



# An overview of OpenFOAM

Arash Alizad Banaei/PDC



# What is OpenFOAM?

Open Source Field Operation and Manipulation

A computational fluid dynamics (CFD) software

Solves equations of fluid motion with finite volume method

First developed by OpenCFD Ltd

Open source

The source code is written in C++

# OpenFOAM versions

- There are two main variants of OpenFOAM:
  - OpenCFD version :
    - New versions released twice a year (June-December)
    - Versions are named as (vYYMM) like v1912
    - Website: <https://www.openfoam.com>
  - OpenFOAM foundation version :
    - No defined date for new release
    - Versions are named with with two digits like 4.1
    - Website: <https://openfoam.org>

# OpenFOAM versions

## What is the story behind this?

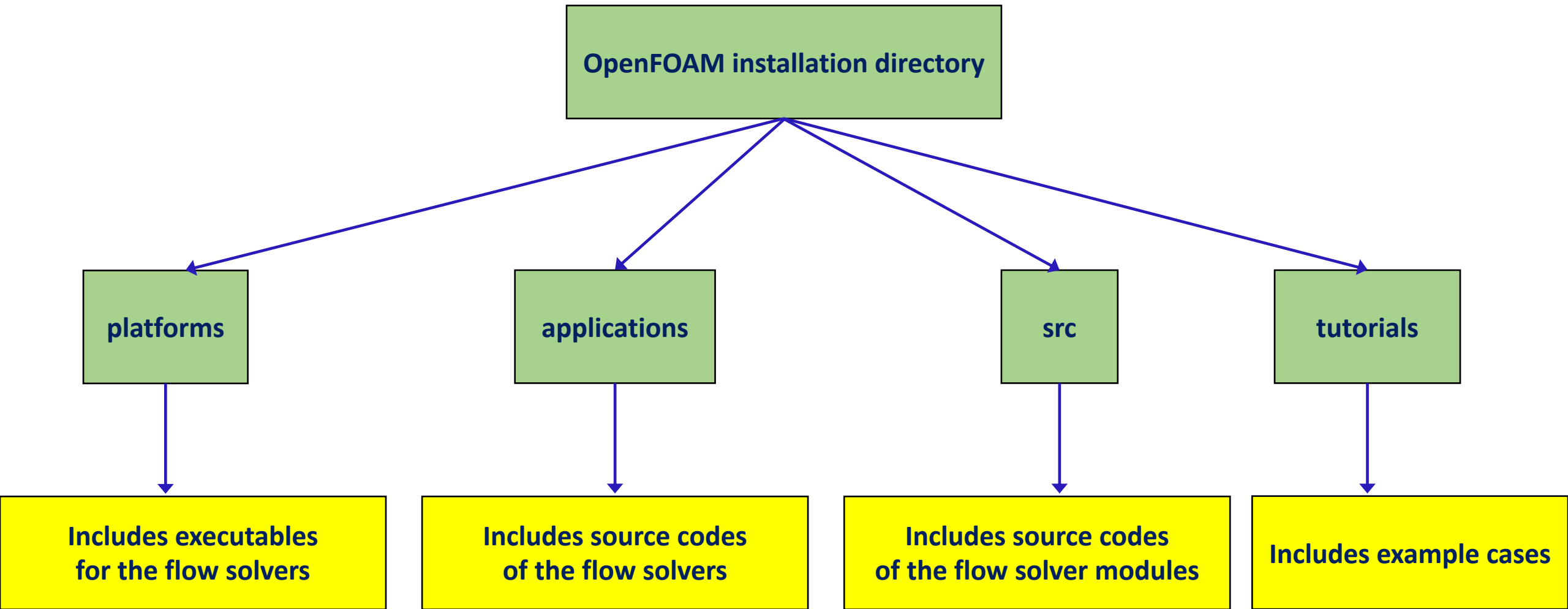
- First version of OpenFOAM was released by OpenCFD in 2004
- OpenCFD kept issuing new releases up to 2.0.1 (2011)
- Silicon Graphics International (SGI) acquired OpenCFD
- OpenFOAM foundation was created with permission of using OpenFOAM trademark
- Since 2016, OpenCFD has started issuing OpenFOAM by itself
- For more information visit : <https://www.openfoam.com/news/history>

# OpenFOAM versions

Which version to use?

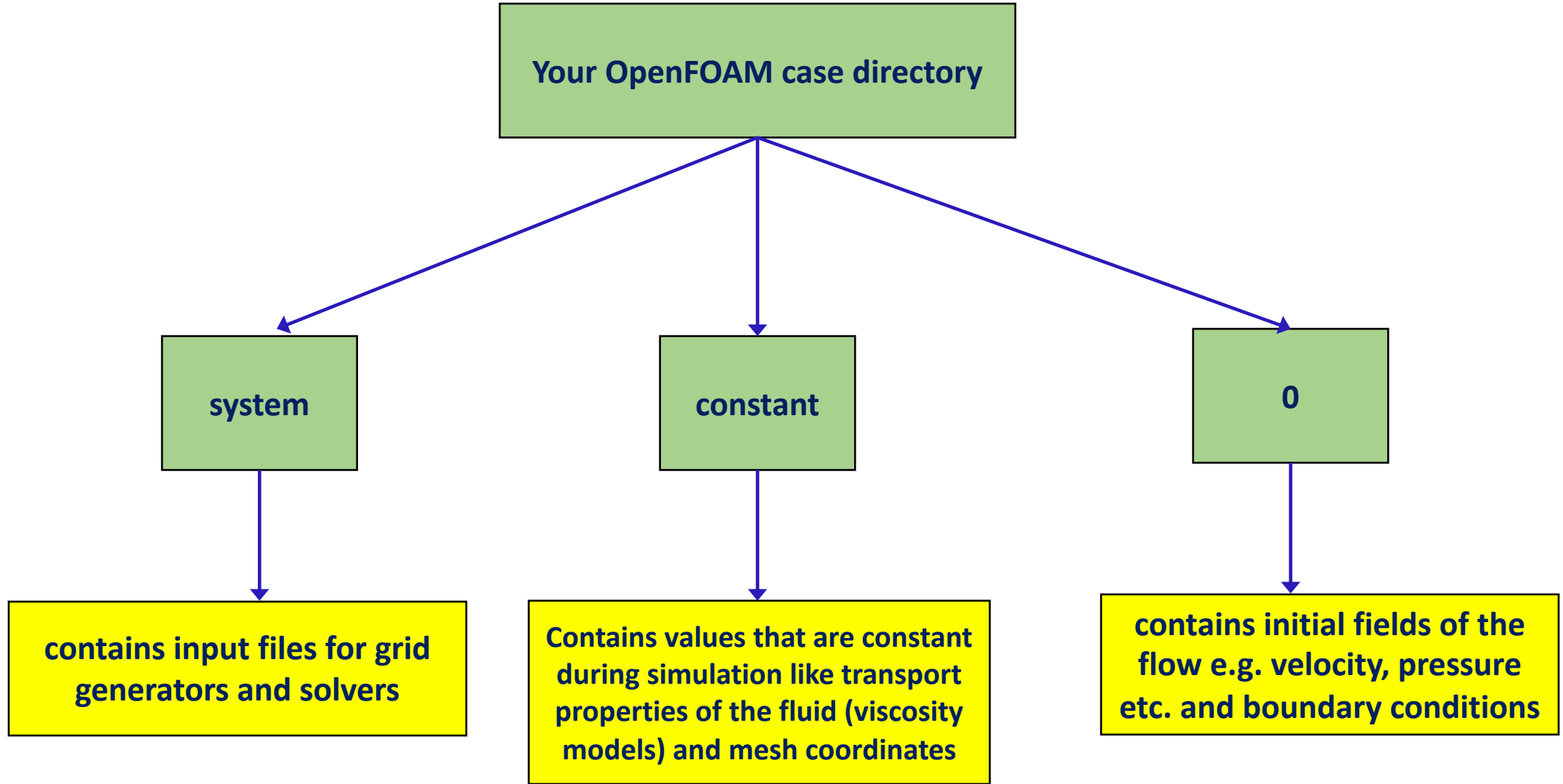
- Depends on the feature you want to use
- Check the website and release notes to see which one fits better to your framework
- If both include the features you need, do some performance and accuracy benchmarks to see which one is better
- Otherwise it is just matter of taste!
- We will be using v1912 for the training

# OpenFOAM environment



# OpenFOAM input files

To setup a case, you need to have 3 directories in your case directory



# OpenFOAM input files

## Input file format

- For input files, OpenFOAM uses a plain text dictionary format with keywords and values

```
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    object       blockMeshDict;
}
// * * * * *

scale  0.1;

vertices —————> keyword
(
    (0 0 0)
    (1 0 0)
    (1 1 0)
    (0 1 0) —————> value
    (0 0 0.1)
    (1 0 0.1)
    (1 1 0.1)
    (0 1 0.1)
);
```



# OpenFOAM executables

- Unlike many other software, OpenFOAM does not have a unique executable. For every solver, mesh generation etc. there is a separate executable!
- You should run the right executable according to the solver you are using!
  - 'simpleFoam': if you use SIMPLE algorithm
  - 'icoFoam': if you use PISO algorithm for laminar flow
  - ...
  - Check the documentation to see recommended solvers for different cases

# OpenFOAM output files

- Similar to the input files, the output files are also in plain text dictionary format

```
internalField    nonuniform List<scalar>  
400  
(  
1.66908e-08  
-0.00577259  
-0.0128362  
-0.0182176  
-0.0201885  
-0.0179911  
-0.0115712  
-0.00137173
```

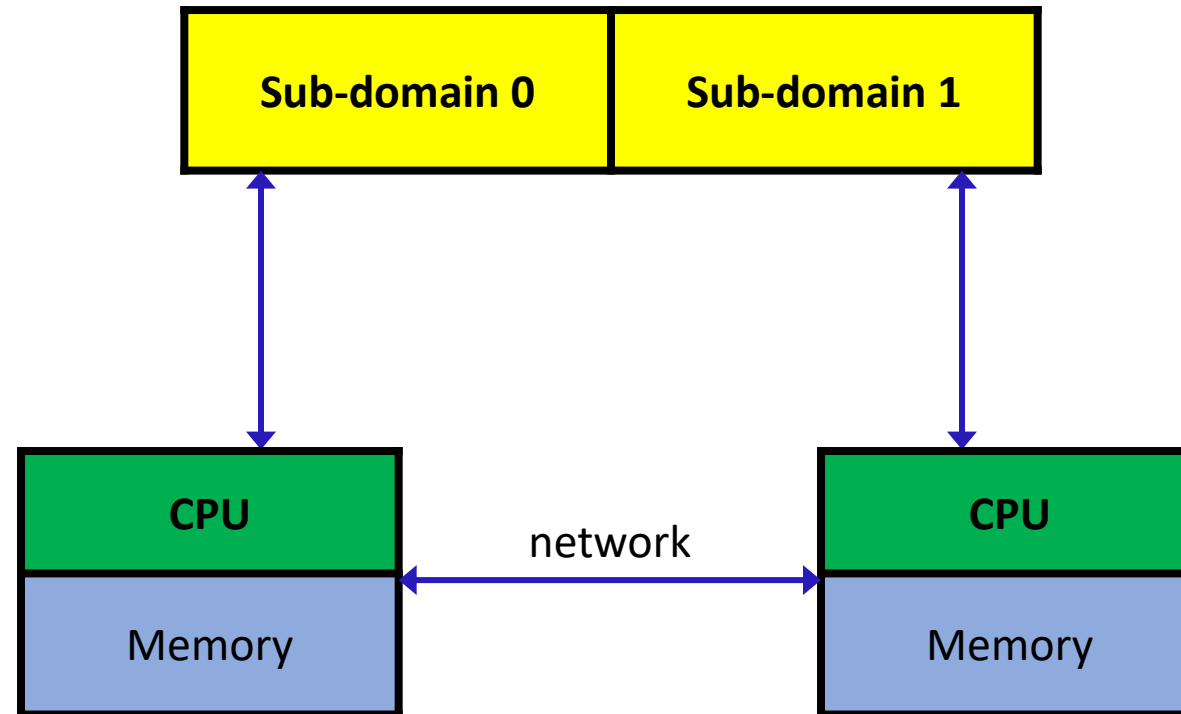
Do not worry if you have not understood anything! We will go through the files with an example

# Postprocessing of the output data

- **ParaView** is the main tool for postprocessing of the output data
- ParaView is a general visualization tool (not part of OpenFOAM)
- It can be used in several ways to visualize OpenFOAM results:
  - Load the output data using ParaView data loader for OpenFOAM
  - Load the output data using the reader module supplied by OpenFOAM
  - Convert output data to VTK format by OpenFOAM conversion module and load it to ParaView
  - Convert output data to VTK in runtime using OpenFOAM function objects
- '**paraFoam**' is an OpenFOAM script to launch ParaView with OpenFOAM reader module

# Parallelisation in OpenFOAM

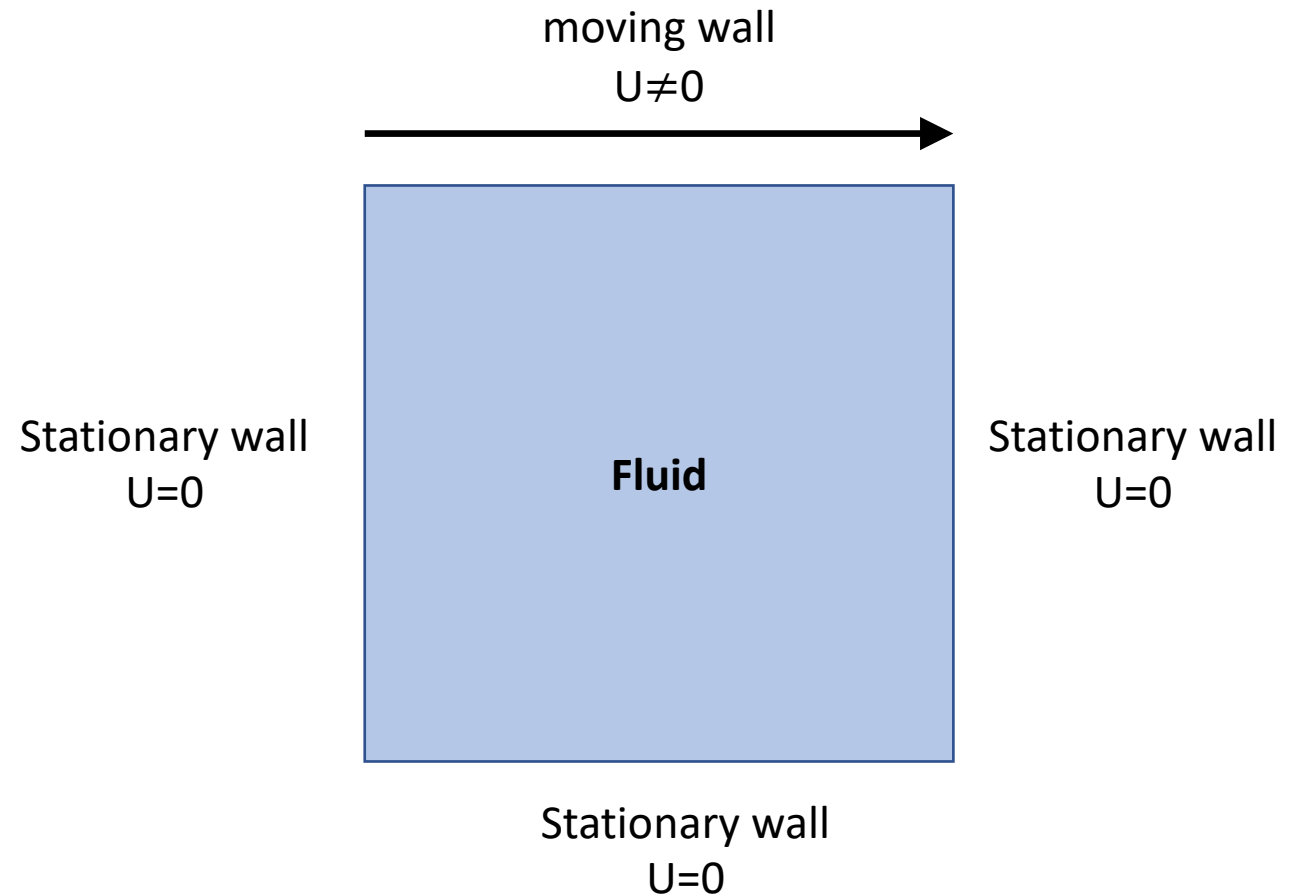
- Parallelisation in OpenFOAM is Message Passing Interface (**MPI**)



# Lid driven cavity

## Geometry and assumptions

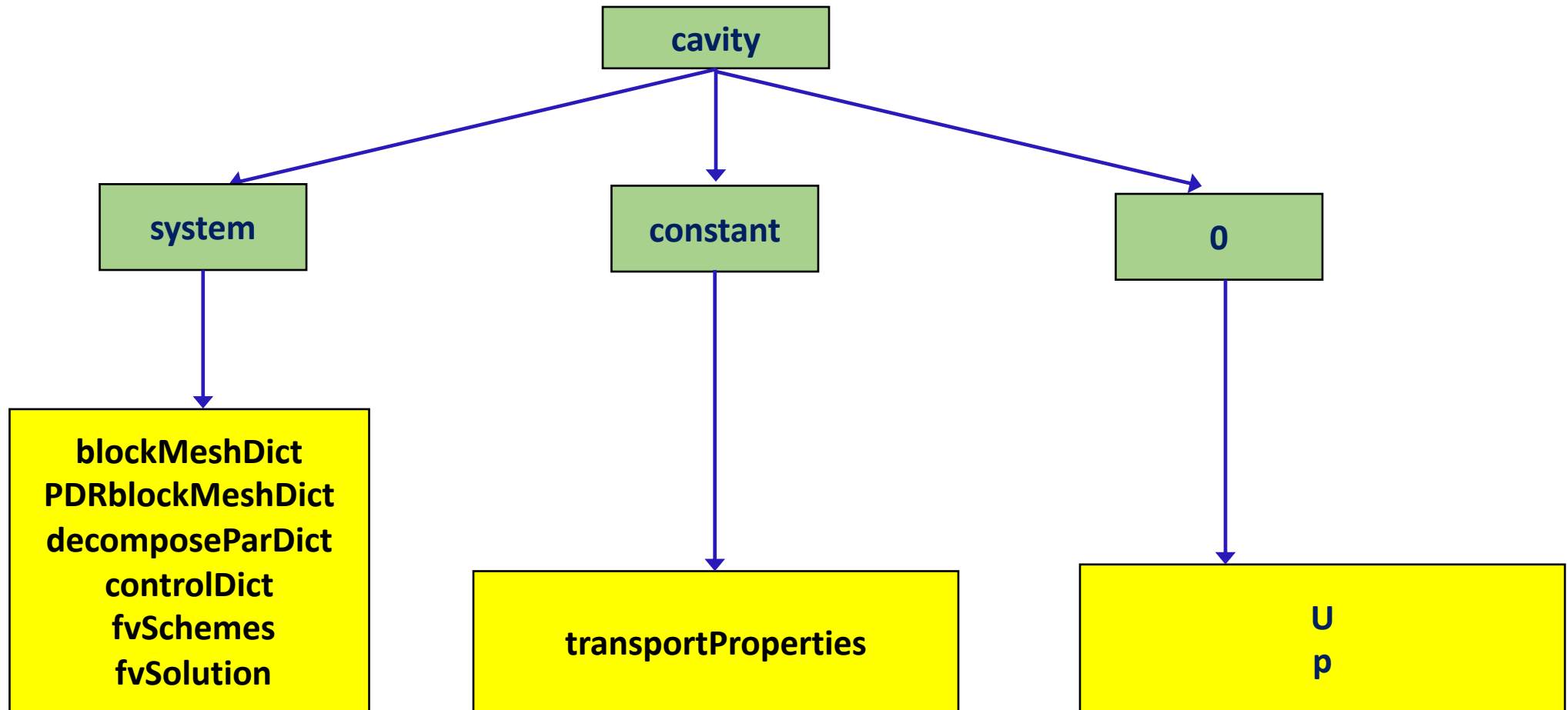
- The flow is 2 dimensional
- The fluid is incompressible  
(density does not change)
- The flow is isothermal (no heat transfer)



# Lid driven cavity

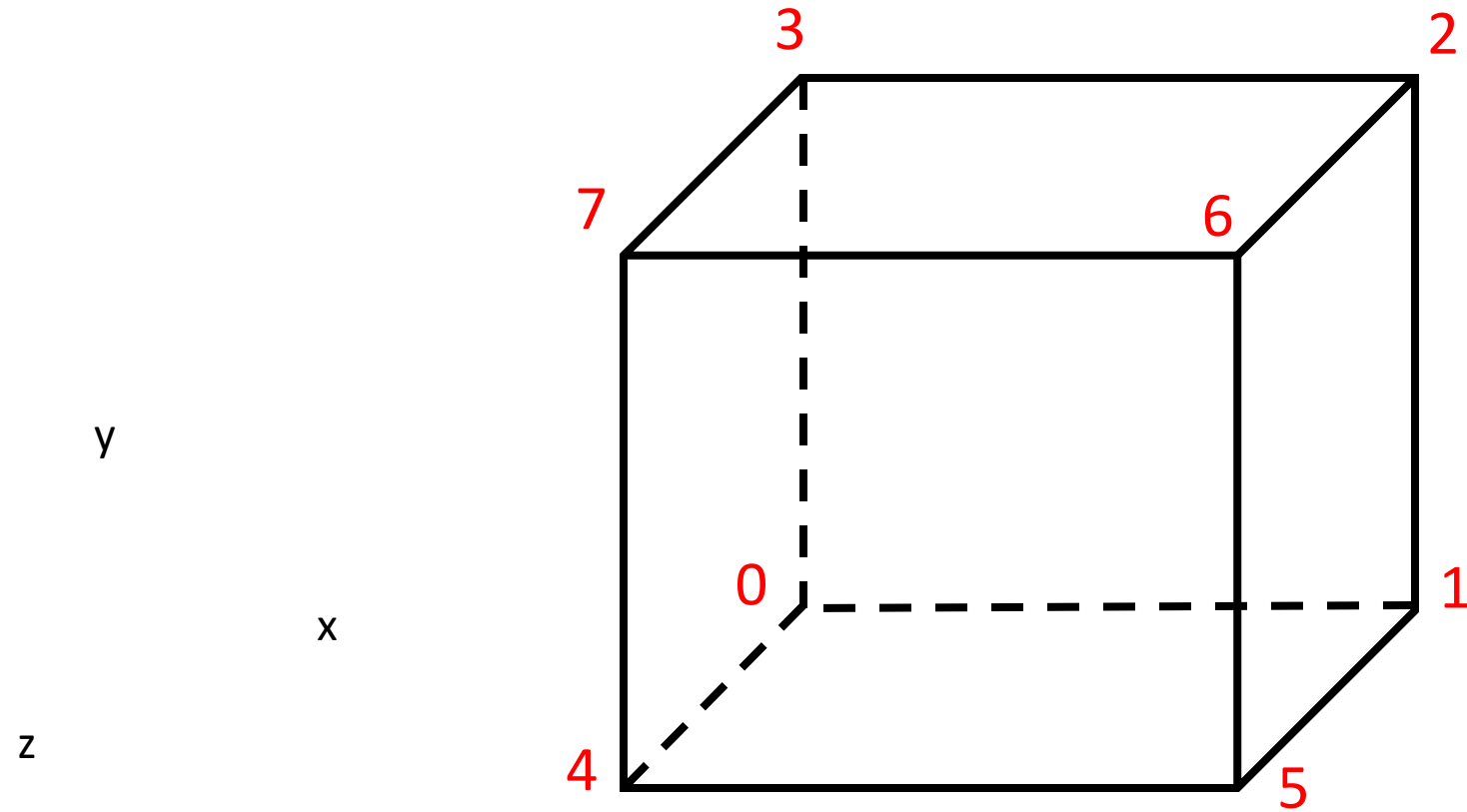
## input files

- Located at tutorials/incompressible/icoFoam/cavity/cavity



# Lid driven cavity

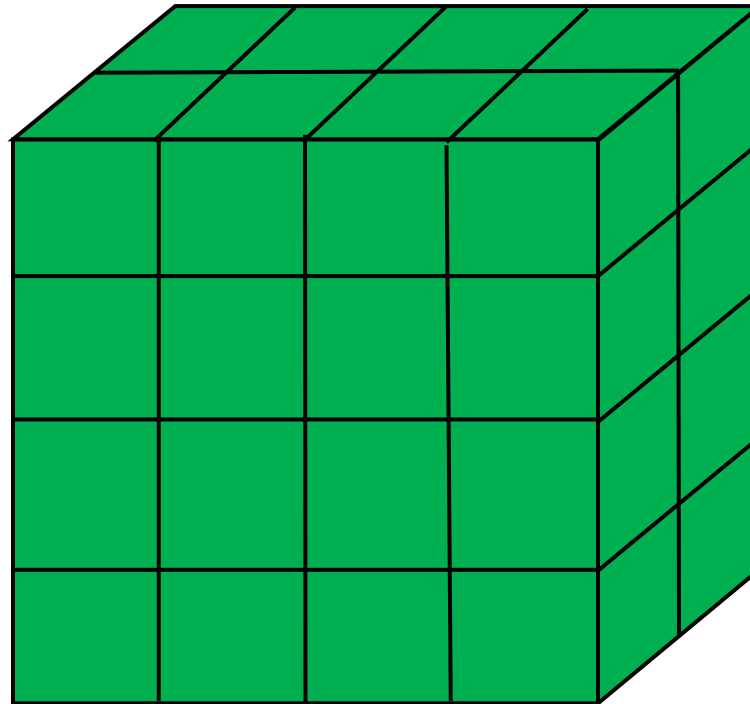
## Geometry



# Lid driven cavity

## blockMesh utility

decomposes the domain geometry into a set of 1 or more three dimensional, hexahedral blocks





# Lid driven cavity

## total workflow of the simulation

