

Computational Thermal Engineering

Day - 7



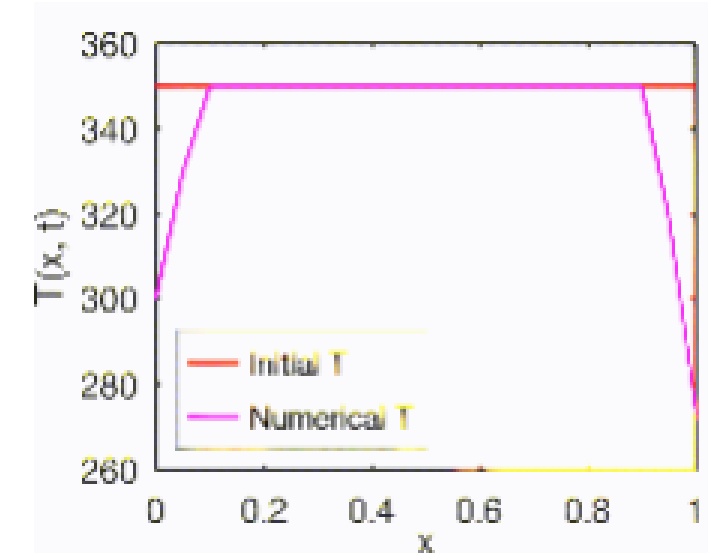
ExaSlate

Develop \equiv Guide \equiv Collab

Quick Recap

Exercise – 6 (Let's solve the Diffusion Equation)

$$T_i^{n+1} = T_i^n + \Delta t \alpha \frac{T_{i+1}^n - 2T_i^n + T_{i-1}^n}{\Delta x^2}$$



`a7_solve_diffusion_sample.m`

1. Resolve the diffusion equation with $\alpha = 1$, $dt = 0.001$, $dx = 0.05$, Dirichlet boundaries ($T_{\text{left}} = 300\text{K}$, $T_{\text{right}} = 273\text{K}$), and write the above numerical solution to extract the results.
2. Change the right boundary condition from Dirichlet to Neumann. Explain about it in few words.
3. Analyze for different time steps (dt) 0.1 and 0.01 and give your comments. Hint: Von Neumann stability analysis.
4. Learning debug skills – fix breakpoints, run, and understand the codes. Explain about it in few words.

Contents

- Introduction to OpenFOAM
- OpenFOAM: Numerical Solution to Diffusion Equation

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.
- OpenFOAM (Version v2306) is implemented in Ubuntu (version 22.04)
- ParaView (version 5.1.0) for post-processing.

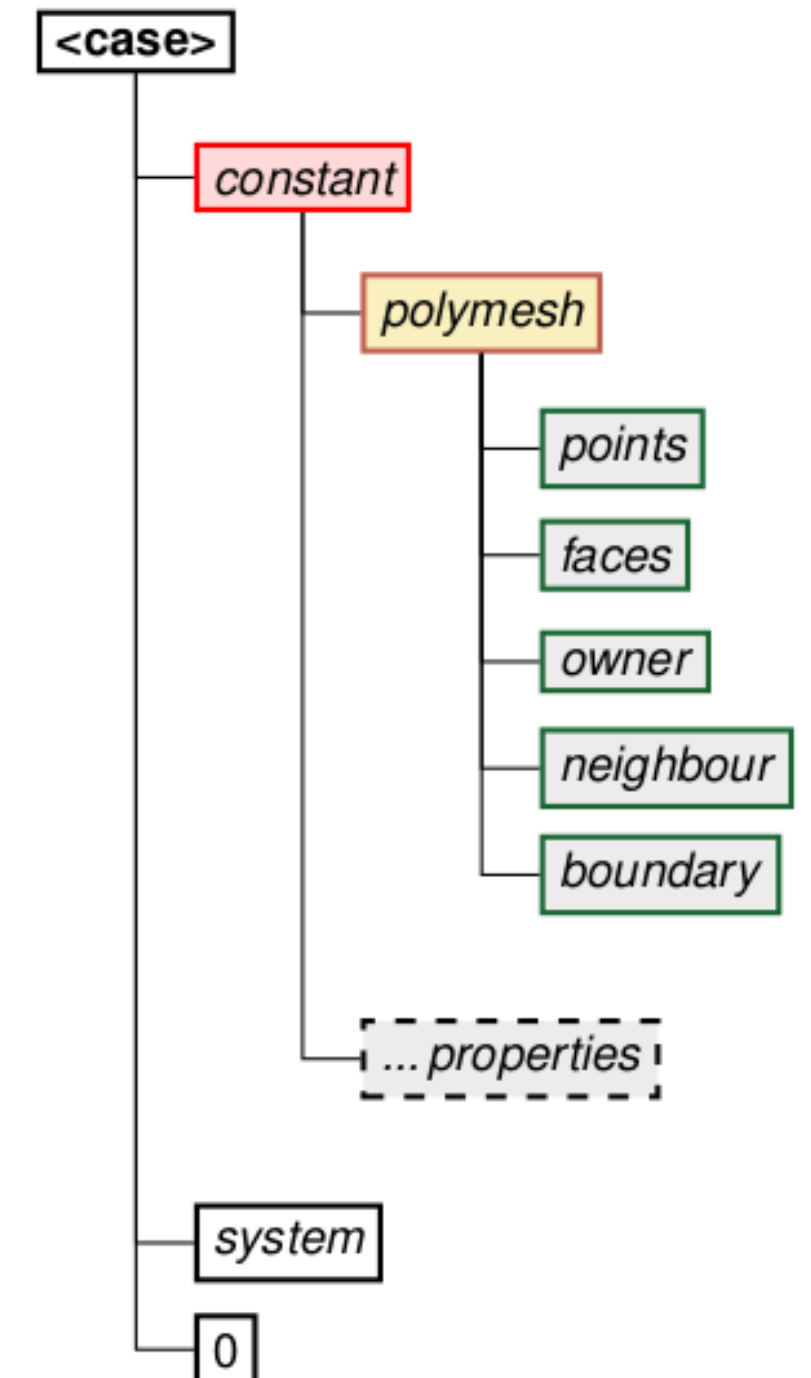
OpenFOAM – Directory Structure

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.



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:

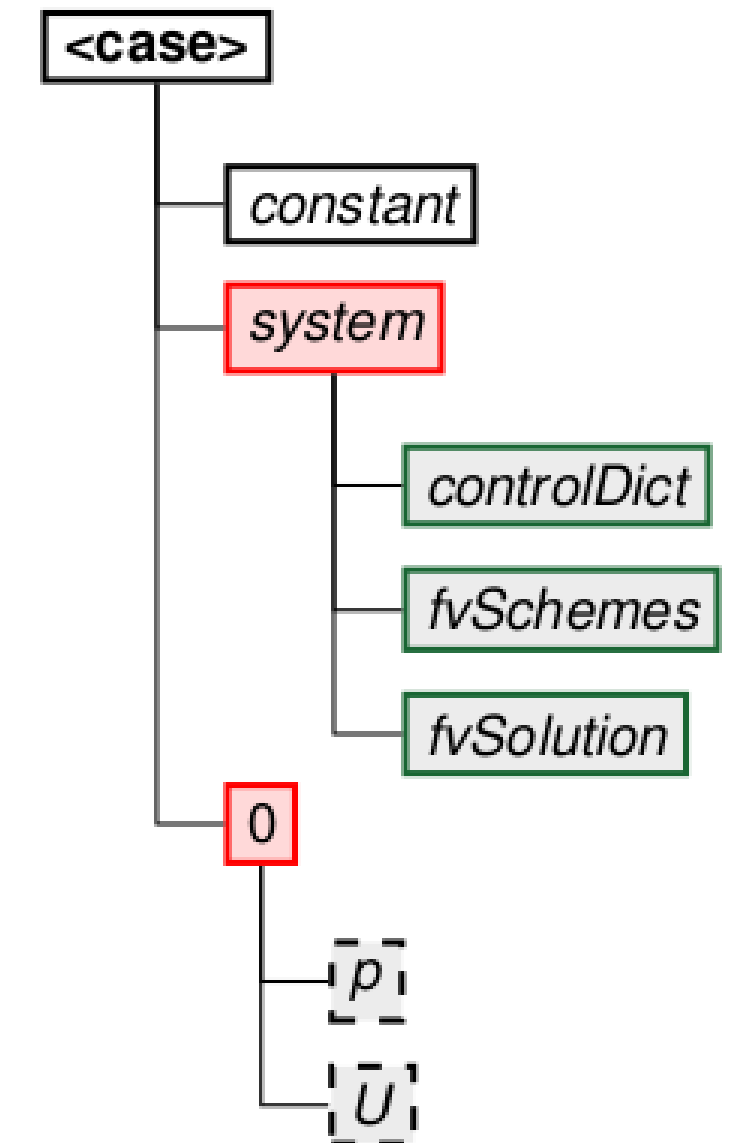
- The principle behind blockMesh is to **decompose the domain geometry** into a set of 1 or more three dimensional, hexahedral blocks. This section describes the **mesh generation utility**, blockMesh, supplied with **OpenFOAM**. The blockMesh utility creates parametric meshes with grading and curved 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



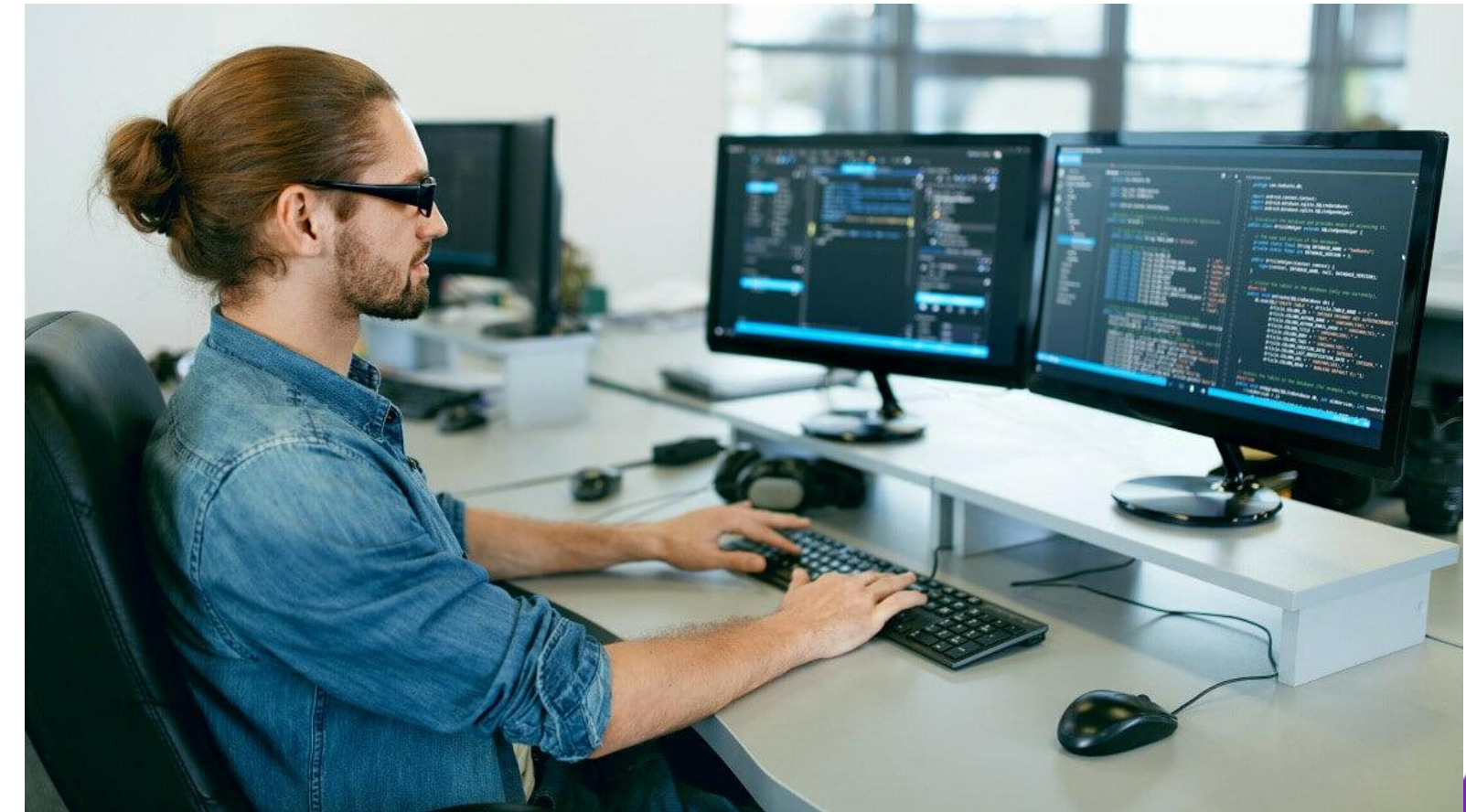
CFD Developers

- 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.



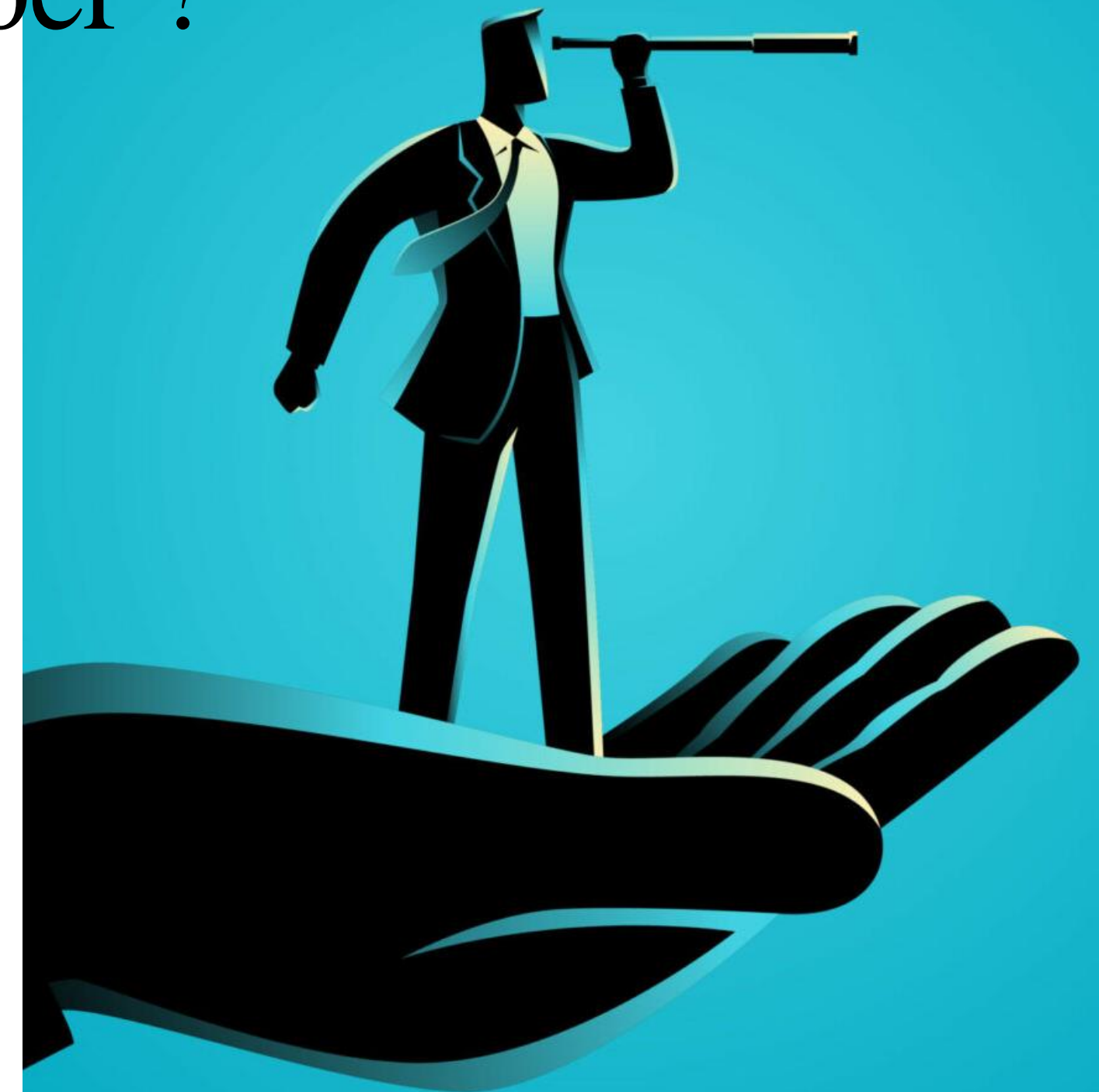
Is it good to be a black box user or a developer ?

- 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.



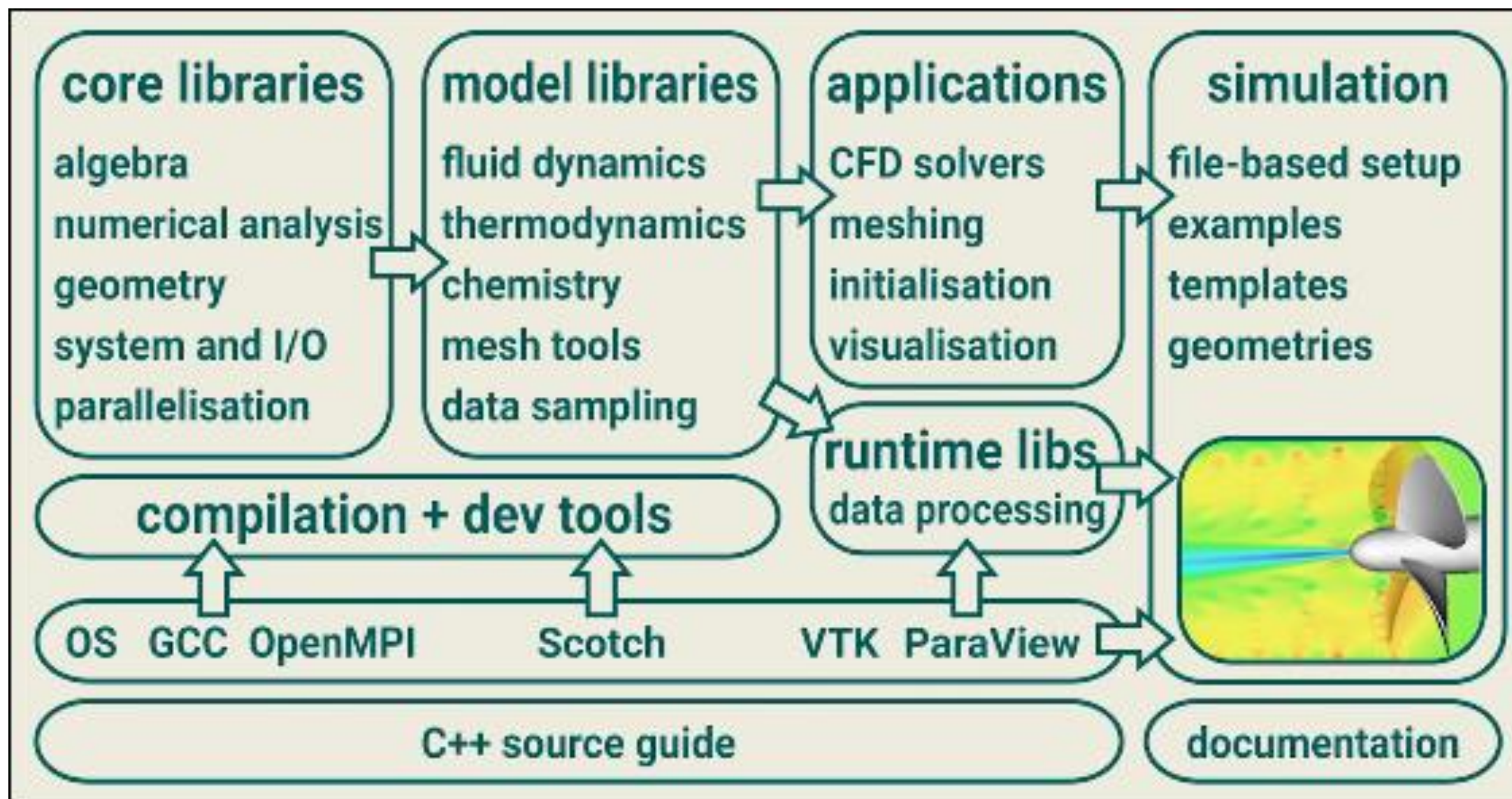
How to grow as a developer ?

- 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



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.

OpenFOAM User GUIDE



User Guide

version 11

11th July 2023

<https://openfoam.org>

Contents

1 Introduction

2 Tutorials

- 2.1 Backward-facing step
- 2.2 Breaking of a dam
- 2.3 Stress analysis of a plate with a hole

3 Applications and libraries

- 3.1 The programming language of OpenFOAM
- 3.2 Compiling applications and libraries
- 3.3 Running applications
- 3.4 Running applications in parallel
- 3.5 Solver modules
- 3.6 Standard solvers
- 3.7 Standard utilities

4 OpenFOAM cases

- 4.1 File structure of OpenFOAM cases
- 4.2 Basic input/output file format
- 4.3 Global controls
- 4.4 Time and data input/output control
- 4.5 Numerical schemes
- 4.6 Solution and algorithm control
- 4.7 Case management tools

5 Mesh generation and conversion

- 5.1 [Mesh description](#)
- 5.2 Mesh files
- 5.3 Mesh boundary
- 5.4 Mesh generation with the blockMesh utility
- 5.5 Mesh generation with the snappyHexMesh utility
- 5.6 Mesh conversion
- 5.7 Mapping fields between different geometries

6 Boundary conditions

- 6.1 Patch selection
- 6.2 Geometric constraints
- 6.3 Basic boundary conditions
- 6.4 Derived boundary conditions

7 Post-processing

- 7.1 ParaView/paraFoam graphical user interface (GUI)
- 7.2 Post-processing command line interface (CLI)
- 7.3 Post-processing functionality
- 7.4 Sampling and monitoring data
- 7.5 Third-Party post-processing

8 Models and physical properties

- 8.1 Thermophysical models
- 8.2 Turbulence models
- 8.3 Transport/rheology models

Index

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).

Machine Learning in CFD



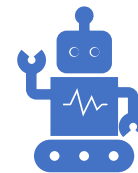
By definition, machine learning is a branch of artificial intelligence (AI) and computer science that focuses on the use of data and algorithms to imitate the way that humans learn, gradually improving its accuracy.



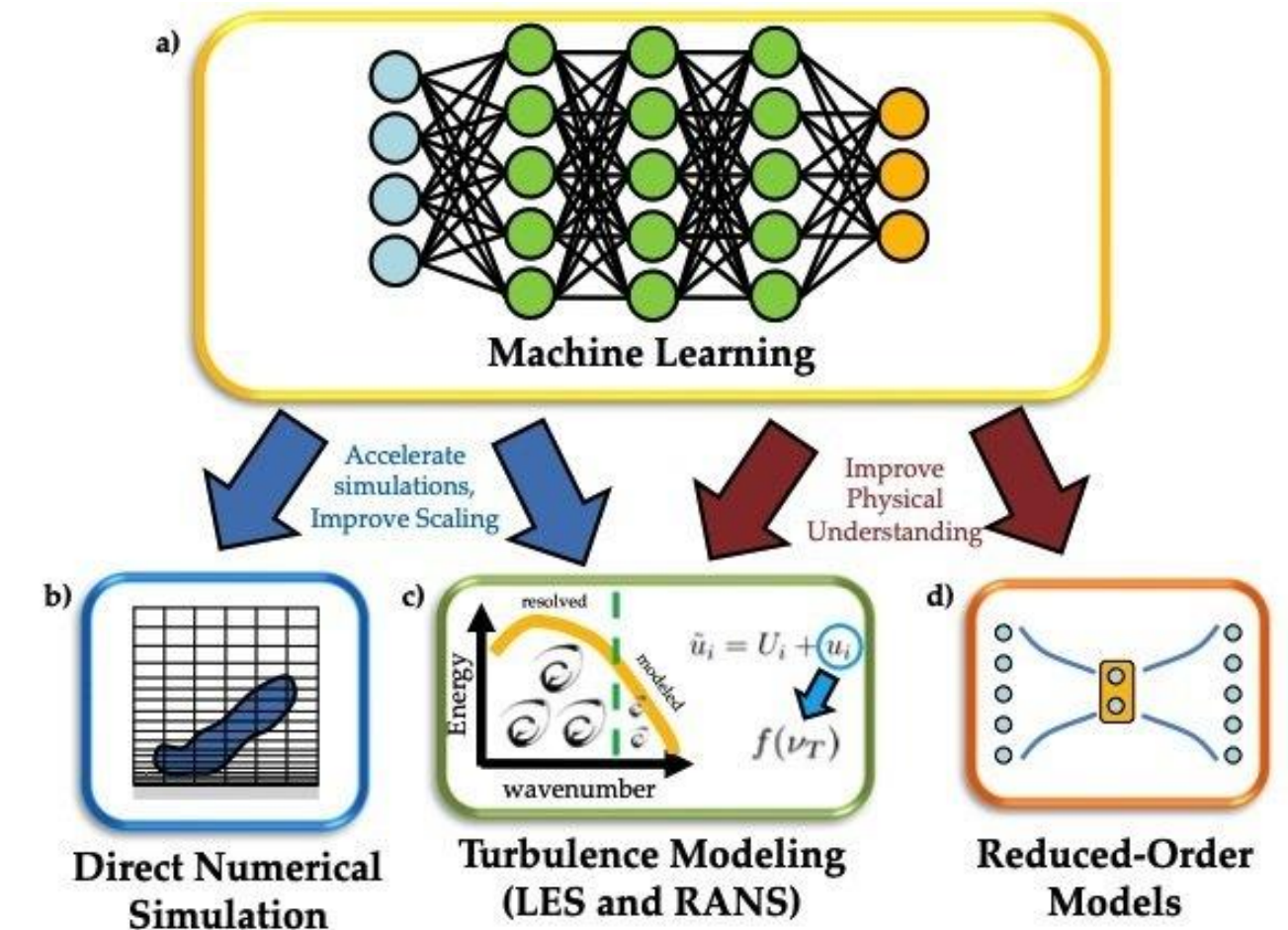
To understand certain critical parameters in any physical phenomenon, data optimisation can be implemented by machine learning algorithms.



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

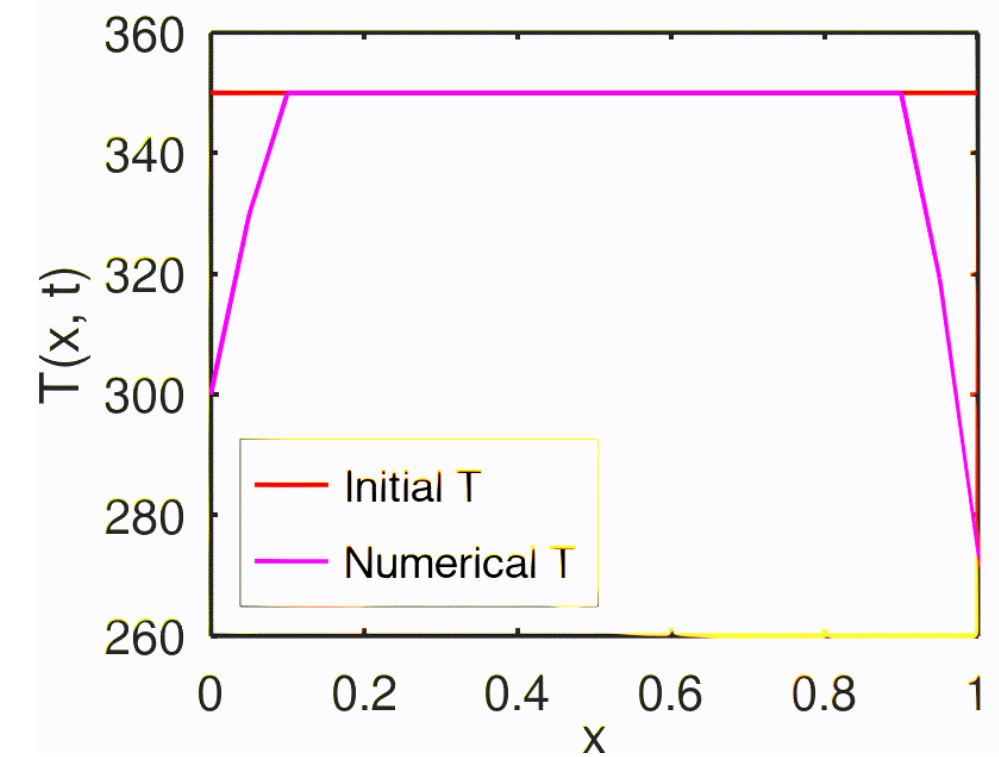
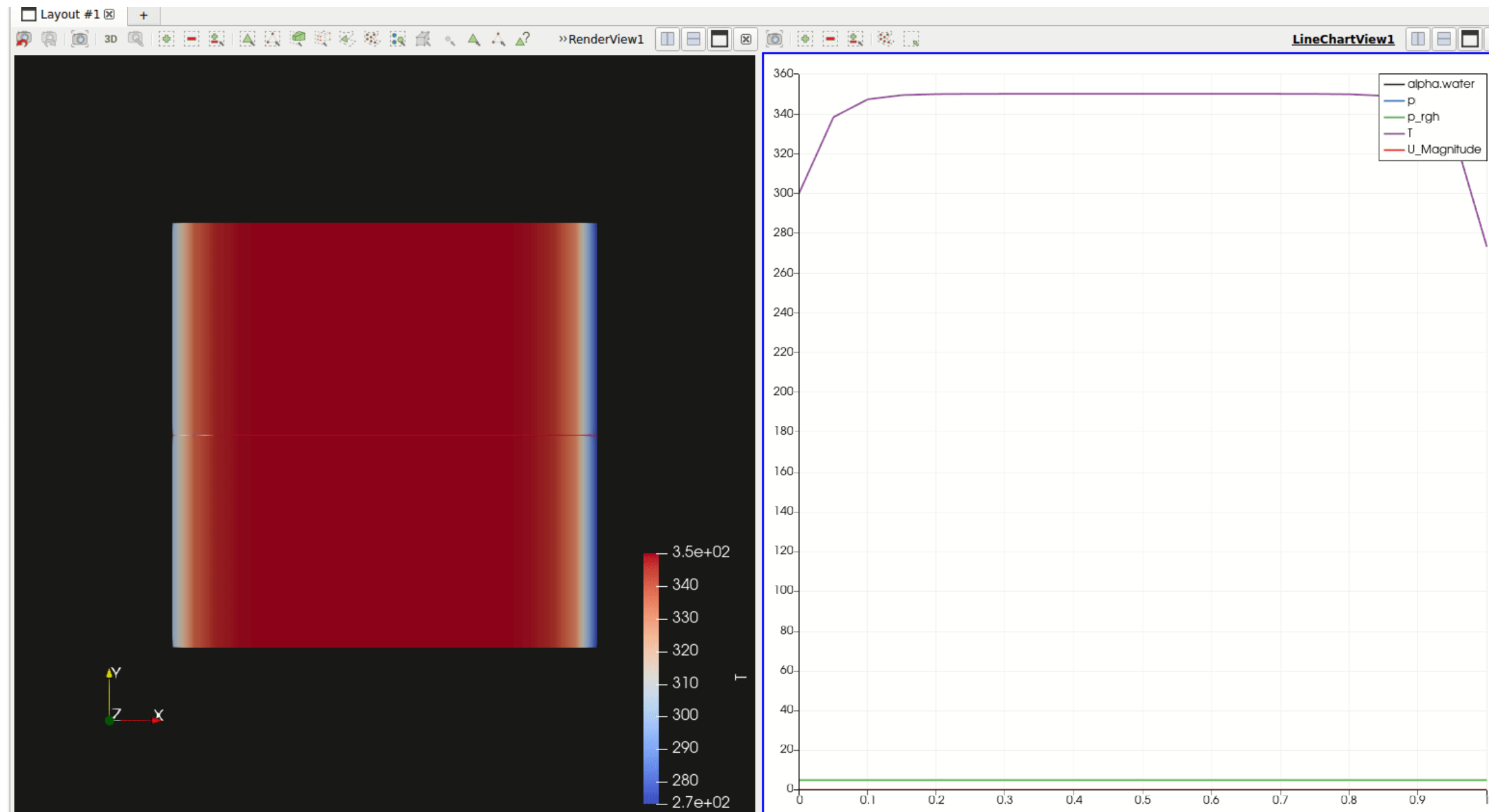


This improvisation will assist steel industry technologies in prospering and creating a better future.



OpenFOAM: Numerical Solution to Diffusion Equation

Paraview – Post Processing tool in OpenFOAM



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

Install “Paraview”

paraview install

About 228,000 results (0.28 seconds)

1

ParaView
https://www.paraview.org > download
Download ParaView
Download **ParaView**. Need help? For general **ParaView** help, visit the Resources page. If you are looking to maximize **ParaView's** capabilities, contact Kitware.

https://www.paraview.org/download/

kitware

ParaView

about solutions resources companion tools customize

download

Download ParaView

Get the Software

You can either download binaries or source code archives for the latest stable or previous release or access the current development (aka nightly) distribution through Git. Specific license information can be found [here](#). This software may not be exported in violation of any U.S. export laws or regulations. For more information regarding Export Control matters please go to https://kitware.com/export_control/index.html.

Version v5.11

ParaView

Full suite of ParaView tools, including the ParaView GUI client, pvpython, pvserver, pvbatch, and bundled MPI.

2

	Sources	Windows	Linux	macOS
ParaView-5.11.2-MPI-Linux-Python3.9-x86_64.tar.gz			2023-09-25 10:40	584.0M
ParaView-5.11.1-MPI-Linux-Python3.9-x86_64.tar.gz			2023-03-31 10:44	584.0M
ParaView-5.11.0-MPI-Linux-Python3.9-x86_64.tar.gz			2022-11-16 14:47	585.1M

Home / Downloads


Name
rod_bundles
google-chrome-stable_current_amd64.deb
rod_bundles.zip
fluentMesh01_Nobinary.msh
ParaView-5.11.2-MPI-Linux-Python3.9-x86_64.tar.gz


3


- Open With Archive Manager Return
- Open With Other Application
- Cut Ctrl+X
- Copy Ctrl+C
- Move to...
- Copy to...
- Move to Trash Delete
- Rename... F2
- Extract Here**
- Extract to...
- Compress...
- Send to...
- Star
- Properties Ctrl+I


Install “Paraview”


 ParaView-5.11.2-MPI-Linux-Python3.9-x86_64

 rod_bundles

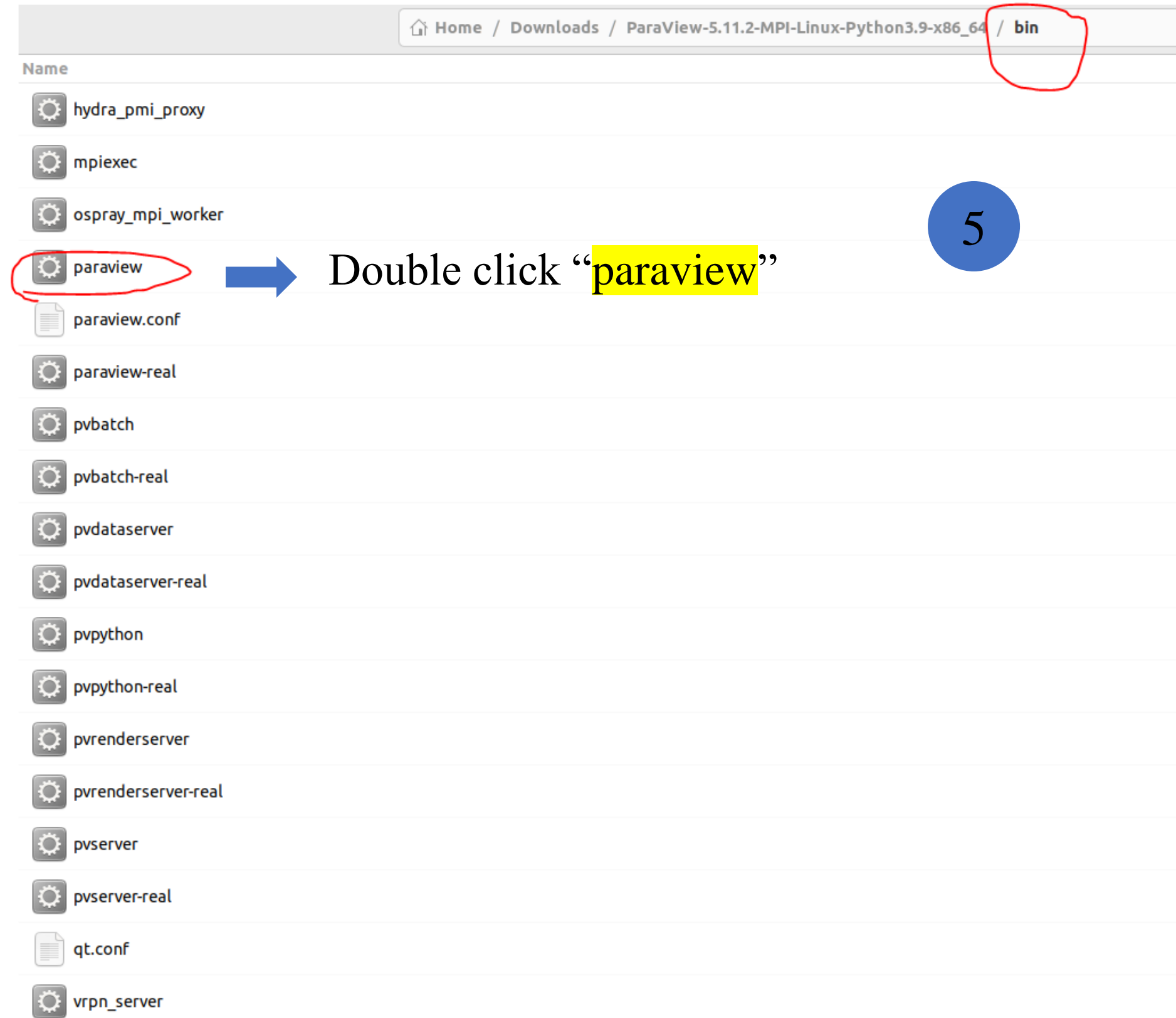
 google-chrome-stable_current_amd64.deb

 rod_bundles.zip

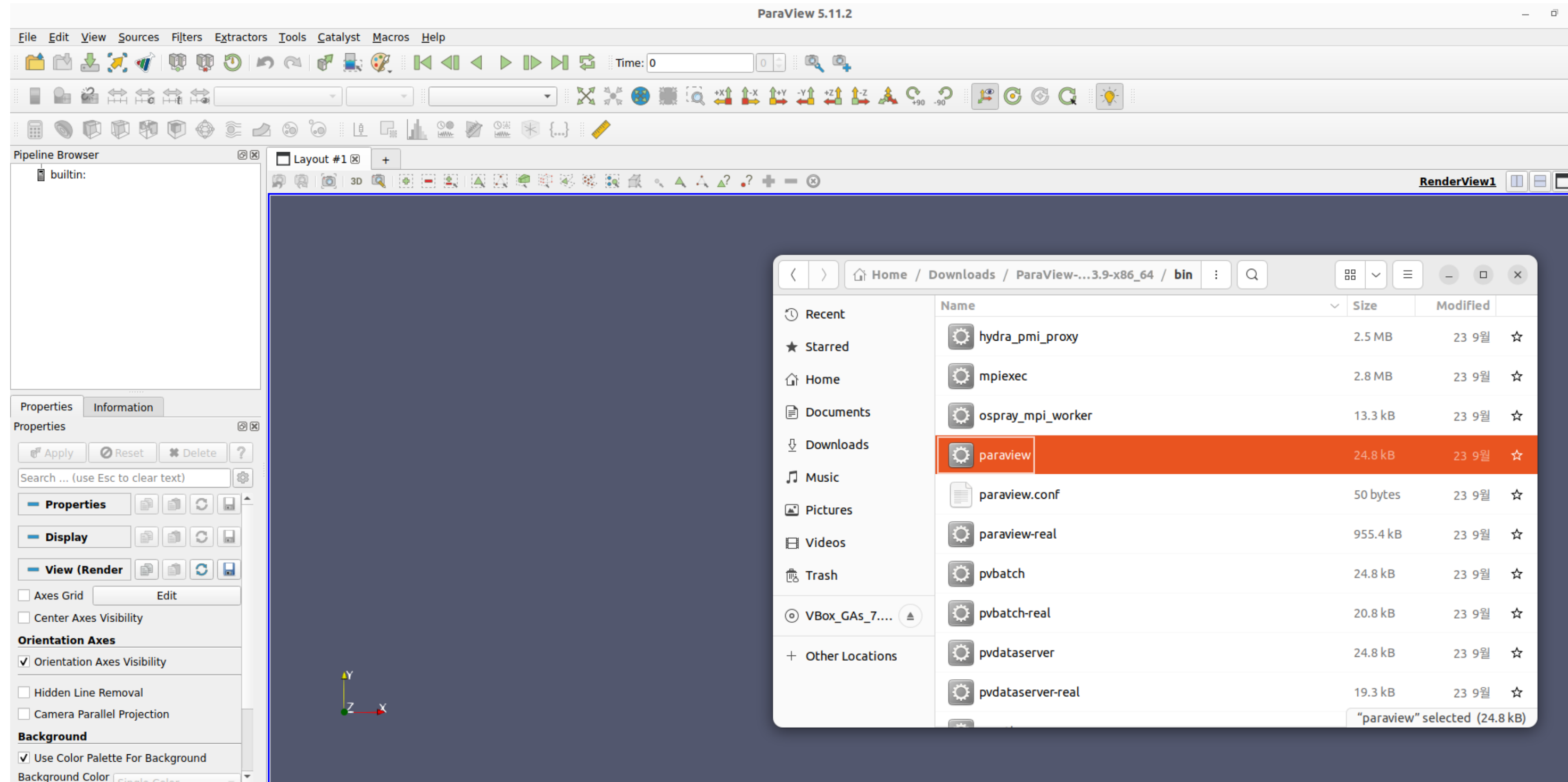
 fluentMesh01_Nobinary.msh

 ParaView-5.11.2-MPI-Linux-Python3.9-x86_64.tar.gz

4





Open “Paraview” → Used for **post-processing** your results







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

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

```

1  Info<< "Reading field T\n" << endl;
2
3  volScalarField T
4  (
5      IObject
6      (
7          "T",
8          runtime.timeName(),
9          mesh,
10         IObject::MUST_READ,
11         IObject::AUTO_WRITE
12     ),
13     mesh
14 );
15
16
17  Info<< "Reading diffusivity DT\n" << endl;
18
19  volScalarField DT
20  (
21      IObject
22      (
23          "DT",
24          runtime.timeName(),
25          mesh,
26          IObject::READ_IF_PRESENT,
27          IObject::AUTO_WRITE
28      ),
29      mesh,
30      dimensionedScalar(dimViscosity, Zero)
31  );

```

Temperature “T” field is created

Diffusivity as a field created

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

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;
}
```



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 -
03 -DNoRepository -ftemplate-depth-100 -I/home/openfoam/OpenFOAM/openfoam/src/finiteVolume/include -I/home/openfoam/OpenFOAM/openfoam/src/meshTools/include -I/home/openfoam/Open
FOAM/openfoam/src/OpenFOAM/include -I/home/openfoam/OpenFOAM/openfoam/src/OSspecific/POSIX/include -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 -
03 -DNoRepository -ftemplate-depth-100 -I/home/openfoam/OpenFOAM/openfoam/src/finiteVolume/include -I/home/openfoam/OpenFOAM/openfoam/src/meshTools/include -I/home/openfoam/Open
FOAM/openfoam/src/OpenFOAM/include -I/home/openfoam/OpenFOAM/openfoam/src/OSspecific/POSIX/include -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

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



Kumaresh0402

Delete DAY7-OpenFOAM_diffusion_equation/CASE1_diffusionFOAM/system/as

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

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

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

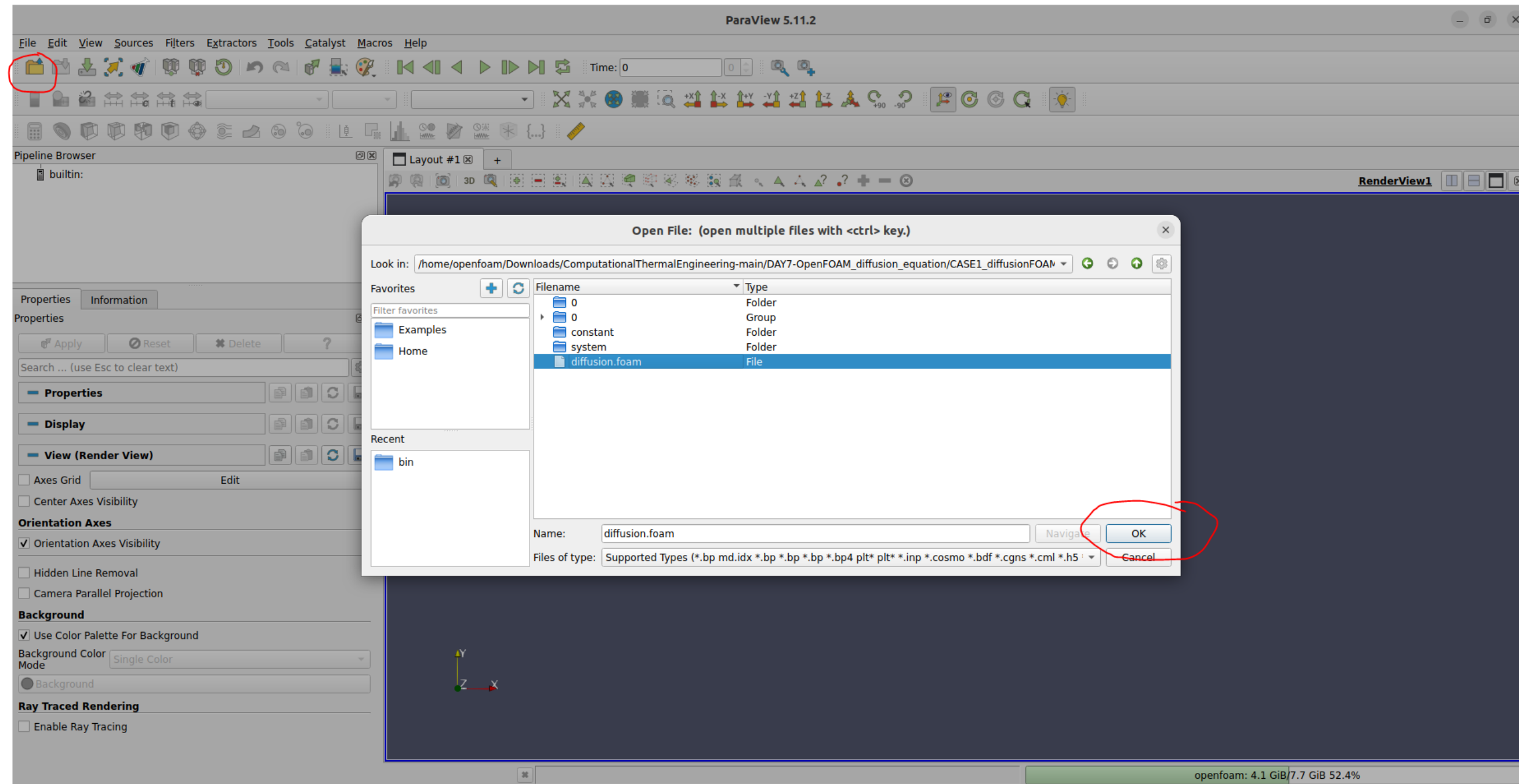
DICPCG: Solving for T, Initial residual = 1, Final residual = 1.90148e-16, No Iterations 1
DICPCG: Solving for T, Initial residual = 3.21798e-15, Final residual = 3.21798e-15, No Iterations 0
```

2

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$ 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$ █
```

Open the “case” in paraview




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

Project – 1 (Diffusion Equation using OpenFOAM)



Kumares0402 1 minute ago Maintainer

edited ▼  Tip
...

Based on DAY 7 presentation, repeat all steps we discussed during the session

- Make sure OpenFOAM is installed on your systems.
- Install ParaView.
- Copy solver and test case to the working directory.
- Build/compile the solver.
- Run the test case.
- Visualize the results.
- Share screenshots of results here.



THANK YOU