

# Applied Computational Fluid Dynamics using OpenFOAM

DAY 6

Creating the new OF solver

---

Value Added Course  
AEC  
Spring 2025

*Kumaresh*

# Contents

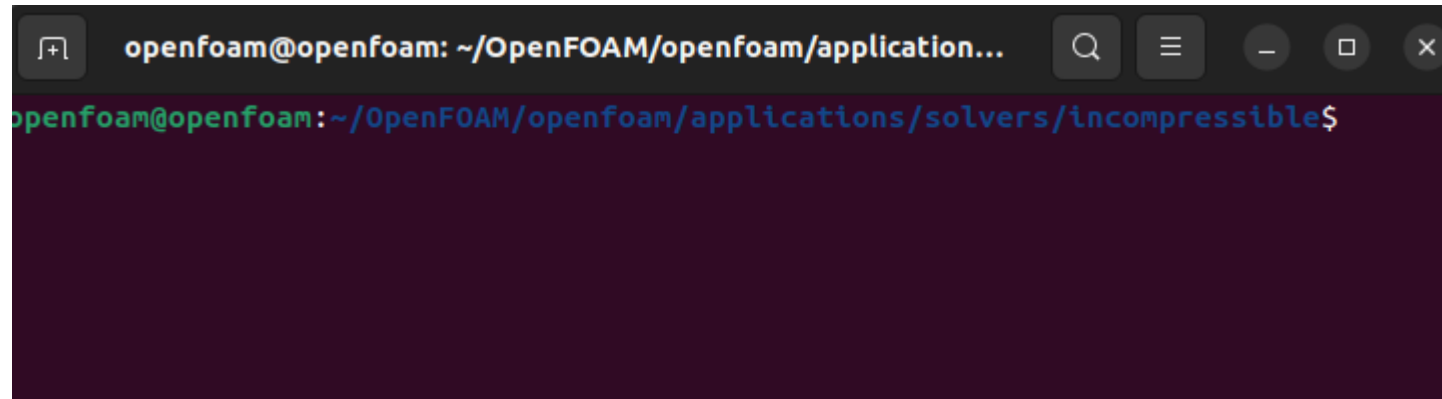
- Creating a new OpenFOAM solver based on icoFoam
- Exercise – 9

# Following things to do:

1. Copy default icoFoam solver
2. Adding the temperature field in the icoFoam solver
3. Copy default cavity tutorial based on icoFoam
4. Add a new file for initial and boundary conditions
5. Add respective files in fvSchemes and fvSolution
6. Run your new case file
7. Upload your new solver, case files, and results in GITHUB

# 1. Copy default icoFoam solver

From the location:

A terminal window with a dark background. The title bar shows 'openfoam@openfoam: ~/OpenFOAM/openfoam/application...'. The prompt is 'openfoam@openfoam:~/OpenFOAM/openfoam/applications/solvers/incompressible\$'.

To make your own solver, firstly do the following:

- Change the default icoFoam name into myIcoFoam.
- Change the source file name from icoFoam.C into myIcoFoam.C
- Open Files, and do the following:

myIcoFoam.C

EXE = \$(FOAM\_USER\_APPBIN)/myIcoFoam

## 2. Adding the temperature field in the icoFoam solver

Under createFields.H

```
//Add here...
dimensionedScalar DT
(
    "DT",
    dimViscosity,
    transportProperties
);
```

```
Info<< "Reading field T\n" <<endl;
volScalarField T
(
    IOobject
    (
        "T",
        runTime.timeName(),
        mesh,
        IOobject::MUST_READ,
        IOobject::AUTO_WRITE
    ),
    mesh
);
```


Under myIcoFoam.C

```
//add these lines...
    fvScalarMatrix TEqn
    (
        fvm::ddt(T)
        + fvm::div(phi, T)
        - fvm::laplacian(DT, T)
    );

    TEqn.solve();
//done adding lines...
```

### 3. Copy default cavity tutorial based on icoFoam

From the location:

A terminal window with a dark background. The title bar shows the user 'openfoam@openfoam' and the current directory '~/OpenFOAM/openfoam/tutorials/in...'. The terminal text shows the user at the prompt 'openfoam@openfoam:~/OpenFOAM/openfoam/tutorials/incompressible/icoFoam/cavity\$'.

```
openfoam@openfoam: ~/OpenFOAM/openfoam/tutorials/in...
openfoam@openfoam:~/OpenFOAM/openfoam/tutorials/incompressible/icoFoam/cavity$
```

Modify your solver name as myCavityCaseFile

# 4. Add a new file for initial and boundary conditions

Under constant/transportProperties

Add DT (new variable created)

```
1 /*----- C++ -----*/
2 |=====|
3 | \ / | F i e l d | OpenFOAM: The Open Source CFD Toolbox
4 | \ / | O peration | Version: v2306
5 | \ / | A nd | Website: www.openfoam.com
6 | \ / | M anipulation |
7 |=====|
8 FoamFile
9 {
10     version      2.0;
11     format        ascii;
12     class         dictionary;
13     object         transportProperties;
14 }
15 // *****
16
17 nu              0.01;
18
19 DT              0.002;
20
21 // *****
```

Add “T” field in “0” file

```
/*----- C++ -----*/
|=====|
| \ / | F i e l d | OpenFOAM: The Open Source CFD Toolbox
| \ / | O peration | Version: v2306
| \ / | A nd | Website: www.openfoam.com
| \ / | M anipulation |
|=====|
FoamFile
{
    version      2.0;
    format        ascii;
    class         volScalarField;
    object        T;
}
// *****

dimensions      [0 0 0 1 0 0 0];

internalField    uniform 300;

boundaryField
{
    movingWall
    {
        type      fixedValue;
        value      uniform 350;
    }

    fixedWalls
    {
        type      fixedValue;
        value      uniform 300;
    }

    frontAndBack
    {
        type      empty;
    }
}
// *****
```

# 5. Add respective files in fvSchemes and fvSolution

## Under system/fvSchemes

```
8 FoamFile
9 {
10     version      2.0;
11     format        ascii;
12     class         dictionary;
13     object        fvSchemes;
14 }
15 // *****
16
17 ddtSchemes
18 {
19     default        Euler;
20 }
21
22 gradSchemes
23 {
24     default        Gauss linear;
25     grad(p)        Gauss linear;
26 }
27
28 divSchemes
29 {
30     default        none;
31     div(phi,U)     Gauss linear;
32     div(phi,T)     Gauss upwind;
33 }
34
35 laplacianSchemes
36 {
37     default        Gauss linear orthogonal;
38     laplacian(DT,T) Gauss linear corrected;
39 }
40
41 interpolationSchemes
42 {
43     default        linear;
44 }
45
46 snGradSchemes
47 {
48     default        orthogonal;
49 }
50
51
52 // *****
```

## Under system/fvSolution

```
solvers
{
    p
    {
        solver          PCG;
        preconditioner   DIC;
        tolerance        1e-06;
        relTol           0.05;
    }

    pFinal
    {
        Sp;
        relTol           0;
    }

    T
    {
        solver          PBiCGStab;
        preconditioner   DILU;
        tolerance        1e-6;
        relTol           0.1;
    };

    U
    {
        solver          smoothSolver;
        smoother         symGaussSeidel;
        tolerance        1e-05;
        relTol           0;
    }
}

PISO
{
    nCorrectors        2;
    nNonOrthogonalCorrectors 0;
    pRefCell            0;
    pRefValue           0;
}

// *****
```



## 6. Run your new case file

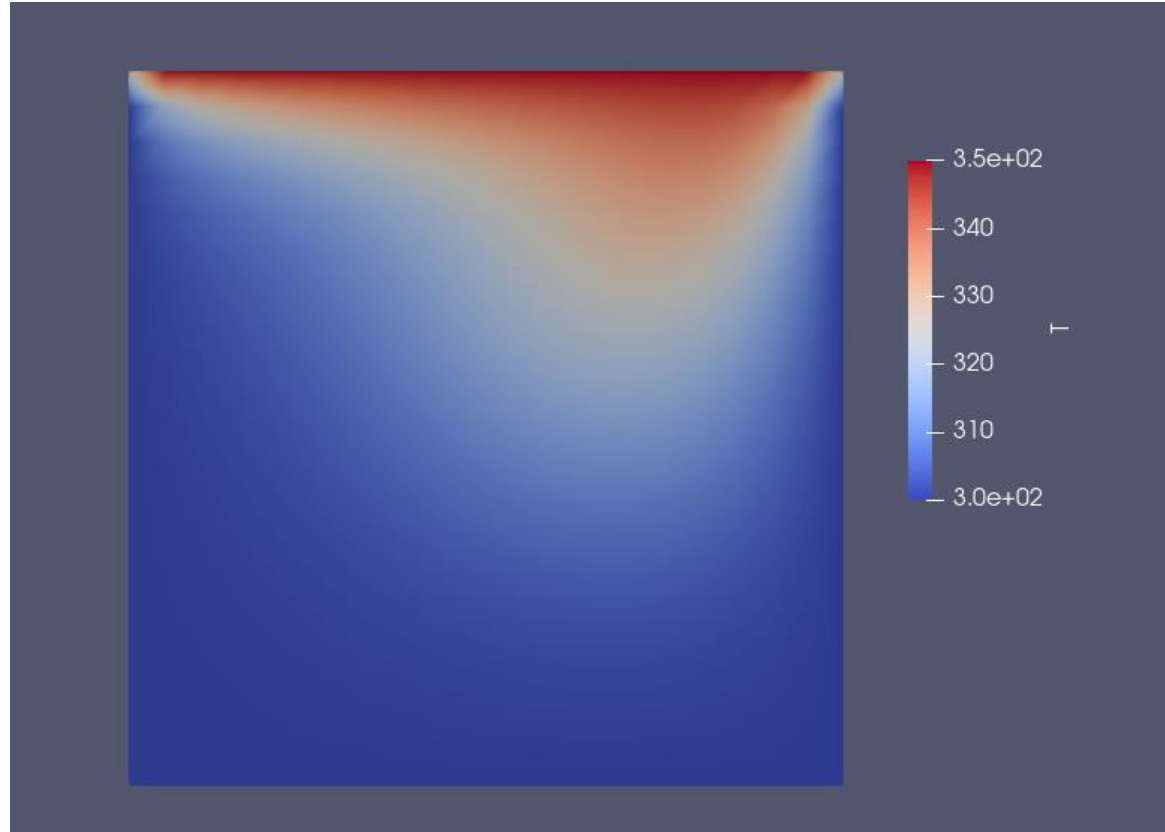
```
Time = 0.4998
Courant Number mean: 0.00444052 max: 0.0170421
smoothSolver: Solving for Ux, Initial residual = 1.5517e-09, Final residual = 1.5517e-09, No Iterations 0
smoothSolver: Solving for Uy, Initial residual = 2.04474e-09, Final residual = 2.04474e-09, No Iterations 0
DICPCG: Solving for p, Initial residual = 1.12694e-06, Final residual = 1.80179e-08, No Iterations 1
time step continuity errors : sum local = 6.5149e-13, global = -9.77893e-21, cumulative = -2.73277e-19
DICPCG: Solving for p, Initial residual = 1.57304e-08, Final residual = 1.57304e-08, No Iterations 0
time step continuity errors : sum local = 5.69014e-13, global = -1.06475e-20, cumulative = -2.83925e-19
DILUPBiCGStab: Solving for T, Initial residual = 2.59838e-05, Final residual = 5.04678e-10, No Iterations 1
ExecutionTime = 2.86 s ClockTime = 3 s

Time = 0.4999
Courant Number mean: 0.00444052 max: 0.0170421
smoothSolver: Solving for Ux, Initial residual = 1.56123e-09, Final residual = 1.56123e-09, No Iterations 0
smoothSolver: Solving for Uy, Initial residual = 2.05631e-09, Final residual = 2.05631e-09, No Iterations 0
DICPCG: Solving for p, Initial residual = 5.45261e-07, Final residual = 5.45261e-07, No Iterations 0
time step continuity errors : sum local = 1.96933e-11, global = 9.276e-21, cumulative = -2.74649e-19
DICPCG: Solving for p, Initial residual = 5.61721e-07, Final residual = 5.61721e-07, No Iterations 0
time step continuity errors : sum local = 2.02877e-11, global = 2.35267e-20, cumulative = -2.51122e-19
DILUPBiCGStab: Solving for T, Initial residual = 2.597e-05, Final residual = 5.04411e-10, No Iterations 1
ExecutionTime = 2.86 s ClockTime = 3 s

Time = 0.5
Courant Number mean: 0.00444052 max: 0.0170421
smoothSolver: Solving for Ux, Initial residual = 1.41142e-09, Final residual = 1.41142e-09, No Iterations 0
smoothSolver: Solving for Uy, Initial residual = 1.86002e-09, Final residual = 1.86002e-09, No Iterations 0
DICPCG: Solving for p, Initial residual = 1.02705e-06, Final residual = 1.63329e-08, No Iterations 1
time step continuity errors : sum local = 5.92229e-13, global = -6.97975e-21, cumulative = -2.58102e-19
DICPCG: Solving for p, Initial residual = 1.42803e-08, Final residual = 1.42803e-08, No Iterations 0
time step continuity errors : sum local = 5.18098e-13, global = 2.40213e-20, cumulative = -2.34081e-19
DILUPBiCGStab: Solving for T, Initial residual = 2.59562e-05, Final residual = 5.04142e-10, No Iterations 1
ExecutionTime = 2.86 s ClockTime = 3 s

End
openfoam@openfoam:~/Documents/myCavityCaseFile$
```


## 7. Upload your new solver, case files, and results in GITHUB



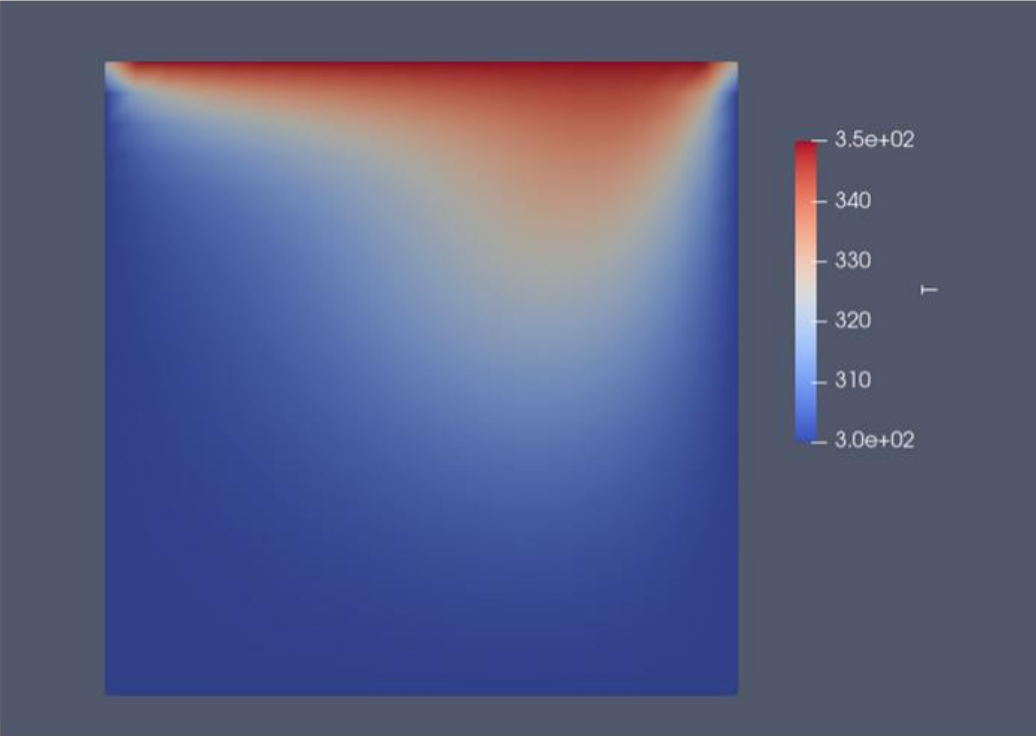
# Exercise – 9


## [Exercise-9] Creating a new OpenFOAM solver based on icoFOAM by adding a new variable (Temperature) #12


kummi0402 started this conversation in General


kummi0402 now Maintainer


1. Copy default icoFoam solver
2. Adding the temperature field in the icoFoam solver
3. Copy default cavity tutorial based from icoFoam
4. Add a new file for initial and boundary conditions
5. Add respective files in fvSchemes and fvSolution
6. Run your new case file
7. Upload your new solver, case files, and results in GITHUB




1




Category 

 General


Labels 

None yet


1 participant





Notifications


 Unsubscribe


You're receiving notifications because you authored the thread.


 Lock conversation

 Transfer this discussion

 Pin discussion

 Pin discussion to General

 Create issue from discussion

 Delete discussion