GROBDESIGN

Tutorial 3

CEILING COOLING (RADIANT COOL) IN OFFICE ROOM USING FDS SOFTWARE

GROBDESIGN PVT LTD, JAIPUR 2018

Tutorial 3 ceiling cooling (radiant cool) in office room using FDS software

In this tutorial, a room is cooled down using chilled beams installed at ceiling surface of roof. Total 3 beams are installed. The purpose of this tutorial is to show the effect of chilled beam on room cooling. The solver is FDS software. Beams have 7C surface temperature. No inflow or outflow boundary condition is set in this tutorial. It is introduction purpose tutorial.

The steps are followed in this tutorial are listed below::

Step I: create header syntax file to start program in FDS software.

```
&HEAD CHID='trial1'/
```

Note: trial1 is user defined name of FDS function/ file.

Step II: create syntax for simulation flow time.

```
&TIME T_END=200.0/
```

Note: 200 sec is simulation flow time, which is solved in FDS software.

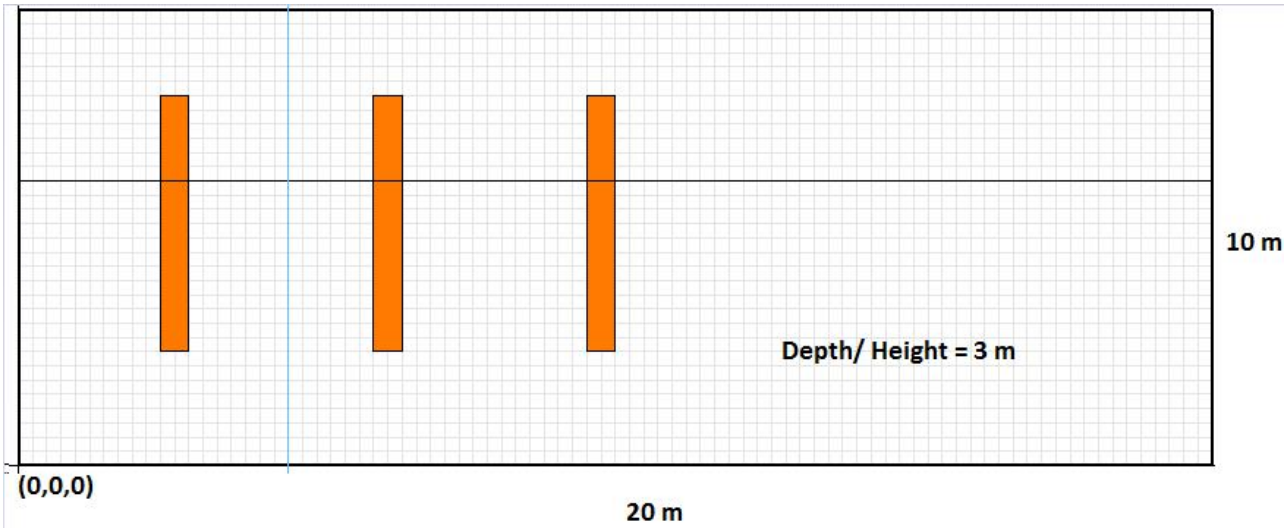
Step III: create syntax for initial temperature of domain.

```
&MISC TMPA=35.0/
```

Note: 35 C is initial room temperature, which is provided in this tutorial. Following three syntax is must for every FDS function.

Step IV: Create syntax for geometry making of fluid domain

```
&MESH ID='MESH', IJK=84,32,11, XB=0.0,20.0,0.0,10.0,0.0,3.0/
```



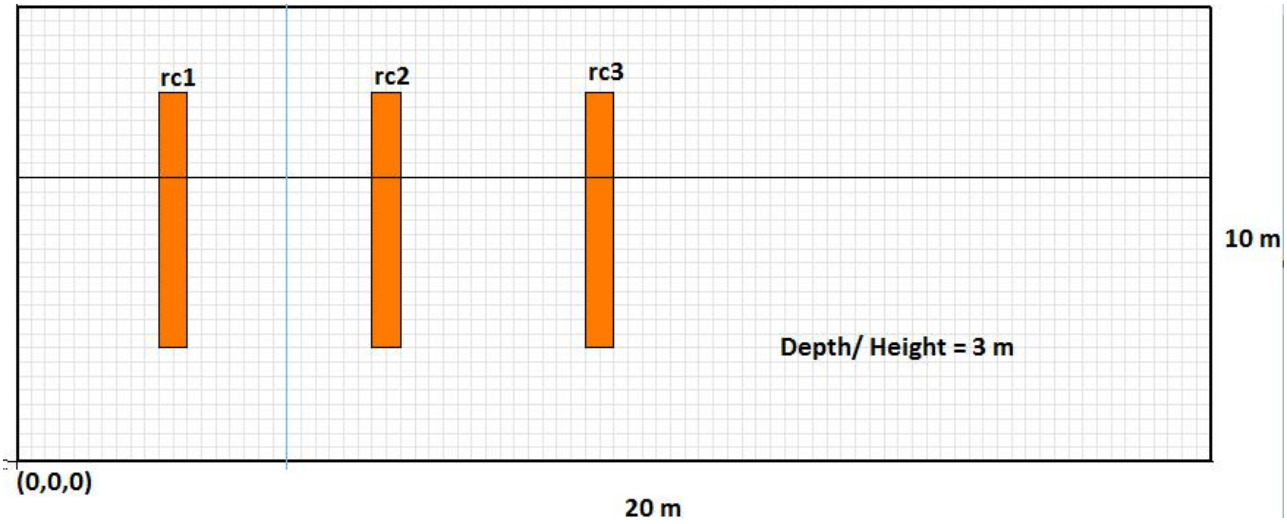
Note: **MESH ID** represent user name of mesh created in FDS. **IJK** represent cell created in domain in X-dir, Y-dir and Z-dir respectively. Like in current problem, 84 cells are created in X-direction, 32 cells are created in Y-direction and 11 cells are created in Z-direction. **XB** represent start and END point of rectangular domain in following syntax

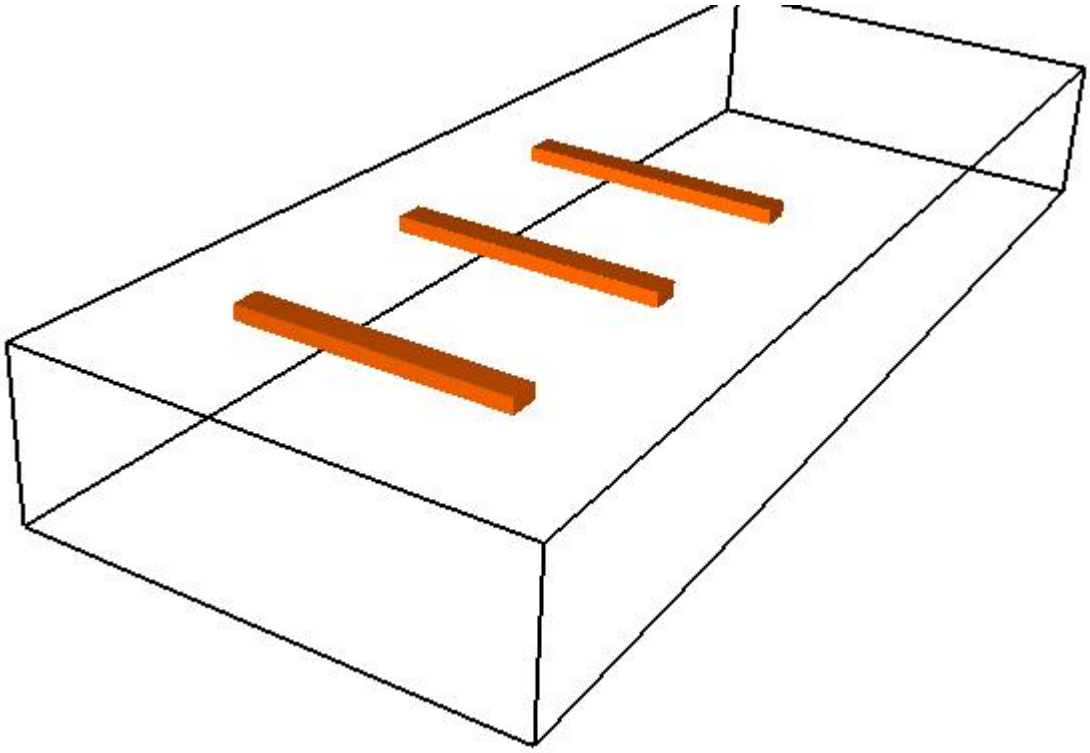
XB=X1,X2,Y1,Y2,Z1,Z2/

Here 1 represent starting point and 2 represent end point. XB is always start from origin.

Step V: create chilled beams using OBST command in syntax

```
&OBST ID='rc1', XB=2.5,3.0,2.0,6.5,2.5,2.75, SURF_ID='cool1'/
&OBST ID='rc2', XB=6.25,6.75,2.0,6.5,2.5,2.75, SURF_ID='cool1'/
&OBST ID='rc3', XB=10.0,10.5,2.0,6.5,2.5,2.75, SURF_ID='cool1'/
```





Note: **OBST** is obstacle created in FDS and it can be 3D or 2D depend on fluid problem. Rc1, rc2 and rc3 represent user name of **OBST** created in current problem. **XB** represent the geometrical presentation in syntax of FDS software. These OBSts are created on top surface of fluid domain (see figure). **SURF ID** represent fluid boundary conditions in FDS software. In this case user defined surf-id named "cool1" is created which is discussed in next step.

Step VI: create surf ID syntax

```
&SURF ID='cool1',
      RGB=255,102,0,
      HEAT_TRANSFER_COEFFICIENT=10.0,
      TMP_FRONT=7.0/
```

Note: As discussed in previous step, cool1 is CFD boundary condition applied on rc1, rc2 and rc3. **RGB** represent color provided to selected **OBST**. Cool1 is convective heat transfer boundary condition which has free stream temperature of 7 C (**TEMP_FRONT**) with 10 SI unit of **heat transfer coefficient**.

Step VII: create surface contour for field variable (velocity and temperature)

```
&SLCF QUANTITY='TEMPERATURE', PBY=5.0/
&SLCF QUANTITY='VELOCITY', PBY=5.0/
```

Note: **SLCF** syntax represent surface contour. **QUANTITY** represent type of field variable and **PBY** represent user defined plane for virtual surface creation.
PBY=5.0 represent 5 m distance from Y-direction. The measurement is stared from origin.

Step VIII: end the function by using syntax

```
&TAIL /
```

Final FDS code for current problem

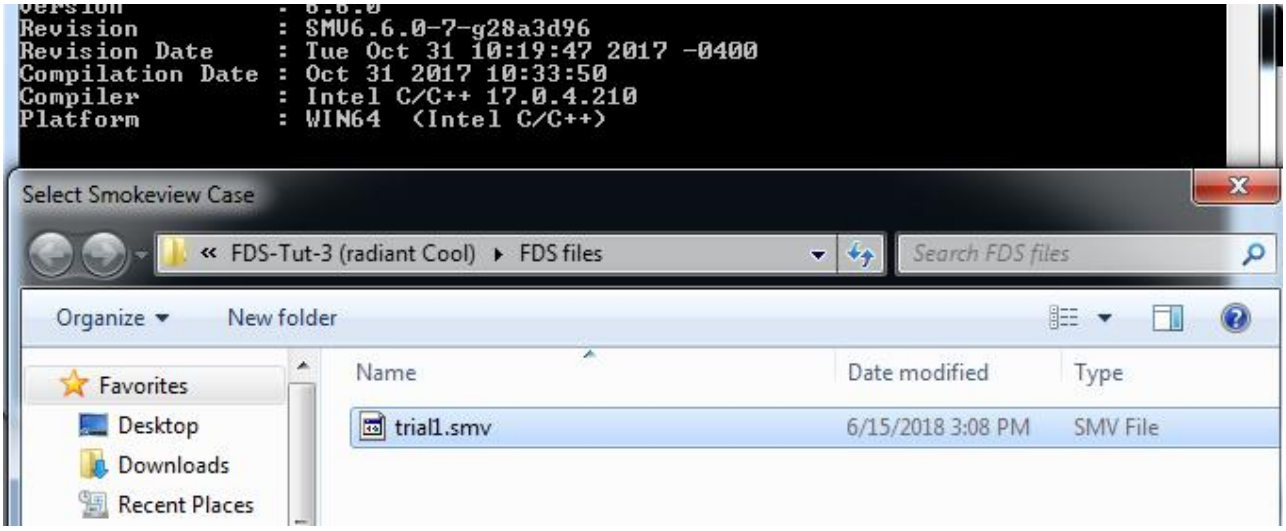
```
&HEAD CHID='trial1'/
&TIME T_END=200.0/

&MISC TMPA=35.0/

&MESH ID='MESH', IJK=84,32,11, XB=0.0,20.0,0.0,10.0,0.0,3.0/

&SURF ID='cool1',
RGB=255,102,0,
HEAT_TRANSFER_COEFFICIENT=10.0,
TMP_FRONT=7.0/

&OBST ID='rc1', XB=2.5,3.0,2.0,6.5,2.5,2.75, SURF_ID='cool1'/
&OBST ID='rc2', XB=6.25,6.75,2.0,6.5,2.5,2.75, SURF_ID='cool1'/
&OBST ID='rc3', XB=10.0,10.5,2.0,6.5,2.5,2.75, SURF_ID='cool1'/

&SLCF QUANTITY='TEMPERATURE', PBY=5.0/
&SLCF QUANTITY='VELOCITY', PBY=5.0/


&TAIL /
```

Note: save this function in notepad and save with exe format of *.fds

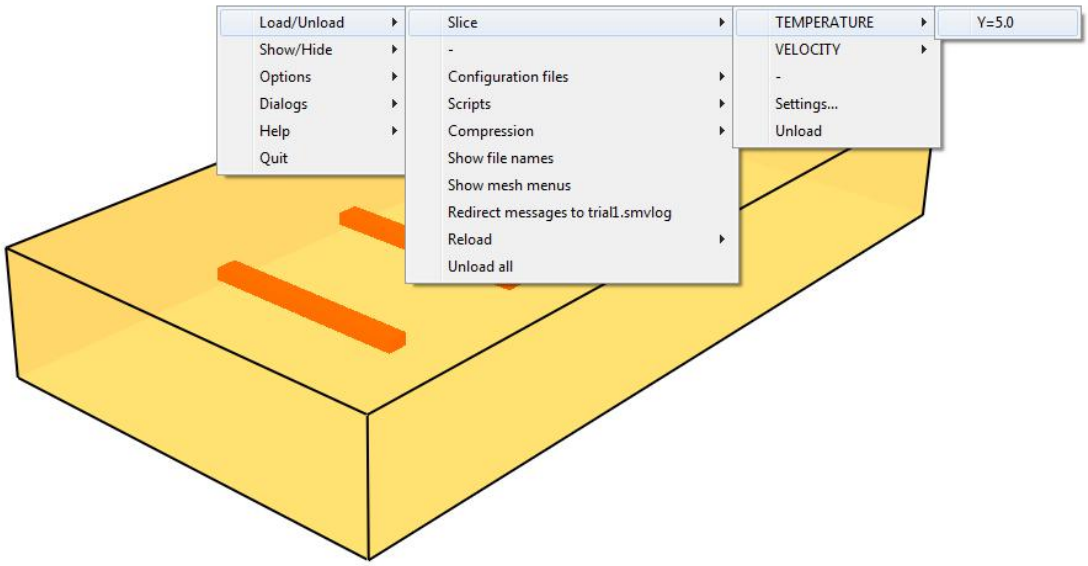
Step IX: run the code by using CMD command prompt in window platform.

```
C:\Users\home\Desktop\FDS-Youtube\FDS-Tut-3 (radiant Cool)\FDS files>fds trial1.
fds
```

Step X: run smoke view after run the FDS file by using CMD command prompt



Step XI: run surface contour using Load/unload in smoke-view



END