Tutorial: BwUniCluster 3.0/HoreKa
Hot Room (not parallel)

In this tutorial we will learn about the simulation of convective heat transfer in OpenFOAM on a single core by submitting a job. We will do it using the example hotRoom from the series of built-in OpenFOAM tutorials.

1. **Tutorial Case (Serial Case)**

Before examining the files we should first copy the tutorial case from the FOAM_TUTORIALS directory and paste it into our workspace (denoted by a point):

```
$>   cd <workdir>
$>   module load cae/openfoam/v2112
$>   source $FOAM_INIT
$>   cp -R $FOAM_TUTORIALS/heatTransfer/buoyantPimpleFoam/hotRoom .
```

2. **Creating the Mesh**

Firstly, let us examine the mesh in the hotRoom tutorial whose objective is to visualize the circulation of air in a room after steady state has been achieved. The mesh in this case represents a 10m x 10m x 5m room equipped with a heat source in the exact centre of its floor.

In order to create the mesh using data from the *blockMeshDict* file, we will use the following command:

```
$>   cd hotRoom
$>   blockMesh
```

Initial conditions should be located in the 0 directory. We create the 0 directory with the command:

```
$>   cp -R 0.orig 0
```

Apart from describing the basic structure of the mesh, the blockMeshDict file also defines boundaries, also known as patches, for which special conditions will be set. In our case these areas are: floor, ceiling and fixedWalls, see file blockMeshDict. To view the file you can use the command:

```
$>   more blockMeshDict
```

The mesh is shown in Fig. 1. Figures 2, 3 and 4 show the different boundary patches.

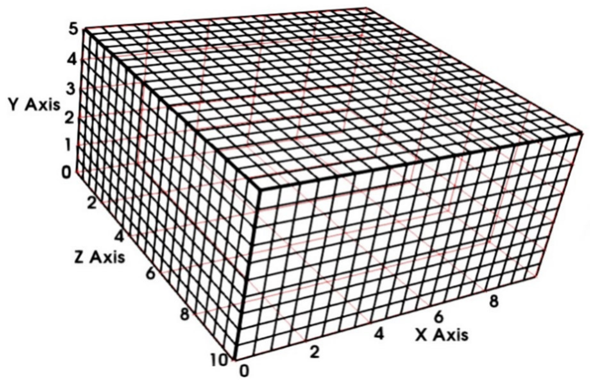


Fig.1 The hotRoom mesh created using blockMesh

Fig.2 Boundary patch "fixedWalls"

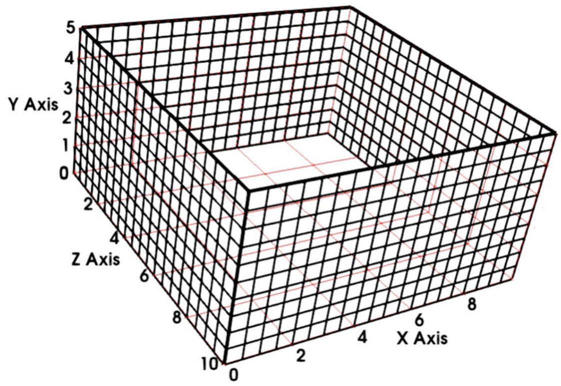


Fig.3 Boundary patch "ceiling"

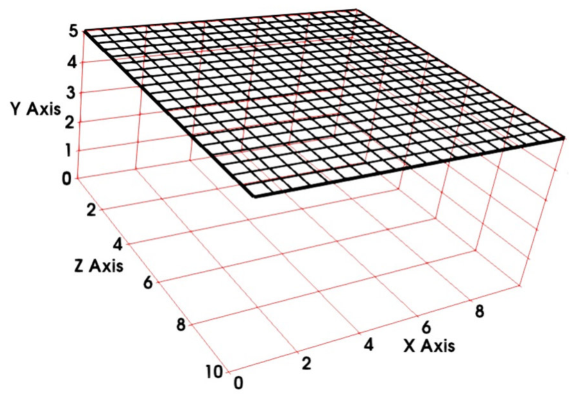
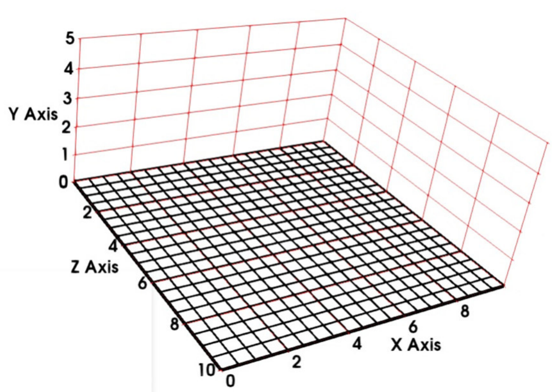


Fig.4 Boundary patch "floor"



After using the blockMesh command and creating the mesh, it is now time to set the initial fields using the setFields command:

```
$>   setFields
```

In this case we will be creating a heat source of 600 °C in the centre of the floor region:

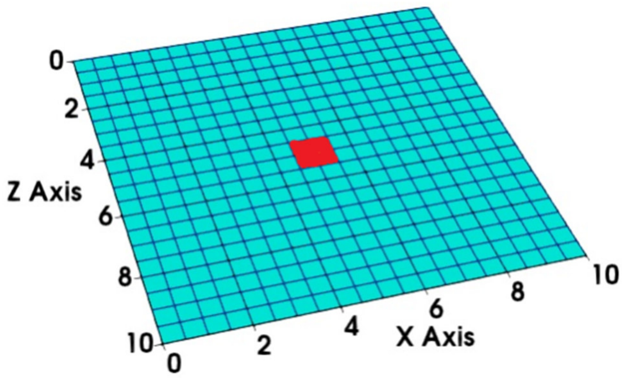


Fig.5 The heat source in the centre of the floor

In case we want to conduct a parallel calculation we should also use the decomposePar command after setting the initial fields. Conducting parallel jobs will be explained later.

Now it is time to submit the job for calculation. Assuming we are conducting the simulation on a single core the batch job file (name e.g. my_dev_single_job.sh) should look something like this:

//

#!/bin/bash	header of the file
#SBATCH --partition dev_single	job will be submitted to queue dev_single
#SBATCH --nodes=1	one node will be used
#SBATCH --ntasks=1	ntasks means „the number of cores" to use
#SBATCH --time=00:30:00	the job will run for maximum 30 minutes
#SBATCH --mem=8000mb	the job may use max. 8 Gb
#SBATCH --job-name=hotRoom	this name is given by the user
module purge module load cae/openfoam/v2106-impi source $FOAM_INIT	loading OpenFOAM on the execute core
buoyantPimpleFoam	the name of the OpenFOAM-solver

//

We can then submit the job using the name of the job-file with the following command:

```
$>   sbatch my_dev_single_job.sh
```

3. **Results**

After the simulation has been completed we can download the results onto our personal computer and visualize them in Paraview. This is the recommended way for visualizing results with small- or medium sized grids which is the most common case.

Now we can see what the steady state looks like (after 2000 seconds). The air in the room gets heated up right above the heat source which causes it to move upwards towards the ceiling. There it cools down which makes it move towards the floor only to be heated up again which drives the circulation:

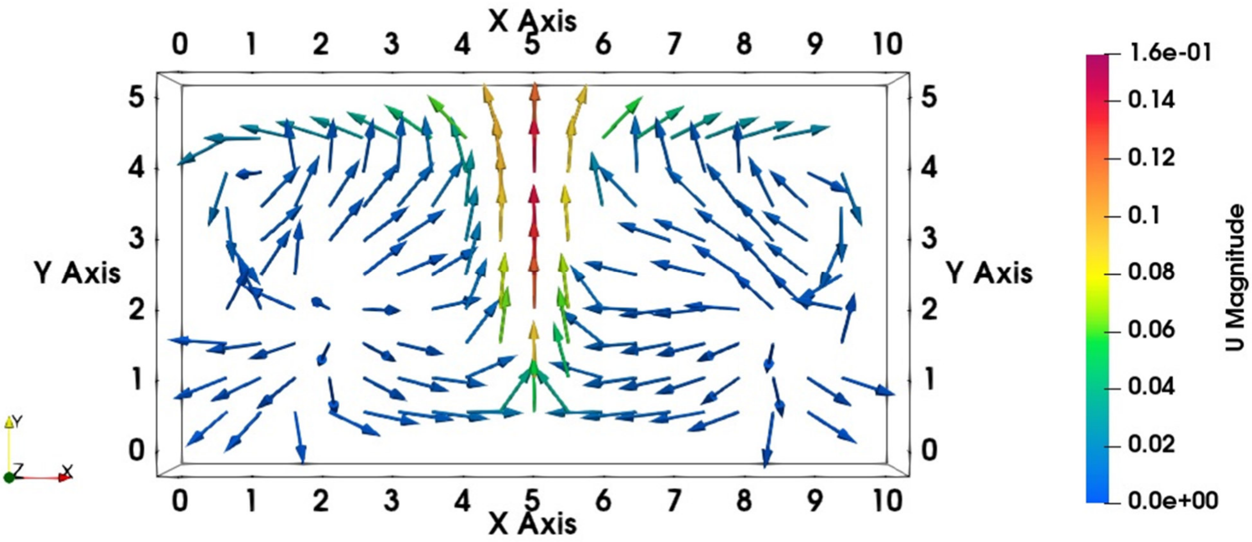


Fig.6 Velocity vectors on a vertical plane going through the centre of the room

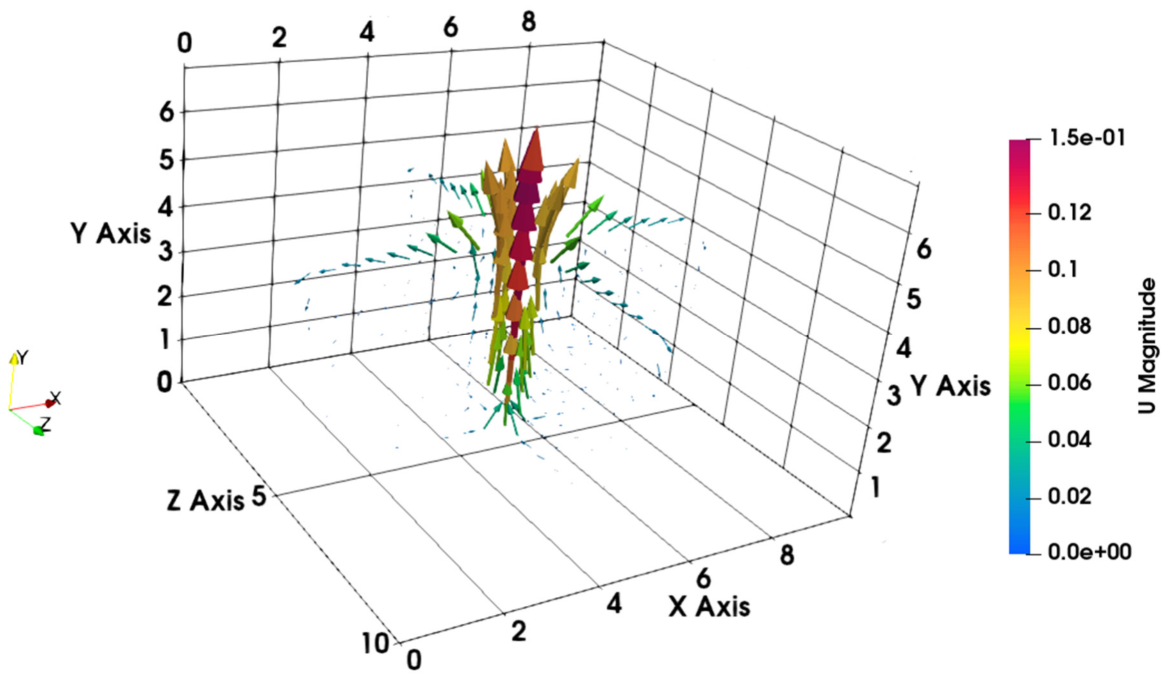


Fig.7 Velocity vectors in two perpendicular planes in the centre of the room

The ceiling of the room is the only surface through which heat can be transferred, which is depicted in the following diagramm:

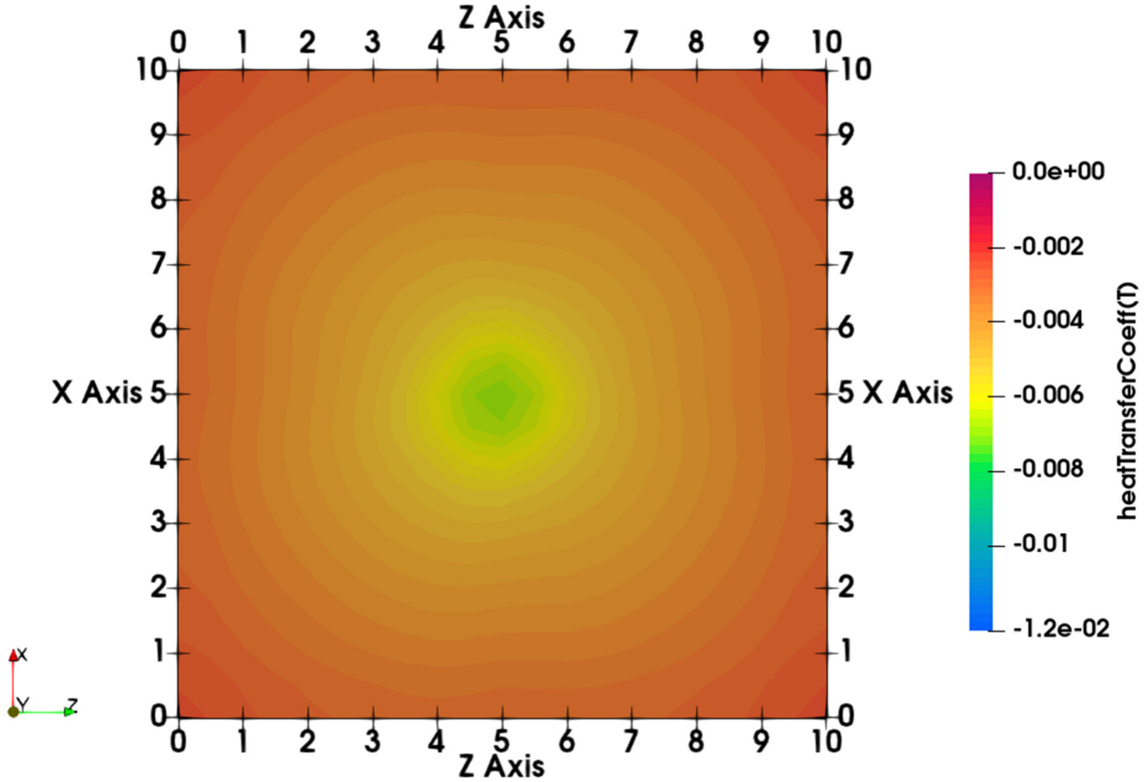


Fig.8 Distribution of the heat transfer coefficient along the ceiling

Another parameter we would like to look at is the effective thermal diffusivity alphat which indicates the speed of heat conduction. It is higher in areas where the heat trasfer occures at a faster rate which is reflected in the following diagramm:

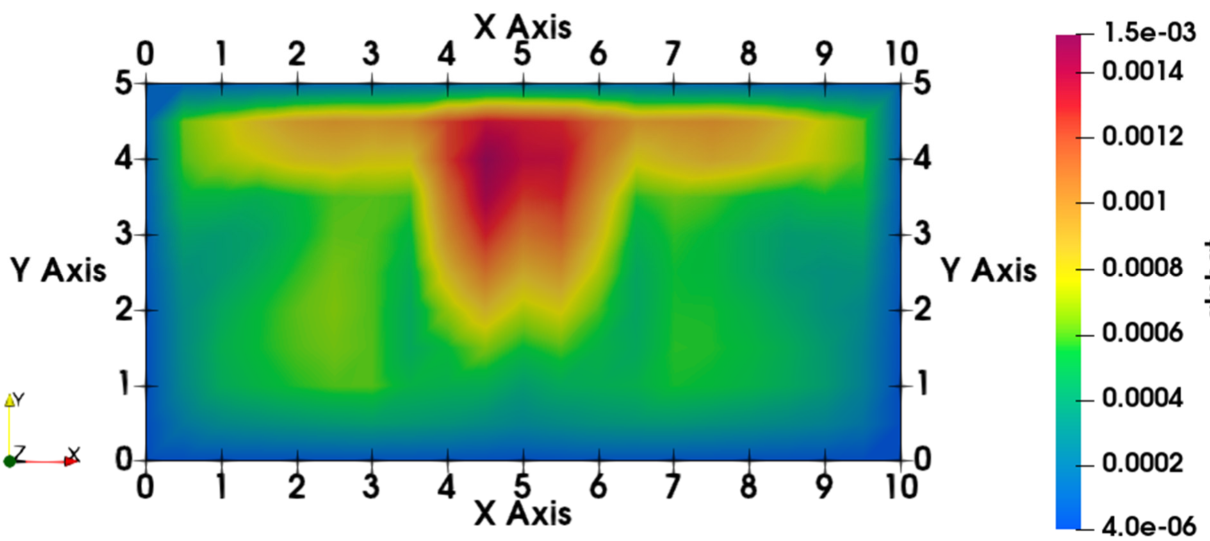


Fig.9 Distribution of the thermal diffusivity on a Z-normal plane in the centre of the room