**Group 1:**

* Create Section Property

The first functionality of the code is the creation of section properties in the format that the new SAFIR preprocessor (GmSAFIR) demands, by giving only the dimensions of the section properties. The section properties are created automatically respecting the center of gravity in order to be in the correct form for the mechanical analysis. The available cross sections are the following: I, Tee, Cruciform, Angle, Double Angle, Channel, Double Channel, Cruciform with 2 Angles and Cruciform with 4 Angles. Additionally, a library for the HEA, HEB, IPE profiles is provided.

* Transform FDS output to .txt files

Here, we select one .csv file from the FDS analysis that contains the temperature in various Devices. The GUI produces automatically equal number of .txt files with the number of Devices. Each file is given the name of the Device and contains the time – temperature data in the format that SAFIR supports.

* Post process temperature files

The division of the time in the FDS analysis is different from the way that SAFIR works. In this functionality we select multiple .txt files that are produced with the previous procedure and given a specific total time domain and the desired time step, new files are produced considering a simple linear regression for the time – temperature data.

* Given a prototype .in file, create same.in files with different time-temperature curve

Here we produce .in files (SAFIR input for thermal analysis) with same section properties (material, convection coefficient, etc.) with a prototype .in file and the only difference is the time-temperature curve. First, we select the prototype .in file, then the .txt file of the prototype .in file and then the rest .txt files of the section properties that we want to produce. Important notice: Always select first the .txt file of the prototype simulation, necessary in order to know which data we want to change.

* Temperature evaluation in structural point with weight coefficients

Here we select a .csv file that contains the devices’ ID that are referred to one structural point and the weight coefficients. Then, we select the .txt files with the corresponding time-temperature curve of each device. The GUI will generate a .txt file with the evaluation of the time temperature curve in the specific xyz coordinates of the structural point. Details for the generation of the .csv file are given in the description of the next Tab (Mechanical analysis).

**Group 2:**

This Tab provides options to import the geometry from the FEM software to FDS format and create Devices in critical locations of a frame structure. Initially, we need a .geo file where the structure is created in the local coordinate system. Two tools to rotate and transport the structure are provided, in order to define the structure to the global coordinate system of the simulation in Pyrosim – FDS. The button “create devices in FDS” opens a pop-up window where we must insert the FDS file where we have formulated the CFD problem. Then three options are available for creating the devices for monitoring the temperature:

* Critical length for creating the devices

By default, the devices will be created in the nodes (start-end) of each frame and in the middle. Here we have the option to create more devices in the frame by selecting a critical length. The algorithm divides the frame in equal parts considering the value of the critical length. By selecting the required quantity (“Temperature or Flux”) the devices are inserted in the FDS file in the correct format.

* Choose a member and create device in each cell

With this option we can choose a specific frame member and create devices along the member in distances equal with the cells of the CFD mesh that it belongs. Particularly, it identifies the starting point in which mesh it belongs and make the calculation considering this mesh.

* Choose a node to create devices for a structural point with weight coefficients

Lastly, if the structural point doesn’t coincide with the center of the cell (CFD mesh) and we want a more accurate evaluation of the temperature in the structure we can use this function. We select a specific node of the structure, the GUI detects in which mesh of the FDS simulation it belongs, it creates *n* nodes around the structural point and according to their distance with the structural node, *n* weight coefficients are evaluated (sum = 1). The number *n* depends on the position of the structural point, more details are provided in the technical manual. The software produces a .csv file with the ID of the new devices and their weight coefficients and writes the same devices in the .fds file. The ID of the new devices has a specific format where the first characters until the first underscore (\_) are the ID of the initial structural point. In this way when we will use the functionality of the first Tab to evaluate the temperature in the exact point location, we can easily detect how many devices are generated for the selected structural point.