OPENFOAM

(OPEN SOURCE FIELD OPERATION AND MANIPULATION)

DEVELOPED BY – CFD DIRECT

REPORT BY – GIRISH NIGAMANTH RAGHUNATHAN

NUMERICAL SCHEME & FEATURES

- OpenFOAM is a free open source CFD software which uses C++ for performing numerical solutions which are done using a finite volume technique. [1]
- Capable of solving Multiphase flows, Reynolds averaged simulations & Large Eddy simulations, Compressible/Thermal Flows, Porous Media and conjugate heat transfer problems. [2]
- The software is supported across all platforms (Windows, Linux and Mac) and is has free license for all users. [1]
- Uses cloud computing technique and there occupies much lesser memory space than any other CFD solver. [3]
- The software lacks in-built GUI but can be used alongside several other third party software (like paraView) for the purpose of geometry creation & post-processing. [4]

PROS AND CONS (COMPARED TO FLUENT) STRENGTHS

- Open source software with 100% free license and readily available online as a free download. [4]
- Availability of a complete documentation of the source code attached with capability of changing the same. [5]
- Cloud based computing (massive parallel computing capability) making it much faster than Fluent.
- Tutorial files available along with the software and contains all information needed for its running.[4]

WEAKNESSES

- Lack of in-built GUI making it difficult for providing boundary conditions, material properties etc.
- Tedious installation process in the windows platform as it requires a linux based console for running. (Based of personal experience)
- Lack of generic solving models and reliance on code based solving making it tough to learn.
- Difficult to get customer support [6].

AVAILABILITY, COST & USER BASE

- Free open source software which comes up with unrestricted access to the same. [1]
- Readily available online for download from the authentic website. [4]
- Supported across all platforms including Linux. [4]
- Used in a lot of research applications owing to the transparency in the source code making it easier to establish a control over the solution. [3]
- Used in automotive industries in recent times owing to the lack of any license cost.

RECOMMENDATION

- Given its ready availability and quickness due to its computing technique, it easier to perform simple calculations using OpenFOAM compared to Fluent.
- However for solving complex multiphase problems the lack of readily available modules makes it difficult to be performed using OpenFOAM.

REFERENCES

- [1] http://cfd.direct/openfoam/user-guide/introduction/#x3-20001
- [2] http://cfd.direct/openfoam/features/
- [3] http://www.totalsimulation.co.uk/wp/why-openfoam/
- [4] http://www.openfoam.com/download/install-windows.php
- [5] http://cpp.openfoam.org/v4
- [6] https://www.hamakor.org.il/wp-content/uploads/2013/06/OF-presentation-mil-oss-sanitized.pdf