

CFD Through OpenFOAM II

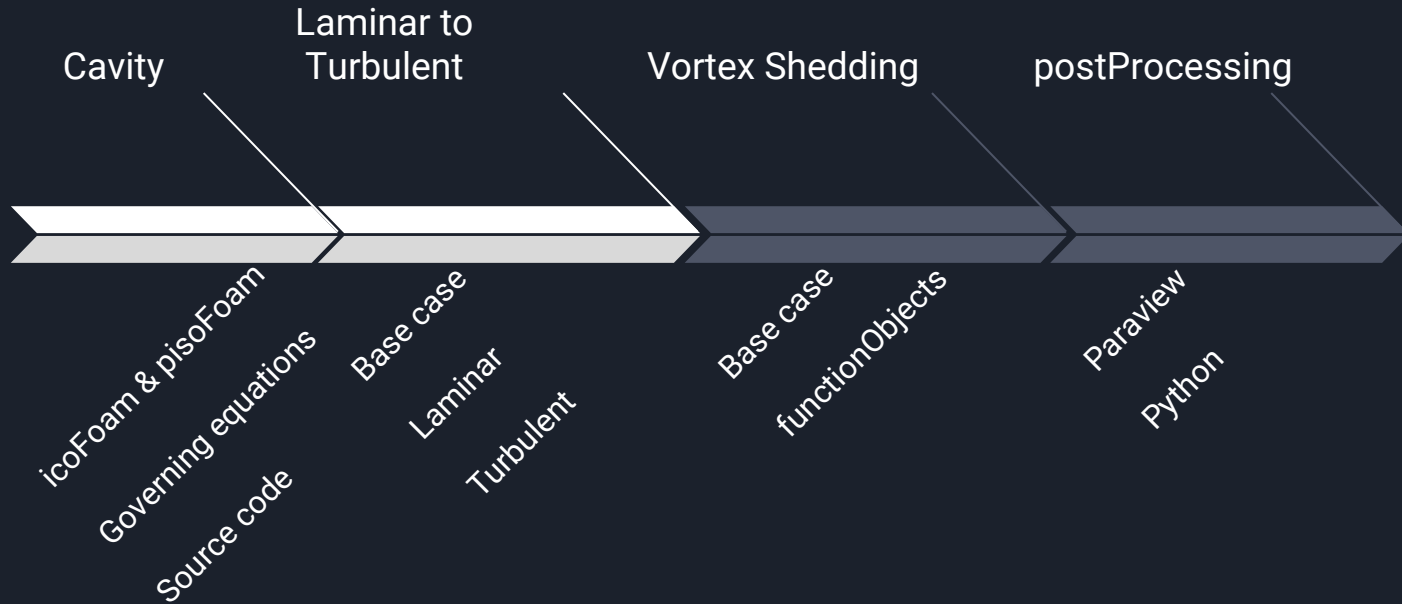
CACDS Workshop, March 13-15, 2018

Taher Chegini : tchegini@uh.edu

Amit Amritkar: armakritk@central.uh.edu



Roadmap





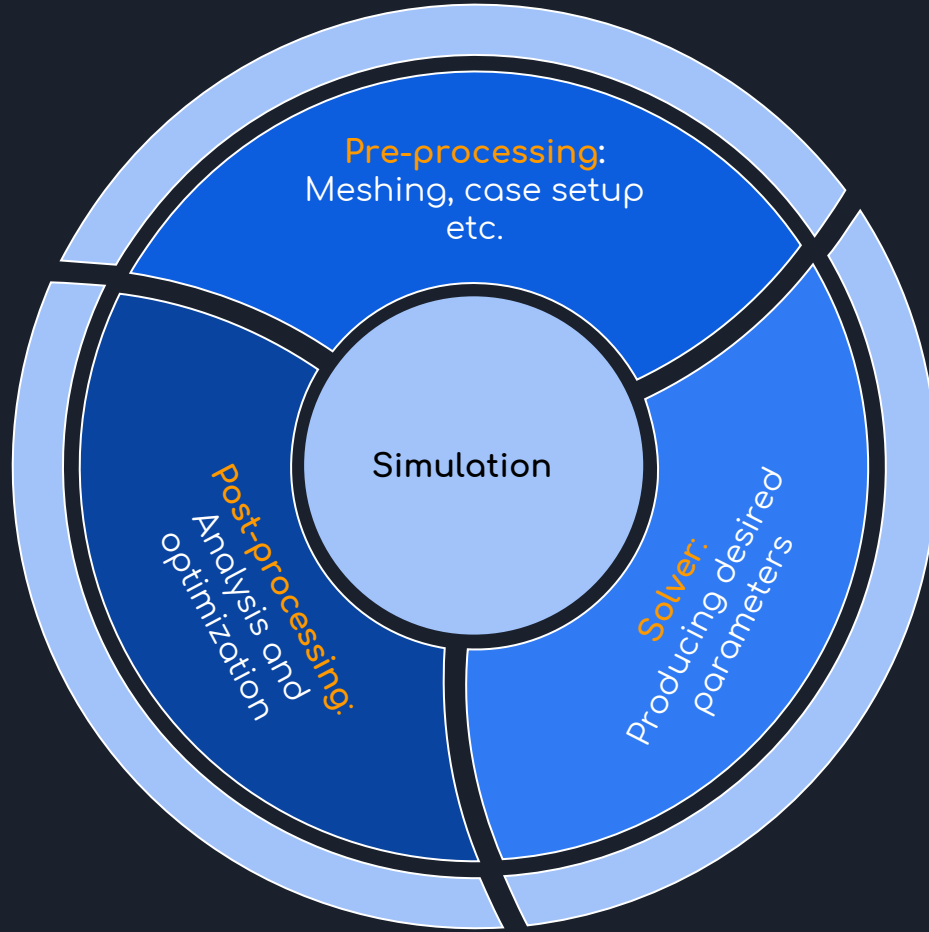
Acquaintanceship

01 Previous workshop?

02 Linux users? Open Source vs. Closed Source, Terminal vs. Mouse

03 CFD experience? FVM, Numerical Methods, Scientific Computing, Visualization

04 OpenFOAM users? C++, Source Code, Parallel Programming





Geometric patch types

- **patch**: generic type containing no geometric or topological information about the mesh, e.g. used for an inlet or an outlet.
- **wall**: for patch that coincides with a solid wall e.g. wall functions in turbulence modelling.
- **symmetryPlane**: for a planar patch which is a symmetry plane.
- **symmetry**: for any (non-planar) patch which uses the symmetry plane (slip) condition.
- **empty**: for solutions in in 2 (or 1) dimensions (2D/1D)
- **wedge**: for 2 dimensional axi-symmetric cases, e.g. a cylinder
- **cyclic**: enables two patches to be treated as if they are physically connected

<https://goo.gl/7vzHUS>



Basic boundary conditions

fixedValue: value is specified

fixedGradient: normal gradient is specified

zeroGradient: normal gradient is zero

calculated: value calculated from other fields

mixed: mixed fixedValue/fixedGradient condition

<https://goo.gl/7vzHUS>



Hands-on



```
$ run  
  
$ mkdir UHOF  
  
$ cd UHOF  
  
$ git clone https://github.com/taataam/UHOFWorkshop.git  
  
$ cd UHOFWorkshop/validation_cases
```