Applied Computational Fluid Dynamics Using OpenFOAM

Day - 4

Value Added Course
College/University: AEC
Spring 2025

ExaSlate
Develop = Guide = Collab

ExaSlate
Develop = Guide = Collab

Contents

Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop • Guide • Collab

OpenFOAM - Introduction

- OpenFOAM® stands for **Open Source Field Operation and Manipulation**.
- OpenFOAM® is first and foremost a C++ library used to solve partial differential equations (PDEs), and ordinary differential equations (ODEs).
- It comes with several ready-to-use or out-of-the-box solvers, pre-processing utilities and post-processing utilities.
- It is licensed under the GNU General Public License (GPL).That means it is freely available and distributed with the source code.

Kumaresh Selvakumar
kumaresh@exaslate.com

OpenFOAM – Directory Structure

1. *constant*: This directory contains the information which remains constant throughout the simulation. It contains the following:

   1.1 *polymesh*: Contains all the mesh information including:

   (a) points → nodal positions

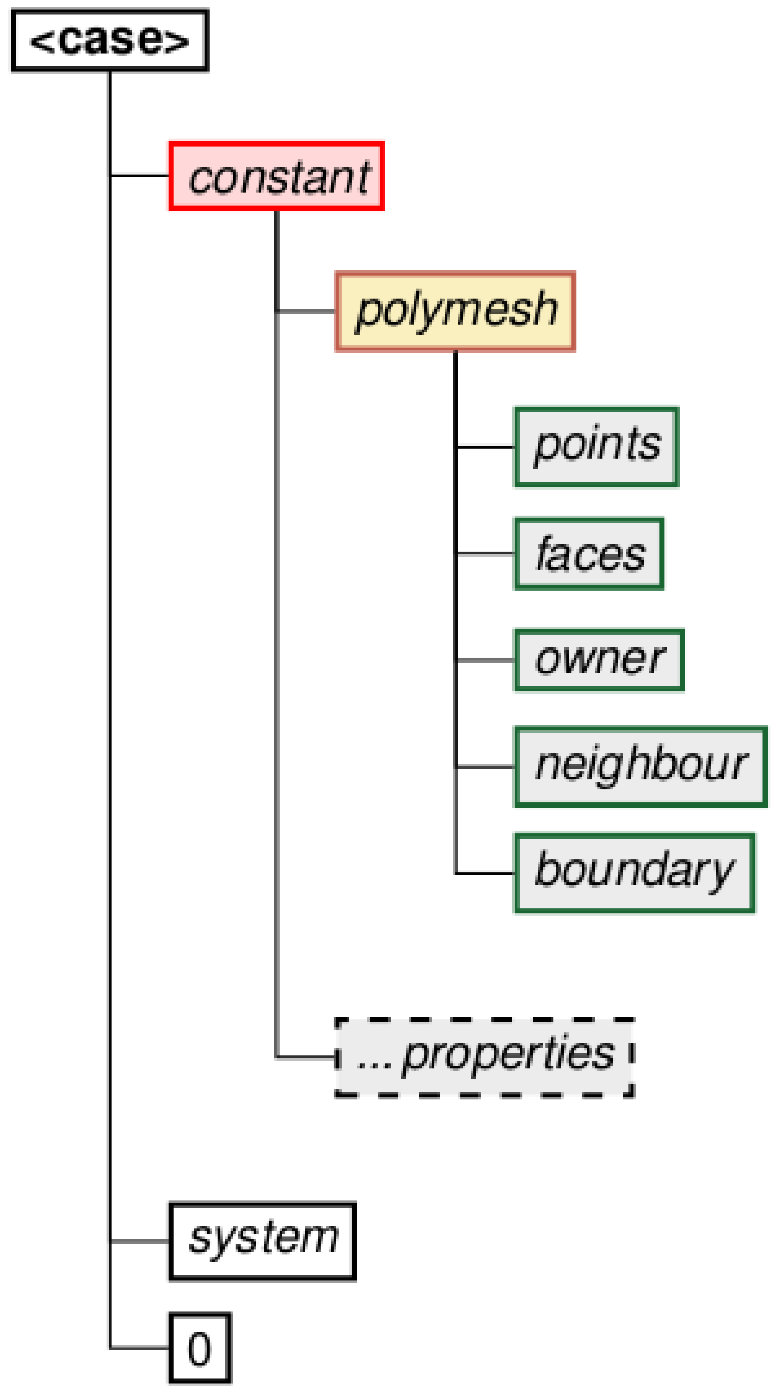
   (b) faces → face connectivity neighbor boundary

   (c) owner → owner cell labels

   (d) neighbor → neighbor cell labels

   (e) boundary → boundary information

   1.2 *properties*: Files which specify physical properties for a particular application. Eg: gravity, viscosity, thermal and transport properties etc.



Kumaresh Selvakumar
kumaresh@exaslate.com

OpenFOAM – Directory Structure

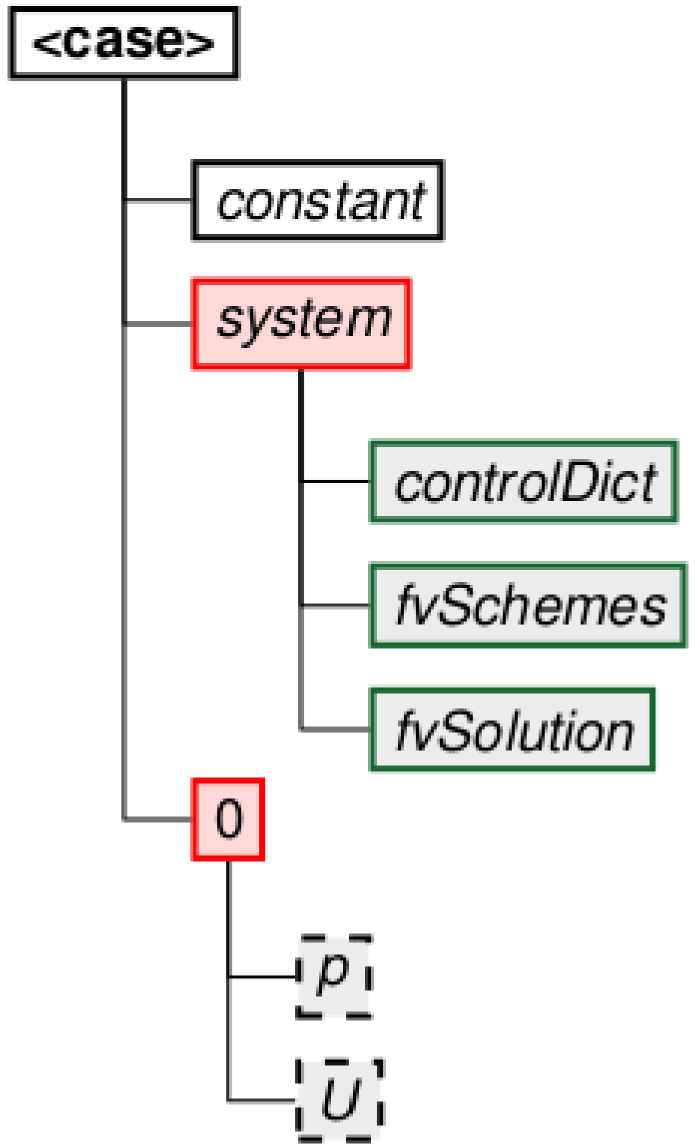
2. *system*: This directory contains all the parameters associated with the solution procedure. It contains at least the following files:

2.1 *blockmeshDict:*

- The principle behind blockMesh is to **decompose the domain geometry** into a set of 1 or more three dimensional, hexahedral blocks. This section describes the **mesh generation utility**, blockMesh, supplied with **OpenFOAM**. The blockMesh utility creates parametric meshes with grading and curved edges. BlockMesh reads this dictionary, generates the mesh and writes out the mesh data to **points** and **faces**, **cells** and **boundary files** in the same directory.

2.2 *controlDict:* Specifies the run control parameters such as start/ end time, time step, write interval etc.
2.3 *fvSchemes:* Contains the finite volume discretization schemes used for the solution procedure such as spatial and temporal discretizations.
2.4 *fvSolution:* Contains equation solvers, algorithm controls and tolerances for the implicit solvers.

3. *0*: The '0' directory corresponds to zero time. It contains the initial and boundary conditions for variables (i.e. **pressure p**, **velocity U**) in individual files



Kumaresh Selvakumar
kumaresh@exaslate.com

CFD Developers

- Developers are the ones who write code on the back end.

- To make it simple for CFD amateur students, let's say ANSYS Fluent and Star CCM are written by developers (by writing code at the back end), so we could easily use them as black box (GUI) tools.

- Developers are the key to the development of technology.

Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

Is it good to be a black box user or a developer ?

- A developer can handle anything from scratch. That brings confidence by capable of solving any puzzles.

- Understanding the logic is an art, so being a developer helps know exactly what and how to do.

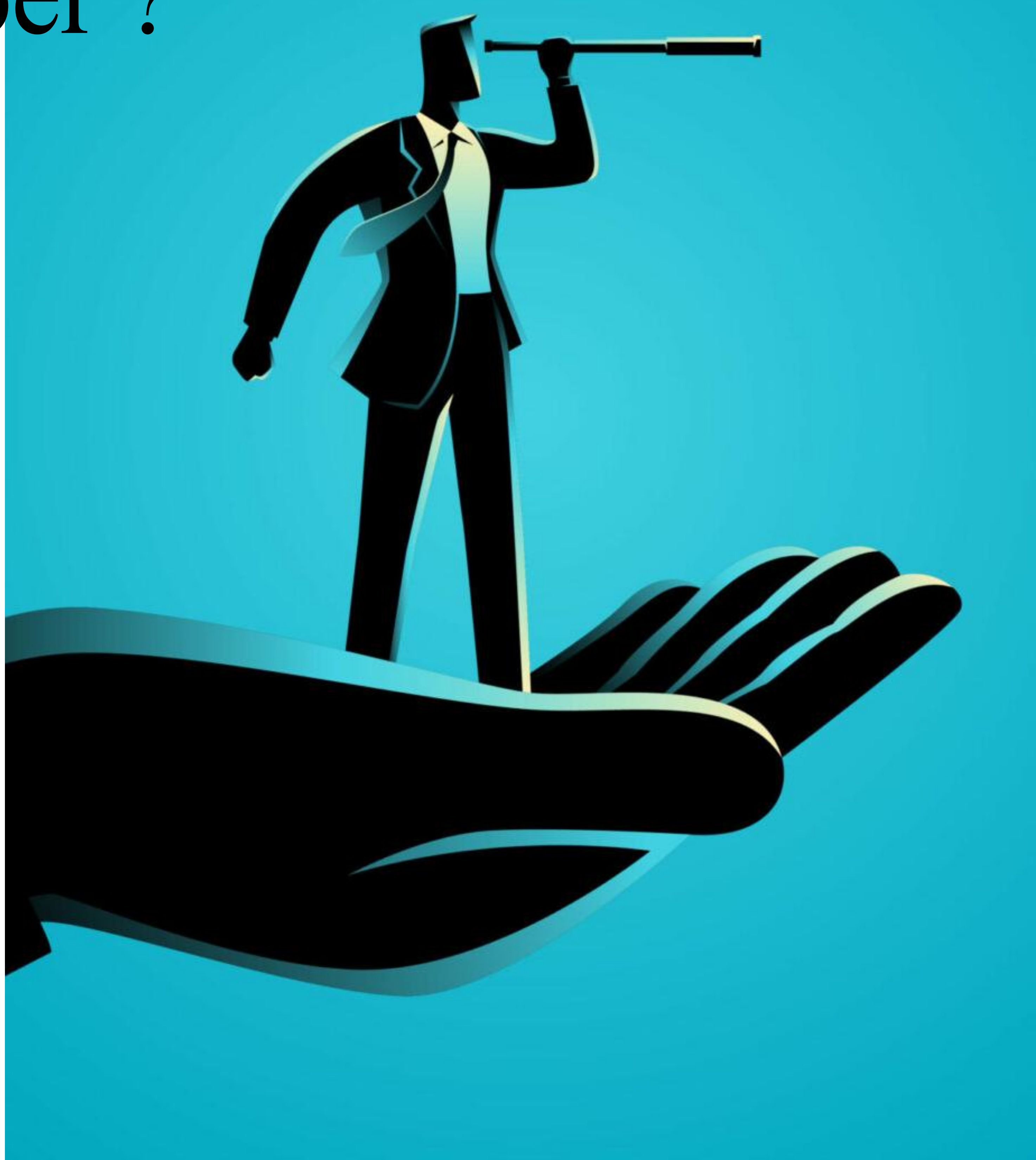
- It's good to be a black box user in the beginning to understand the basic ideas/physics, but in order to route your future, trust me, developers can handle it better.

Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop - Guide - Collab

How to grow as a developer ?

- Work with passion
- Face the hurdles and don’t get tired of it
- Never lose the joy of programming
- Test your skills in open-source projects
- Read code and do code reviews
- Learn from your colleagues and mentors
- Share your knowledge

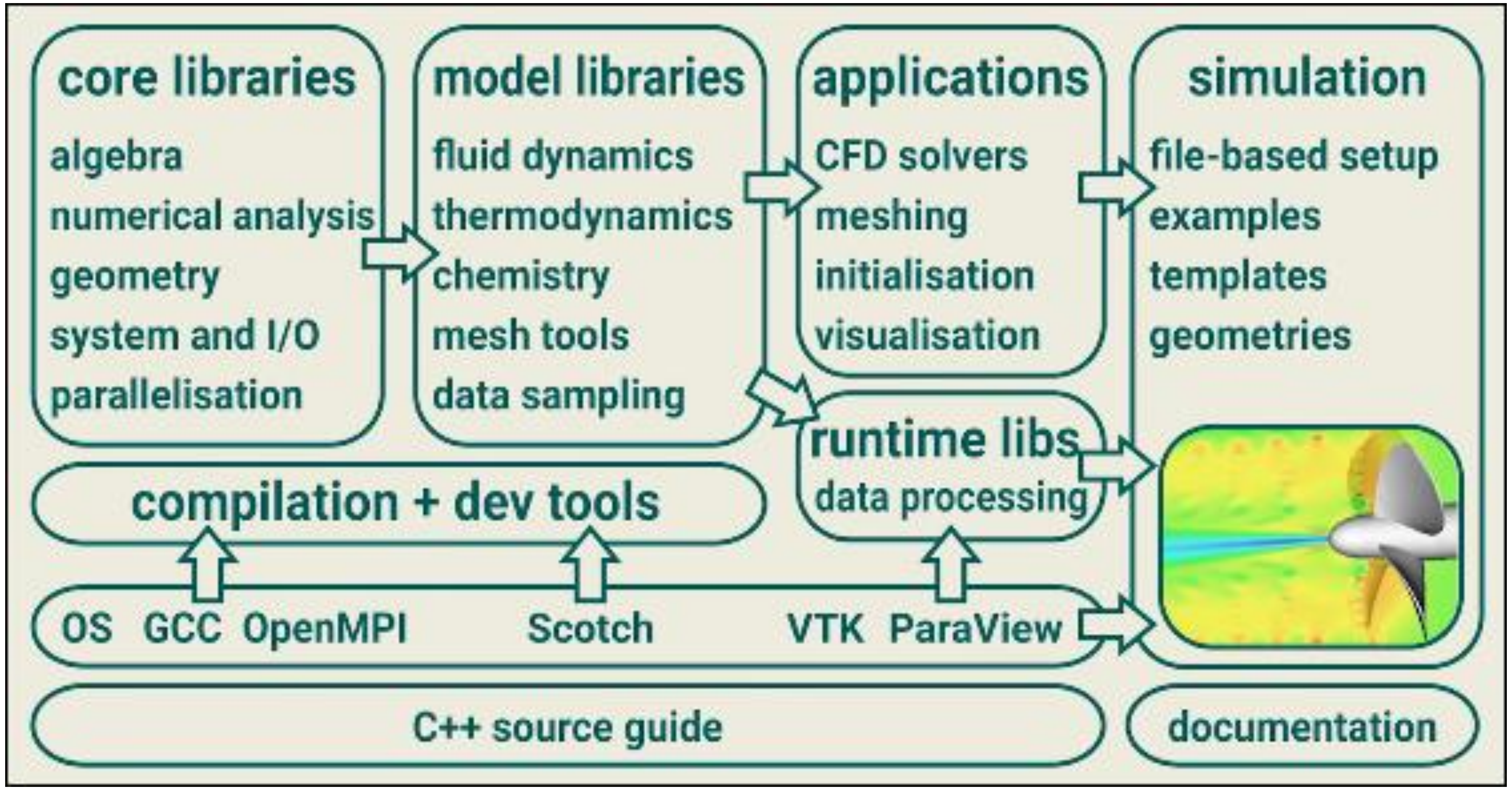


Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

How to grow as a developer ? → OpenFOAM

- OpenFOAM is a software framework (or toolbox) that you can use to develop Finite Volume Method based solvers for general continuum mechanics problems mostly for fluid flow and heat transfer.



- It is a huge library of about 1.5 million lines of C++ code located in hundreds of files.

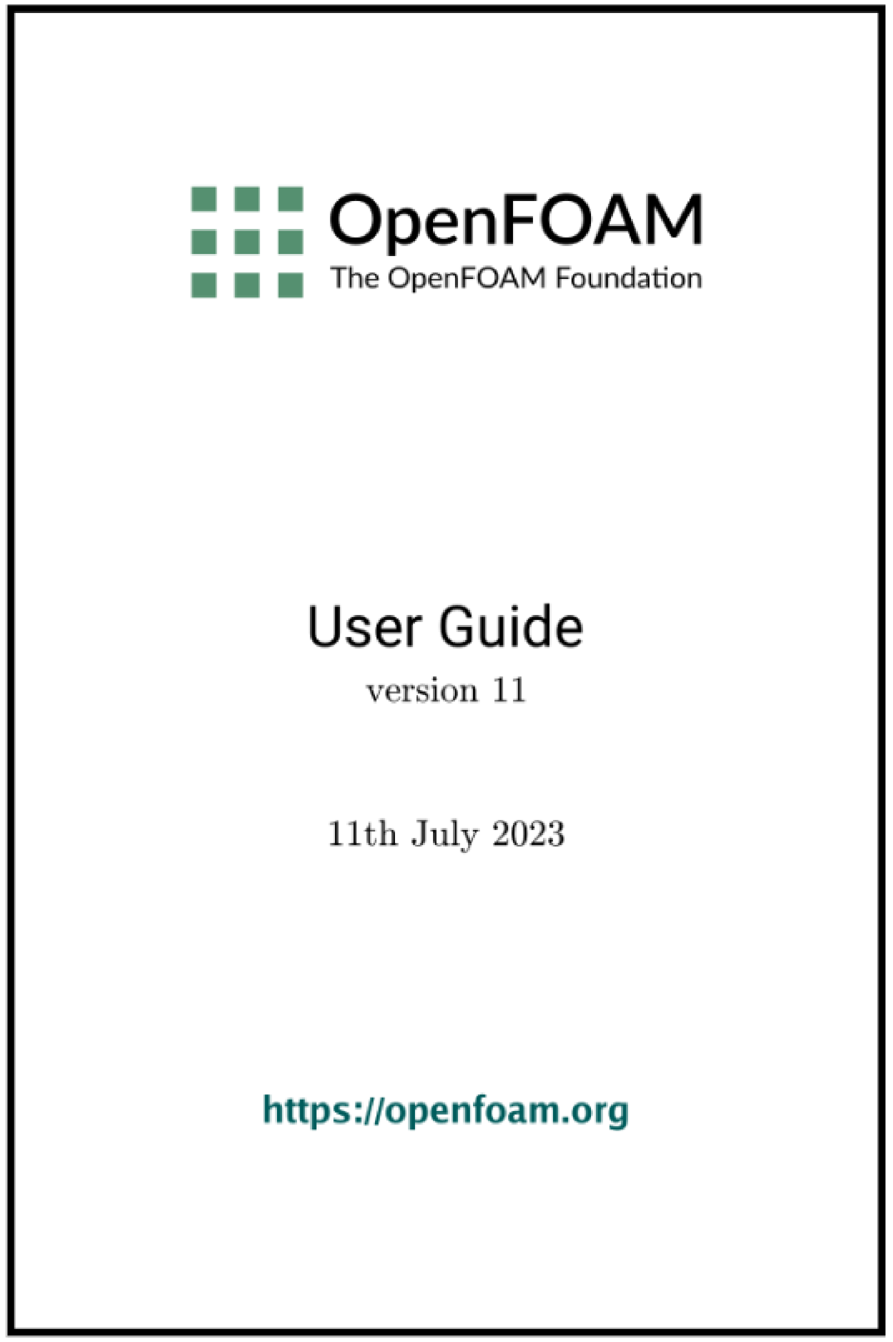
- You can use this toolbox to develop Computational Fluid Dynamics (CFD) solvers.

Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

OpenFOAM User GUIDE

The OpenFOAM User Guide provides an introduction to OpenFOAM, through some basic tutorials, and some details about the general operation of OpenFOAM. OpenFOAM is a collection of approximately 150 applications built upon a collection of approximately 150 software libraries (modules).


OpenFOAM
The OpenFOAM Foundation

User Guide
version 11

11th July 2023

https://openfoam.org

Contents

Kumaresh Selvakumar
kumaresh@exaslate.com

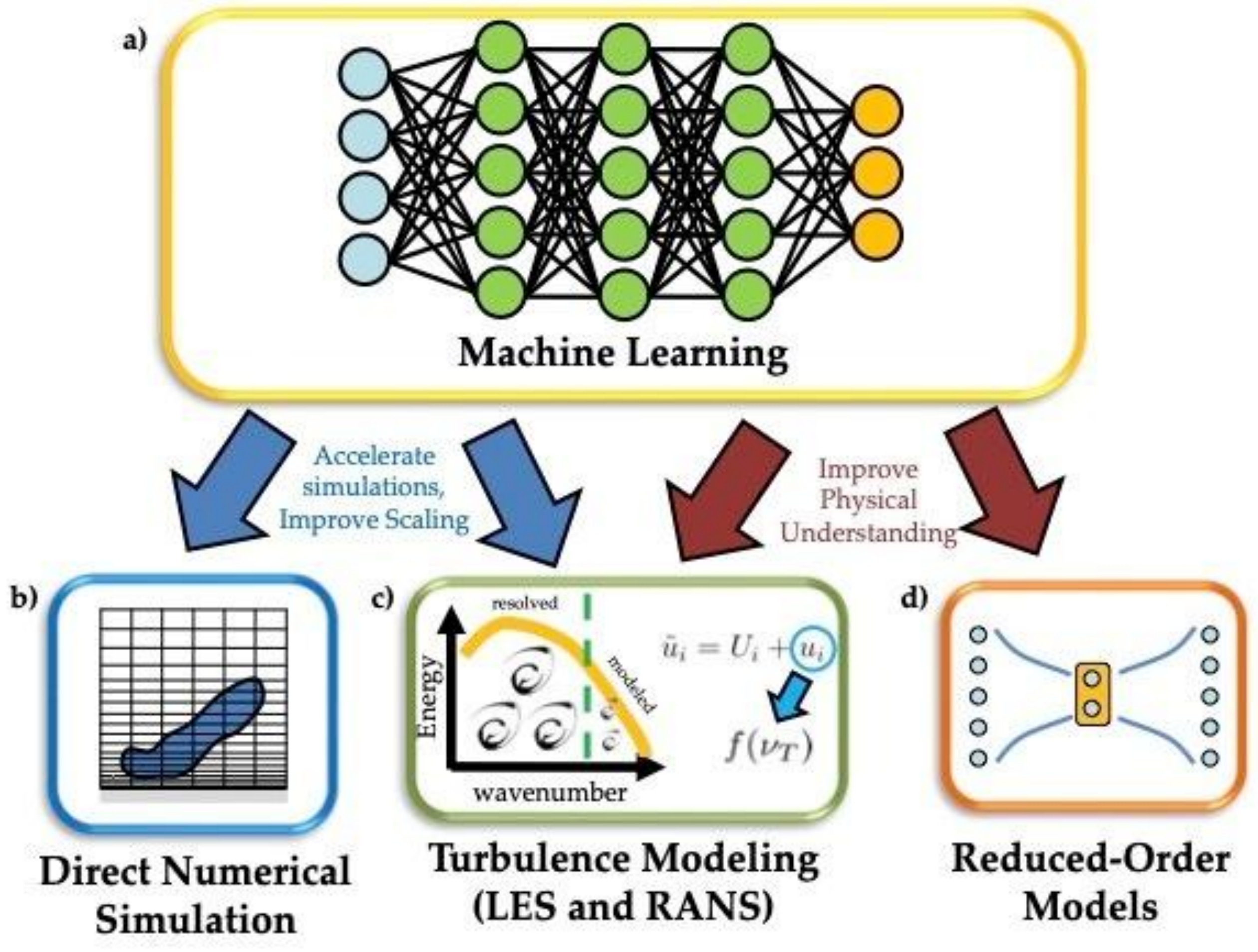
Machine Learning in CFD

By definition, machine learning is a branch of artificial intelligence (AI) and computer science that focuses on the use of data and algorithms to imitate the way that humans learn, gradually improving its accuracy.

To understand certain critical parameters in any physical phenomenon, data optimisation can be implemented by machine learning algorithms.

The data optimisation saves simulation time to interpret the phenomenon by improving accuracy. This is in the current developing stage of this technology world.

This improvisation will assist steel industry technologies in prospering and creating a better future.



Kumaresh Selvakumar
kumaresh@exaslate.com

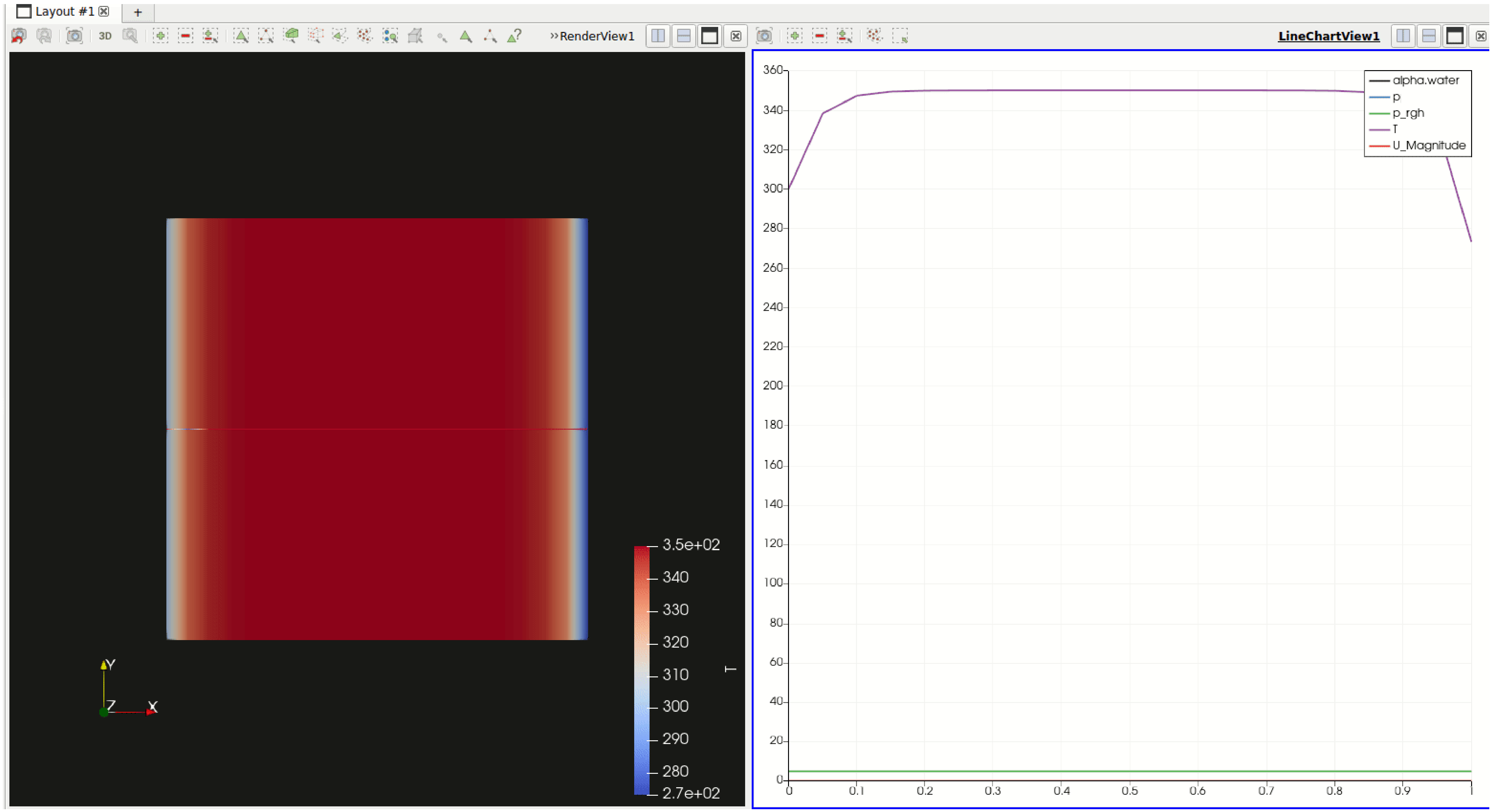
ExaSlate
Develop Guide Collab

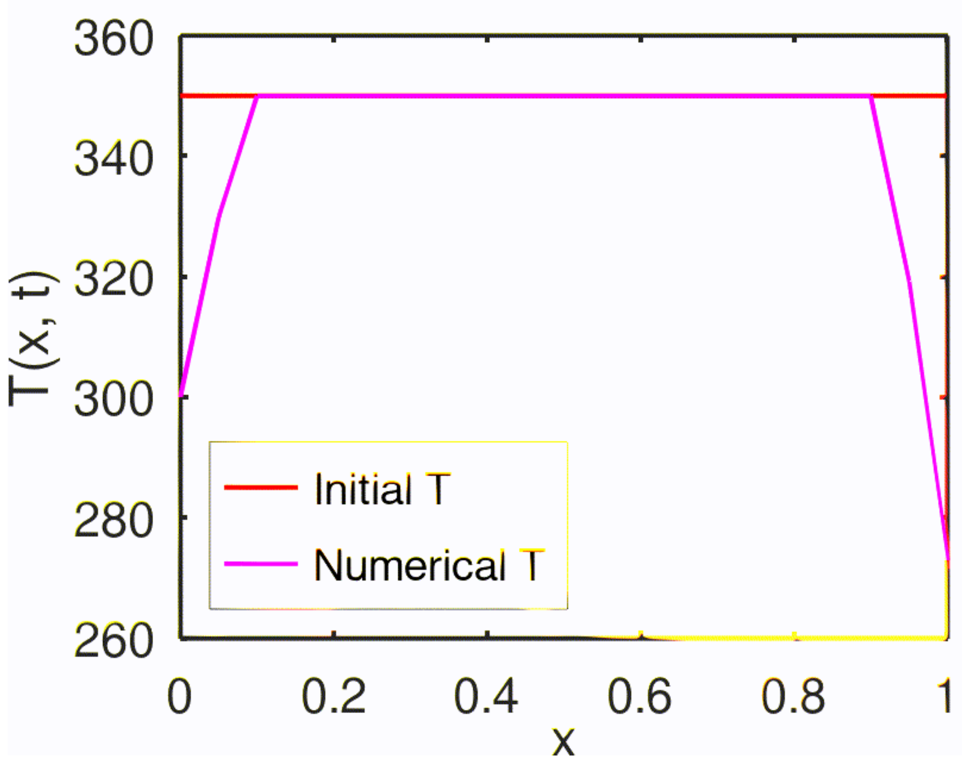
Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

OpenFOAM: Numerical Solution to Diffusion Equation

Paraview – Post Processing tool in OpenFOAM



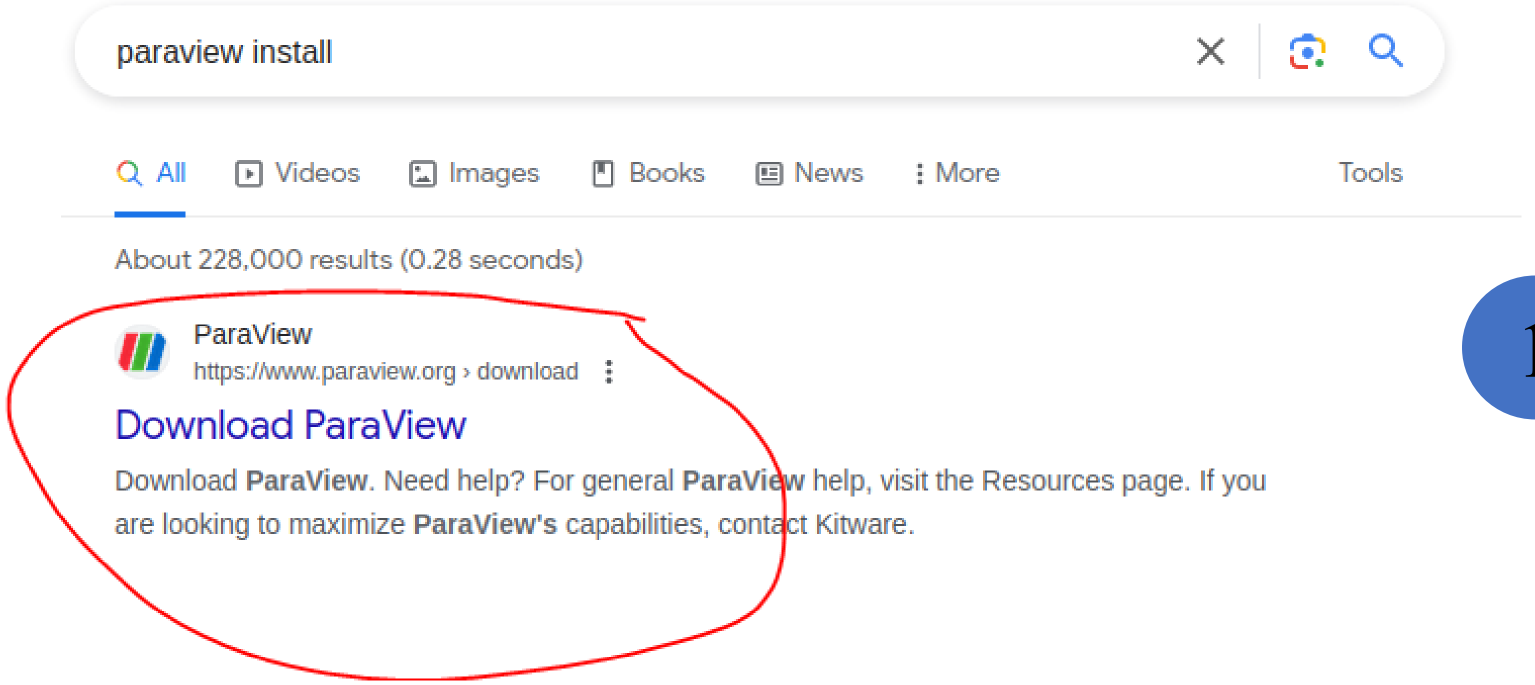


$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

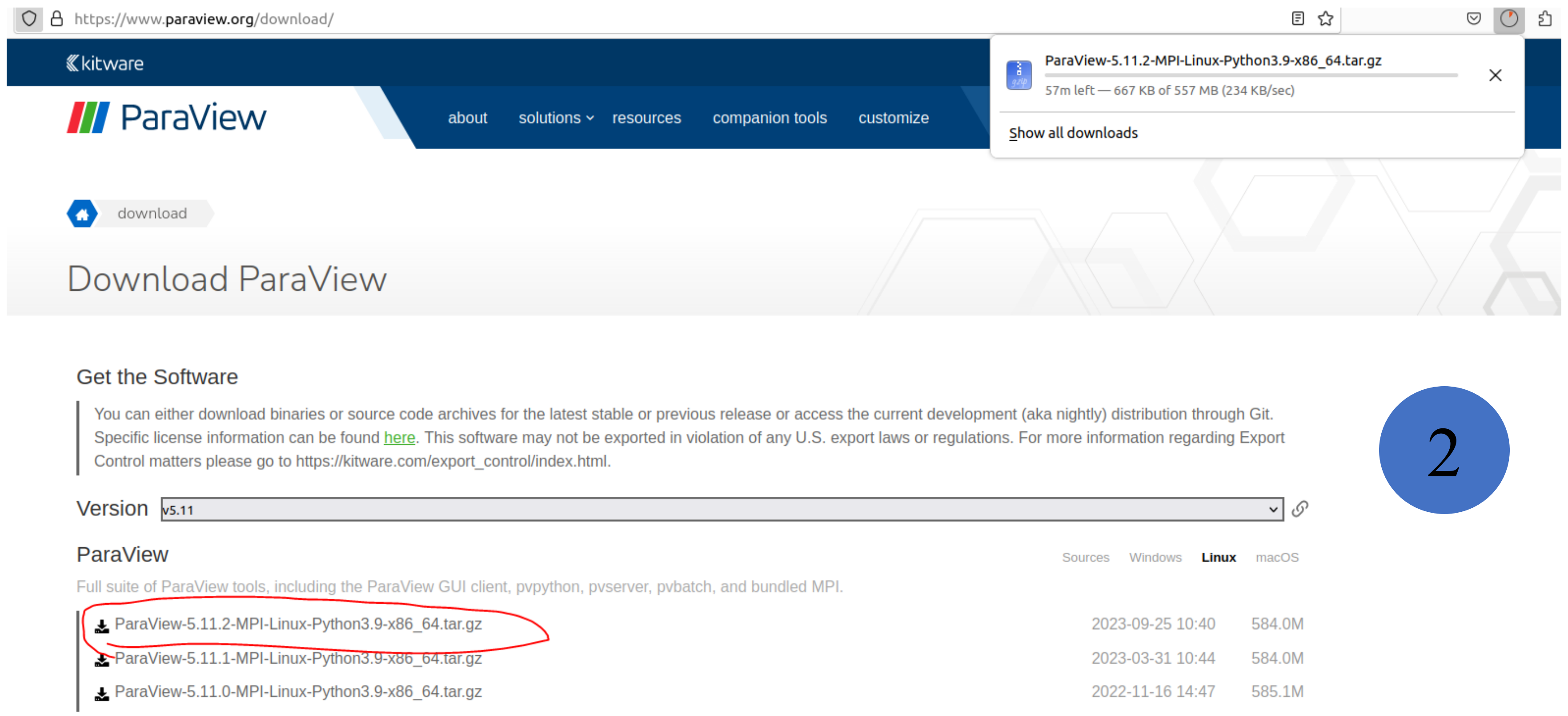
Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

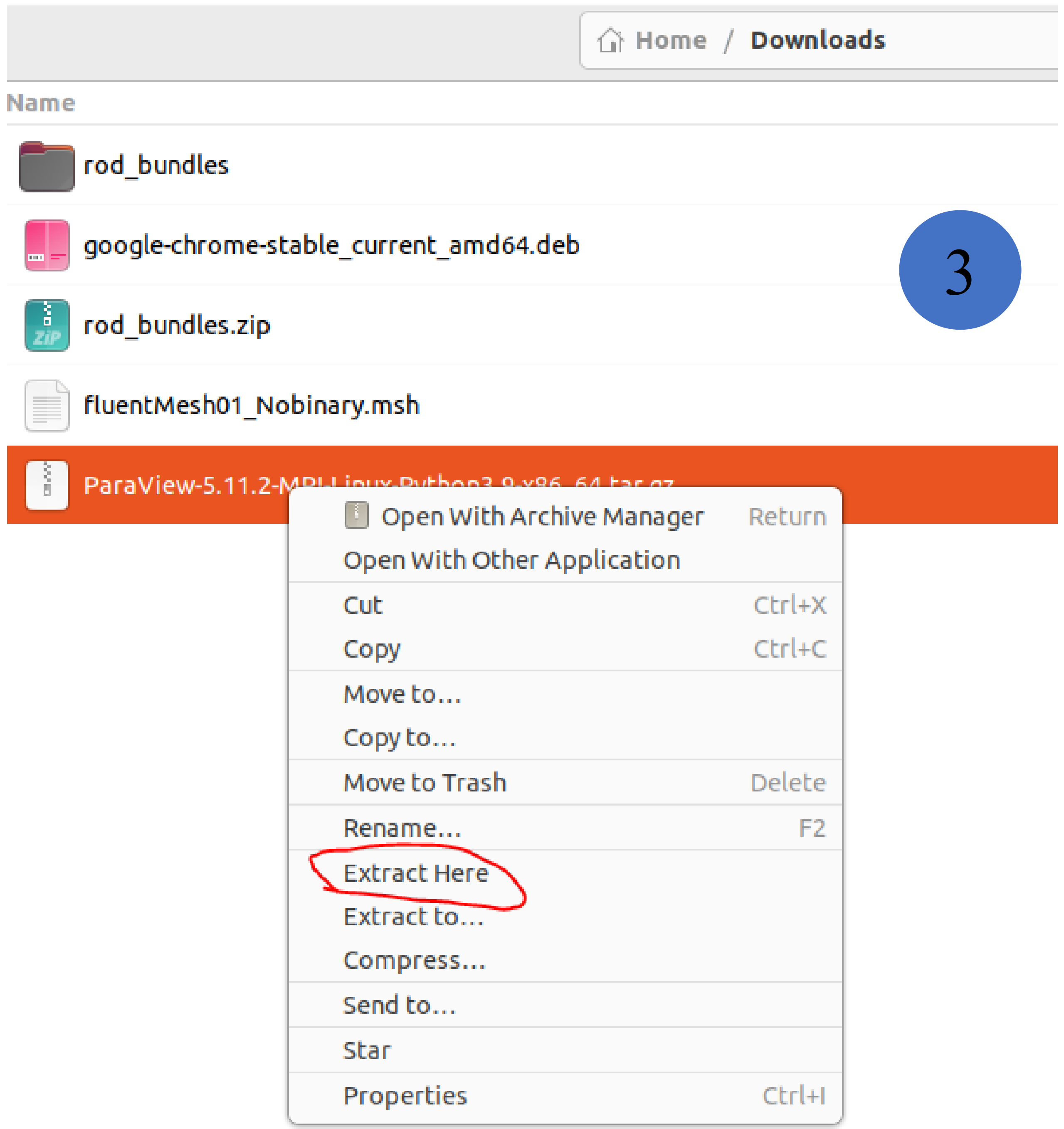
Install “Paraview”



1



2

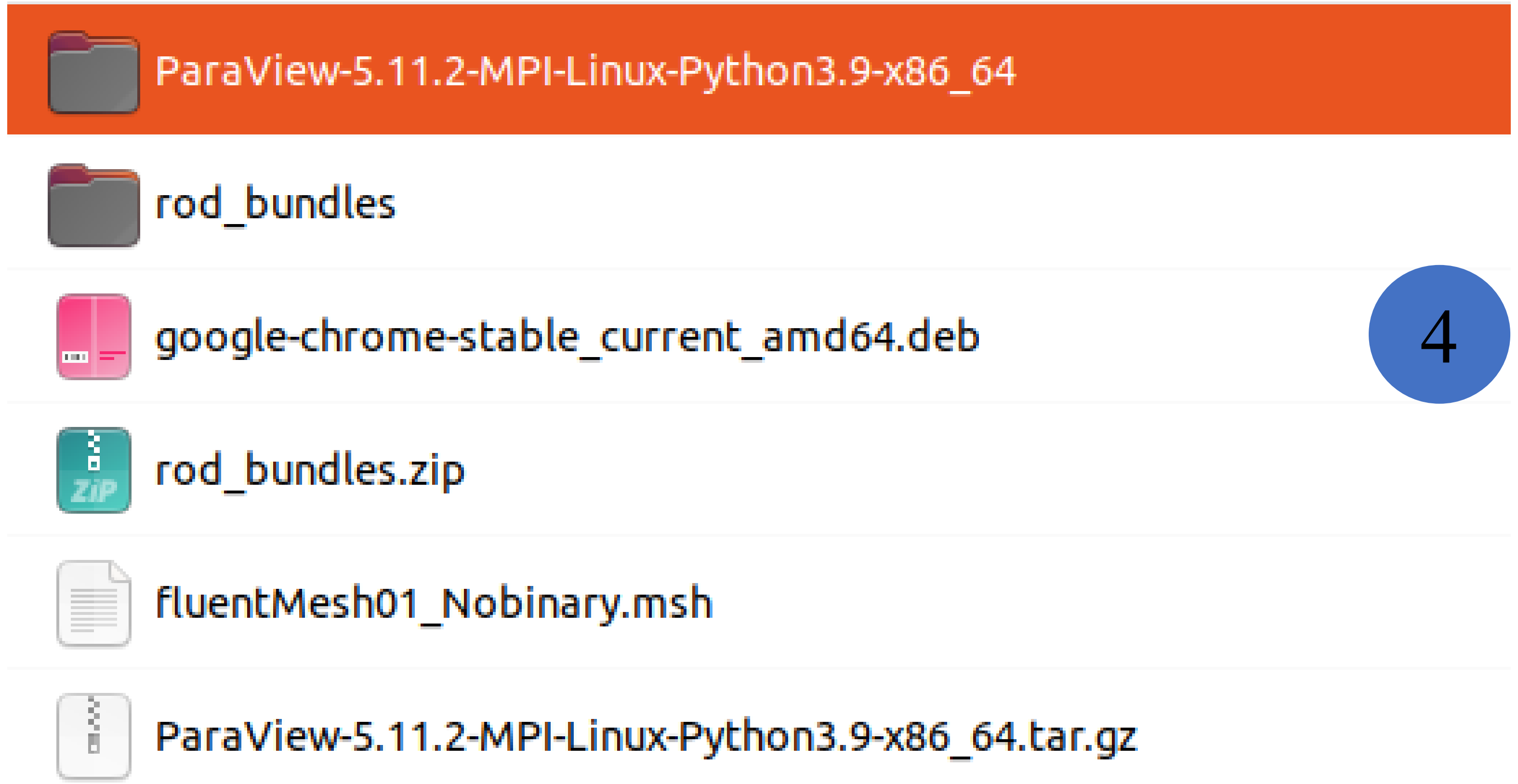


3

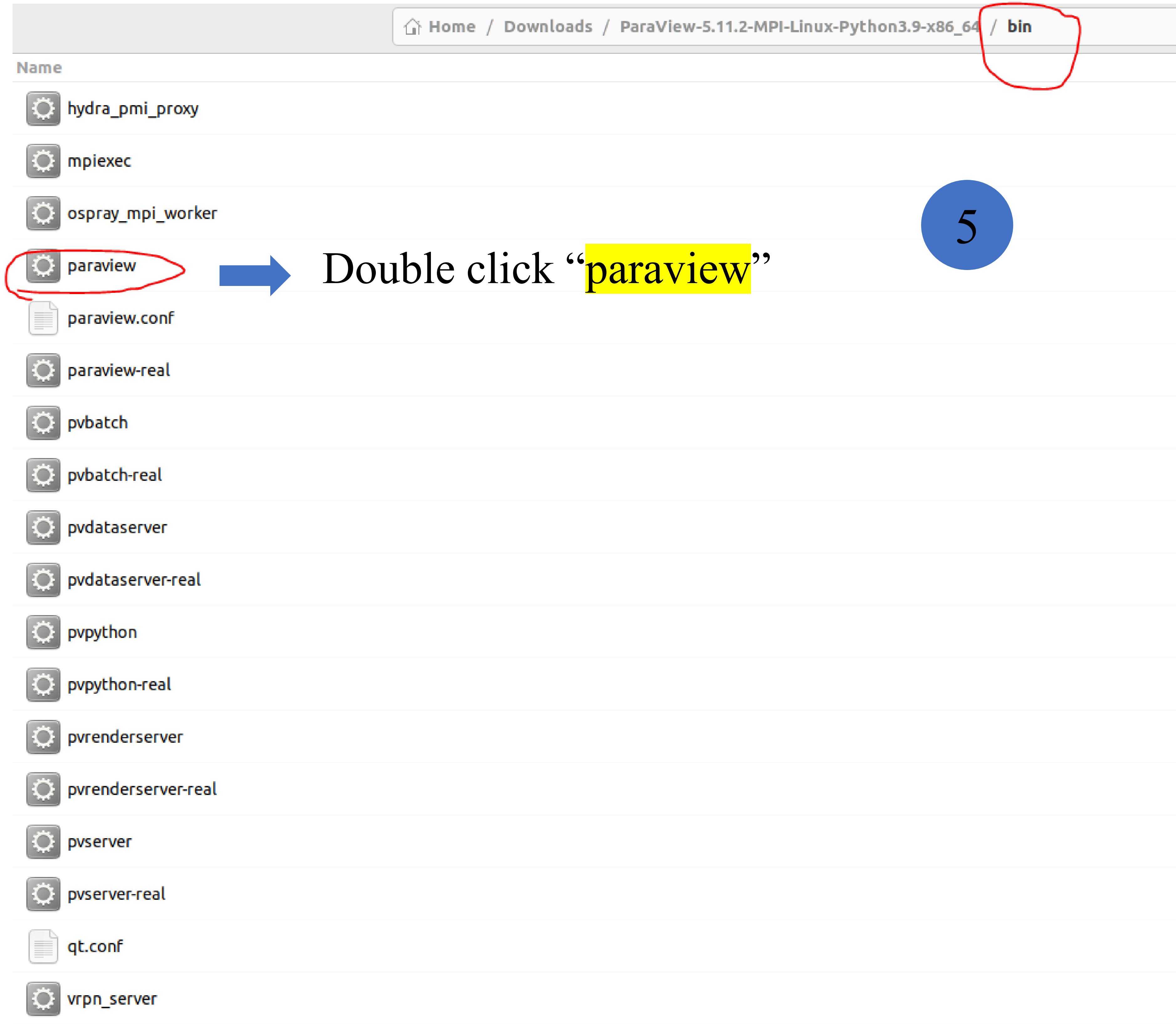
Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

Install “Paraview”



4



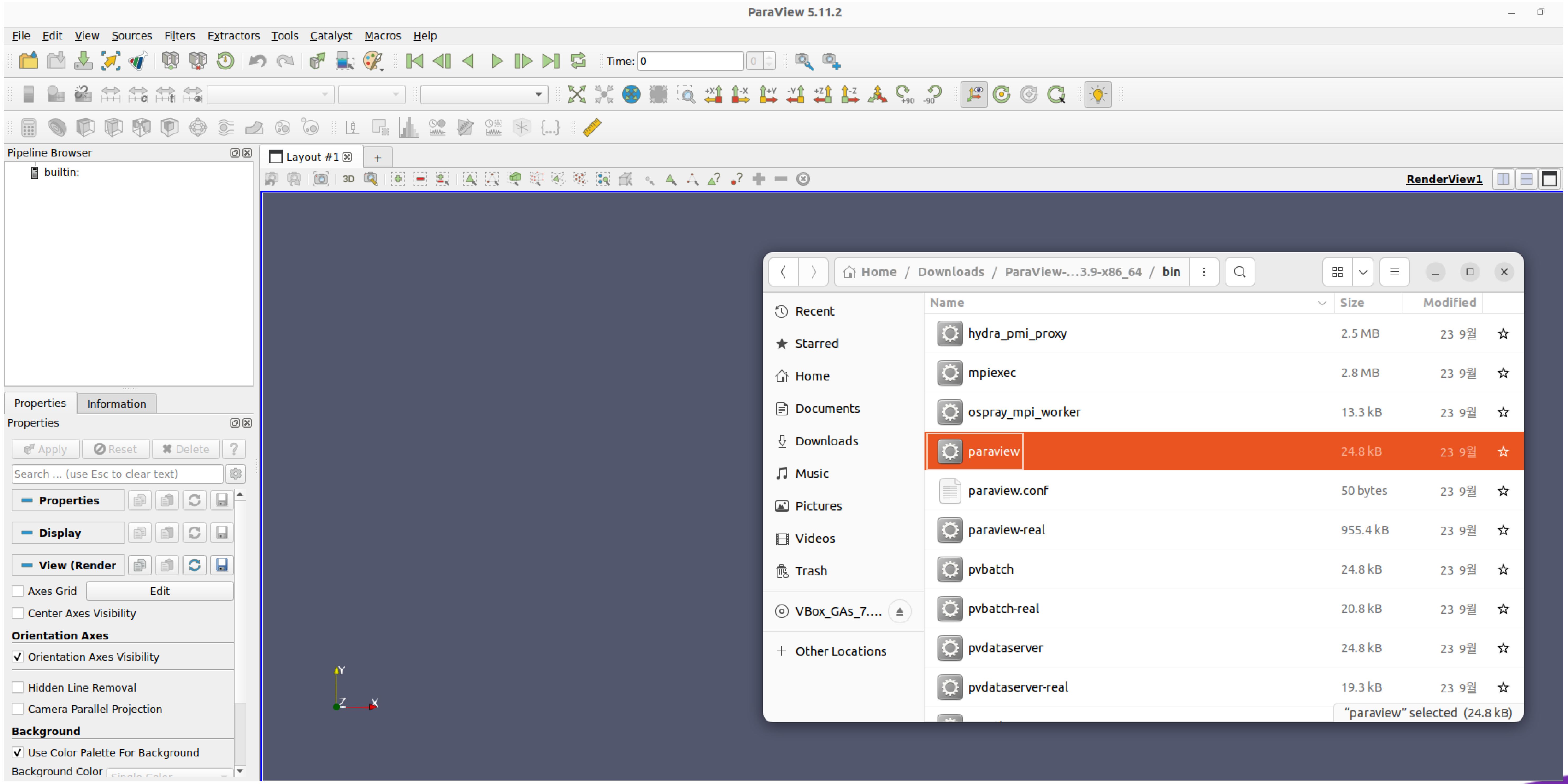
5

Double click “paraview”

Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

Open "Paraview" → Used for post-processing your results



Kumaresh Selvakumar
kumaresh@exaslate.com

Extract files from GitHub

applied-cfd-using-openfoam-ksr-spring-2025 / day-7 /

Add file

kummi0402 Add files via upload

966f948 · 1 minute ago History

Name	Last commit message	Last commit date
..		
CASE1_diffusionFOAM.rar	Add files via upload	1 minute ago
diffusionFoam.rar	Add files via upload	1 minute ago
readme.md	Update readme.md	11 minutes ago

readme.md

Exercise - 7: OpenFOAM: Numerical Solution to Diffusion Equation

- solver
  - Run 'wmake' command inside 'diffusionSolver'
- case
  - blockMesh // this command will generate the mesh in OpenFOAM
  - diffusionFoam // this command will run the diffusion solver compiled
  - touch diffusion.foam // this command will generate the .foam extension to run the post-processing
- post-processing
  - Open 'paraview'
  - Generate the contour and line plot

Kumaresh Selvakumar
kumaresh@exaslate.com

solver file ➔ diffusionFoam/createFields.H

ComputationalThermalEngineering / DAY7-OpenFOAM_diffusion_equation / diffusionFoam / **createFields.H**

Kumaresh0402 Add files via upload

Code Blame 49 lines (43 loc) · 846 Bytes Code 55% faster with GitHub Copilot

```
Info<< "Reading field T\n" << endl;

volScalarField T
(
    IOobject
    (
        "T",
        runTime.timeName(),
        mesh,
        IOobject::MUST_READ,
        IOobject::AUTO_WRITE
    ),
    mesh
);


Info<< "Reading diffusivity DT\n" << endl;

volScalarField DT
(
    IOobject
    (
        "DT",
        runTime.timeName(),
        mesh,
        IOobject::READ_IF_PRESENT,
        IOobject::AUTO_WRITE
    ),
    mesh,
    dimensionedScalar(dimViscosity, Zero)
);
```

Temperature “T” field is created

Diffusivity as a field created

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

solver file ➔ diffusionFoam/diffusionFoam.C

```
#include "fvCFD.H"
#include "fvOptions.H"
#include "simpleControl.H"

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

int main(int argc, char *argv[])
{
    argList::addNote
    (
        "Laplace equation solver for a scalar quantity."
    );

    #include "postProcess.H"

    #include "addCheckCaseOptions.H"
    #include "setRootCaseLists.H"
    #include "createTime.H"
    #include "createMesh.H"

    simpleControl simple(mesh);

    #include "createFields.H"

    // * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

    Info<< "\nCalculating temperature distribution\n" << endl;

    while (simple.loop())
    {
        Info<< "Time = " << runTime.timeName() << nl << endl;

        while (simple.correctNonOrthogonal())
        {
            fvScalarMatrix TEqn
            (
                fvm::ddt(T) - fvm::laplacian(DT, T)
             ==
                fvOptions(T)
            );
```

Necessary “header” files

Matrix is created to solve “FVM”

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

```
        while (simple.correctNonOrthogonal())
        {
            fvScalarMatrix TEqn
            (
                fvm::ddt(T) - fvm::laplacian(DT, T)
             ==
                fvOptions(T)
            );

            fvOptions.constrain(TEqn);
            TEqn.solve();
            fvOptions.correct(T);
        }

        //#include "write.H"
        runTime.write();

        runTime.printExecutionTime(Info);
    }

    Info<< "End\n" << endl;

    return 0;
}
```

Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

Compile diffusionFoam "solver"

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

```
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$ ls
CASE1_diffusionFOAM  diffusionFoam
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$ cd diffusionFoam/
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/diffusionFoam$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/diffusionFoam$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/diffusionFoam$ wmake
Making dependencies: diffusionFoam.C
g++ -std=c++11 -m64 -pthread -DOPENFOAM=2306 -DWM_DP -DWM_LABEL_SIZE=32 -Wall -Wextra -Wold-style-cast -Wnon-virtual-dtor -Wno-unused-parameter -Wno-invalid-offsetof -Wno-attributes -Wno-unknown-pragmas -
O3  -DNoRepository -ftemplate-depth-100  -I/home/openfoam/OpenFOAM/openfoam/src/finiteVolume/lnInclude -I/home/openfoam/OpenFOAM/openfoam/src/meshTools/lnInclude -iquote. -IlnInclude -I/home/openfoam/Open
FOAM/openfoam/src/OpenFOAM/lnInclude -I/home/openfoam/OpenFOAM/openfoam/src/OSspecific/POSIX/lnInclude   -fPIC -c diffusionFoam.C -o Make/linux64GccDPInt32Opt/diffusionFoam.o
g++ -std=c++11 -m64 -pthread -DOPENFOAM=2306 -DWM_DP -DWM_LABEL_SIZE=32 -Wall -Wextra -Wold-style-cast -Wnon-virtual-dtor -Wno-unused-parameter -Wno-invalid-offsetof -Wno-attributes -Wno-unknown-pragmas -
O3  -DNoRepository -ftemplate-depth-100  -I/home/openfoam/OpenFOAM/openfoam/src/finiteVolume/lnInclude -I/home/openfoam/OpenFOAM/openfoam/src/meshTools/lnInclude -iquote. -IlnInclude -I/home/openfoam/Open
FOAM/openfoam/src/OpenFOAM/lnInclude -I/home/openfoam/OpenFOAM/openfoam/src/OSspecific/POSIX/lnInclude   -fPIC -Xlinker --add-needed -Xlinker --no-as-needed  Make/linux64GccDPInt32Opt/diffusionFoam.o -L/h
ome/openfoam/OpenFOAM/openfoam/platforms/linux64GccDPInt32Opt/lib \
    -lfiniteVolume -lfvOptions -lmeshTools -lOpenFOAM -ldl  \
     -lm -o /home/openfoam/OpenFOAM/openfoam-v2306/platforms/linux64GccDPInt32Opt/bin/diffusionFoam
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/diffusionFoam$
```

→ **Compiled "diffusionFoam" solver**

Kumaresh Selvakumar
kumaresh@exaslate.com

case file

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

ComputationalThermalEngineering / DAY7-OpenFOAM_diffusion_equation / **CASE1_diffusionFOAM** /

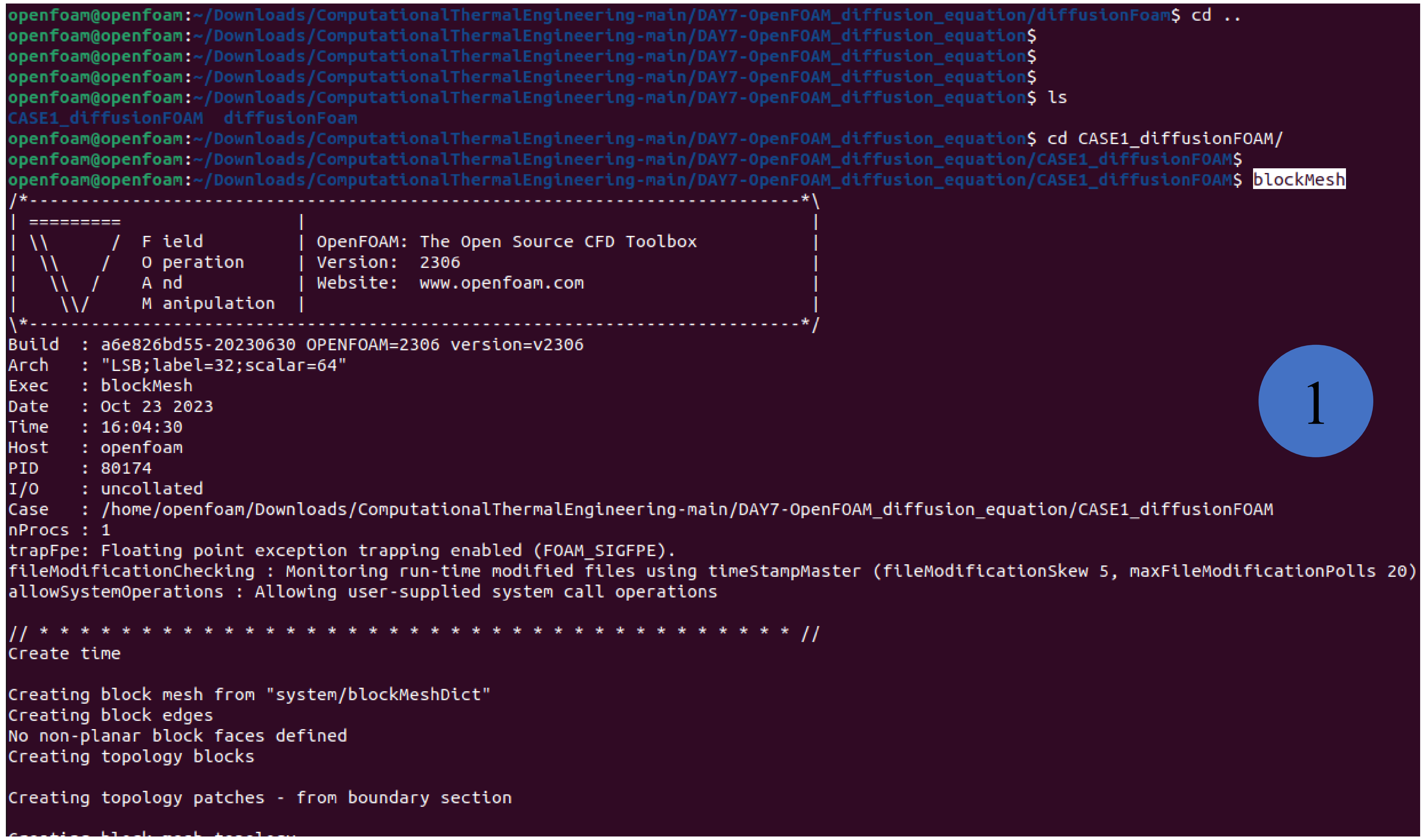
Kumaresh0402 Delete DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM/system/as

Name		Last commit message
..		
0	Initial conditions	Delete DAY7-OpenFOAM_diffu
constant	Properties	Delete DAY7-OpenFOAM_diffu
system	Mesh, schemes, solvers, algorithm controls and tolerances for the implicit solvers.	Delete DAY7-OpenFOAM_diffu

Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop = Guide = Collab

Compile the “case” file named – **CASE1_diffusionFOAM**



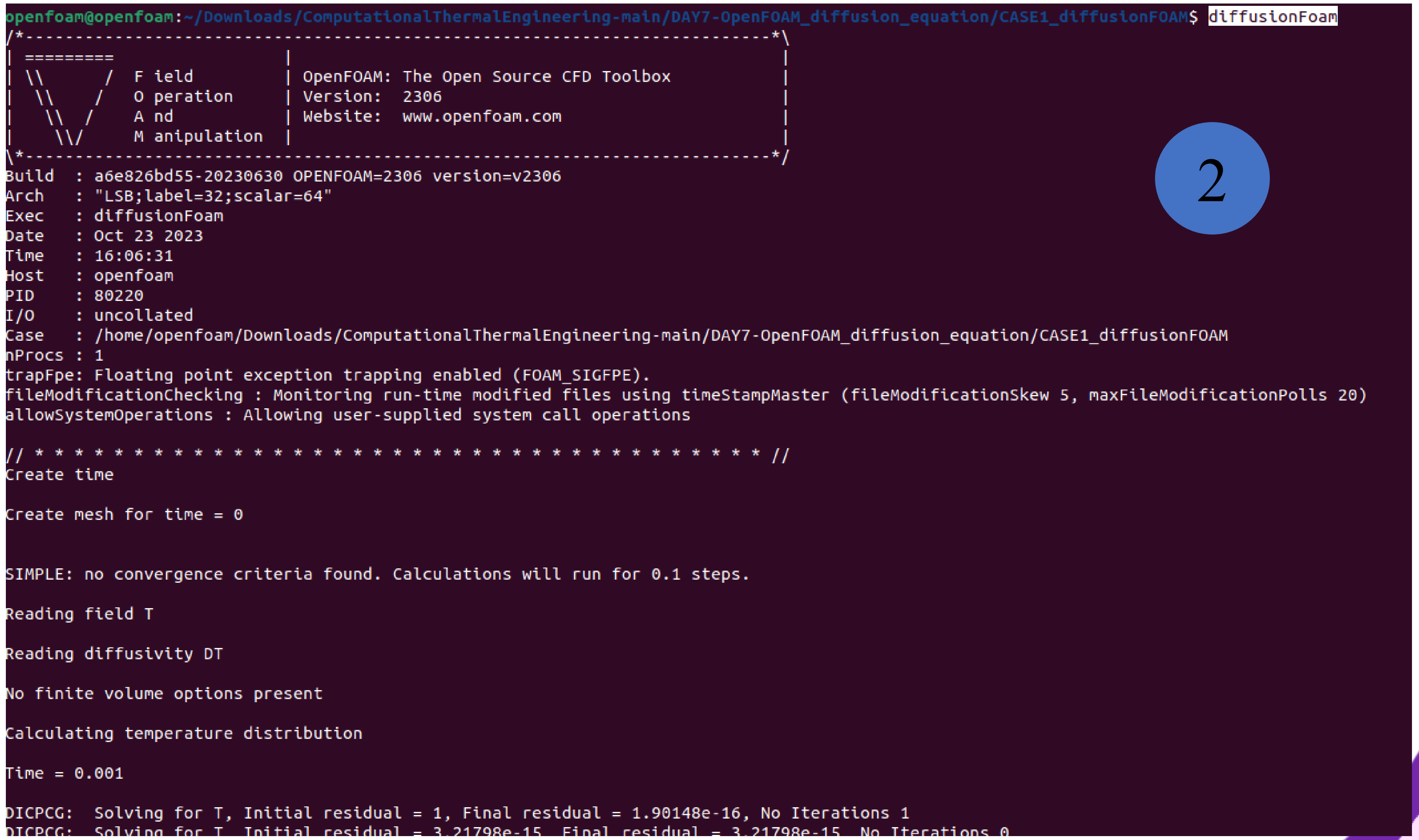
```
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/diffusionFoam$ cd ..
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$ ls
CASE1_diffusionFOAM  diffusionFoam
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation$ cd CASE1_diffusionFOAM/
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$ blockMesh
/*---------------------------------------------------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2306                                  |
|   \\  /    A nd           | Website:  www.openfoam.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
Build  : a6e826bd55-20230630 OPENFOAM=2306 version=v2306
Arch   : "LSB;label=32;scalar=64"
Exec   : blockMesh
Date   : Oct 23 2023
Time   : 16:04:30
Host   : openfoam
PID    : 80174
I/O    : uncollated
Case   : /home/openfoam/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM
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

Creating block mesh from "system/blockMeshDict"
Creating block edges
No non-planar block faces defined
Creating topology blocks

Creating topology patches - from boundary section
```

1



```
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$ diffusionFoam
/*---------------------------------------------------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2306                                  |
|   \\  /    A nd           | Website:  www.openfoam.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
Build  : a6e826bd55-20230630 OPENFOAM=2306 version=v2306
Arch   : "LSB;label=32;scalar=64"
Exec   : diffusionFoam
Date   : Oct 23 2023
Time   : 16:06:31
Host   : openfoam
PID    : 80220
I/O    : uncollated
Case   : /home/openfoam/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM
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

Create mesh for time = 0


SIMPLE: no convergence criteria found. Calculations will run for 0.1 steps.

Reading field T

Reading diffusivity DT

No finite volume options present

Calculating temperature distribution

Time = 0.001

DICPCG:  Solving for T, Initial residual = 1, Final residual = 1.90148e-16, No Iterations 1
DICPCG:  Solving for T, Initial residual = 3.21798e-15, Final residual = 3.21798e-15, No Iterations 0
```

2

Kumaresh Selvakumar
kumaresh@exaslate.com

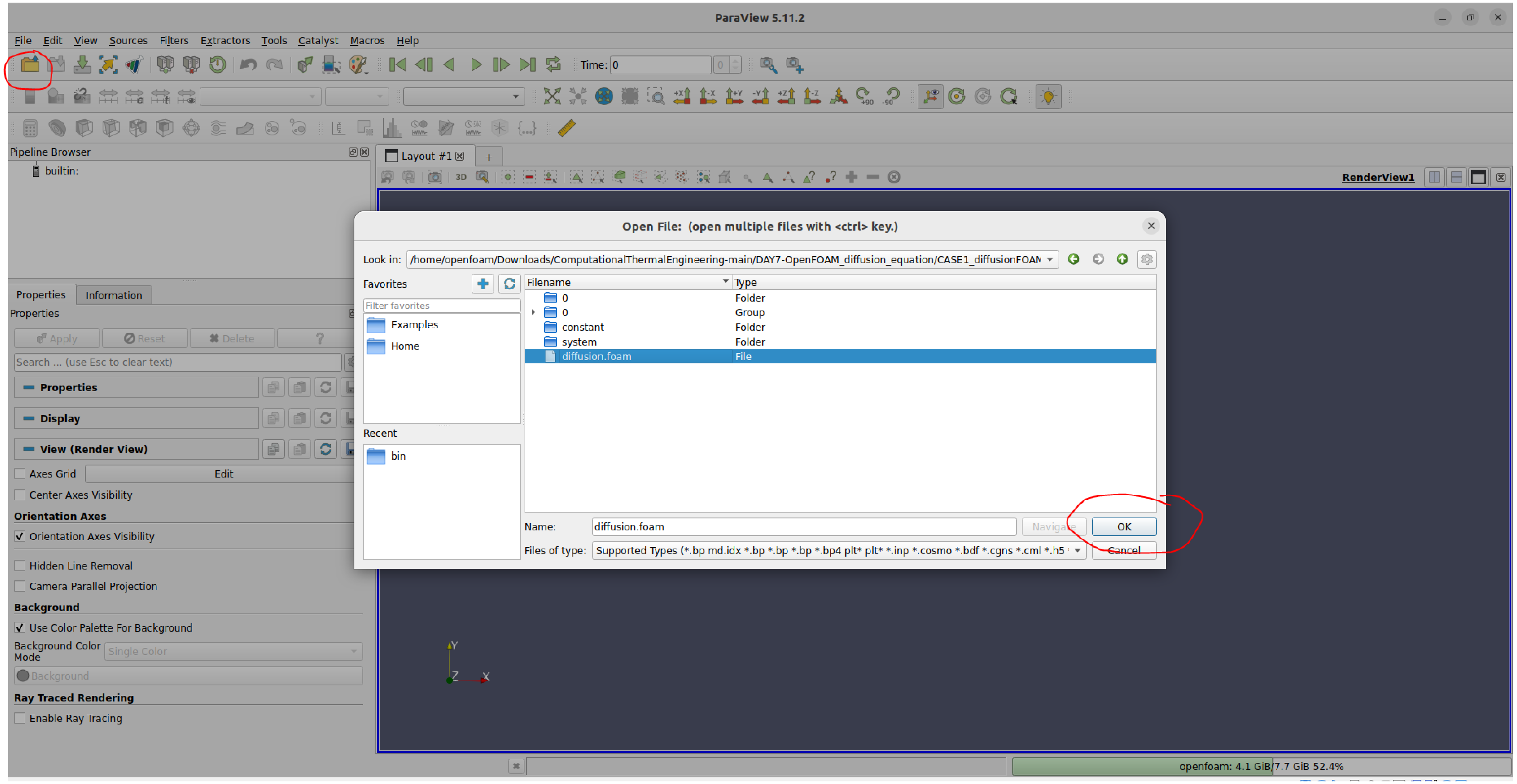
Compile the “case” file named – **CASE1_diffusionFOAM**

```
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$ ls
0      0.004  0.008  0.012  0.016  0.02   0.024  0.028  0.032  0.036  0.04   0.044  0.048  0.052  0.056  0.06   0.064  0.068  0.072  0.076  0.08   0.084  0.088  0.092  0.096  0.1
0.001  0.005  0.009  0.013  0.017  0.021  0.025  0.029  0.033  0.037  0.041  0.045  0.049  0.053  0.057  0.061  0.065  0.069  0.073  0.077  0.081  0.085  0.089  0.093  0.097  constant
0.002  0.006  0.01   0.014  0.018  0.022  0.026  0.03   0.034  0.038  0.042  0.046  0.05   0.054  0.058  0.062  0.066  0.07   0.074  0.078  0.082  0.086  0.09   0.094  0.098  system
0.003  0.007  0.011  0.015  0.019  0.023  0.027  0.031  0.035  0.039  0.043  0.047  0.051  0.055  0.059  0.063  0.067  0.071  0.075  0.079  0.083  0.087  0.091  0.095  0.099
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$ touch diffusion.foam
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$ ls
0      0.004  0.008  0.012  0.016  0.02   0.024  0.028  0.032  0.036  0.04   0.044  0.048  0.052  0.056  0.06   0.064  0.068  0.072  0.076  0.08   0.084  0.088  0.092  0.096  0.1
0.001  0.005  0.009  0.013  0.017  0.021  0.025  0.029  0.033  0.037  0.041  0.045  0.049  0.053  0.057  0.061  0.065  0.069  0.073  0.077  0.081  0.085  0.089  0.093  0.097  constant
0.002  0.006  0.01   0.014  0.018  0.022  0.026  0.03   0.034  0.038  0.042  0.046  0.05   0.054  0.058  0.062  0.066  0.07   0.074  0.078  0.082  0.086  0.09   0.094  0.098  diffusion.foam
0.003  0.007  0.011  0.015  0.019  0.023  0.027  0.031  0.035  0.039  0.043  0.047  0.051  0.055  0.059  0.063  0.067  0.071  0.075  0.079  0.083  0.087  0.091  0.095  0.099  system
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
openfoam@openfoam:~/Downloads/ComputationalThermalEngineering-main/DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM$
```

3

Kumaresh Selvakumar
kumaresh@exaslate.com

Open the “case” in paraview



Kumaresh Selvakumar
kumaresh@exaslate.com

ExaSlate
Develop · Guide · Collab

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

Project 1 - Solving diffusion equation in OpenFOAM #10

kummi0402 started this conversation in General

kummi0402 2 weeks ago Maintainer edited

> Make sure OpenFOAM is installed on your system.
> Install ParaView.
> Copy solver and test case to the working directory.
> Build/compile the solver.

1. wclean
2. wmake

> Run the test case by using following commands:

1. blockMesh
2. diffusionFoam
3. touch a.foam

> Visualize the results.
> Share screenshots of results here with clear description

↑ 1

Kumaresh Selvakumar
kumaresh@exaslate.com

THANK YOU