RET | ENGINEERING
TEXAS A&M UNIVERSITY
Research Experience for Teachers

Engineering Mechanics: Simulating 3-D Models with OpenFOAM

Dr. Goong Chen, Stefan San Miguel, and Michelle A. Williamson
Department of Mathematics, Texas A&M University

Abstract

Application problems that capture students' interest and represent "real world" situations such as the crash of the Malaysian Airline Flight MH-370 are seldom explored within the high school classroom. Science, technology, engineering and math fields (STEM) have many situations where knowledge of 3-D modeling and coding is needed to simulate the physics of the "real world." Computer-generated 3-D models using open-source programs such as OpenFoam have a great potential for use, exploration, and development in the 9th - 12th mathematics classrooms. Through exploration, students can visualize and virtually interact with the geometry of simple and complex models and the physical law constraints of those models in "real world" situations.

Research Objectives

The goal of the RET program sponsored by Texas A&M University's ETID Department is to provide secondary educators the opportunity to engage in research and curriculum development with faculty mentors. Throughout this research, the objective is to use OpenFOAM to model simple and complex flow models.

Research Goals:

- Learn how to effectively use OpenFOAM
- Refine meshes of the simple objects.
- Model incompressible, multi-phase, and buoyancy driven flows.
- Create simple 3-D models with BlockMesh (an OpenFOAM tool) specifically; box, box with hole in the middle, column of water breaking over a dam.
- Create 3-D models with snappyHexMesh (an OpenFOAM tool) specifically; object floating on water, motorbike, and the Boeing 777 aircraft.
- Use ParaView (an OpenFOAM processing tool) to render simple and complex physics of geometric shapes.

http://etidweb.tamu.edu/hsieh/ResExp-Teachers

Methodology

The Open Field Operation and Manipulation (OpenFOAM) Toolbox is an open source computational fluid dynamics software package. OpenFoam comes with a wide variety of features which include tools for generating 3-D meshes to solving complex solid dynamics. It also includes pre- and post-processing subprograms to compile numerical data and create visual representations. This research is divided into four main parts: effectively learning how to use OpenFOAM, using BlockMesh to generate simple geometries, using ParaView to render data into an interactive 3-D model, and utilizing snappyHexMesh to generate complex geometries. This process required starting with simple shapes and working up to the complex geometry of an airplane.

Results and Conclusions

OpenFoam was effectively utilized to explore physical applications of simple and complex 3-D generated models. The tool, ParaView, rendered 3-D meshes compiled using blockMesh to map water pressure, velocity, and stress through a timed fluid dynamics flow to simulate real-world conditions. SnappyHexMesh built complex geometries such as the Boeing 777.

ParaView: a post-processing tool and rendering engine used in OpenFOAM

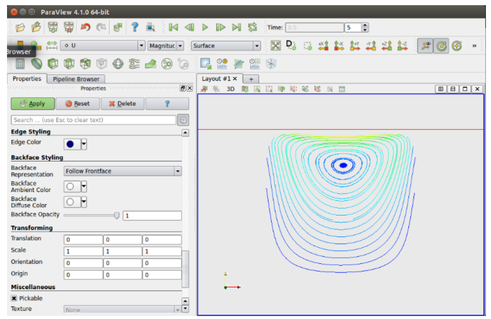


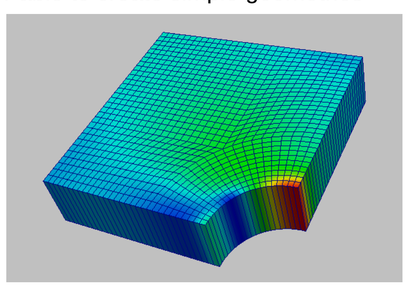
Figure 1 (to the left)

A trace streamer displaying the magnitude and directions of the velocity as a liquid flows into a cell from the top, once full, it then continues to flow across the top of the cell.

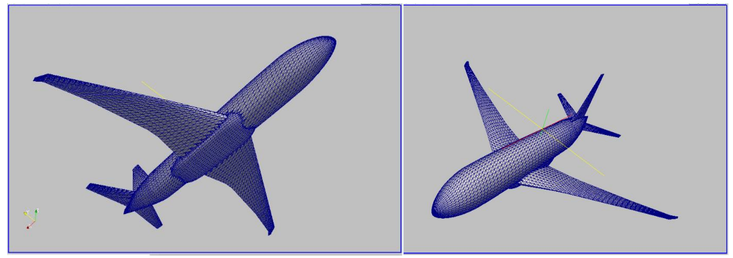
BlockMesh – a multi-block mesh generator able to create simple geometries

Figure 2 (to the right)

A stress analysis of a 3-D generated plate with a hole through the center. Due to the symmetry of the plate, it is only necessary to view a quarter of this model.



snappyHexMesh – a mesh generator that meshes surfaces from CAD (Computer Aided Design) to create complex geometries.



Figures 3 & 4 (above**)** show two views of the Boeing 777's complex geometry.

Engineers and mathematicians alike use computer fluid dynamics and rendering software like OpenFOAM to explore the physical dynamics of solids interacting with other objects, such as in the study Aerodynamics and Fluid dynamics.

OpenFOAM offers students the opportunity to learn computer programing, data visualization, and 3-D mesh construction. These opportunities allow them to connect mathematical concepts to both "real-world" and current engineering applications. For example, OpenFOAM can be used to explore the geometry of individual parts of a physical model and how they impact the efficiency of a design. Understanding how to code, design, and test fluid dynamics flows provides students with a hands-on learning advantage. Students gain access to STEM (Science, Technology, Engineering, and Math) research and future employment opportunities.

References

Chen, Goong et al. 'Malaysia Airlines Flight MH370: Water Entry Of An Airliner'. *Notices Amer. Math. Soc.* 62.04 (2015): 330-344. Web.

Openfoam.com,. 'The Open Source Computational Fluid Dynamics (CFD) Toolbox'. N.p., 2015. Web. 2 July 2015.

Waller, H, C Santos, and R Jones. *Openfoam*. OpenCFD, 2015. Print.

Acknowledgements: This material is based upon work supported by the Research Experiences for Teachers Program under National Science Foundation under Grant No. 1300779. Any opinions, findings, and conclusions or recommendations expressed in this material are those of the author(s) and do not necessarily reflect the views of the National Science Foundation.