

## 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 time-consuming to operate but is open source

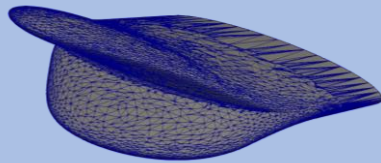
## Difficulties of OpenFOAM

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

## Solution

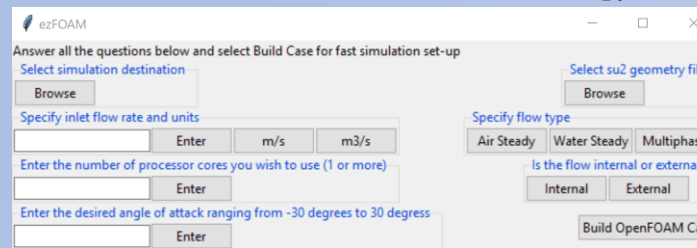
- EasyCFD is an automated and open source software workflow which makes CFD simulations much simpler to set up and run
- Make OpenFOAM easy and accessible
  - Maximize work done by computer and minimize work done by human user without sacrificing accuracy

## Stage 1: Geometry Preparation through enGrid

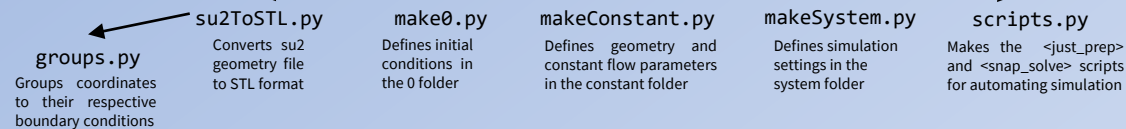


Mesh generation software saves faces in su2 format to define boundary conditions, which OpenFOAM can't do

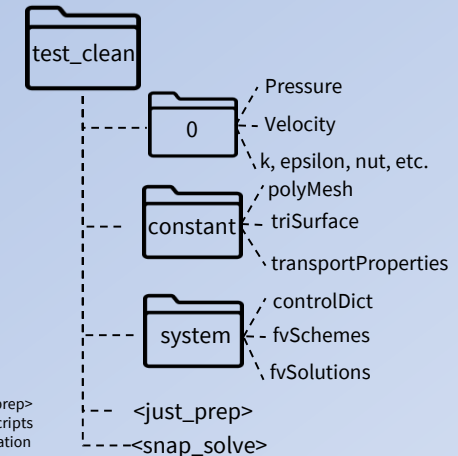
## Stage 2: Case Construction through ezFOAM.py



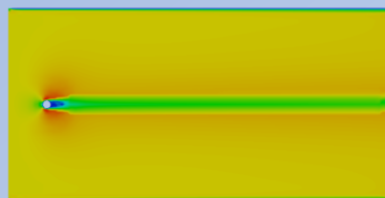
main.py Uses inputs from ezFOAM to call functions below



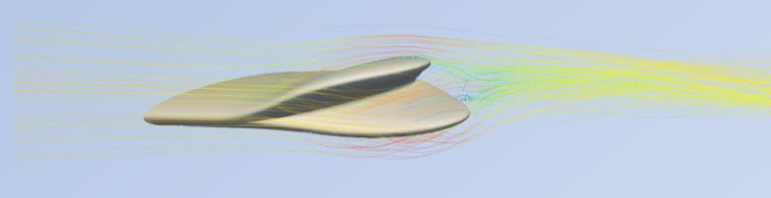
## Stage 3: Simulating through blueCFD-Core



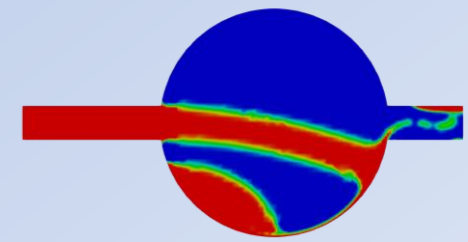
## Sample Results



Velocity (m/s)  
0.0e+00 0.5 1 1.5 2 2.7e+00  
Air around a cylinder



Velocity (m/s)  
0.0e+00 1 1.5 2 2.5 3 3.5 4 4.5 5 5.5 5.63e+00  
Water around a wingtip



Phase Fraction  
0.0e+00 0.2 0.3 0.4 0.5 0.6 0.7 0.8 1.0e+00  
Multiphase case of water filling an empty container

