GROBDESIGN

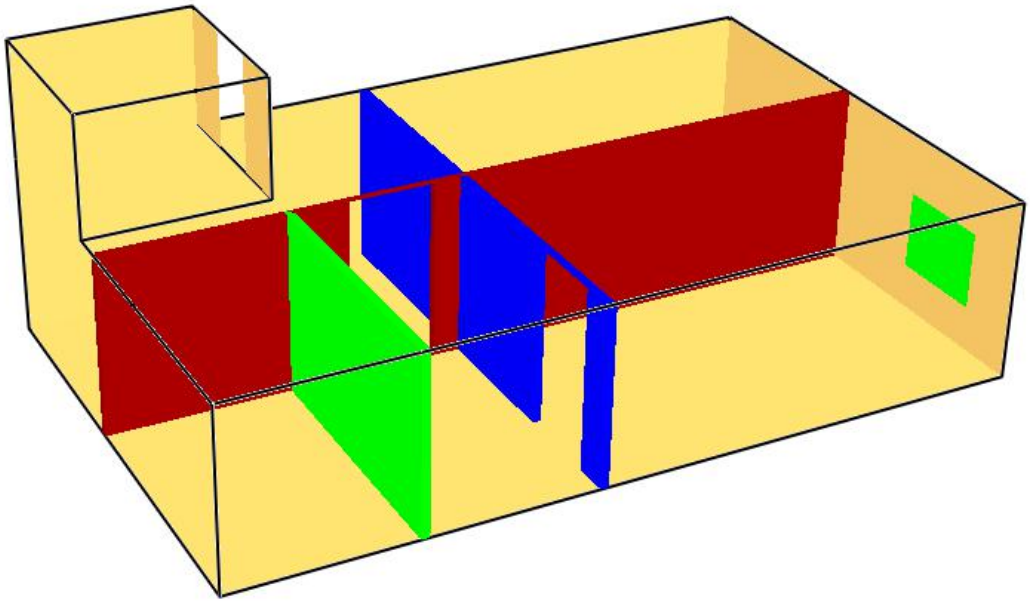
Tutorial 1

SIMPLE FLUID FLOW ANALYSIS USING FDS (FIRE DYNAMICS SIMULATOR) TOOL

GROBDESIGN PVT LTD, JAIPUR 2018

Tutorial 1 Simple fluid flow analysis using **FDS** (fire Dynamics Simulator) tool

In this tutorial a window is created which is treated as inflow of air with velocity of **2.5 m/s** having temperature of **5 C**. The outflow conditions is treated at top of the office, and the boundary condition is set as **open to atmosphere**



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

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

```
&HEAD CHID='office'/
```

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

Step II: create syntax for simulation flow time.

```
&TIME T_END=15.0
```

Note: 15 sec is simulation flow time, which is solved in FDS software.

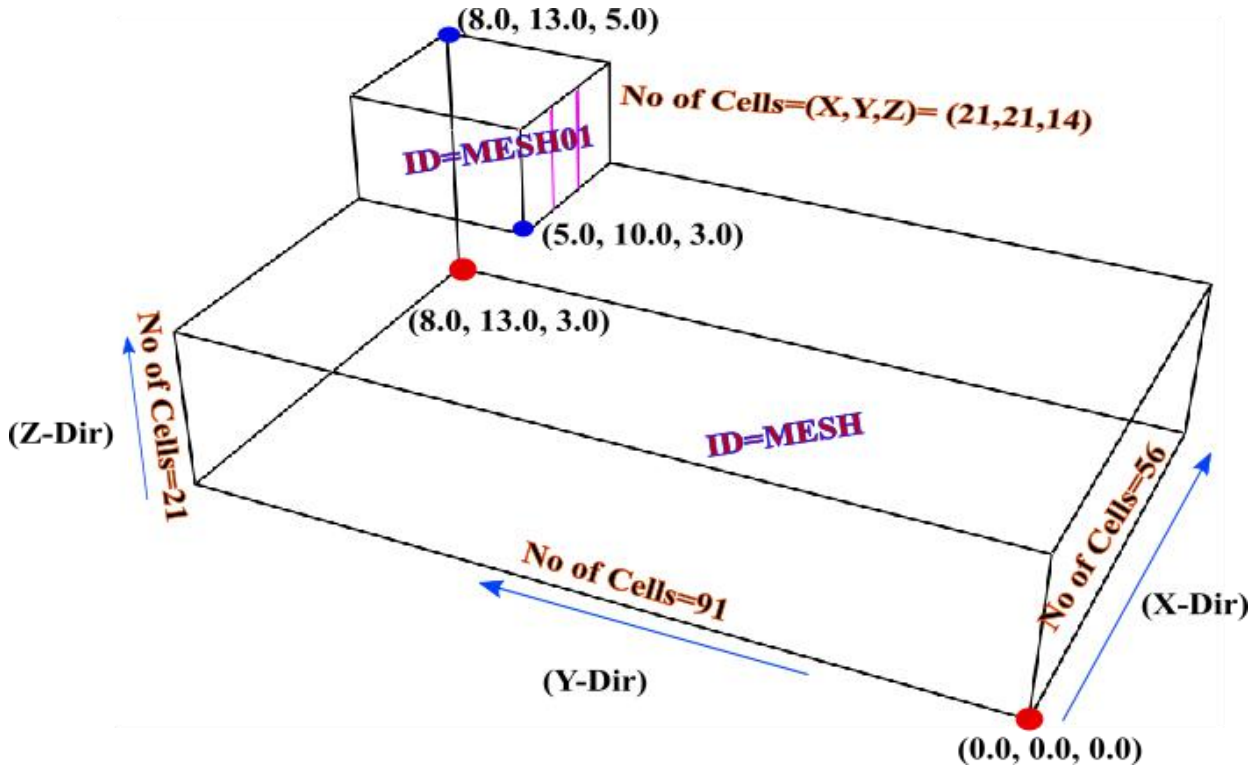
Step III: create syntax for initial temperature of domain.

```
&MISC TMPA=45.0/
```

Note: 45 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', RGB=51,0,0, IJK=56,91,21, XB=0.0,8.0,0.0,13.0,0.0,3.0/
&MESH ID='MESH01', COLOR='RED', IJK=21,21,14, XB=5.0,8.0,10.0,13.0,3.0,5.0/
```



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, 56 cells are created in X-direction, 91 cells are created in Y-direction and 21 cells are created in Z-direction. **XB** represent start and END point of rectangular domain in following syntax

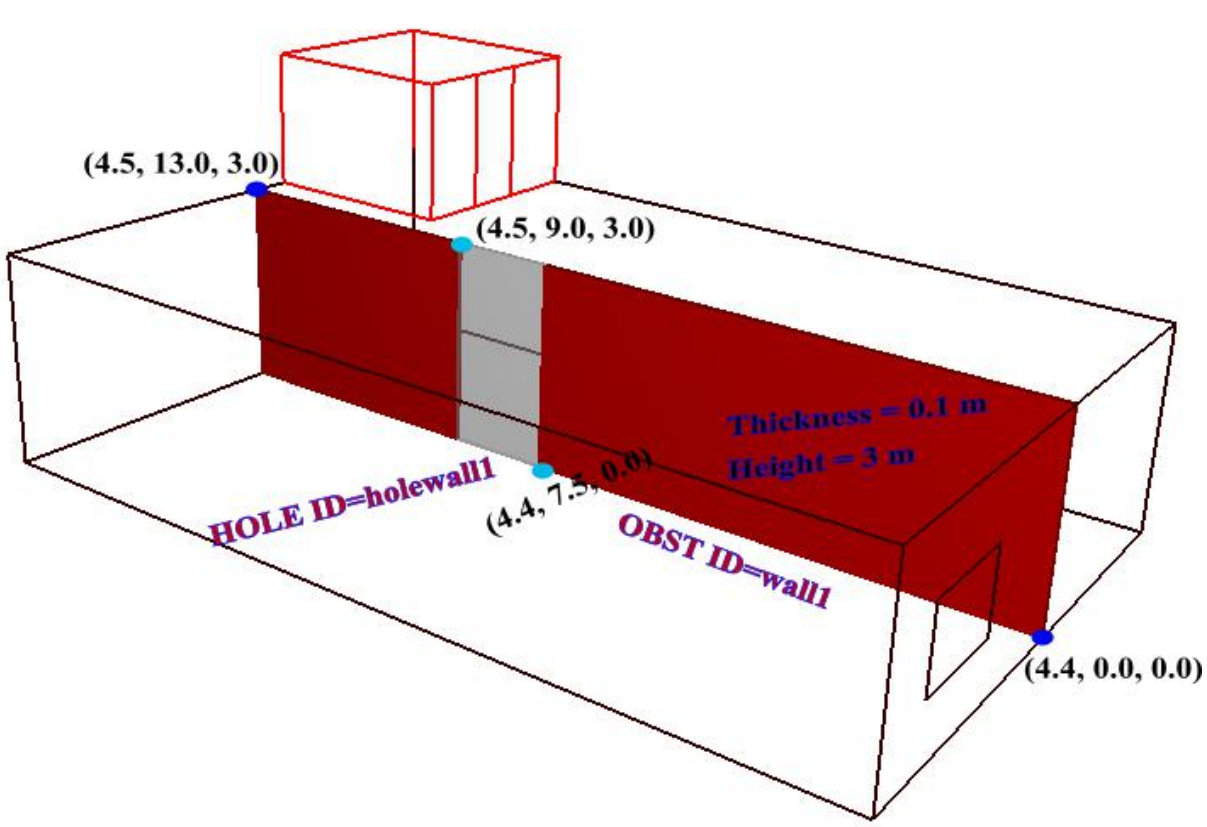
XB=X1,X2,Y1,Y2,Z1,Z2/

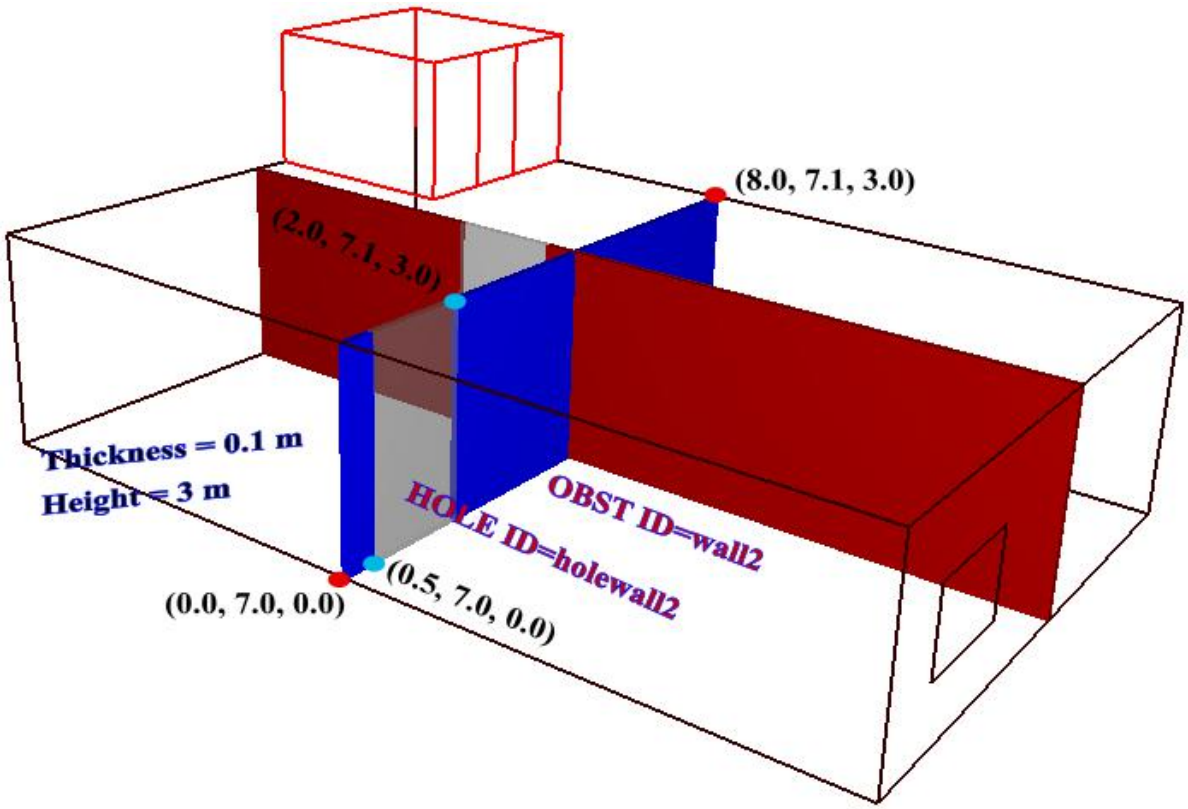
Here 1 represent starting point and 2 represent end point. XB is always start from origin.

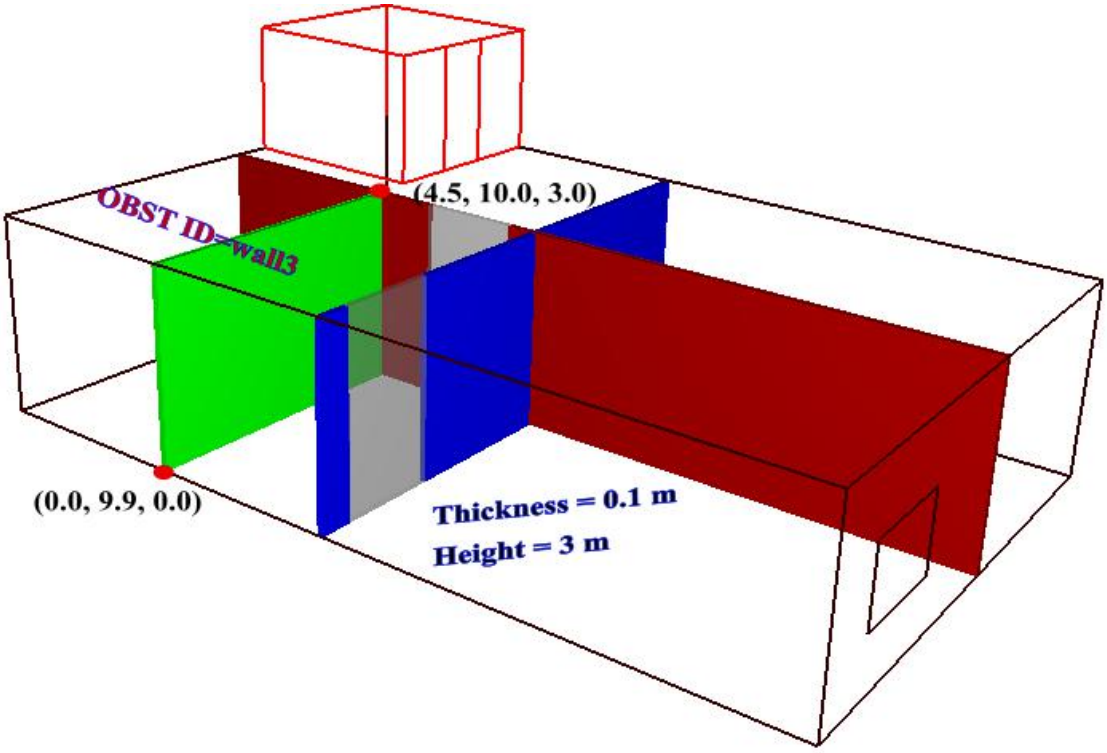
Step V: create walls and holes in walls using OBST command in syntax

```
&OBST ID='wall1', XB=4.4,4.5,0.0,13.0,0.0,3.0, RGB=152,0,0,
SURF_ID='INERT'/
&HOLE ID='Holewall1', XB=4.4,4.5,7.5,9.0,0.0,3.0/
&OBST ID='wall2', XB=0.0,8.0,7.0,7.1,0.0,3.0, COLOR='BLUE',
SURF_ID='INERT'/
&HOLE ID='Holewall2', XB=0.5,2.0,7.0,7.1,0.0,3.0/
&OBST ID='wall3', XB=0.0,4.5,9.9,10.0,0.0,3.0, COLOR='GREEN',
SURF_ID='INERT'/
```

Note: **OBST** is obstacle created in FDS and it can be 3D or 2D depend on fluid problem. Wall1, wall2 and wall3 represent user name of **OBST,** created in current problem. **XB** represent the geometrical presentation in syntax of FDS software. These OBSts are created at different location of domain (see figure). **SURF ID** represent fluid boundary conditions in FDS software. In this case software generated surf-id named “inert” is created inert has no CFD interference in simulation. **HOLE ID** is used to create hole in **OBST ID** in FDS software

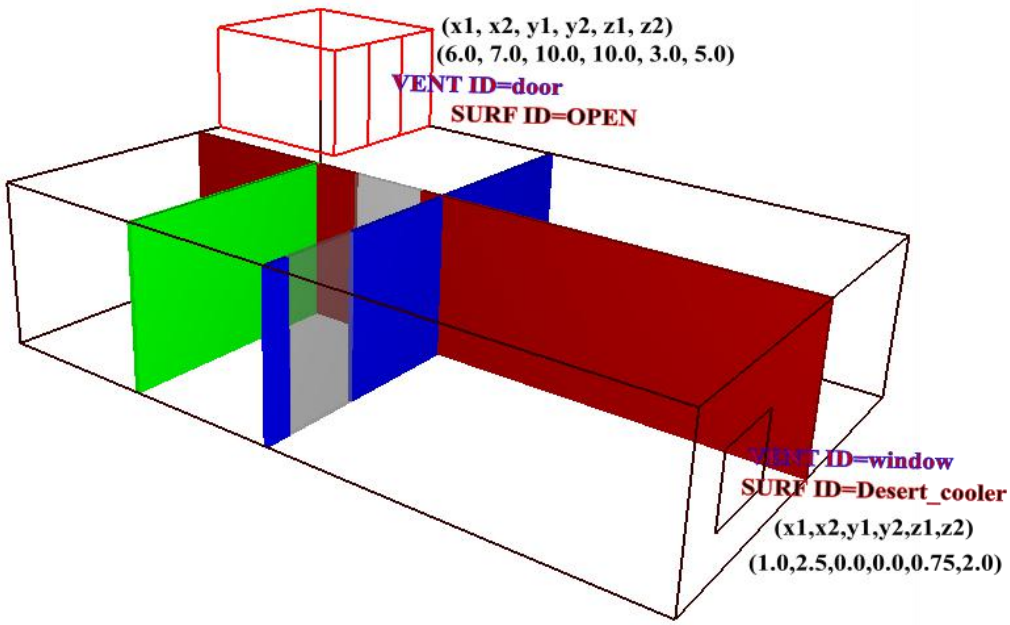

(4.5, 13.0, 3.0)
(4.5, 9.0, 3.0)
Thickness = 0.1 m
Height = 3 m
HOLE ID=holewall1
(4.4, 7.5, 0.0)
OBST ID=wall1
(4.4, 0.0, 0.0)


(8.0, 7.1, 3.0)
(2.0, 7.1, 3.0)
Thickness = 0.1 m
Height = 3 m
OBST ID=wall2
HOLE ID=holewall2
(0.0, 7.0, 0.0)
(0.5, 7.0, 0.0)



Step VI: create door and window at wall surface using VENT ID syntax

```
&VENT ID='door', SURF_ID='OPEN', XB=6.0,7.0,10.0,10.0,3.0,5.0/
&VENT ID='window', SURF_ID='Desert_cooler', XB=1.0,2.5,0.0,0.0,0.75,2.0, COLOR='GREEN'/
```



Note: **VENT ID** is special type of CFD boundary conditions, which are applied on surface of mesh/ **OBST**. In this problem one door and one window is created using these **VENT ID** syntax. Door has software created **SURF ID "OPEN"**, where as window has user created **SURF ID "Desert Cooler"**. **COLOR** represent grren color to window surface.

Step VII: create surf ID syntax

```
&SURF ID='Desert_cooler',
      RGB=26,204,26,
      TMP_FRONT=5.0,
      VEL=-2.5/
```

Note: As discussed in previous step, Desert Cooler is CFD boundary condition applied on **VENT ID** window. **RGB** represent color provided to selected **VENT ID. TMP_FRONT** represent 5 C temperature at window with 2.5 m/s velocity at window. The default fluid medium is AIR in FDS software.

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

```
&SLCF QUANTITY='TEMPERATURE', VECTOR=.TRUE., PBZ=1.5/
&SLCF QUANTITY='VELOCITY', VECTOR=.TRUE., PBX=6.5/
```

Note: **SLCF** syntax represent surface contour. **QUANTITY** represent type of field variable and **PBY/ PBZ/ PBX** represent user defined plane for virtual surface creation. VECTOR TRUE means vector is also present in contour in smoke-view PBZ=1.5 represent 1.5 m distance from Z-direction. The measurement is stared from origin.

Step IX: end the function by using syntax

&TAIL /

Final FDS code for current problem

```
&HEAD CHID='office'/
&TIME T_END=15.0/

&MISC TMPA=45.0/

&MESH ID='MESH', RGB=51,0,0, IJK=56,91,21, XB=0.0,8.0,0.0,13.0,0.0,3.0/
&MESH ID='MESH01', COLOR='RED', IJK=21,21,14,
XB=5.0,8.0,10.0,13.0,3.0,5.0/

&OBST ID='wall1', XB=4.4,4.5,0.0,13.0,0.0,3.0, RGB=152,0,0,
SURF_ID='INERT'/
&HOLE ID='Holewall1', XB=4.4,4.5,7.5,9.0,0.0,3.0/

&OBST ID='wall2', XB=0.0,8.0,7.0,7.1,0.0,3.0, COLOR='BLUE',
SURF_ID='INERT'/
&HOLE ID='Holewall2', XB=0.5,2.0,7.0,7.1,0.0,3.0/

&OBST ID='wall3', XB=0.0,4.5,9.9,10.0,0.0,3.0, COLOR='GREEN',
SURF_ID='INERT'/

&VENT ID='door', SURF_ID='OPEN', XB=6.0,7.0,10.0,10.0,3.0,5.0/
```

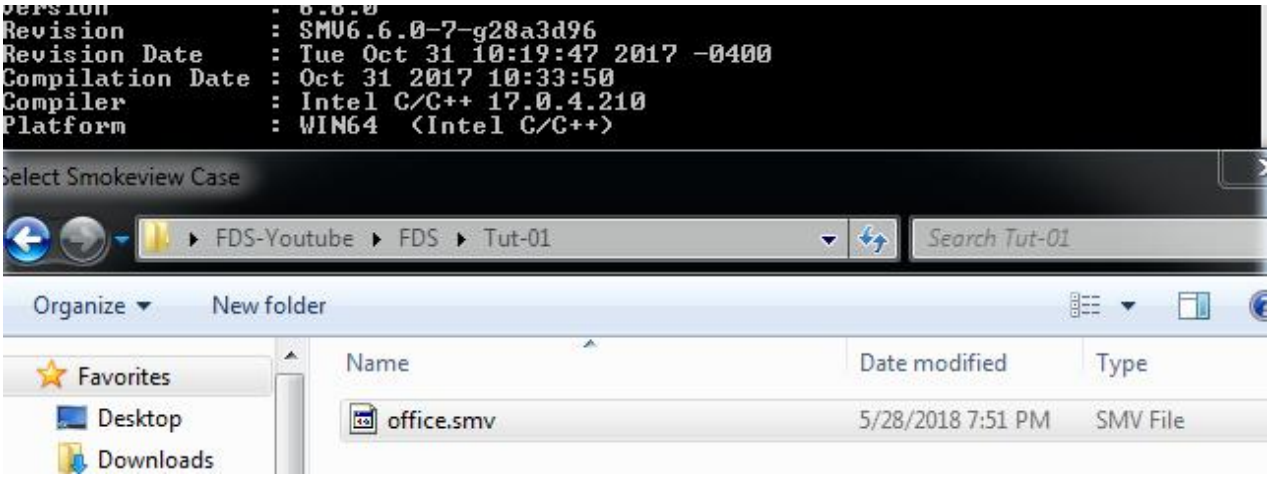
```
&VENT ID='window', SURF_ID='Desert_cooler', XB=1.0,2.5,0.0,0.0,0.75,2.0,
COLOR='GREEN'/

&SURF ID='Desert_cooler',
      RGB=26,204,26,
      TMP_FRONT=5.0,
      VEL=-2.5/

&SLCF QUANTITY='TEMPERATURE', VECTOR=.TRUE., PBZ=1.5/
&SLCF QUANTITY='VELOCITY', VECTOR=.TRUE., PBX=6.5/


&TAIL /
```

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

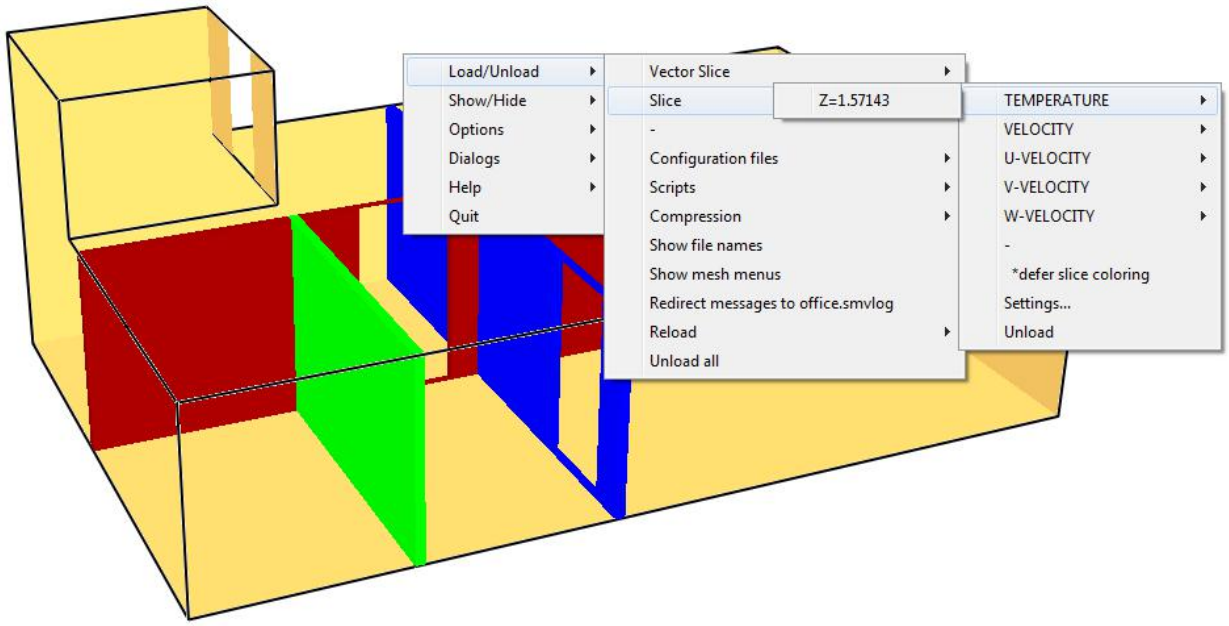
Step X: run the code by using CMD command prompt in window platform.

Go to folder using CD command and then type **fds office.fds.**

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



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



END