This files explains the different steps that have to be performed to generate the input files that are necessary for a 3D structural analysis, starting from existing input files that have been previously defined for a 2D analysis

Step 1 : calculate the torsional stiffness in the section

We assume that the starting point is an input file created to calculate the temperatures, say C2T.IN.

1. Copy the existing C2T.IN file and rename it in C2TOR.IN
2. In this new file, change the comment lines (optional)
3. In this new file, replace the commands

```
  TEMPERAT
     TETA       0.9
  TINITIAL     20.0
  MAKE.TEM
by the command
   TORSION
```

4. In this new file, change the name of the file which will be created, for example, replace "c2t.tem" by "c2tor.tor"
5. Delete all the `FRONTIER` and related commands (there is no fire curve applied here). The `SYMMETRY` command now follows immediately the definition of the elements.
6. Delete the `TIME` …. `ENDTIME` commands (there is no need of time steps because the calculation is done only once).
7. Use of symmetries
    a. If the whole section is modelled (no symmetry is used), IT IS ABSOLUTELY REQUIRED to fix the value of the solution to 0 in at least one node of the axis of symmetry. If not, the torsional stiffness will be correctly evaluated, but the warping function at all nodes will be offset by an arbitrary value and this will create amazing results in torsion during the mechanical analysis.
    b. `YSYM` cannot be used in a 3D problem. It means that, if the temperatures where previously calculated using the `YSYM` command because the .TEM file was to be used in a 2D mechanical analysis, it is now necessary to recalculate the temperatures and to replace this `YSYM` command by a `REALSYM` command.
    c. If one part of the section only is discretised and 1 or 2 `REALSYM` commands are used, it is necessary to fix the value of the solution to 0 at all the nodes which are on the axes of symmetry. If there is another axis of symmetry in the solution, but this axis is not used in a `REALSYM` command, the values of the solution must also be fixed to 0 on this axis. This is the example of a square section if ¼ of the section is discretised, then the solution must also be fixed to 0 on the diagonal of the square section.

8. In the MATERIAL command, you must enter the material mechanical properties for each material (the same properties as the one you will use in the mechanical analysis. Don't forget the Poison's ratio).

9. Run SAFIR with this new file. The CPU time is extremely short. The end of the .OUT file looks like this:

```
  G * Ip =   27259615.3846154
According to the principle of virtual works,
     GJ = 0.919990E+08

EIw = 0.856407E+07

 THE STIFFNESS MATRIX HAS BEEN TRIANGULARISED     1 TIMES.
```

GJ is the elastic torsional stiffness of the section at room temperature. In reality, this value will decrease because of cracking even at room temperature (may-be not so much for a column loaded in compression) and will decrease even further during the fire because of the temperature increase and the subsequent decrease in material stiffness. it is not possible to calculate this evolution as a function of time in SAFIR. Only a constant value can be entered. The stiffness can, for instance, be divided by an arbitrary factor of 2. This has to be made in the .TEM file. This means that the simulation is valid provided that torsion is not the may load path. This is not so restrictive in most usual building elements. This would be restrictive in, for example, a cantilever beam submitted to a pure torsion moment.

Step 2. Introduce the torsion properties in the .TEM file.

1) Open the .TEM file which had been created during the temperature analysis and open the new file which has been created during this torsion analysis, "c2tor.tor" in this case. They look very similar (the number of fibres, NFIBERBEAM, must be the same in the 2 files), with the difference that the value of the warping function and of the torsional stiffness is present in the torsion file.
2) From the .TOR file, copy the group of lines starting with

```
          W
```

and finishing with

```
According to the principle of virtual works,
     GJ = 0.919990E+08
```

3) Insert this group of lines in the .TEM file just before this line

```
HOT
```

The .TEM file is now ready to be used in a 3D mechanical analysis. If an YSYM command was used when the file was to be used in a 2D calculation, the number of points has now been multiplied by 2, and the information on torsion is now present in the file.

Step 3. Modify the IN file for the mechanical analysis.

1) Copy and rename the .IN file, for example C2M.IN into C3M.IN
2) If a first step, don't modify the `NPTTOT` value. SAFIR will later let you know to which value it has to be increased. The same holds for `LARGEUR11` and `LARGEUR12`.
3) Even if one single member, i.e. a beam or a column, is analysed, the number of nodes has to be increased by 1, here from 21 to 22 (see section 4.2 of the user's manual to see why this node is needed and how you can define the coordinates). In more complex structures, it is possible that more additional nodes are needed.
4) `NDIM` goes from 2 to 3.
5) `NDDLMAX` goes from 3 to 7.
6) The nodes which had 3 DoF have now 7 DoF, see the `FROM` command.
7) Add a `FROM` command which says that the node(s) that you have added, see point 3, has 0 DoF.
8) The number of fibres has to be adapted in the `NFIBERS` command (probably multiply it by 2 if you previously used an `YSYM` command in the thermal calculation).
9) The nodes which were previously present have to be given a third coordinate. In order to have a traditional view with DIAMOND, it is preferable that the new co-ordinate be the Y co-ordinate, see table below. Giving 0 to all these Y coordinates means no initial imperfection in this direction.

2D	=>	3D
X	=>	X
-	=>	Y
Y	=>	Z

10) Give the coordinates to the node(s) that you have added, see point 3
11) Modify the `BLOCK` commands taking into account the new DoF's. they go in the order: X, Y, Z, tetaX, tetaY, tetaZ, warping. Note that 1 line is needed for each blocked node, and 1 blank line at the end of the series.
12) Add the fourth node (the one that you have added) to every beam finite element.
13) Modify the loads in order to reflect the new DoF's. Note that DISTRBEAM have 3 components, pX, pY and pZ. NODELOAD have 6 components, FX, FY, FZ, MX, MY, MZ.

Step 4 : run SAFIR

You may have to adapt some values for the dimensions of the arrays, but SAFIR will let you know. When using Diamond, don't forget to use the TOOL-ViewPoint Tool.