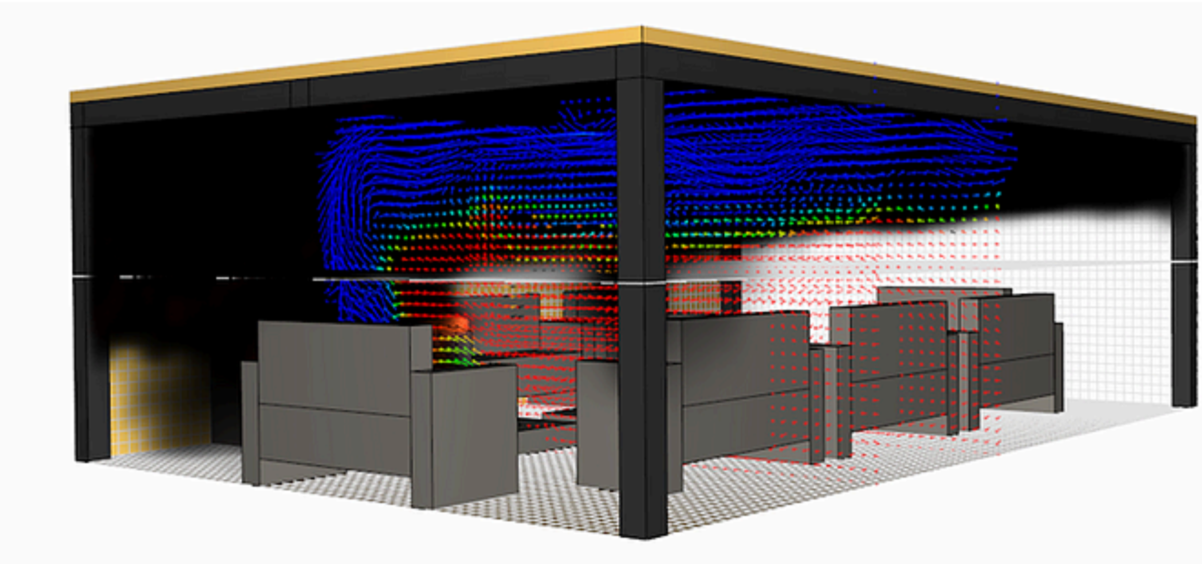
OpenFIRE: Real Fire



User's Manual

v1.0.0

Aatif Ali Khan

Introduction

OpenFIRE tool is able to perform heat transfer (HT) analysis for both idealized fire scenarios as well as real fires. This chapter describes how OpenFIRE tool can be used to perform the heat transfer analysis for idealised fires scenarios such as standard fire, parametric fires, localised fires. It also discusses how to interface CFD (FDS) with OpenSEES to perform all three analysis (fire modelling, heat transfer analysis and mechanical analysis) by using a middleware tool (FSDM) – an integral part of OpenFIRE. The middleware generates a part of script files for all three analyses with minimal human efforts and reduces chances of errors while handling a large amount of data for structural analysis especially when it is required to perform structural analysis of tall buildings (WTC and Plasco building) where structural model may contain thousands of elements.

FSDM allows user to produce common parts of the scripts for all three analyses simultaneously, which further are used to perform sequential analysis. Unlike FSDM, OpenFIRE for idealised fire models is able to perform the HT analysis directly. OpenFIRE includes a number of modules which are written in Python programming language. Graphical User Interface (GUI) is developed for each module and subsequent sections explain how it can be used to perform structural analysis for both idealised and field models.

1. OpenFIRE for CFD-FEM coupling

1.1.OpenFIRE Middleware (FSDM) Executable

As the middleware tool is written in the Python programming language, it can be run on Windows, Linux or Mac OS. The relevant executable file can be downloaded from [Ref]. Once execute the program, it shows multiple options for the modules to be used as shown in Figure 1. Some of these modules are discussed later in this chapter.

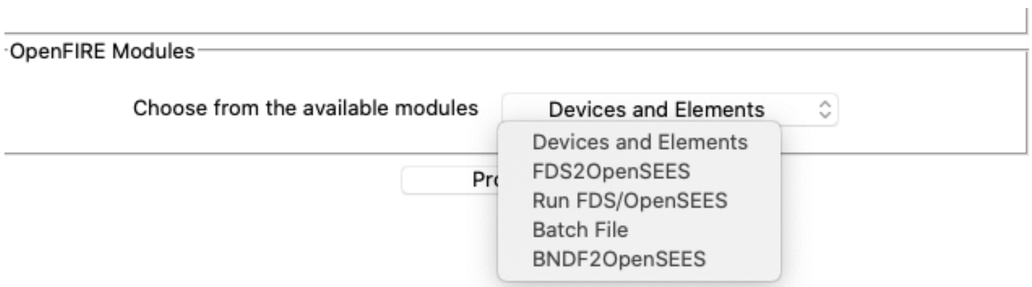


Figure 1: Modules in current OpenFire Middleware (FSDM)

Available Modules in FSDM

Modules included in the current version of FSDM middleware package are listed below:

Device and Elements: It generates *devices* (quantity to be measured and location) for the FDS script and entities for OpenSEES. It also produces a part script for structural analysis which includes the details of the *elements*.

FDS2OpenSEES: When *Device* (Refer Chapter Coupling) approach is used to get the output from the FDS, this module converts the data in the OpenSEES format.

Run FDS and OpenSEES: The installed FDS and OpenSEES can be run directly from this module of the middleware.

BNDFBatchFile: When *BNDF* method is used to measure a quantity in FDS, this module of the middleware generates a batch file that can be run in *Terminal* to produce an output of the FDS in ascii format.

BNDF2OpenSEES: When BNDF approach is used, this module will utilise the output from the fds2ascii program, and finally output in OpenSEES format can be obtained. Refer FDS User's manual [Ref] to get the data from fds2ascii.

The last methods are optional, therefore these are not discussed in this chapter however readers are recommended to refer FDS User's manual [Ref] to understand how and when BNDF approach can be beneficial to capture data from FDS.

1.2. Devices and Elements

Generally, it is not required to define structural components in FDS, however measuring devices are needed to be placed at the required location where data needs to be exploited from FDS to use as boundary conditions for the thermal and structural analysis in OpenSEES. FDS is written in Fortran [Ref] so the input script file must be written in such format. On the other hand, the input files for OpenSEES to perform heat transfer and structural analysis are required to be written in **.tcl* format. As the link between fire model and structural model is the *Device* location. It is worth noting that only the fire compartment would be generated for fire modelling rather than whole building therefore **it is necessary to keep the global coordinates same for both models.**

1.2.1. FDS Devices and other basic inputs

FDS requires location and orientation of the measuring devices such as adiabatic surface temperature (AST), heat flux (HF), heat transfer coefficients (HTC). This part of the module helps to write a script for the device locations based on the structural geometry (Figure 2). Once a directory is selected, user needs to define *quantity* which is required as a boundary condition for heat transfer analysis (Figure 3 shows an example of input data for defining a device in FDS). The current version of middleware has three quantities (AST, HF, and HTC) to obtain time-variant data for HT analysis because OpenSEES requires only these three quantities to carry out HT analysis.

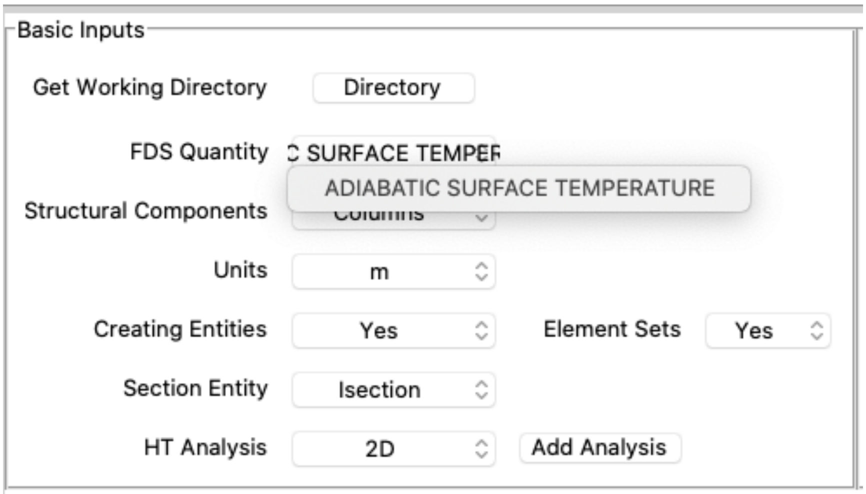


Figure 2: Basic Inputs for the Devices in FDS and OpenSEES Entities

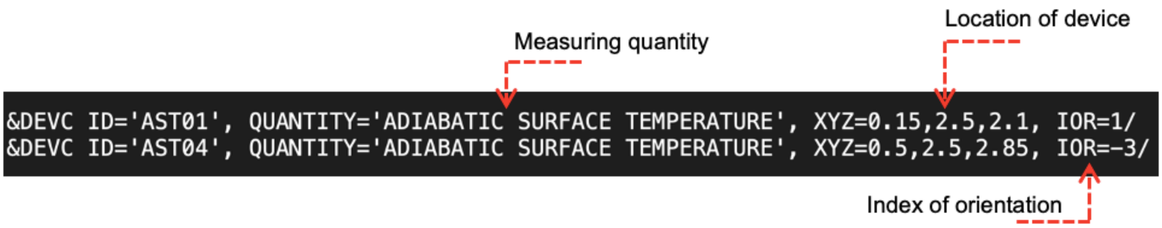


Figure 3: Defining a measuring device in FDS

Generally, a structure is composed of columns, beams, trusses, slabs, or combinations of them. It is required to define structural components where device needs to be installed as shown in Figure 4. The device's location can be set based on the level of accuracy in terms of thermal gradients is required for structural analysis. For example, if structural analysis requires different time-temperature history for every 0.5m then this module will generate the devices accordingly (devices are installed at every 0.5m). After selecting the type of structural component, information related to structural component needs to be filled in the input entry boxes of the corresponding frame (Figure 5 shows the frames for columns and longitudinal beam). It is worth noting while generating structural model in FDS, by default, **Z-axis is taken as vertical direction** (gravity is along the Z-axis) therefore, it is recommended to follow this in structural model. Furthermore, as discussed earlier **global coordinates should be same** for the fire compartments and building model. As shown in Figure 5, for the column it is needed to provide the X, Y coordinates for the devices and initial, height of the columns, and the distance between each device as an increment to place the devices. Finally, the orientation of the devices (-1, -2, -3, 1, 2, and 3 for -X, -Y , -Z , +X, +Y, and +Z, respectively) is to be defined based on the outward position of the installed devices (Please refer FDS User's Manual []). Similar information is provided for other structural components. Once all input data is provided, an FDS input file containing the information of *devices* is generated in the selected directory. Follow the below steps to define the devices for fire simulation.

Steps

- Chose the working directory
- Assign the type of data needs to be extracted such as AST, HTC or HF. Only one quantity can be assigned at one time.
- Choose the type of structural component where a device needs to be placed such as columns, beams, trusses or slabs.
- Choose the units of structural components (mm or m). It is noteworthy, though structural frame may be crated in "mm" but FDS devices and heat transfer entities (section) are generated in standard SI unit (m).
  User has to specify if heat transfer entities (explained in next section) needs to be created . If only FDS devices needs to be created then select "*No*" from the drop menu. The default value is "*Yes*" as shown in Figure 3. In some instances, it might be required to define more devices than entities or needs to install two types of devices such as AST and HTC, in such cases, the user may select "No" to avoid creating duplicate entity. Similar option is for creating the element sets for structural analysis (Section 1.3.1).
- OpenSEES is capable to perform both 2D and 3D heat transfer analysis (refer Chapter 3). By default, 2D analysis is selected which is generally preferable for large geometry to save computational cost. By pressing "Add analysis" button, type of analysis would be added to the script file.
- Based on the chosen "*Structural Component*", the data in the relevant frame needs to be filled. For example, Figure 4 shows the frame for columns where user has to provide the values of X and Y coordinate. The '*initial value of Z'* suggests Z coordinate for the bottom of the column or location from where output data from FDS is required. Provide the total height of the column. By filling the increment value, the program would evenly place devices separated with the suggested increment value until the height of the column.
- Based on the face of structural components where device is to be placed, the orientation of the devices (outward direction) needs to be provided for each component.

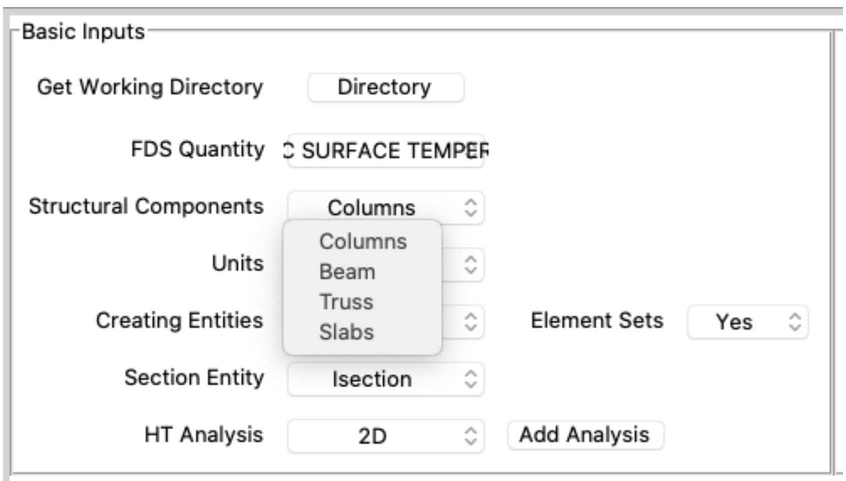


Figure 4: Structural components

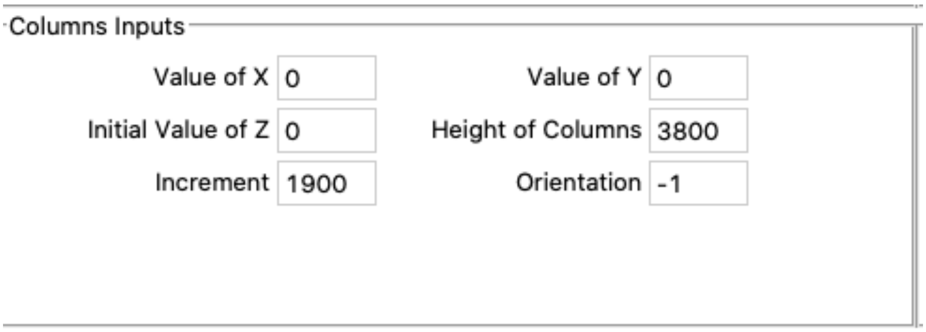


Figure 5: Inputs of structural components

1.3. OpenSEES heat transfer entities

To perform HT analysis in OpenSEES, structural components are defined as entities corresponding to the section type such as I-section entities for beam and columns, block or brick entities for slabs [OpenSEES Website Link]. The input file for HT analysis should be written as TCL (**.tcl*) script. Figure 6 represents a typical heat transfer script file as an example.

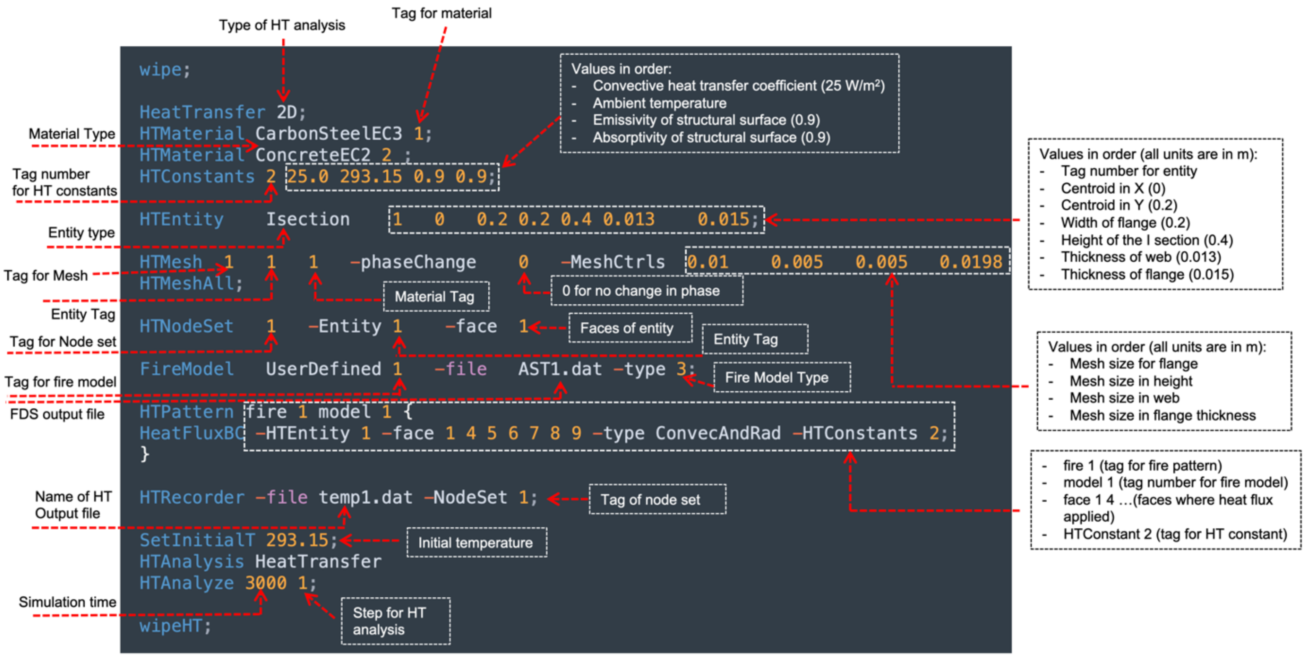


Figure 6: An example of HT input file in OpenSEES

As shown in Figure 7, some basic information needs to provide in OpenSEES HT script before generating heat transfer entities such as type of material, heat transfer constants, faces where heat fluxes are applied and so on. Readers are recommended to visit *OpenSEES for Fire* for more details [Ref]. To perform heat transfer analysis, in the current version of OpenSEES four different types of boundary conditions can be applied with the combinations of the quantities obtained from FDS as shown in Table 1.

Table 1: Boundary conditions for heat transfer analysis

Method	Boundary condition	Value of convective heat transfer Coefficient	Type of fire in OpenSEES HT file
1	AST	Fixed	1
2	HF	Fixed	3
3	AST+ HTC	Varying	2
4	HF + HTC	Varying	4

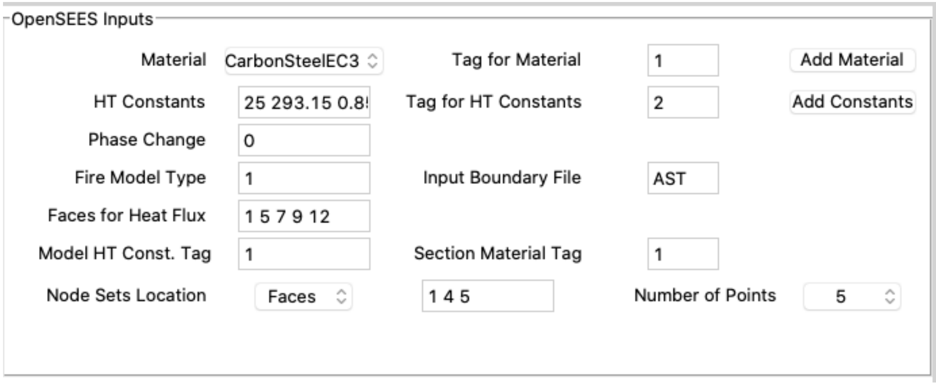


Figure 7: Section Details for structural components

In the first method, only time-variant adiabatic surface temperatures with a fixed value of the convective heat transfer coefficient are applied. By applying the AST as a boundary condition, the time-temperature history at the surface is obtained. In the second method, the time-varying heat fluxes from FDS with a fixed value of HTC are used as boundary conditions for HT analysis. In the third and fourth methods, the time-varying HTCs obtained from FDS are added to AST and HF, respectively.

Once preliminary details for conducting heat transfer analysis such as material type, heat transfer constants, fire model and thermal boundary conditions etc are defined, it is required to provide inputs for the section type and dimensions to generate HT entities as shown in Figure 7. The inputs for defining a section in Figure 7 are self-explanatory. This part of the module creates an entity in the heat transfer model corresponding to each device in the FDS model. The AST obtained from individual devices in FDS will then be applied to corresponding entity in heat transfer model as thermal boundary conditions to obtain the time temperature history. This time temperature output generated by the heat transfer model can be used as thermal load to conduct a thermomechanical analysis of the structure. Steps for generating the heat transfer model which is compatible with the FDS model are listed below :

Steps

- After providing the data for devices and selecting the analysis and entity type for heat transfer analysis, define the material type of the structural components such as concrete or steel for CarbonSteelEC3 as defined in Eurocode 3. The script of OpenSEES should have such information and for each material type, a tag number should be assigned. In the entry boxes as shown in Figure 7, provide type of material for the sections that would be used in analysis and assign a tag number to each material and add them in script by pressing the assigned button (Figure 7).

- Now, provide the values of heat transfer constants that are used in heat flux equation such as convective heat transfer coefficient, emissivity, absorptivity, Stephen-Boltzmann constant, and assign a tag number to each group of constants. Then, press "Add Constants" to append them in the script file.

- Some material such as concrete contains moisture which may evaporate (change its phase) while high temperature reaches within. Therefore, it is required to explicitly mention if phase change is required in the analysis or not. If material changes its phase during a fire provide "1" to the entry box otherwise "0" (default value).
- To enable a coupling between heat transfer and thermomechanical model, locations, where temperature data is required for conducting thermomechanical analysis are defined as node sets by using 'HTNodeSet' command as shown in Figure X3. The current version of FSDM allows two methods to define location: nodes at entity faces and nodes at user defined locations. Each HT entity has various "faces" as shown in Figure 6. In the entry box "Location for Nose Sets" define "faces" of the entity for the nodal data where temperature history can be recorded. User can provide as many faces separating them by "space or tab". When 'User Defined' option is selected, location where data needs to be extracted needs to be defined. Sd shown in Figure 8. Temperature data along the depth of the entity may required to perform structural analysis, in such cases in the current version of the OpenSEES allows to select any of the three options; 2, 5 , for example if "5" is selected, temperature data at five locations along the depth of entity will be provided after heat transfer analysis.
- There are various fire models available in OpenSEES to perform HT analysis such as standard fire, localised fire (refer OpenFIRE for idealised fires). The boundary condition from the CFD analysis or experiment is referred as "*User-defined*" in OpenSEES (Figure 6). Each fire model should be assigned its *type*. The current version of OpenFIRE supports four *type*s of user defined thermal boundary conditions as shown in Table 1; for example, if AST is being applied as boundary condition then select '1' as type. Please refer the latest version of OpenSEES while filling the entry box for fire model.
- The output from the FDS would be in the form of time varying heat flux or time-temperature history (Refer Section **FDS 2 OpenSEES**). For heat transfer analysis, the output obtained from FDS is used as input in the form of thermal boundary conditions. The input file name needs to be provided in the entry box. In the current version, as one entity is created for each device, therefore, only one type of boundary file name can be assigned. However, the user can replace it based on the type of BC is required for the HT analysis.
- While performing the HT analysis, *faces* of the entities which are exposed to fire need to be defined to calculate heat fluxes on those faces (Figure 9). Some instances, it might be required to calculate the heat fluxes on specific faces of the entity. For such cases, only particular face can be assigned; for example, if a slab is exposed to fire from bottom, the thermal boundary conditions are applied only at the bottom face "face 1" of the slab. User can provide as many faces as required depending upon the number of exposed faces separating them by "space or tab".

- When basic information for HT script file is filled, the data for the specific section type (I-section, block) needs to be provided such as dimensions of the entities, location of the centroid, mesh sizes. The section type would depend on the type of structural component for example slab may be represented by block type entity and beam or columns may be represented by Isection type entity. Figure 7 shows the 'data input frames' for "Isection" and "Block" type entities.

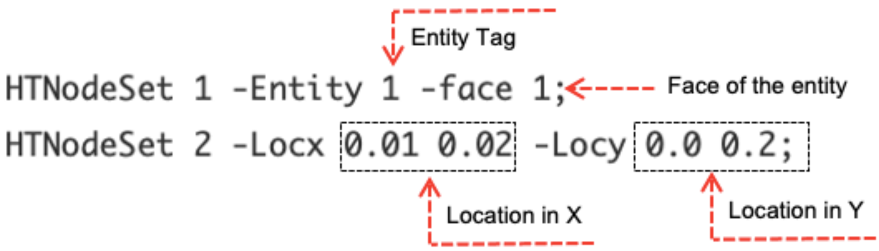


Figure 8: Node Set creation

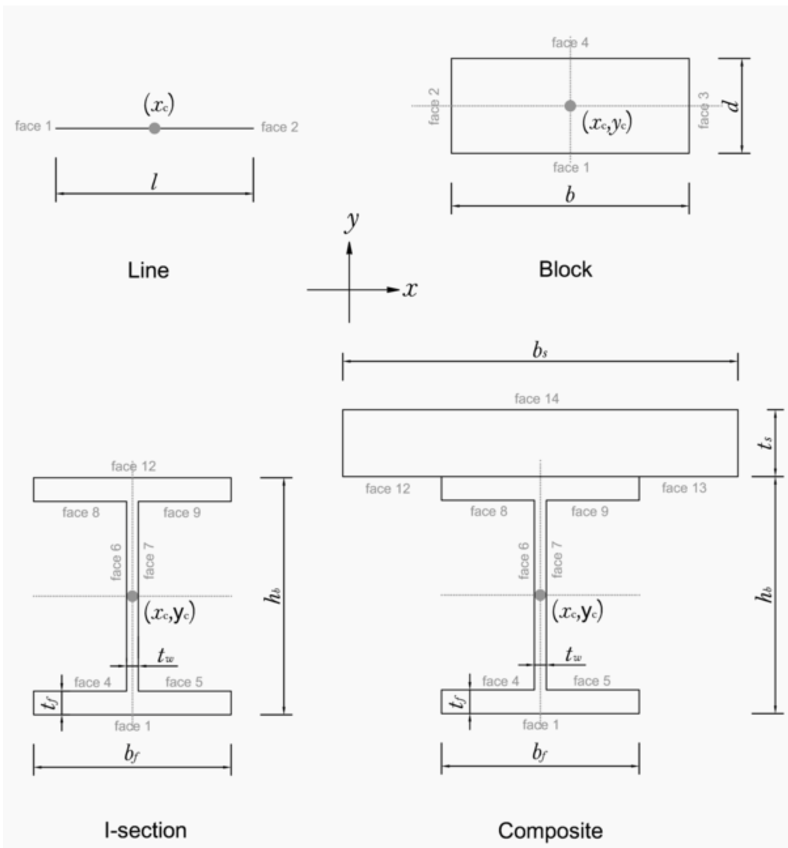


Figure 9: Faces for HT Entities [Ref OpenSEES for fire]

1.3.1. **Elements sets**

This module of the FSDM maps the specific time-temperature history from heat transfer analysis to the thermo mechanical analysis.

Similar to any other finite element (FE) software, structural model in OpenSEES requires components to be represented by *different type of elements*. Various types of elements are available in OpenSEES such as beam-column elements, shell elements, and continuum (solid) elements. Beam-column elements are computationally inexpensive and are most suitable for understanding global behaviour of very large structures . In a typical thermomechanical model, slabs are generally represented by shell elements, and other components such as beam, columns, and trusses may be represented by beam-column elements. In OpenSEES, there is no coupling feature available to map the data from a heat transfer analysis to a thermomechanical analysis. The output data from the HT analysis is to be used as an input for thermomechanical analysis. This output data is applied as the thermal loads to the structural elements, therefore it is required to identify the right elements for specific HT output recorder file. Once HT analysis is finished, a number output files would be generated based on the number of entities and nodesets required along the depth of the entity. Another module (section #) joined the temperature data of all node sets and combine them in the format required for structural analysis in OpenSEES. Now, the output files containing the time-temperature history for each entity containing the all nodesets data would be applied to the structural elements present at the same geometrical location as in the heat transfer model or previously in the CFD model. For a large structure identifying and mapping the correct entity output to the structural model at specific locations is cumbersome and may increase the chances of human error. Therefore, it is necessary to find elements associated to a particular geometrical location to which the entity output is assigned as thermal loads.

As an entity is created for each device in CFD calculation. When defining these devices, the elements at that specific location can be identified using a module of the FSDM. This module generates a part of the script file for thermomechanical model which contains the data associated with the application of thermal loads (specific entity output) to the corresponding structural elements as shown in X ("temp1.dat" is the output from the HT analysis). In this way, accurate mapping of the data from HT analysis can be performed with minimal chances of error. Below are the steps to create a part of structural analysis script file which allows to maps the data by applying the HT output as thermal loads to the structural model.

Steps

- Firstly, define if element set needs to be generated or not. By default, its value is "Yes" as shown in Figure 4, however, if structural analysis needs not to be performed or only HT entities are required to generate, the option "No" can be chosen from the drop menu.
- Once structural geometry is created in OpenSEES, it creates elements and associated nodes for each element. Elements and nodes data can be extracted from the structural model script. Three different files should be saved namely; nodes file, beam-column elements and shell elements (if exist) in either **.dat* or **.tcl* format.

- Save these files in the working directory, and browse the nodes file by clicking the relevant button as shown in Figure 10 and generate nodes. A node file containing the nodes number and corresponding coordinates (*x, y* and z) would be generated, and other data would be removed. Similarly, for beam-column elements and shell elements files, some of temporary files would be created.
- Based on the input coordinates for each CFD measuring *device*, this part of the module searches the nodes within the specified range. The module also identified the elements within this range, to which an output file is generated after HT analysis. This output file would be applied to all elements within that range (Refer Website) as shown in Figure 11 as an example for fire load file ("temp1.dat") on an element.
- To apply thermal loads on beam-column or shell elements, temperature field for these element should be defined. Generally, data points are defined across the depth of the section to capture a thermal gradient. As shown in Figure 12, in the current version of OpenSEES, maximum nine data points can be used to represent a thermal gradient for a beam-column element which creates eight temperature zones. Lesser number of data points can also be used where thermal gradient is not significant and could be calculated using linear interpolation between data points.
- Once above-mentioned data has been filled for devices, entities and element sets, press the ***save*** button as shown in Figure 13. It will generate three script files namely; CFD script file containing the device location, HT script file containing the entities and other relevant information including mesh and fire model, boundary condition files name, an input file for structural analysis containing the element load with corresponding file name containing the output of HT analysis.
- To finalize the HT analysis file, define the initial temperature, simulation time and time-step for HT analysis (Figure 13). Pressing "Save HT File" would generate the full script of HT analysis.

Now, CFD file can be updated with other details such as the geometry of the fire compartment, fire load, fire size, reaction and so on [FDS manual]. Similarly, structural analysis script file can be updated with other necessary details. This framework streamlines the process of mapping the data from CFD to FEM (OpenSEES). Next section describes how the data from FDS is transferred to OpenSEES for HT analysis.

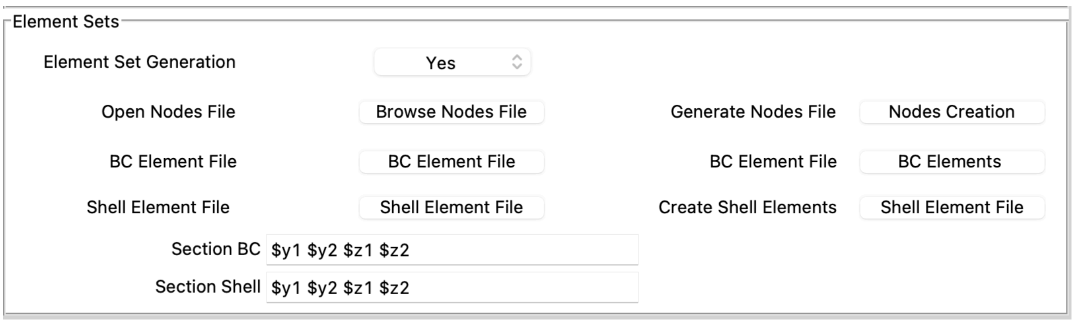


Figure 10: Element set generation for structural components

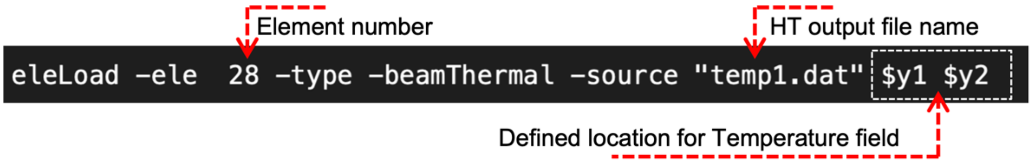


Figure 11

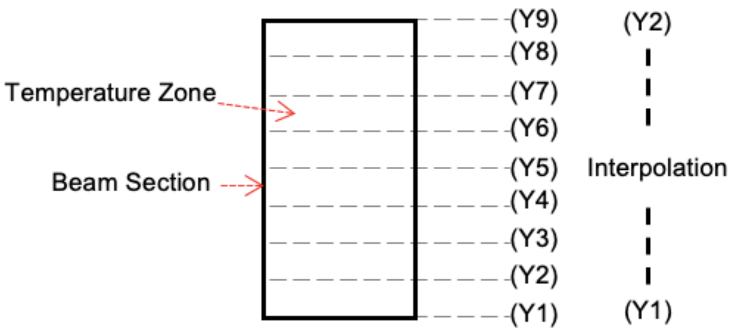


Figure 12

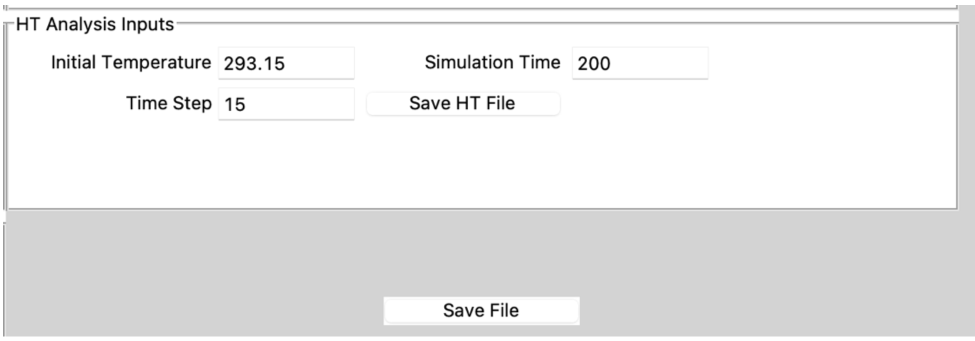


Figure 13: Saving the file and generating the three script files

```
eleLoad -ele 28 -type -beamThermal -source "temp1.dat" $c1 $c2 $c3 $c4 $c5
eleLoad -ele 29 -type -beamThermal -source "temp1.dat" $c1 $c2 $c3 $c4 $c5
eleLoad -ele 30 -type -beamThermal -source "temp1.dat" $c1 $c2 $c3 $c4 $c5
eleLoad -ele 31 -type -beamThermal -source "temp1.dat" $c1 $c2 $c3 $c4 $c5
eleLoad -ele 32 -type -beamThermal -source "temp1.dat" $c1 $c2 $c3 $c4 $c5
eleLoad -ele 33 -type -beamThermal -source "temp1.dat" $c1 $c2 $c3 $c4 $c5
eleLoad -ele 34 -type -beamThermal -source "temp1.dat" $c1 $c2 $c3 $c4 $c5
eleLoad -ele 35 -type -beamThermal -source "temp1.dat" $c1 $c2 $c3 $c4 $c5
eleLoad -ele 36 -type -beamThermal -source "temp1.dat" $c1 $c2 $c3 $c4 $c5
```

Figure X: Element set where HT output would be applied

1.4. FDS2OpenSEES

Using this module data from the FDS can be transferred to OpenSEES. The output data obtained from FDS is not in the format of OpenSEES, pre-processing work needs to be done to covert the output which can be suitable to perform HT analysis. For HT analysis in OpenSEES, it requires a separate file containing time varying data of a quantity such as a temperature or heat flux for each HT entity. On the other hand, FDS provides a single "*.*csv*" file containing all data for all output devices (if DEVC approach is used). This module allows to generate the necessary input file for each entity.

Figure 14 shows the frame of the module that allows to generate files that are used as a boundary condition for HT analysis. Below are the steps to convert FDS output into OpenSEES format.

Steps

- Firstly define the directory where the output file of FDS is saved.
- As discussed in the previous section that four types of boundary conditions can be used in the current version of OpenSEES (Refer to Table 1). This module can generate the necessary files for each boundary condition presented in Table 1. As shown in Figure 15, from drop menu (Boundary Condition) user can select the type of input files require. It is worth noting that the FDS output data would be in the same sequence as written in the FDS script file. Therefore, it is recommended to keep a sequence to avoid any error while handling a very large data. In the current version of this module, firstly all AST devices are placed followed by HF, and HTC. User must maintain the sequence if all four boundary conditions are required in the analysis. Generally, the method where only ASTs are taken as boundary condition provides better or similar results comparing to others (Refer Validation Case).
- Browse the FDS file and reformat by clicking the "Update File", it will generate another file after removing the information which is not required by OpenSEES such as header files, units description.
- Provide number of devices (or entities) that are installed for each type of quantity (AST, HF, HTC), and press "Save" button. It will generate as many files as the number of entities (separate time-temperature history for each entity) which can be used as a boundary condition for HT analysis.

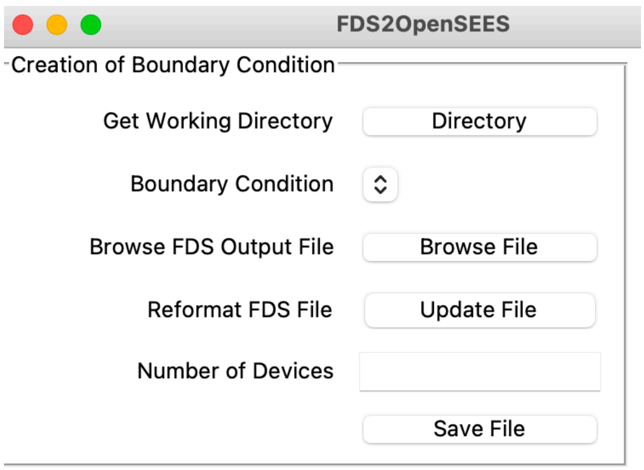


Figure 14: FDS to OpenSEES module

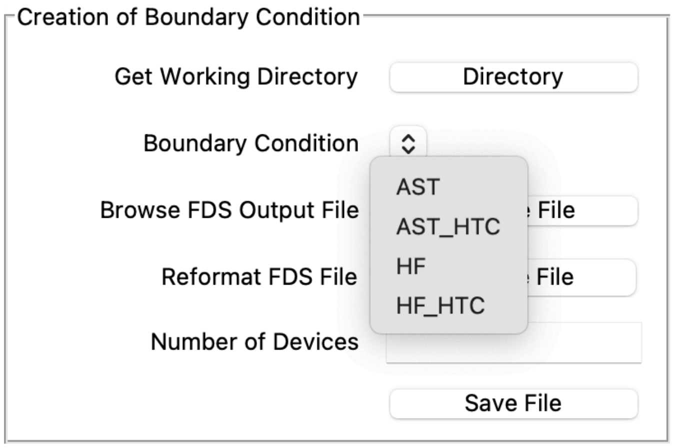


Figure 15: Boundary conditions for HT analysis in OpenSEES

1.5.Run FDS and OpenSEES

This module of the middleware has two frames to run both FDS and OpenSEES. First, make sure that FDS is installed (Figure 16). User can download FDS from the NIST website [Ref]. This module allows running both software from one frame, although OpenSEES would be run after fire simulation is finished. Below are the steps to run both software.

Run FDS

- Select the directory where the FDS script file ("**.fds*") is saved.
- Write the file name in the entry box without the file extension.
- Based on the number of processor and cores in the machine, assign the number of processors and cores to run the fire simulation. To utilize this option, split the FDS mesh at least in the number of processor user wants to assign for the calculation (refer FDS manual to install and save) [Ref]. Once this information has been provided run FDS by pressing "Run FDS".

Run OpenSEES

- Select the directory where the OpenSEES HT analysis script file ("**.tcl*") is saved.
- Save the files (generated by using FDS2OpenSEES module) in the same folder where HT analysis script file is saved (generated by "Device and element" module).
- Write the file name in the entry box without the file extension.
- Make sure that OpenSEES program is saved in the computer at specified location.
- Run OpenSEES.
- Once HT analysis is finished, it will generate output files containing the time-temperature history to perform structural analysis.

HT analysis provides input files for subsequent structural analysis (mechanical analysis). These files will be used as a thermal load to the specific elements. A part of the script for structural analysis which is defining the specific file for the corresponding element is already written by the module "Device and Elements" (refer section Device and Elements). It finalises the process of mapping data from FDS to OpenSEES.

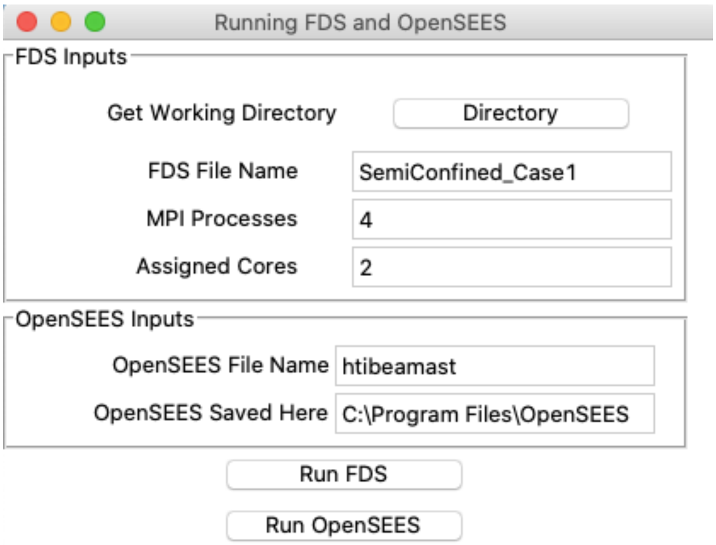


Figure 16:Frame for running FDS and OpenSEES

2. <u>Idealised Fire Models</u>

As discussed in Chapter 2, there are various fire models available to evaluate the structural fire resistance. In OpenSEES, these models can be applied as a boundary condition for thermo-mechanical analysis. This middleware of OpenFIRE (OpenFIRE for Idealized fire) is able to perform heat transfer analysis and provide the results for subsequent structural analysis for idealized fire scenarios. It includes some of the widely accepted fire models such as standard fire, localised fires, natural fire. Additionally, a few models which are suggested to represent the travelling fire are also included in this module. Following paragraph explain how a user can perform structural analysis by choosing a particular fire model. Figure 17 shows the frame of the module (OpenFIRE for Idealized fire).

2.1.Fire models in OpenSEES

In the current version of '**OpenFIRE for Idealised fire**' following models are present which are discussed one by one (Figure 18). In the future, more models will be added in this program.

- Standard fire
- Localised fire
- Localised SFPE
- Natural fire
- Travelling Fire

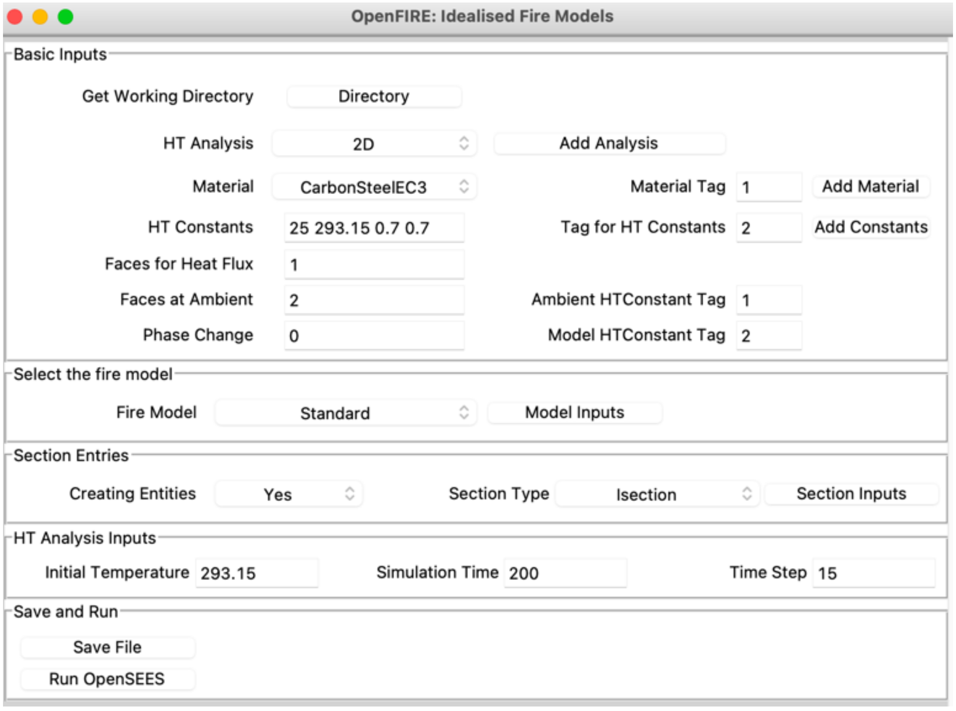


Figure 17:OpenFire for idealized fire models

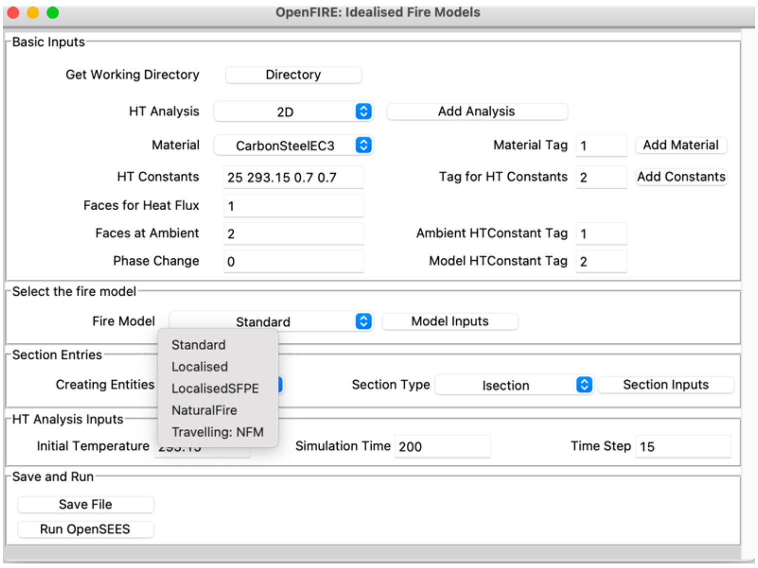


Figure 18: Fire models in the current version of OpenSEES

2.2.Basis Inputs

The foremost thing to do structural analysis is to provide the basic information for the analysis such as material type, heat transfer constants, type of analysis as discussed earlier in this Chapter. Once a directory is selected where the output needs to be saved the other information needs to be filled. This section explains each term present the "Basic Inputs" frame as shown in Figure 17.

Steps

- OpenSEES allows performing both types of analysis (2D and 3D). From the drop menu, type of analysis can be selected. In a 2D analysis, HT entities are independent to one another, while in 3D analysis translational heat transfer is also considered and each entity is connected to the adjacent entity. After choosing the analysis type, "Add Analysis" button will generate an HT input file containing the analysis type and further information will keep appending to this file.
- Type of material can be chosen from the drop menu such as carbon steel, concrete, and assign a "tag" number to each material for identification in the heat transfer analysis.
- Heat flux needs to be applied for HT analysis and calculating structural temperatures therefore the constants for heat flux calculations must be provided such as convective heat transfer coefficient, ambient temperature, emissivity , absorptivity. These values can be written with a 'space' separation between each value.
- When assigning a HT entity for analysis, it is required to assign a name of faces where heat flux boundary is to be applied for the analysis. Figure 9 shows the faces of each entity. Separating each face number with a "space" multiple faces can be defined in the entry box.
- Assign the faces of the boundaries which would be exposed to ambient, and provide a tag number to the values of heat transfer constants assigned at the ambient face.
- Provide '0' for no change in phase and '1' for any phase change (refer section in OpenSEES)
- Now, assign the tag number for the heat transfer constants which is assigned to the faces which are exposed to fire and where the boundary condition from the fire models would be applied.

2.3. Adding Fire Models

2.3.1. Standard Fire

Among fire models, 'Standard Fire' is the most conservative fire scenario where single temperature is applied to the whole structural member within a compartment (refer Chapter 2). When Standard Fire model is chosen, no other inputs are required. Press "Add Model" to save the model in HT script file.

2.3.2. Localised fire model

Once the localised fire model is chosen, it is required to fill inputs for the analysis as shown in Figure 19.

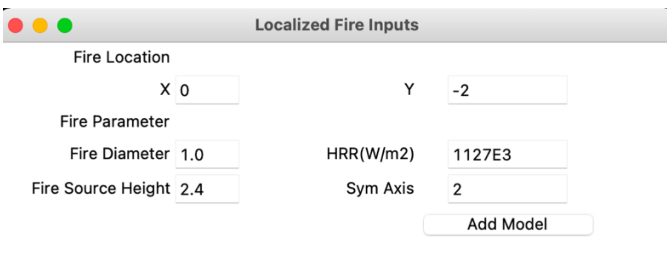


Figure 19: Inputs for localised fires

Steps

- Provide the location of the fire such as values of X and Y. The size of fire is defined as the diameter of the fire [Eurocode and Hasemi]. If the source of fire is not circular, calculate the value of the diameter of fire (D) from Equation X.1.

$$D = \sqrt{\frac{4\pi}{A}}$$

  Where A is the area of the fire.

- The heat generation is defined in terms of heat release rate (HRR) in W/m^2.
- Height is defined as the distance between the structural component (beam or ceiling) from the fire source.
- Symmetrical axis defines the symmetry of the fire. (Need to check)
- Once all this information is provided add model for the analysis.

2.3.3. Localised SFPE model

2.3.4. Natural Fire model

This model is similar to the localised fire model except it requires smoke temperatures as well. It added heat fluxes generated by the smoke as well while performing the heat transfer analysis (Figure 20).

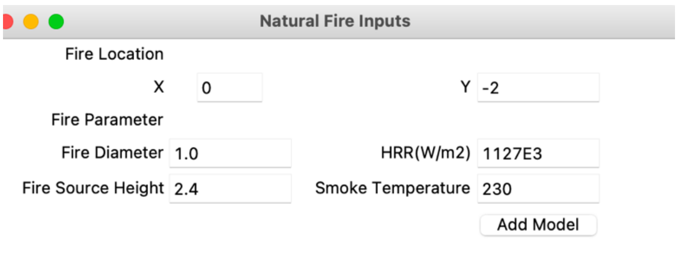


Figure 20: Inputs for natural fire model

2.3.5. Travelling Fire model

To be added

2.4.Section Detail

As discussed earlier, in OpenSEES, heat transfer analysis is performed using various kinds of section that defines structural components such as I-section, block, brick as shown in Figure 9. From the frame of ‘Section Entries’, the type of the entity can be chosen. As shown in Figure 21, the current version of OpenFIRE includes Isection, block, brick and Isection3D (I section for 3D analysis) [Refer website]. Researcher can further add more entities in future.

2.5.Entities details (steps)

- Once section type is chosen, the inputs for an entity type are required to fill in as shown in Figure 22. In the input frame, dimensions and mesh details of the section need to be provided.

- Type of material of the section is also required to be assigned before adding the entity for the analysis.
- To get the output after heat transfer analysis for structural analysis, user needs to define the location where the output is required such as at the face of an entity, or any defined location by providing the coordinates and/or by using *user-defined* location as shown in Figure 22.
- The section details where temperatures field along the depth of an element is required should be provided for structural analysis as shown in Figure 22.
- Press "Make Node Set" button in Nodeset frame, it will the data to the HT analysis input file.
- Now, provide the initial condition such as initial temperature, simulation time and time step for the analysis. The total time of the simulation would the product of the "Simulation Time" and "Time Step".
- Now, before running the analysis save the file from the "Save File" button and then press "Run OpenSEES" to carry out heat transfer analysis. It will generate the output files in the chosen directory.

Similar to Isection, inputs for other section entities can be provided for analysis. It is recommended to refer the website (Opensees for fire) for the details required for each type of entity.

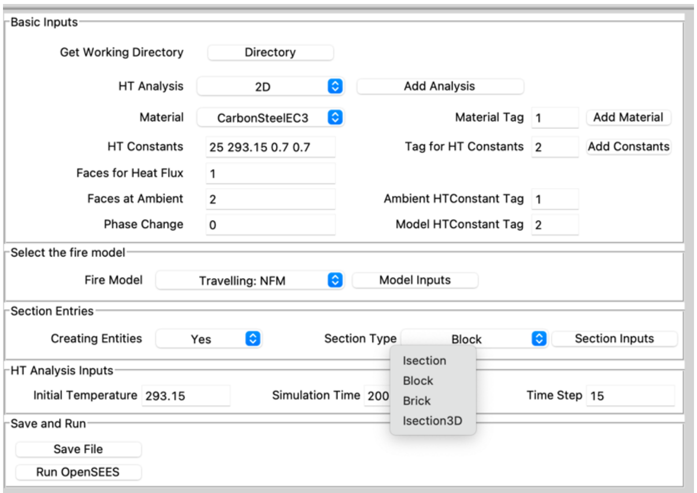


Figure 21: Type of entities

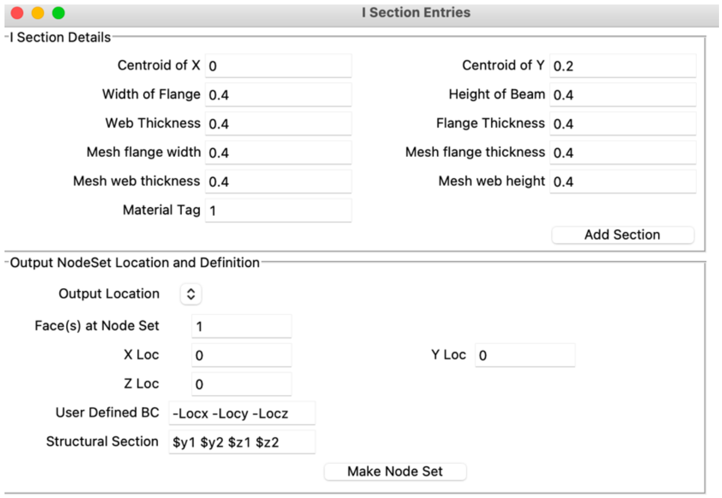


Figure 22: Inputs for a HT entity and creation of node set

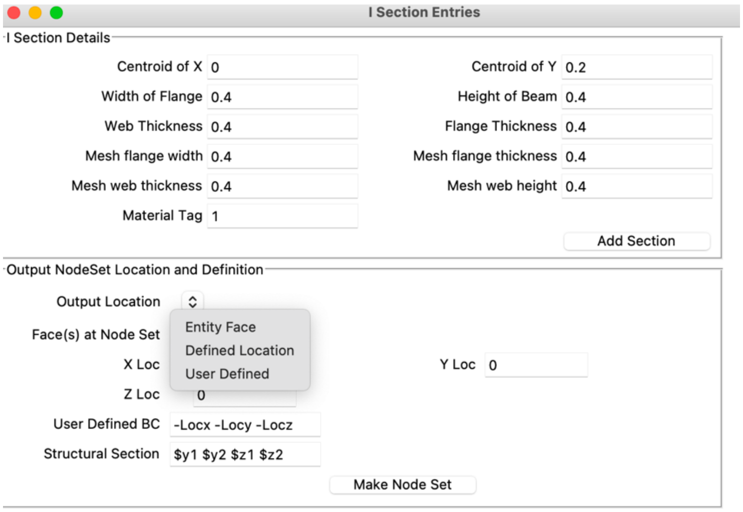


Figure 20: Entity and NodeSet details

Conclusion

OpenFIRE for idealised fire models allows to perform the HT analysis directly because these fire models are already embedded in OpenSEES.