

# Applied Computational Fluid Dynamics using OpenFOAM

Mr. Kishore Rangarajan

**KCT**

*Mon & Thu : 5 PM to 7 PM*

Dr. Lakshman Anumolu

Dr. Kumaresh Selvakumar

Mr. Shyam Sundar J (TA)

**ExaSlate**

Course co-ordinators

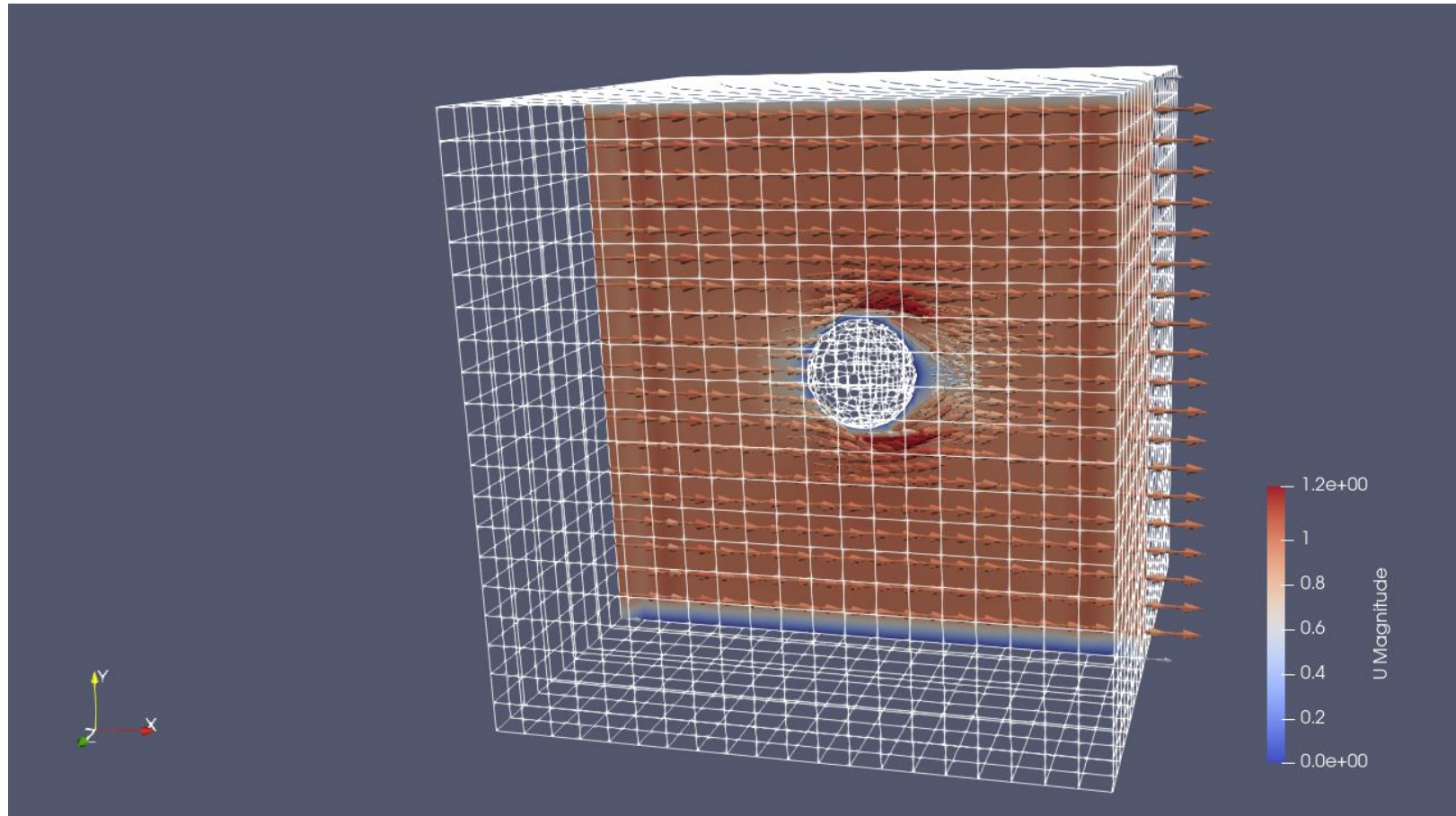
- Mr. Blesscin
- Mr. Dhanush

# Overview

- Introducing snappyHexMesh
  - Flow over a sphere
- How to fix minor and common case setup issues

# Flow over a Sphere

# Flow over a Sphere



# Case Setup Minor Issues

```
openfoam~/OpenFOAM/openfoam-v2312/run/applied-cfd-using-openfoam-kct-fall2024/Day-8/flowOverASphere_fixesRequired $ blockMesh

=====
\\  F i e l d      | OpenFOAM: The Open Source CFD Toolbox
\\  O p e r a t i o n | Version: 2312
\\  A n d             | Website: www.openfoam.com
\\  M a n i p u l a t i o n |
=====
\\*-----*\\

Build : 1d8f0d55f7-20231221 OPENFOAM=2312 version=v2312
Arch  : "LSB;label=32;scalar=64"
Exec  : blockMesh
Date  : Sep 22 2024
Time  : 21:50:04
Host  : 329c562e37b8
PID   : 1182
I/O   : uncollated
Case  : /home/openfoam/OpenFOAM/openfoam-v2312/run/applied-cfd-using-openfoam-kct-fall2024/Day-8/flowOverASphere_fixesRequired
nProcs : 1
trapFpe: Floating point exception trapping enabled (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)
allowSystemOperations : Allowing user-supplied system call operations

// *****
Create time

--> FOAM FATAL IO ERROR: (openfoam-2312)
Lookup:stopAt enumeration endtime is not in enumeration: (endTime noWriteNow writeNow nextWrite)

file: system/controlDict at line 17 to 49.

    From bool Foam::Enum<EnumType>::readEntry(const Foam::word&, const Foam::dictionary&, EnumType&, bool, bool) const [with EnumType = Foam::Time::stopAtControls]
    in file lnInclude/Enum.C at line 259.

FOAM exiting
```

# Exercises-8

- <https://github.com/exaslate-learn/applied-cfd-using-openfoam-kct-fall2024/discussions/9>