

Special Topics in CFD

DAY 8

Creating the new OF solver

Kumares

Contents

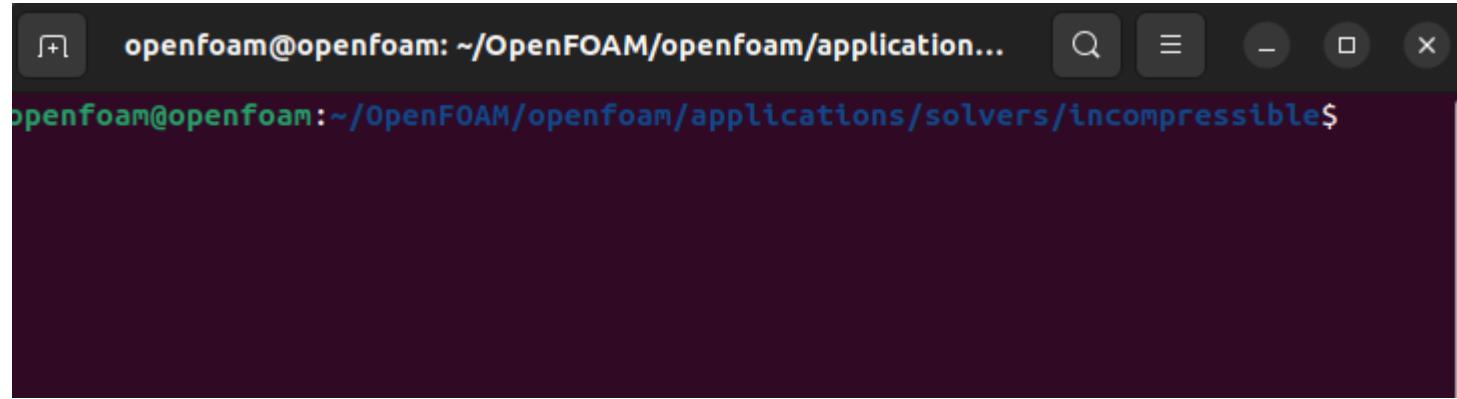
- Creating a new OpenFOAM solver based on icoFoam
- Exercise – 6

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 screenshot of a terminal window titled "openfoam@openfoam: ~/OpenFOAM/openfoam/application...". The window shows a command line with the text "openfoam@openfoam:~/OpenFOAM/openfoam/applications/solvers/incompressible\$". Below the title bar, there are standard window control buttons for minimize, maximize, and close. The main area of the terminal is dark, and the text is white.

```
openfoam@openfoam:~/OpenFOAM/openfoam/applications/solvers/incompressible$ cp icoFoam myIcoFoam
```

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 screenshot of a terminal window titled "openfoam@openfoam: ~/OpenFOAM/openfoam/tutorials/in...". The window shows a command line with the text "openfoam@openfoam:~/OpenFOAM/openfoam/tutorials/incompressible/icoFoam/cavity\$". The background of the terminal is dark purple.

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 | \\ / Field | OpenFOAM: The Open Source CFD Toolbox
4 | \\ / Operation | Version: v2306
5 | \\ / And | Website: www.openfoam.com
6 | \\ / Manipulation | |
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++ -----*/
| ====== |
| \\ / Field | OpenFOAM: The Open Source CFD Toolbox
| \\ / Operation | Version: v2306
| \\ / And | Website: www.openfoam.com
| \\ / Manipulation | |
/*
FoamFile
{
    version    2.0;
    format      ascii;
    class       volScalarField;
    object      T;
}
// *****
dimensions      [0 0 0 1 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-06;
        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

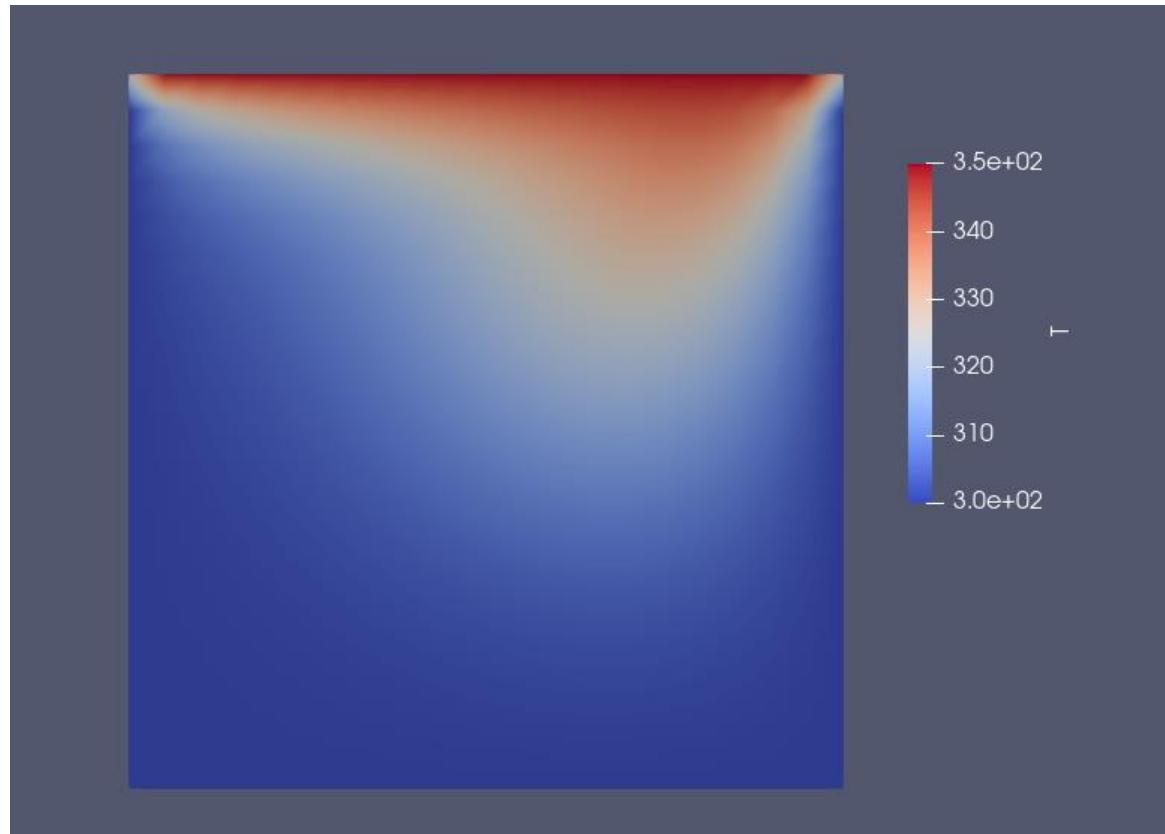
Time = 0.5

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 – 6

[Exercise-6] : Creating a new OpenFOAM solver based on icoFoam by adding a new variable (Temperature) #9

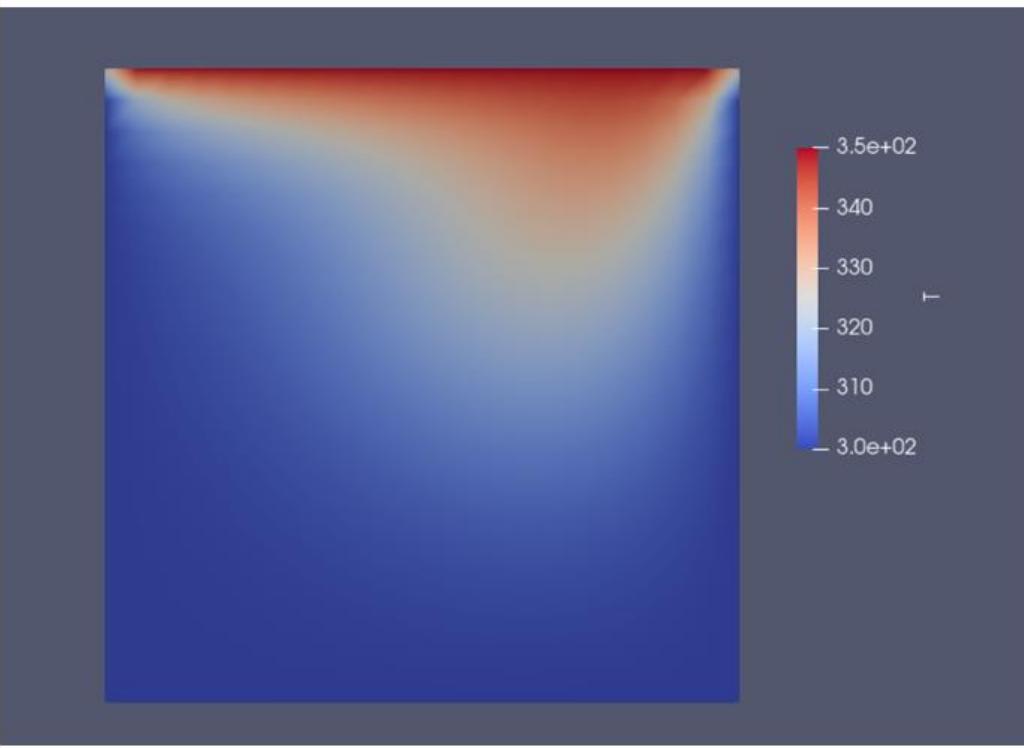
Edit

Kumares0402 started this conversation in General

Kumares0402 5 minutes ago Maintainer edited ...

Do the following:

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



Category General

Labels None yet

1 participant

Notifications

Unsubscribe You're receiving notifications because you're watching this repository.

Lock conversation Transfer this discussion Pin discussion Pin discussion to General Create issue from discussion Delete discussion