Porous zone	Yes	Yes
Pressure outlet, pressure inlet, velocity inlet	Yes	Yes
Mass flow inlet	Yes	Yes
Exhaust fan, inlet vent, outlet vent, intake fan	No	Yes
Outflow, pressure far field	No	Yes

E. Solver setup

Table 6. Solvers and schemes available in OpenFOAM and Fluent

	OpenFOAM	**ANSYS Fluent**
Graphical user interface (GUI)	Not yet (It makes OpenFOAM is not popular for ordinary user)	Yes
Laminar flow, transient, and steady state	Yes	Yes
Turbulence models		
k-e, k-w, (wall functions)	Yes	Yes
Large Eddy Simulation	Yes	Yes
Detached eddy simulation Spalart-Allmaras	Yes	Yes
Direct Numerical Simulation	Yes	Yes
Numerical Interpolation scheme		
Spatial 1. order upwind (upwind)	Yes	Yes
Spatial 2. order upwind (Quick)	Yes	Yes
Time 1. order (Euler)	Yes	Yes
Time 2. order (CrankNicholson)	Yes	Yes
Pressure velocity coupling		
Simple	Yes	Yes
PISO	Yes	Yes
Coupled (Pimple)	Yes	Yes

F. Post processing

Table 7. Post processing utilities available in OpenFOAM and Fluent

	OpenFOAM	**ANSYS Fluent**
Data Conversion	foamDataToFluent foamToEnsight foamToEnsightParts	Fluent to CFX

	foamToFieldview9 foamToGMV foamToVTK smapToFoam	
Loop Calculation	Yes	Yes
Patch (average and integration)	Yes	Yes
Sample & Probe	Yes	Yes
Stress Component	Yes	Yes
Velocity Field (Mach, Co, enstrophy, vorticity)	Yes	Yes
Wall Function (Grad, Heat Flux, Shear stress, yPlusLES, yPlusRAS)	Yes	Yes
and so on		

IV. Community and development

Along the development time of open source CFD software, it is also followed by the growing of the open source communities in many countries. Many reasons are standing behind this issue. For example, having contact to the community will ease the exchange of the information regarding to the numerical research for industrial purposes.

Each community mostly has special research interest, for example to solve special physical problem in turbo machinery in hydropower industry, multiphase flow for oil and gas industry, turbulence research in wind turbine technology or even to solve the financial problem for trading and banking.

Every year, open source community on CFD has organized international conference for discussing the research and their progress. Started from the first conference in London UK in 2007, then in 2008 they continued their meeting in Berlin Germany. Afterwards, Barcelona Spain became preferred place for 2009 meeting.

Some experts and professionals in open source CFD also prepare the international workshop for OpenFOAM. This workshop is organized by The OpenFOAM Workshop Committee from different institutions in many countries. Here, in Table 8, is listed of places where the workshops were organized[6].

Table 8. The International open source CFD codes - OpenFOAM workshop

Workshops	Places and Dates
First OpenFOAM Workshop	Zagreb, January 26-28, 2006.
Second OpenFOAM Workshop	Zagreb, June 7-9, 2007
Third OpenFOAM Workshop	Milano, July 9-11, 2008
Fourth OpenFOAM Workshop	Montreal, June 1-4, 2009
Fifth OpenFOAM Workshop	Gothenburg, June 21-24 2010

Many countries in their research institutions and universities boost the effort and energy to develop the open source CFD softwares. Some countries those show significant progress on this research are Germany, UK, Sweden, Denmark, Italy, India, Japan, China and Brazil.

Danish CFD community

In recent years, CFD has been extensively used within Danish industry as a computer aided engineering (CAE) tool. However the use of commercial programs for CFD simulations burdens in many companies. Among the reasons are the lack of competition, which yields to an expensive commercial license price, and the fact that it is virtually impossible to exchange data between companies using different programs.

In 2008, Danish research industry and the universities start a national initiative to promote open source CFD in Denmark. Leading by eight Danish companies and two universities, they have combined their efforts in development of the Danish open source software platform for CFD. OpenFOAM is intensely projected as an alternative to commercial CFD programs.

Those 8 institutions build a group meeting called **OSFRI meeting (styregruppemøde)[7].** They have established a project to speed up the development of the open source CFD programs. The project will run for a period of 18 months and has received a grant of DKK 3.44 million from the Danish Agency for Science, Technology and Innovation. Two universities (Aalborg Universitet and Technical University of Denmark) are the representation from education institution in this project. From industry side, they are FORCE Technology, Grundfos Management, GEA Niro, MAN Diesel, ODS Lloyd's register, Babcock & Wilcox Vølund, Haldor Topsøe and FLSmidth Airtech.

Swedish CFD community

Research in open source CFD leads by Fluid Mechanics research division, Chalmers University of Technology, Gothenburg. Prof. Håkan Nilsson established a special PhD course in CFD with OpenSource software OpenFOAM since 2006[8]. His turbomachinary group develops codes to simulate the cavitations and rotating flow. Open source CFD course is also teaches in Lund University, Lulea University of Technology and Mälardalen University by some CFD experts.

Niklas Nordin and his research group in Scania Group Industry have developed the powerful solver for combustion, called dieselFoam. Swedish companies like Volvo, SKF and Sab begin to use the open source CFD code in their industrial purposes.

German CFD community

Many research institutions and universities in German have started to work with OpenFOAM. They arrange regular meetings (Stammtisch) of OpenFOAM users in almost every half year in many places. Some area of the meeting points such as Stuttgart / Heilbronn area, Munchen, Erlangen, Berlin, Braunschweig, and Rostock.

TU Munchen, leads by Holger Marschall from Chemical Engineering Department and open source CFD community have started to initiate a FOAM Documentation Project elaborating the worldwide research in open source CFD. By these activities, it is a hope that all the research can be well documented in one place. This project can be visited on their website http://www.foam-documentation.org.

UK CFD community

Original development of OpenFOAM started in the late 1980s at Imperial College, London, motivated by a desire to find a more powerful and flexible general simulation platform than the defacto standard at the time, FORTRAN[9]. Since then it has evolved by exploiting the latest advanced features of the C++ language. The predecessor, FOAM, was sold by UK Company Nabla Ltd. before being released open source in 2004. Since 2004, OpenCFD Ltd develops and maintains the official release of open source CFD OpenFOAM to the public[10]. So far a number of universities in UK are working with OpenFOAM in their research such as University of Exeter, University of Strathclyde, and Kingston University London.

Norway CFD community

Universitetet i Stavanger will start the Master Course in the CFD and Heat Transfer in the next spring semester 2010. This course becomes special because the Open Source CFD (OpenFOAM) will be used as a modeling tool.

IRIS - Risavika Gas Centre (Gas and Energy Division) works on the numerical simulation of multiphase flow with heat transfer for CO_2 capture and sequestration project. This institution has published papers on this research and has developed a new case and tutorial for compressibleInterFoam + heat transfer. Petroleum Reservoir Division of IRIS in Bergen has started to work on porous medium problem. The goal of this research is to study the CO_2 storage under geological formation.

NTNU and SINTEF have established research collaboration on multiphase flow by using OpenFOAM. Their Laboratory works on the validation of the numerical simulation with the experimental results. The projects currently run are bubble column chemical reactor and multiphase flow simulation in the offshore production line (gas - oil – sand).

V. Challenges

The Open Source CFD code that growing very fast, has a good prospect in the future. Continuous researches have been conducted in developing the open source CFD. It is predicted that in a short time, open source CFD will have same peer to peer capability against commercial CFD software. However, big numerical scientific efforts are required when using open source programs compared to the commercially available programs. Working with open source CFD codes faces less support, difficultness in numerical coding and lack of proper manuals, so that greater resources are required to familiarize the users in the programs.

REFERENCES

1. Suzzi, D., *Diesel Nozzle Flow and Spray Formation: Coupled Simulations with Real Engine Validation*, in *Institute of Aerospace Thermodynamics*. 2009, Universität Stuttgart: Stuttgart. p. 118.
2. FLUENT, A. *A Brief History of CFD*. 2009 [cited 2009; Available from: http://www.fluent.com/about/cfdhistory.htm.
3. *OpenFOAM User Guide Version 1.6.* 2009.
4. *CFD Online Forum*. 2009 [cited 2009 October]; Available from: http://www.cfd-online.com/.
5. Andersen, C., *Numerical investigation of a BFR using OpenFOAM*, in *AAU - Institute of Energy Technology*. 2008, Aalborg University: Esbjerg. p. 57.
6. Wikki_Ltd. *OpenFOAM Workshops and Training*. 2008 [cited 2009; Available from: http://foamcfd.org/.
7. OSFRI. *Open source CFD i Dansk Industri*. [cited 2009; Available from: http://www.osfri.dk/.
8. Nilsson, H. *Håkans Homepage*. 2009 [cited 2009; Available from: http://www.tfd.chalmers.se/~hani/.
9. Wikipedia. *OpenFOAM*. 2009 3 August 2009 [cited 2009 October]; Available from: http://en.wikipedia.org/wiki/OpenFOAM.
10. OpenCFD_Ltd. *OpenFOAM official website*. 2009 [cited 2009 October]; Available from: http://www.opencfd.co.uk/.