FDS7

User's manual

Since version 7, FDS allows creating a transfer file to be used in subsequent SAFIR thermal analyses.

This document describes the specific command line that has to be inserted in the input file of FDS to generate the transfer file.

1) The compartment and the fire source are first described in the input file. Three command lines are of particular interest for the transfer file.

   The command "MESH". It has 2 arguments:
   - "IJK" gives the number of divisions of the FDS domain in the X, Y and Z directions;
   - "XB" gives the limits of the FDS domain as X_{min}, X_{max}, Y_{min}, Y_{max}, Z_{min}, Z_{max}.

   For example, with the command line underneath, the FDS domain extends from 0 to 7 m in the X and Y direction and from 0 to 3.5 m vertically. The size of the cells is 0.10 in each direction.

   ```
   &MESH   IJK = 70, 70, 35   XB = 0.0, 7.0, 0.0, 7.0, 0.0, 3.5 /
   ```

   The command "TIME" gives the final time of the FDS simulation.
   For example, with the command line underneath, the FDS simulation will run for an hour.

   ```
   &TIME T_END = 3600. /
   ```

   The command "RADI" dictates, according to the table underneath, the number of directions in which the radiation intensities will be computed and written in the transfer file. Note that the number that is indicated in this command is not the number of directions.

NUMBER_RADIATION_ANGLES	RESULTING NUMBER OF INTENSITIES
12	16
24	24
30	32
36	40
48	48
52	56
56	64
64	72
80	80
86	88
90	96
100	104

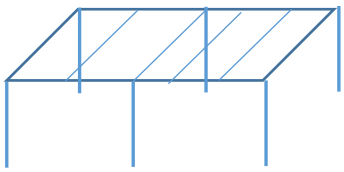
   For example, with the following command, the radiation intensities will be written in 104 directions

   ```
   &RADI NUMBER_RADIATION_ANGLES = 100 /
   ```

2) A specific command line is introduced for the creation of the transfer file. This is the command "RADF". This command has several arguments:

- "XB" gives the limits of the transfer domain as X_{min}, X_{max}, Y_{min}, Y_{max}, Z_{min}, Z_{max}.. The transfer domain may not extend outside the FDS domain described by "XB" in the "MESH" command. It is good practice to have the boundaries of the transfer domain aligned with the cells of the CFD domain.
- "T_RADF_BEGIN" (optional) gives the first time for which information will be written in the transfer file. It is good practice to write T_RADF_BEGIN = 0 in order to have initial conditions written in the transfer file. By default, T_RADF_BEGIN is set to the start time of the simulation, T_BEGIN.
- "T_RADF_END" (optional) gives the last time for which information will be written in the transfer file. The value given for "T_RADF_END" in the command "FDStoSAFIR" may not be greater than the value given for "T_END" in the command "TIME". By default, T_RADF_END is set to T_END.
- "DT_RADF" (optional) gives the time step after which information will be written in the transfer file. It is good practice that the result of (T_RADF_END – T_RADF_BEGIN) / DT_RADF be an integer. By default, DT_RADF, the time interval, is the end time minus the start time divided by 5.
- I_STEP, J_STEP and K_STEP (optional) allow skipping cells in the transfer file in order to reduce the size of this file.
  For example, if there are 20 cells in the transfer domain in the direction X, utilisation of "I_STEP=3" will make FDS to write the information for cells 1, 4, 7, 10, 13, 16 and 19. The information will not be written for cell 20.

  Note that it is possible to have several RADF commands in one single FDS analysis, each defining a different transfer domain. This will create several transfer files that can be successively renamed as "cfd.txt" to perform successive thermal analyses for different section types.
  For example, in the following structure made of a skeleton of beams and columns, it could be worth, to reduce the size of the transfer files, to create 3 transfer domains:
  - One, rather flat that would contain the beams;
  - One, rather thin that would contain the 3 columns at the front;
  - One, rather thin, that would contain the 3 columns at the back.



3) The time limits and intervals for writing the information in the transfer file are described in the DUMP line.

- "T_RADF_BEGIN" (optional) gives the first time for which information will be written in the transfer file. It is good practice to write T_RADF_BEGIN = 0 in order to have initial conditions written in the transfer file. By default, T_RADF_BEGIN is set to the start time of the simulation, T_BEGIN.

- “T_RADF_END” (optional) gives the last time for which information will be written in the transfer file. The value given for “T_RADF_END” in the command “FDStoSAFIR” may not be greater than the value given for “T_END” in the command “TIME”. By default, T_RADF_END is set to T_END.
- “DT_RADF” (optional) gives the time step after which information will be written in the transfer file. It is good practice that the result of (T_RADF_END – T_RADF_BEGIN) / DT_RADF be an integer. By default, DT_RADF, the time interval, is the end time minus the start time divided by 5.

  Example:

```
&RADF XB=2.3, 6.6,   1.3, 5.3,   0, 3.5,  I_STEP=3, J_STEP=3, K_STEP=3/
&DUMP T_RADF_BEGIN=0., T_RADF_END=30., DT_RADF=10.
```