EasyCFD: Automated Open Source Workflow for Computational Fluid Dynamics

University of New Hampshire NH College of Engineering and Physical Sciences

Kevin Cole

Advisor: Ivaylo Nedvalkov

Difficulties of OpenFOAM

- User must know simulation and OpenFOAM case format thoroughly to correctly set up
- No graphical user interface, only command line ٠
- Easy to lose track of changes made to simulation settings
- DEKA avoids these difficulties through their own ٠ workflow which utilizes some commercial software

Solution

- · EasyCFD is an automated and open source software workflow which makes CFD simulations much simpler to set up and run
- Make OpenFOAM easy and accessible
 - Maximize work done by computer and minimize work done by human user without sacrificing accuracy

/ ezFOAM Answer all the questions below and select Build Case for fast simulation set-up Select su2 geometry file Select simulation destination test_clean Browse Browse Pressure Specify inlet flow rate and units Specify flow type Ente Air Steady Water Steady Multiphase m/sm3/s Velocity 0 Enter the number of processor cores you wish to use (1 or more) s the flow internal or external k, epsilon, nut, etc. Enter External Enter the desired angle of attack ranging from -30 degrees to 30 degrees Mesh generation software saves faces in su2 format to ; polyMesh Build OpenFOAM Case define boundary conditions, which OpenFOAM can't do Enter triSurface constant transportProperties Uses inputs from ezFOAM main.py to call functions below ; controlDict system - fvSchemes su2ToSTL.py makeSystem.py , fvSolutions make0.py makeConstant.py scripts.py Defines initial Converts su2 Defines geometry and Defines simulation Makes the <just_prep> <just prep> groups.py geometry file conditions in constant flow parameters settings in the and <snap_solve> scripts Groups coordinates to STL format the 0 folder in the constant folder system folder for automating simulation <snap solve> to their respective boundary conditions Sample Results Velocity (m/s) Phase Fraction

0.0e+00 1 1.5 2 2.5 3 3.5 4 4.5 5 5.56.3e+00

Water around a wingtip

0.2 0.3 0.4 0.5 0.6 0.7 0.8

Multiphase case of water filling an empty container

Objective

To break the status-quo in education and industry that computational fluid dynamics (CFD) software is difficult and expensive to use

- SolidWorks Flow Simulation and ANSYS FLUENT are popular yet cost thousands of dollars and are black boxes
- OpenFOAM has a steep learning curve and is timeconsuming to operate but is open source

Stage 1: Geometry Preparation through enGrid



Stage 3: Simulating through blueCFD-Core

Acknowledgements: Sofia Lemons, John Mannisto

Velocity (m/s)

Air around a cylinder

1.5

2.7e+00

0.5

Visit our YouTube channel for tutorial videos:

0.0e+00



1.0e+00