Summer Fellowship Report

On

Cooling of sphere - Natural and Forced Convection

Submitted by

Vignesh.S.P

Under the guidance of

Prof.Kannan M. Moudgalya
Chemical Engineering Department
IIT Bombay

Vignesh.S.P

July 9, 2019

Acknowledgment

The success of this project required a lot of guidance and help from many people and I am extremely privileged to have got this along the completion of this project. I respect and thank Prof.Kannan M. Moudgalya, Chemical Engineering Department and Prof.Shivasubramanian Gopalakrishnan, Mechanical Engineering Department, for providing me an opportunity to do the project work in FOSSEE, IIT Bombay and giving me all the support and guidance which made me complete the project duly.

I'm extremely thankful to my mentors Mr. Sathish Kanniappan and Ms. Deepa Vedartham for their support and mentorship through out the course of completion of the project. I'm thankful to my mentors for providing equipment support which was very helpful.

I'm thankful and fortunate enough to get constant encouragement, support and guidance from all the personnel's from FOSSEE, IIT Bombay.

Abstract

This case study aims to numerically simulate the behavior of cooling of a heated sphere in both natural and forced convective environment. The flow behavior in the fluid domain and the temperature in the solid sphere is analysed. Conjugate Heat Transfer between conduction in solid sphere and convection in fluid domain is used to simulated the case study. The thermal source for the simulation is provided by the source terms in the energy equation using fvOptions.

Contents

List of Figures

List of Tables

Chapter 1

Introduction

The coupled heat transfer between conduction in solid and convection in fluid is termed as conjugate heat transfer. This methodology is employed here to simulate cooling of sphere under natural convection and forced convection. The comparison between the two modes of convection is done and the results are studied. Heat to the solid zone is given using fvOptions utility which enables user to specify heat

1.1 OpenFOAM

OpenFOAM (for "Open-source Field Operation And Manipulation") is a C++ toolbox for the development of customized numerical solvers, and pre/post-processing utilities for the solution of continuum mechanics problems, most prominently including computational fluid dynamics. OpenFOAM is freely available and open source, licensed under the GNU General Public Licence. The licence is designed to offer freedom, in particular it encourages users of the software to make modifications or developments. Overset Grid in OpenFOAM is available in versions released only after 2017. It is being actively developed and several solvers and validation has been done.

1.2 Conjugate Heat Transfer

This model, based on a strictly mathematically stated problem, describes the heat transfer between a body and a fluid flowing over or inside it as a result of the interaction of two objects. The physical processes and solutions of the governing equations are considered separately for each object in two subdomains. Matching conditions for these solutions at the interface provide the distributions of temperature and heat flux along the bodyflow interface, eliminating the need for a heat transfer coefficient.
The mass conservation equation is as follows

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho U) = 0$$

where ρ is density of the fluid, U is velocity field.

The momentum conservation equation is as follows

$$\frac{\partial \rho U}{\partial t} + \nabla.(\rho U U) = -\nabla p + \nabla.\rho\nu\tau + F$$

where U is the velocity field, ρ is the density, p is the pressure, ν is the kinematic viscosity, τ is the stress tensor and F is the combined sources term.
The energy equation for the fluid is as follows

$$\frac{\partial \rho E}{\partial t} + \nabla.(\rho U E) + \nabla.(U p) = -\nabla.q + \nabla.(\tau.U) + \rho r + \rho g.U$$

where E is the total energy, r is specific heat source, g is acceleration due to gravity.
Then finally the equation governing the heat conduction in solid regions is as follows

$$\frac{\partial \rho h}{\partial t} = \nabla.(\alpha\ grad(h))$$

where h is specific enthalpy, ρ is the density of solid, $\alpha = k/c_p$ is thermal diffusivity and k is thermal conductivity and c_p is specific heat capacity.

The coupling of temperature between solid and fluid regions are done using baffle patches which are shared by both regions. Boundary condition for sharing the temperature information is also given at these baffle patches.

Chapter 2

Case Setup

2.1 Geometry and Mesh

The geometry of the case study is a rectangular duct in which the spherical solid is placed near the inlet. The sphere is place at (0,0,0) co-ordinate with an diameter of 0.05m in a rectangular duct of 0.5m height and 1m length. Thickness is not considered since the case is simulated in 2D. The sphere encompasses the solid region with structured hexagonal mesh with 4500 cells. The fluid zone is the rectangular duct without the solid zone which is off 4500 cells. This combined mesh is separated into regions using 'splitMeshRegions' utility which will create the necessary baffles (shared patches) between solid and fluid regions. The mesh can be seen in figure 2.1

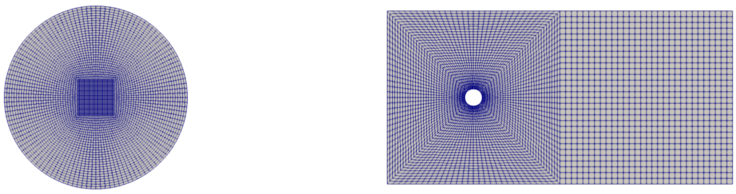

Figure 2.1: Separated mesh regions

2.2 Boundary conditions

The boundary conditions employed for the computational domain is given in table 2.1. Boundary condition for natural and forced convection is similar only there is no velocity at the inlet in natural convection. The Solid region has one patch named solidZone_to_fluidZone which will be given same temperature boundary condition as in fluidZone_to_solidZone. Boundary condition for the baffles solidZone_to_fluidZone and fluidZone_to_solidZone are same for other fields.

Boundary Name	U	p_rgh	T
inlet	type fixedValue; value uniform (0.001 0 0);	type zeroGradient; value uniform 0;	type fixedValue; value uniform 300;
outlet	type inletOutlet; inletValue uniform (0 0 0);	type fixedValue; value uniform 0;	type inletOutlet; value uniform 300;
'bottom' and 'top'	type fixedValue;	type fixedFluxPressure; value uniform 0;	type zeroGradient; value uniform 300;
'frontFluid' and 'backFluid'	type empty;	type empty;	type empty;
fluidZone_ -to_solidZone	type fixedValue; value uniform (0 0 0);	type fixedFluxPressure; value uniform 0;	type compressible::turbulent -TemperatureCoupled -BaffleMixed; Tnbr T; kappaMethod fluidThermo; value uniform 300;

Table 2.1: Boundary Conditions for Fluid Region - Forced Convection

2.3 Adding Source using fvOptions

The fvOptions functionality in OpenFOAM is flexible framework to add various source terms to the governing equations without the need to rewrite the original source code. The fvOptions framework has been introduced to allow users to select any physics that can be represented as sources or constraints on the governing equations, e.g. porous media, thermal source and body forces. This new fvOptions framework enhances and supercedes the previous run-time selectable sources in version 2.1. The thermal source use in this case is fixedTemperatureConstraint to the solidZone for 1000s starting from 500s of the simulation time step.

```
fixedTemperature
{
    type             fixedTemperatureConstraint;
    active           yes;
    timeStart        500;
    duration         1000;
    selectionMode    cellZone;
    cellZone         heatSource;
    mode             uniform; // uniform or lookup
    temperature      constant 1273;
    // fixed temperature with time [K]
}
```

2.4 Solver and Simulation control

chtMultiRegionFoam solver is used for this simulation. The official definition for this solver is as follows:

Transient solver for buoyant, turbulent fluid flow and solid heat conduction with conjugate heat transfer between solid and fluid regions

The transport and thermal properties for case is setup in thermophysicalProperties file in region directory within constant directory. The fluid properties are as follows

```
mixture
{
    specie
    {
        molWeight          18;
    }
    equationOfState
    {
        rho                1000;
    }
    thermodynamics
    {
        Cp                 4181;
        Hf                 0;
    }
    transport
    {
        mu                 959e-6;
        Pr                 6.62;
    }
}
```

The solid Zone properties are as follows

```
mixture
{
    specie
    {
        molWeight     50;
    }

    transport
    {
        kappa     80;
    }
```

```
    thermodynamics
    {
        Hf          0;
        Cp          450;
    }

    equationOfState
    {
        rho         8000;
    }
}
```

The important control values as given in controlDict is as follows

```
application         chtMultiRegionFoam;
startFrom           latestTime;
startTime           0.001;
stopAt              endTime;
endTime             4000;
deltaT              0.001;
writeControl        adjustableRunTime;
writeInterval       20;
purgeWrite          0;
writeFormat         ascii;
writePrecision      8;
writeCompression    off;
timeFormat          general;
timePrecision       6;
runTimeModifiable   yes;
maxCo               0.6;
maxDi               10.0;// Maximum diffusion number
adjustTimeStep      yes;
```

Chapter 3

Result and analysis

3.1 Temperature Plot

The temperature plot against the length of the duct at different time steps for both natural and forced convection will show cooling behavior. From the plots we can surely see the effect of large temperature reduction in forced convection in comparison to natural convection as in figure 3.1 and 3.2

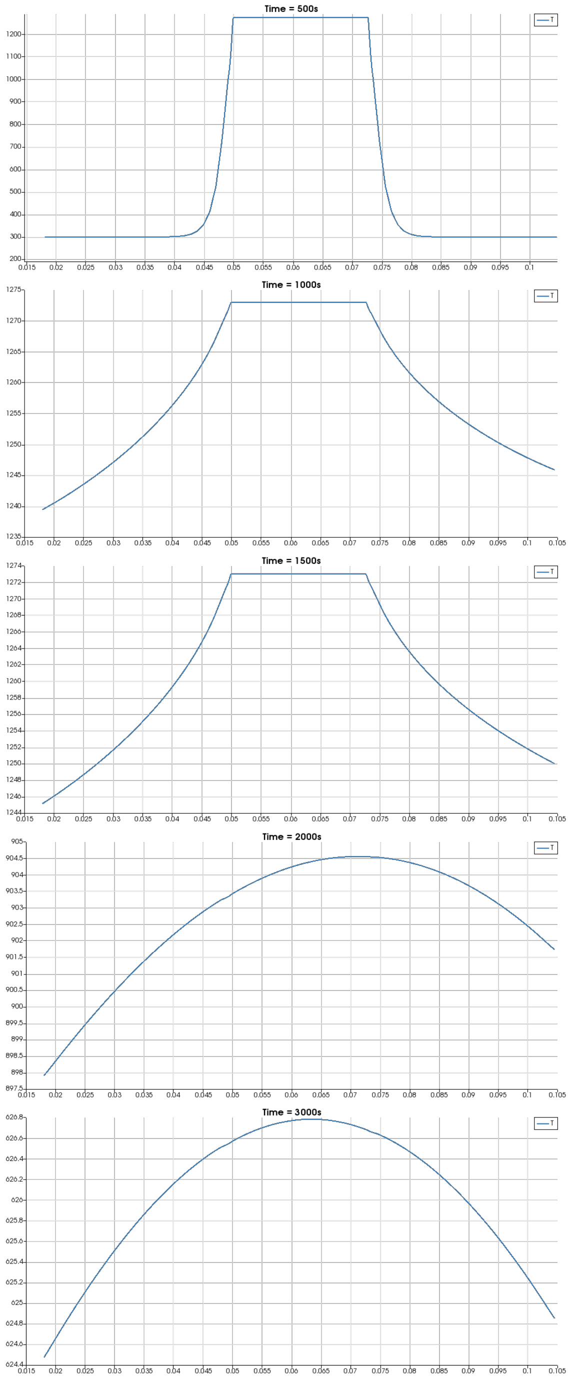

Figure 3.1: Temperature plot of solid zone at different time steps - Natural Convection

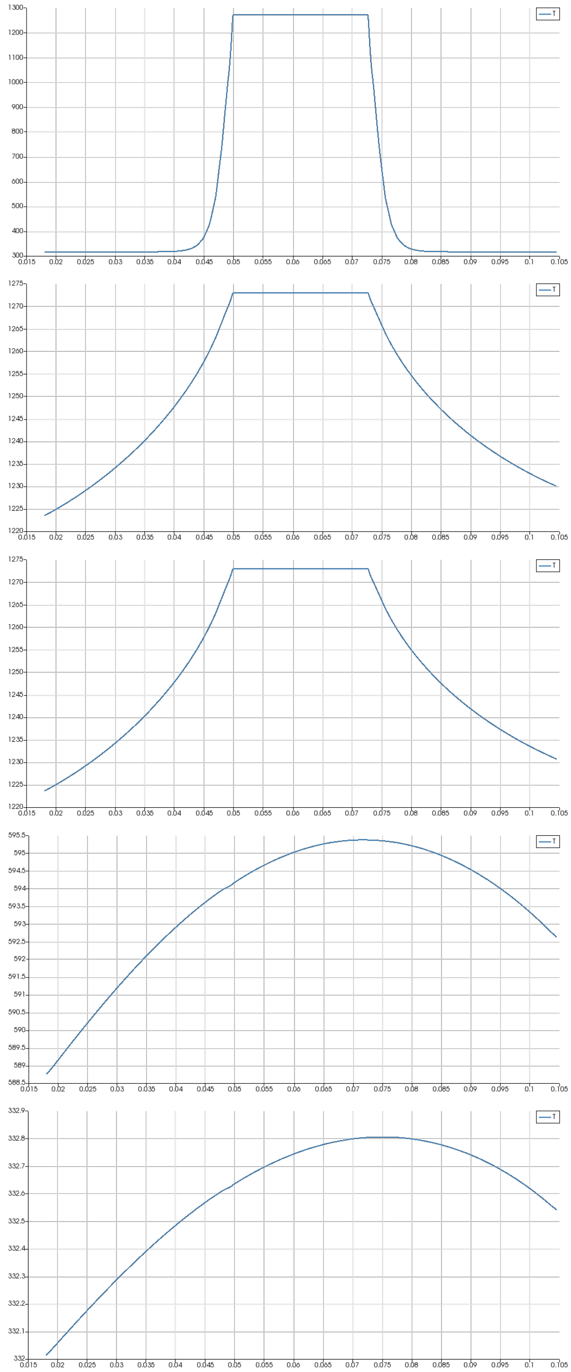

Figure 3.2: Temperature plot of solid zone at different time steps - Forced Convection

Chapter 4

Conclusion

This case study explores the conjugate heat transfer between solids and fluids. By using fvOptions utility, thermal source for simulation is setup without having to recompile the source file of the solver. Comparison between natural and forced convection is studied.

Reference

- chtMultiRegionFoam solver - OpenFOAMwiki.net
- Tutorial on conjugate heat transfer in OpenFOAM
- Implementing chtMultiRegionFoam Solver for Electric Welding in OpenFoam 1.6.x