Heat transport in cavity with convection and diffusion

Albert S. Kim*

Created in Mon Apr 15 13:22:27 HST 2013 and
last modified in Tue Apr 16 15:32:41 HST 2013

1 Flow field, U

Follow the direction below:

1. Make "OpenFOAMcases" directory under your $HOME directory, if you did not.

```
mkdir OpenFOAMcases
cd OpenFOAMcases
```

2. Copy cavity case files and go to the directory:

```
cp -r $HOME/OpenFOAM/OpenFOAM-2.1.1/tutorials/incompressible/icoFoam/cavity
./cavity-org/
cd cavity-org
```

3. To run cavity case

```
blockMesh
icoFoam
```

4. You will see directories generated: 0.1 0.2 0.3 0.4 and 0.5. For Paraview visualization

```
touch cavity-org.foam
```

As long as the extension is ".foam" the file name does not matter.

5. Go to the parent directory and recursively zip cavity-org directory:

```
cd ../
zip -r cavity-org.zip cavity-org/
```

6. Donwload cavity-org.zip using FileZilla and open "cavity-org.foam" using Paraview.

*Associate Professor, Civil and Environmental Engineering, University of Hawaii at Manoa

2 Heat transfer under convection and diffusion: temperature T

We will use the pseudo-steady state flow file from the previous cavity run as a convection field.

Part 1

1. Go to the parent directory of `cavity-org`. If you did not use 'cd' command, you are above `cavity-org` directory.

2. Copy the tutorial `cavity` directory with a different name from `cavity-org` (such as `cavity-heat`) and `pitzDaily` directory to `pitzDaily-org`.

```
cp -r $HOME/OpenFOAM/OpenFOAM-2.1.1/tutorials/incompressible/icoFoam/cavity ./cavity-heat/
cp -r ~/OpenFOAM/OpenFOAM-2.1.1/tutorials/basic/scalarTransportFoam/pitzDaily/ ./pitzDaily-org
```

3. Go to `cavity-heat` directory and execute `blockMesh` only

```
cd cavity-heat
blockMesh
```

Therefore, the generated meshes using `blockMesh` will be identical in `cavity-org` and `cavity-heat` directories.

Part 2

4. In `cavity-heat` directory, if you list files under "system", you will find `controlDict`, `fvSchemes`, and `fvSolution`. Since these are for `icoFoam` simulation, replace them with those for `pitzDaily`:

```
cp ../pitzDaily-org/system/* ./system/
```

5. Copy transportProperties file from pitzDaily to here:

```
cp ../pitzDaily-org/constant/transportProperties ./constant/
```

In this 'transportProperties' file, DT is defined, which indicates heat diffusion coefficient $[m^2/s]$.

6. We will use the flow-field U from the final time of the previous run (under `cavity-org`) as a steady-state flow field for `cavity-heat` run. Copy U file (at time 0.5) from `cavity-org/0` to ./0/ directory.

```
cp ../cavity-org/0.5/U ./0/
```

7. Go to `0` directory and move `p` file to `T` file.

```
cd 0
mv p T
```

8. Open `T` file using an editor and go to line 13.

```
vim +13 T
```

where "+13" moves your cursor to line 13 when vim editor opens `T` file. You will see

```
object p;
```

and then change it to

```
object T;
```

Change the dimension from m^2/s to Kelvin:from

```
dimensions [0 2 -2 0 0 0 0];
```

to

```
dimensions [0 0 0 1 0 0 0];
```

9. In the same file (T), change boundary conditions for fixedWalls from

```
type zeroGradient;
```

to

```
type fixedValue;
value uniform 300;
```

This uniform value means 300 degree Kelvin. But, the unit is basically arbitrary since it is not used for non-dimensionalization.

10. Go to the parent directory and type/execute

```
cd ..
scalarTransportFoam
```

11. For Paraview visualization

```
touch cavity-heat.foam
```

12. Then, zip and download the directory, and visualize transient temperature using Paraview:

```
cd ../
zip -r cavity-heat.zip cavity-heat/
```

3 Analysis

We are using `scalarTransportFoam`. As its name indicated, it simulates a scalar quantity under convection and diffusion. pitzDaily case is for heat transport, which is mathematically identical to mass transport. Two parameters, `DT` and `boundary value (fixed)`, need to be appropriately modified.

$$\frac{\partial T}{\partial t} + \nabla \cdot (\mathbf{U}T) - \nabla^2 (D_T T) = 0$$

where T is temperature, U the is previously calculated flow field in the steady state, and D_T (denoted as DT in OpenFOAM) is the heat-diffusivity.

(Note: If `gmsh` is used for mesh generation, `gmshToFoam` should be used instead of `blockMesh`, and the rest of procedures will be almost identical.)

Link: http://openfoamwiki.net/index.php/ScalarTransportFoam