# OpenFOAM-1.7.0 Capabilities for Thermal Modeling Purpose

## 1. INTRODUCTION:

OpenFOAM stands for Open Source Field Operation and Manipulation and it is a programming environment rather than a CFD code. OpenFOAM can also be defined as a C++ library, primarily to create executables, known as applications. The applications fall into two categories: solvers and utilities. Solvers are each designed to solve a specific problem in continuum mechanics. Utilities are designed to perform tasks that involves with data manipulation. The OpenFOAM distribution contains numerous solvers and utilities covering a wide range of problems. Each solvers has specific capabilities but all are not integrated. It is not readily usable like a CFD code/software. Sometimes, users need to modify the source code and create their won solver and utilities for their particular applications.

## 2. STANDARD SOLVERS:

Solvers are executables, each designed to solve a specific problem in continuum mechanics. There are several solvers to solve different kind of problems. One solver may have a specific capability and the rest do not have that capability. All capabilities are not integrated into all solvers, so all solvers do not have all capabilities. Most of the time, users need to modify the source code of a particular solver and recompile that particular solver to have some specific capabilities that are not readily available with this solver but available with another solver. Table 2.1 represents the various physical phenomena supported by the different solvers available with OpenFOAM related to thermal analysis / heat transfer which are usable for aerothermal analsys. Table 2.2 and 2.3 show the supported boundary conditions by the standard solvers related to thermal modeling in solid and fluid region respectively.

**Table 2.1: Supported physical phenomena by various OpenFOAM solvers**

| Physical Phenomena | Solver Name | | | |
|---|---|---|---|---|
| | buoyantBoussinesq Foam | buoyantFoam | buoyantRadiation Foam | chtMultiRegionFoam |
| Turbulence modeling | Yes[1] | Yes[2] | Yes[2] | Yes[2] |
| Radiation modeling | No | No | Yes[3] | No |
| Conjugate heat transfer modeling | No | No | No | Yes |
| Natural convection modeling | Yes[4] | Yes[5] | Yes[5] | Yes[5] |
| Heat generation/source in the solids | Not applicable[6] | Not appicable[6] | Not applicable[6] | No |

1. All of the one and two equation turbulence models for incompressible flow provided by OpenFOAM will be supported.
2. All of the one and two equation turbulence models for compressible flow provided by OpenFOAM will be supported.
3. P1 and DOM radiation models.
4. Incompressible flow modelling with Boussinesq approximation.

5. Compressible flow modelling.
6. The solver does not have capability to model multiregions.

**Table 2.2: Supported boundary conditions in solid by various OpenFOAM solvers**

| Physical Phenomena | Solver Name | | | |
|---|---|---|---|---|
| | buoyantBoussinesq Foam | buoyantFoam | buoyantRadiation Foam | chtMultiRegionFoam |
| Dirichlet | N/A[1] | N/A[1] | N/A[1] | Yes |
| Numann | N/A[1] | N/A[1] | N/A[1] | Yes |
| Convection | N/A[1] | N/A[1] | N/A[1] | Yes |
| Symmetry | N/A[1] | N/A[1] | N/A[1] | Yes |
| Adiabatic | N/A[1] | N/A[1] | N/A[1] | Yes |
| Periodic | N/A[1] | N/A[1] | N/A[1] | Yes |
| Radiation | N/A[1] | N/A[1] | Yes | N/A[2] |
| Thermal contact | N/A[1] | N/A[1] | N/A[1] | No |
| Time varying parameter | Yes[3] | Yes[3] | Yes[3] | Yes[3] |

1. This solver does not model multi-region, only applicable to fluid region
2. This solver does not include radiation
3. TimeVaryingUniformFixedValue BC is available in current version of OpenFOAM

**Table 2.3: Supported boundary conditions in fluid by various OpenFOAM solvers**

| Physical Phenomena | Solver Name | | | |
|---|---|---|---|---|
| | buoyantBoussinesq Foam | buoyantFoam | buoyantRadiation Foam | chtMultiRegionFoam |
| Dirichlet | Yes | Yes | Yes | Yes |
| Numann | Yes | Yes | Yes | Yes |
| Convection | Yes | Yes | Yes | Yes |
| Symmetry | Yes | Yes | Yes | Yes |
| Adiabatic | Yes | Yes | Yes | Yes |
| Periodic | Yes | Yes | Yes | Yes |
| Wall | Yes | Yes | Yes | Yes |
| Inlet (imposed velocity & pressure) | Yes | Yes | Yes | Yes |
| Outlet (imposed velocity & pressure) | Yes | Yes | Yes | Yes |

## 3. REQUIRED SOLVER CAPABILITY FOR OUR TEST CASES

### 3.1 Simple Thermal Use Case:

#### 3.1.1 Physical Phenomena:

(i) Steady-state & transient both capability
(ii) Multi-region or conjugate heat transfer
(iii) Incompressible flow with Boussinesq approximation
(iv) Turbulence models
(v) Heat source or heat generation in the solids and heat generation can vary with time
(vi) Modified heat transfer at the contact or thermal contact condition

#### 3.1.2 Boundary Conditions:

***Boundary conditions in solid:***

*(a) Temperature boundary conditions:*

(i) Dirichlet or Imposed T
(ii) Neumann or Imposed heat flux
(iii) Convection or Externally cooled
(iv) Symmetry
(v) Adiabatic or Temperature gradient zero
(vi) Periodic
(vii) Thermal contact conditions
(viii) Time varying boundary conditions
(ix) Space dependence boundary conditions

***Boundary conditions in fluid:***

*(a) Temperature boundary conditions:*

All the above mentioned boundary conditions for energy equation in solid is applicable in fluid. Apart from the energy equation boundary conditions the follwoing boundary conditions needs to be handled in fluid.

*(b) Velocity/Pressure boundary conditions:*

(i) Wall conditions
(ii) Inlet: imposed velocity and imposed pressure conditions
(iii) Outlet: imposed velocity and imposed pressure conditions
(iv) Boundary condition can vary depending on a parameter (eg. temperature, pressure etc) value at any cell within the computational domain in the previous time step.
(v) Time varying boundary conditions
(vi) Space dependence boundary conditions

### 3.2 Vertical Tail Plane (VTP) Case:

#### 3.2.1 Physical Phenomena:

(i) Steady-state
(ii) Multi-region or conjugate heat transfer
(iii) Incompressible flow with Boussinesq approximation
(iv) Turbulence models
(v) Radiation

### 3.2.2 Boundary Conditions:

***Boundary conditions in solid:***

*(a) Temperature boundary conditions:*

(i) Dirichlet or Imposed T
(ii) Neumann or Imposed heat flux
(iii) Convection or Externally cooled
(iv) Symmetry
(v) Adiabatic or Temperature gradient zero
(vi) Periodic
(vii) Radiation

***Boundary conditions in fluid:***

*(a) Temperature boundary conditions:*

All the above mentioned boundary conditions for energy equation in solid is applicable in fluid. Apart from the energy equation boundary conditions the follwoing boundary conditions needs to be handled in fluid.

*(b) Velocity/Pressure boundary conditions:*

(i) Wall conditions
(ii) Inlet: imposed velocity and imposed pressure conditions
(iii) Outlet: imposed velocity and imposed pressure conditions

## 4. REFERENCE

OpenFOAM 1.7 User Guide, http://www.openfoam.com/docs/user