SYRTHES 5
Tutorial

I. Rupp, C. Peniguel

2018

AVERTISSEMENT / CAUTION

Summary

bg

Chapter 1

Getting started with anisotropic conduction

3disks3d

1.1 What is the problem ?

We would like to compute the temperature field inside 3 disks heated in their center. According the disk considered, conductivity is isotropic, orthotropic or anisotropic.

1.1.1 Geometrical description

The solid domain constists in three separated disks with a hole in the center.
Radius of internal holes is 0.05 m and the radius of each disk is 0.4 m.
Geometrical characteristics are shown on figure 5.1

1.1.2 Physical description

Density and specific heat are considered identical for the three disks and set to : $\rho = 7700\ kg/m^3$ and $C_p = 460\ J/kg°C$ (that could correspond to steel).

Each disk is affected by a different conductivity :

- For the disk 1, conductivity is isotropic : 25 $W/m/°C$
- For the disk 2, conductivity is orthotropic : 25 $W/m/°C$ along the direction x, 5 $W/m/°C$ along direction y, and 25 $W/m/°C$ along direction z
- For the disk 3, conductivy is anisotropic : $(25,\ 5,\ 25)$ $W/m/°C$ along the axis of a local system of coordinates being at a 45° with respect to the reference system of coordinates.

1.1.3 Initial conditions and boundary conditions

The initial temperature is $20°C$. Boundary conditions are :

- center of the disks : T = $50°C$, h = 1000 $W/m^2/°C$
- other surfaces : adiabatic

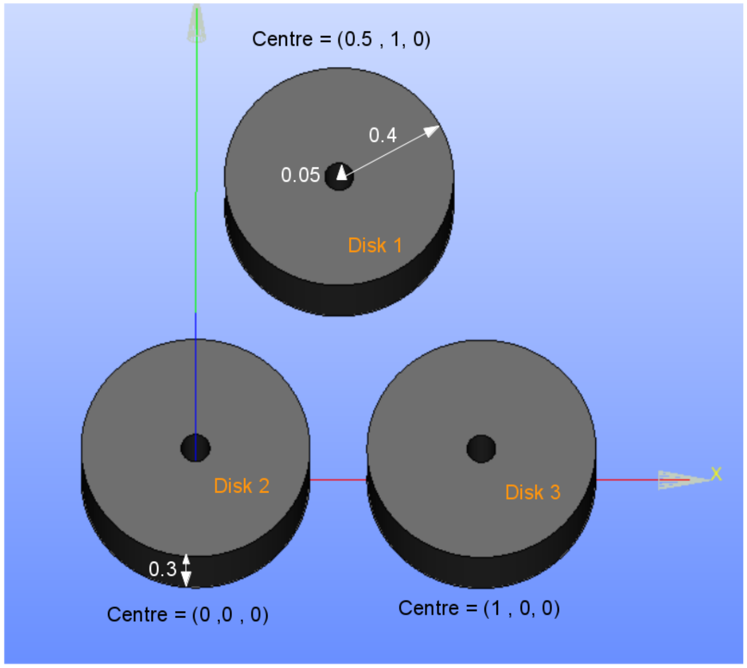

FIGURE 1.1 – Sketch of the problem

1.2 How to do that ?

1.2.1 To organize the study

We propose in this section an organization for the different files of your study. This is only advice, but advanced users can do as they wish...

- create a new directory for your study : `mkdir cas_3disks3d`
- go inside : `cd cas_3disks3d`
- create a new directory for the creation of the mesh : `mkdir salome`

1.2.2 Creating mesh

If the geometry stays very simple and do not create any trouble, attention must however be paid to the references allowing to identify materials and boundary conditions.
We used SALOME to define the geometry and create the mesh.

In the directory `mkdir cas_3disks3d/salome`, run SALOME : `/.../runAppli` (the command is depending on your local installation of SALOME. Your are ready to create your mesh. Save your SALOME-study and export your mesh to MED format in this directory.

In order to define the different boundary conditions and material properties, groups of faces and edges have been created.
For that specific case, conduction mesh counts 23667 nodes and 120722 elements (4-nodes tetrahedra).

You can create your own mesh, but below, are described the characteristics of the mesh provided in the SYRTHES distribution :
`/..../syrthes5.x/arch/`*your_ arch*`/share/syrthes/tests/1-cas_3disks3d`.

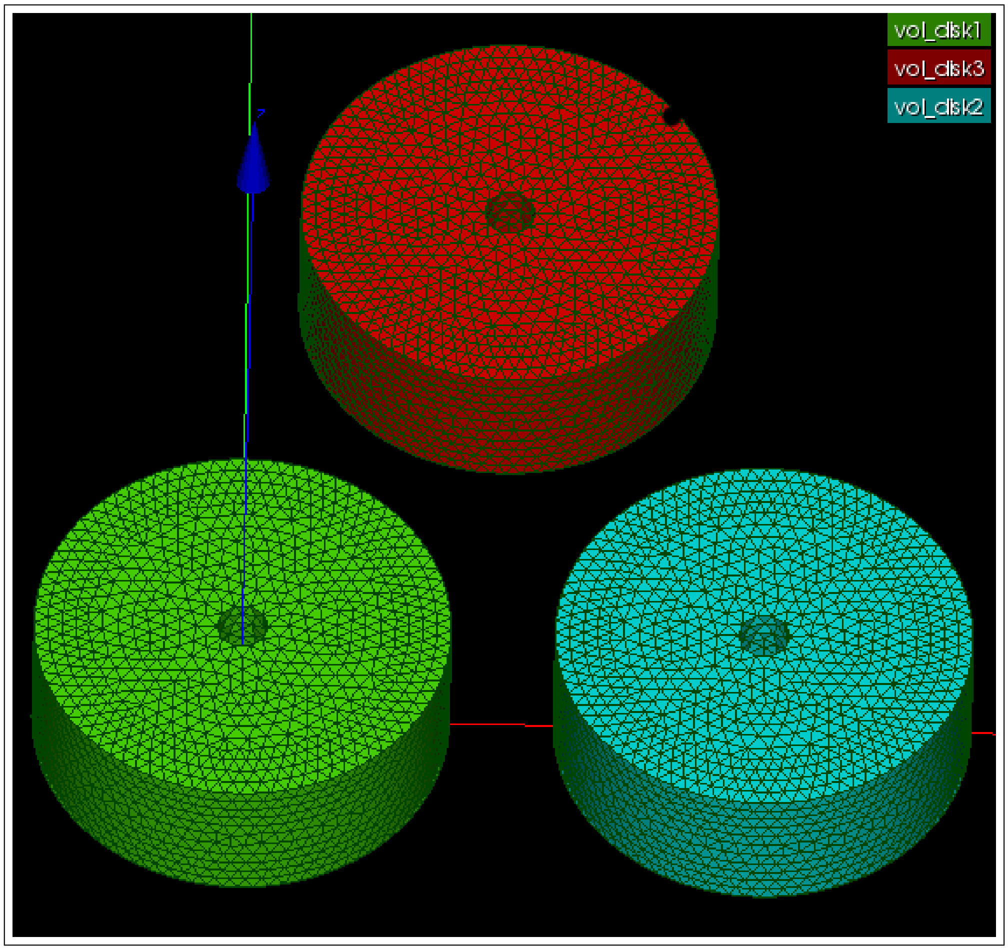

FIGURE 1.2 – Group names for volumes

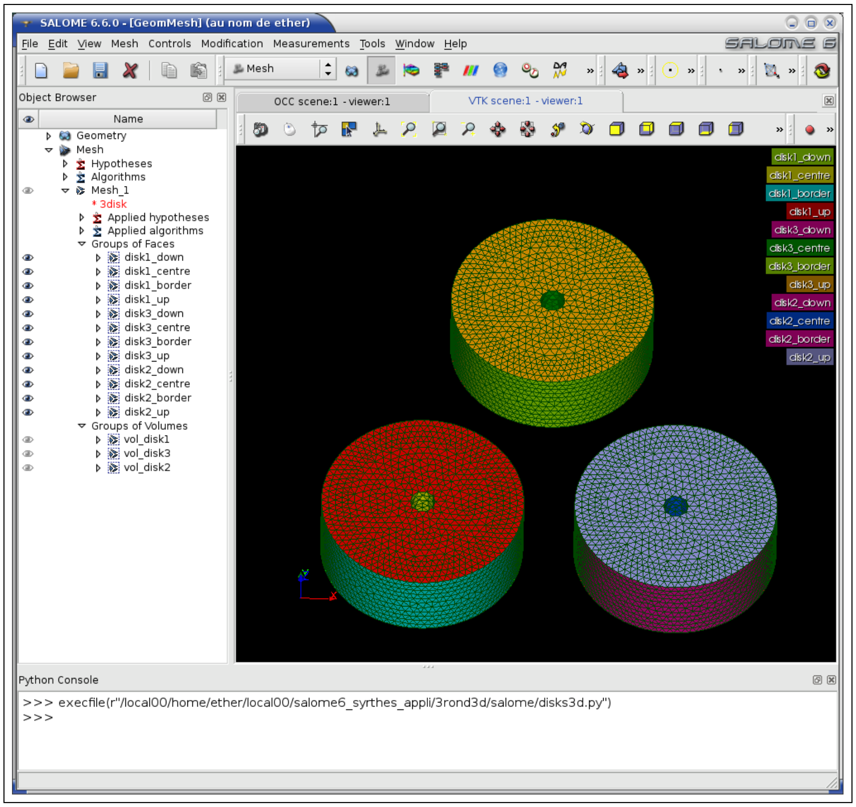

FIGURE 1.3 – Group names for surfaces

1.2.3 Create your SYRTHES-study

Go back to the initial directory : `cas_3disks3d`
If not already done : source the SYRTHES environment (Linux only) :
`source /.../syrthes5.0.1/arch/myarch/bin/syrthes.profile`
Run the SYRTHES-gui : `syrthes.gui`
Create a new case : `syrthes`

Now, all your calculation will be managed by the SYRTHES Graphic User Interface.

1.2.4 Main view

Give a title to your study. The dimension of the problem is 3D.
Save your data file :

1.2.5 File Names

- Click on the next item in the menu on the left : File Names
- Select your conduction mesh : `cas_3disks3d/salome/3disks.med`

A conversion of the file format is done automatically and you should get the message :

FIGURE 1.4 – SYRTHES Managing your cases

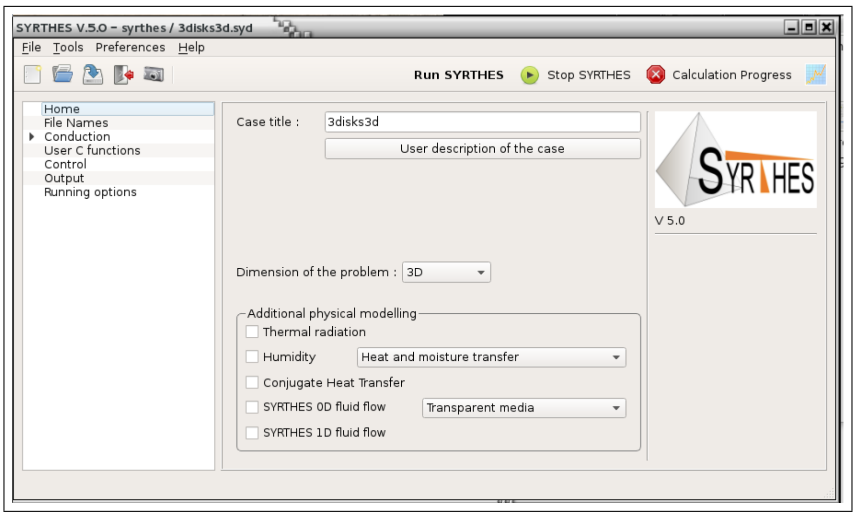

FIGURE 1.5 – SYRTHES Main View

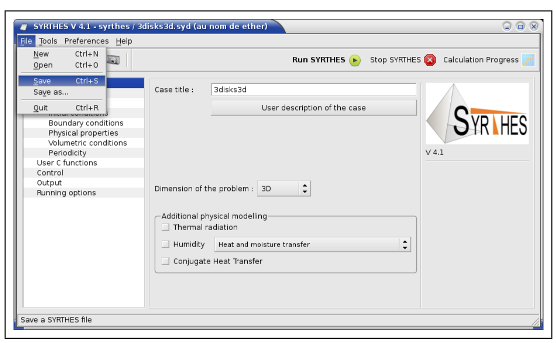

FIGURE 1.6 – SYRTHES Save your data file

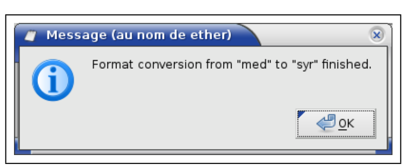

FIGURE 1.7 – SYRTHES File format conversion OK

Finally, give a name for your results files (a name without extension; SYRTHES will create different files with the same radical but different extensions depending on the type of files).

The file Names looks as shown below :

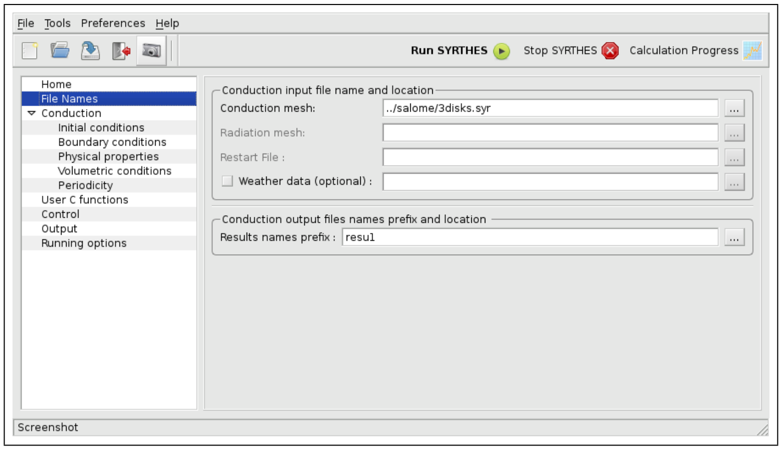

FIGURE 1.8 – SYRTHES File Names window

1.2.6 Input data for conduction computation

1.2.6.1 Group names and references numbers

While SALOME is using group names to distinguish different parts in the mesh, SYRTHES is using reference numbers. Group names and reference numbers are included in the mesh file `3disks.med`. The links between both are given in an additional file (.syr_descr) created while converting the MED file to the SYRTHES file. You can open this ASCII in the SYRTHES GUI : menu “Tools” and “Open Desc”.

WARNING : depending on SALOME version, pairs (group-name , number) could change. So make sure you have a look on the description file and adapt numbers before proceeding.

Here is `3disks.syr_desc` provided file :

```
group_of_faces    10    disk1_centre
group_of_faces    11    disk1_border
group_of_faces    12    disk1_up
group_of_faces    13    disk3_down
group_of_faces    14    disk3_centre
group_of_faces    15    disk3_border
group_of_faces    16    disk3_up
group_of_faces    17    disk2_down
group_of_faces    18    disk2_centre
group_of_faces    19    disk2_border
group_of_faces    20    disk2_up
group_of_volumes    6    vol_disk1
group_of_volumes    7    vol_disk3
group_of_volumes    8    vol_disk2
group_of_faces    9    disk1_down
```

1.2.6.2 Initial conditions

Unroll the conduction menu by clicking on the arrow, and select the first item : Initial conditions

Set the initial temperature ($20°C$), and the list of volumes considered (6 7 8). If all the volumes are affected by the same initial condition, you can put "-1" instead of an explicit list. You can add a comment in the last column (optional).

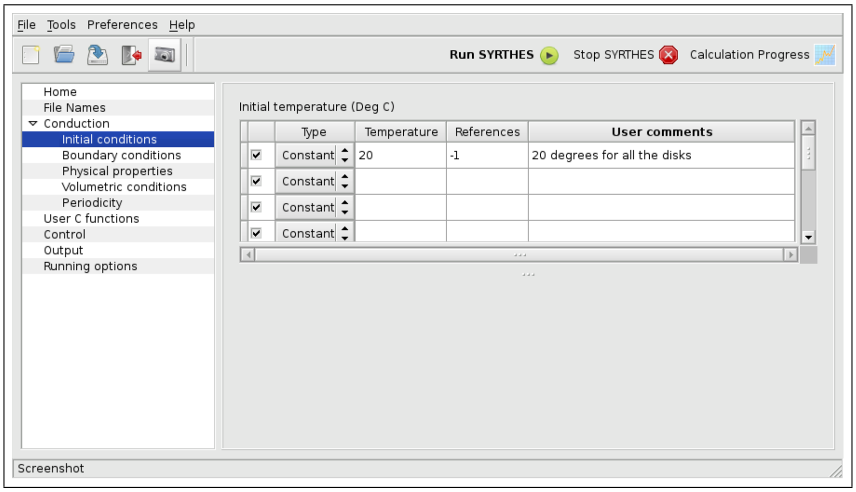

FIGURE 1.9 – SYRTHES - Initial conditions

1.2.6.3 Boundary conditions

We now want to set an heat exchange coefficient at the center of the 3 disks. The values of the boundary condition are the same for the 3 disks, therefore you can define it with only 1 line.

- External temperature $= 50°C$
- Heat exchange coefficient $= 1000\ W/m^2/°C$
- References of the faces concerned : 10 18 14 (disk1_centre,disk2_centre,disk3_centr,)

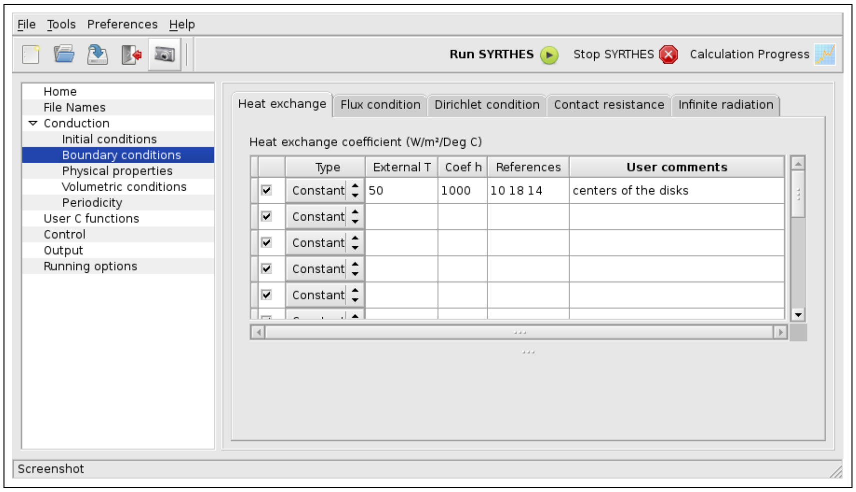

FIGURE 1.10 – SYRTHES - Boundary conditions

1.2.6.4 Physical properties

The disks have different material properties.
For the first disk, conductivity is isotropic : click the Isotropic tab and set the material properties. The elements refererence is 6 (group vol_disk1) for the disk 1.

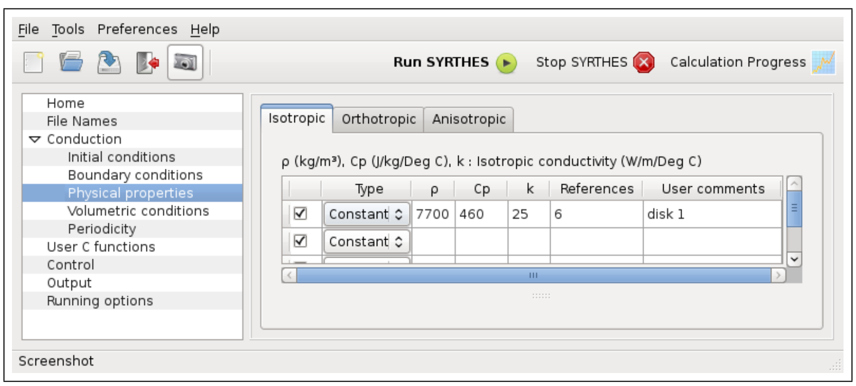

FIGURE 1.11 – SYRTHES - Isotropic conductivity

For the second disk, conductivity is orthotropic : click the Orthotropic tab and set the material properties. The elements reference is 7 (group vol_disk2) for the disk 2.
For the last disk, conductivity is anisotropic : click the Anisotropic tab and set the material properties. The elements refererence is 8 (group vol_disk3) for the disk 3.
In this case we have to define the values of the conductivity in a local system of coordinates.

Now all physical parameters are defined, for this simple case you can jump directly to the Control Window.

1.2.7 Control

We have to set the time step since we are dealing with a thermal transient (the converged solution has no interest, indead all disks will reach a uniform temperature). The time step has to stay reasonnable, if a fair precision is required during the transient, indeed the time error is more or less proportionnal to the time step retained. In the present case, and considering the mesh, a time step from 10s to 100s seems

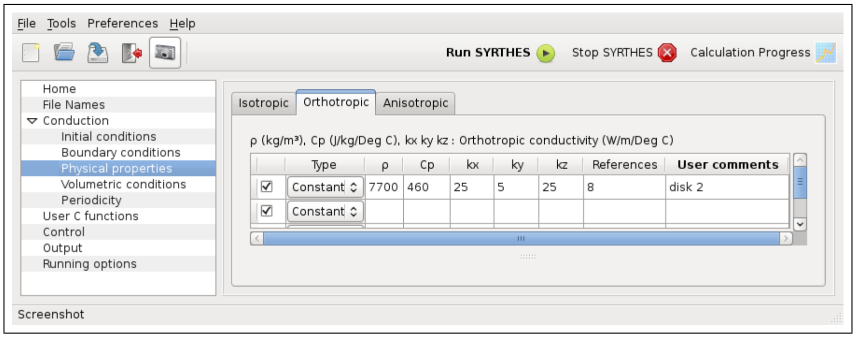

FIGURE 1.12 – SYRTHES - Orthotropic conductivity

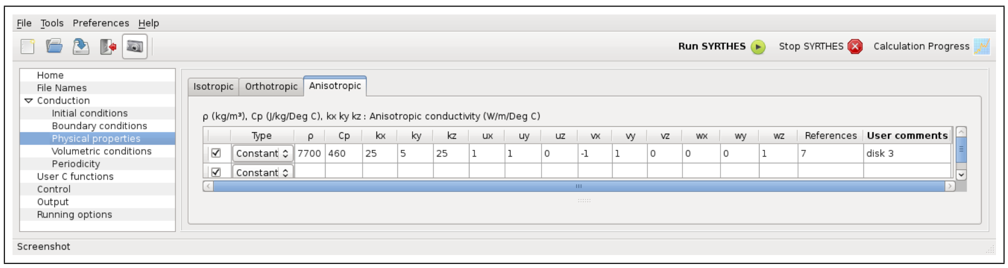

FIGURE 1.13 – SYRTHES - Anisotropic conductivity

reasonnable.
For your first run, you can compute 100 time steps. An average time step for this case is around 50 seconds.

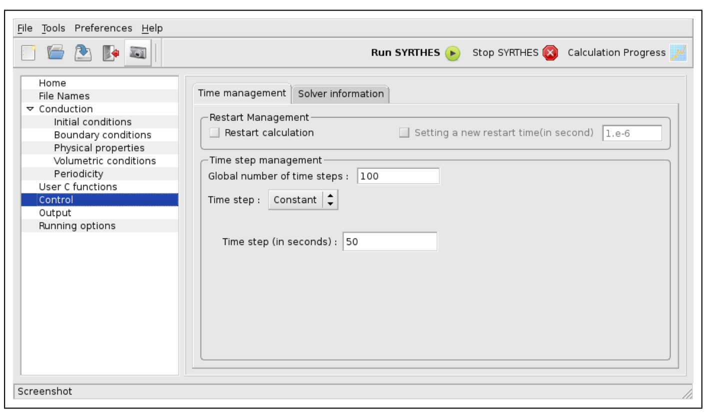

FIGURE 1.14 – SYRTHES - Control window

Generally, there is no need to change the default values provided for the solver.

1.2.8 Output

In this section, we are going to define the type of results to be generated by SYRTHES.
Whatever the options you will have a result file with the temperature on all nodes of the mesh.
But, sometimes, it is also advisable to define some thermal probes : during the run, you will be able to follow the temperature evolution at some strategic points of your domain.
Set the coordinates of points in your domain. Here we have selected just 1 point per disk.
Then, define the frequency you want to save the values of the probes on file : here, 1 means " every time step".

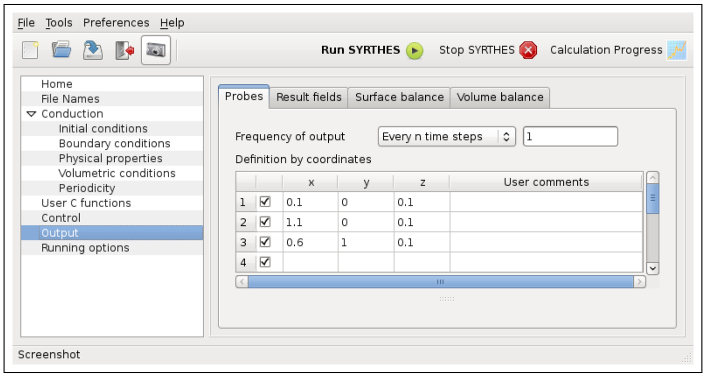

FIGURE 1.15 – SYRTHES - Defining thermal probes

Click on the next tab in the same window (Control) to select "result fields".
At the end of the calculation, you will get of course the temperature field corresponding to the last time step. But, you can also get intermediate results with a choosen frequency. Click "Fields" and set the frequency to 25 : you will get the temperature every 25 time steps.
These temperature fields will be recorded in the ".rdt" file and will be post-processed like the final result (".res" file).

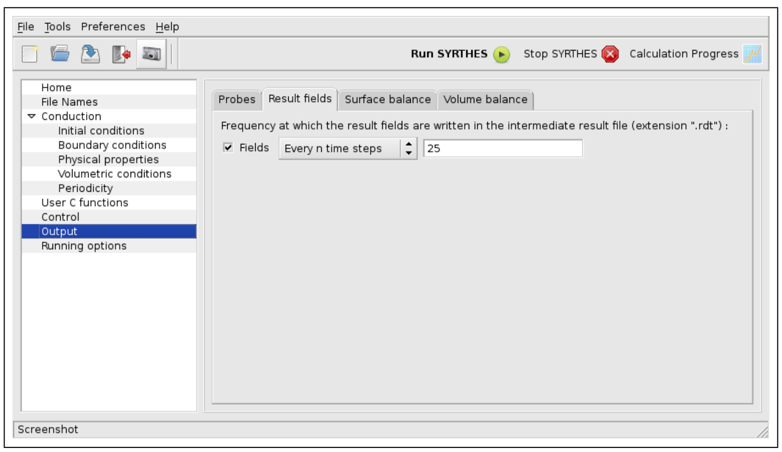

FIGURE 1.16 – SYRTHES - Defining transient result file

1.2.9 Running options

This is the last step : give a name for the listing file. SYRTHES will give you some information about the calculation (option summary, solver convergence,...)
Furthermore, according the kind of computer at your disposal and how big is the case, you can try a parallel computation, setting the number of processors to 3 or 4 (but here, as the element number of this mesh is very low, the benefit in term of CPU time will be very low).

1.2.10 You're ready to run SYRTHES !

Click "Run SYRTHES", the "calculation progress" window appears and you can display the evolution of the temperature at the 3 points defined previously.
The 3 tabs display :

- Listing file browser : displays the 200 last lines of the listings
- Listing file editor : whole SYRTHES listing file

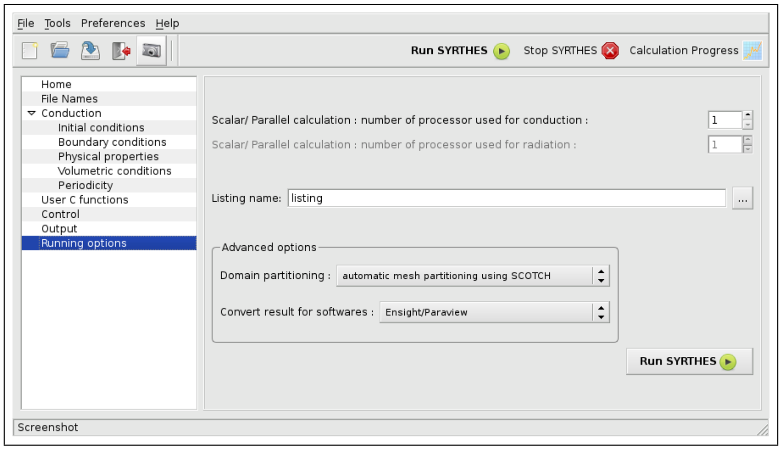

FIGURE 1.17 – SYRTHES - Running options window

- Log : system messages (usefull in case of error during user files compilation)

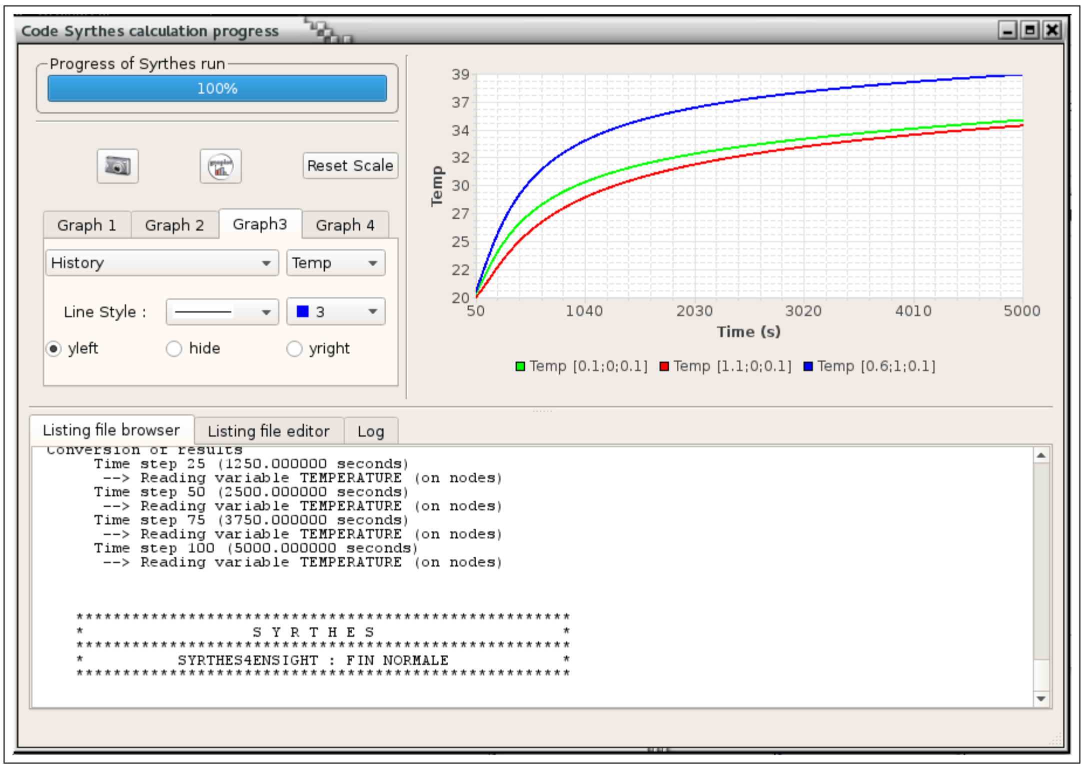

FIGURE 1.18 – SYRTHES Calculation progress window

1.2.11 Analyzing the results

When SYRTHES calculation ends, you can visualize the results using a post-processor. For this example, we are using the opensource sofware Paraview, but you can also use Ensight.

1.2.11.1 Final temperature field

Run paraview and open the file : `cas_3disks3d/syrthes/POST/resu1.ensight.case` It is interesting

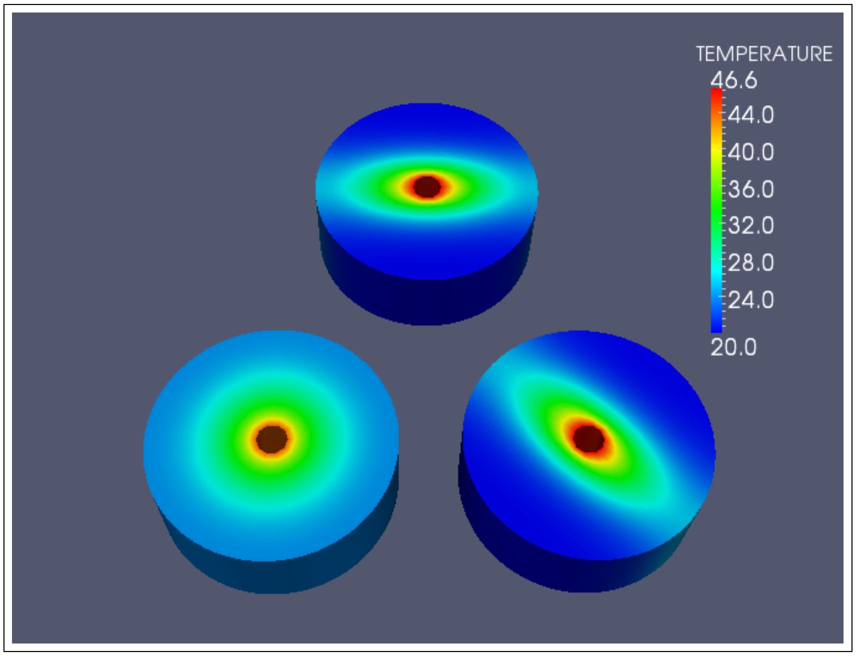

FIGURE 1.19 – Temperature field after 5000 s

to note the behaviour difference when using different material physical properties. The isothermals of the disk 1 stay concentric, while for the others, ellipses appear before being affected by the outer boundary conditions. One may underline that in the fully anisotropic case, the ellipses axis are not aligned with the axis of the reference system of coordinates.

1.2.11.2 Transient temperature field

As we selected transient result file (see figure 1.16), we can also post-process this file.
Run paraview (or ensight) and open the file : `mkdir cas_3disks3d/syrthes/POST/resu1_rdt.ensight.case`.
Then the temperature field can be visualized at different times : 1250 s, 2500 s, 3750 s, and finally 5000 s.

1.2.11.3 Result files

At the end of the run, in the directory where the SYRTHES calculation has taken place, one should be able to find several files :

- `resu1.res` : final result file
  Temperature field at the end of the calculation. This file is automaticaly converted to a post-processor file format (Ensight/Paraview or SALOME-MED) in the `POST` directory.
- `resu1.rdt` : transient result file Temperature field during the transient. This is an optional file (depending on the output options defined). If existing, it is also automatically converted to a post-processor file format (Ensight/Paraview or SALOME-MED) in the `POST` directory.

- `resu1.his` : temperature probes
  This file contains the temperature at the probes locations initially defined. This is an "in columns" kind of file.
  — column 1 : time
  — column 2 : temperature
  — columns 3-5 : coordinates (only columns 3-4 in 2D)
  The values can also be extracted automatically through an option of the SYRTHES GUI. To help users, a gnuplot command file is also automatically generated.
- `resu1.mnx` : min-max values
  The min and max values of each variable of the calculation and the place where these maximum values are reached during the transient. This is an "in columns" file. Content depends on the type of calculation. The content is described at the beginning of the file.
  Each variable is corresponding to a column. This file can be post-processed with a 1D plotter.
  Example :

```
# 1=temps  2=T_min    3=x   4=y   5=z   6=T_max     7=x   8=y   9=z
#         10=rho_min 11=x  12=y  13=z  14=rho_max  15=x  16=y  17=z }
```

  In this file, you will find :
  — column 1 : time
  — column 2 : minimum of temperature
  — columns 3-5 : coordinates where is located the minimum of temperature
  — columns 6 : maximum of temperature
  — columns 7-9 : coordinates where is located the maximum of temperature
  — column 10 : minimum of density
  — columns 11-13 : coordinates where is located the minimum of density
  — columns 14 : maximum of density
  — columns 15-16 : coordinates where is located the maximum of density
- `resu1.add` : additionnal file This file is unused in this study. In other cases, you can calculate some specific entities and save them in this file for further graphical post-processing.

Chapter 2

Conduction and radiation in an oven

plates3d

2.1 Description of the problem ?

We want to estimate the temperature field inside an oven in which 7 disks or rings are placed. This purely theorical oven is supposed to be heated at the top while the bottom is maintained at $20°C$. 3 different materials have been used :

- steel for the oven,
- copper for the rings,
- granite for the disks.

2.1.1 Geometrical description

The solid domain is constituted of an oven and inside, 4 granite disks and 3 copper rings. The oven is 1.3 m high for a diameter of 0.59 m.
Geometrical characteristics are shown on figure 5.1

2.1.2 Physical description

The physical characteristics of the materials are :

- steel : $\rho = 7700\ kg/m^3$, $C\_p = 460\ J/kg/°C$, $k = 25\ W/m/°C$
- copper : $\rho = 8900\ kg/m^3$, $C\_p = 385\ J/kg°C$, $k = 390\ W/m/°C$
- granite : $\rho = 2500\ kg/m^3$, $C\_p = 790\ J/kg°C$, $k = 2.2\ W/m/°C$

2.1.3 Initial conditions and boundary conditions

The initial temperature is $20°C$. Boundary conditions are :

- bottom of the oven : T = $20°C$
- top of the oven : T = $800°C$, h = 1000 $W/m^2/°C$
- other extern surfaces of the oven : adiabatic
- all the surfaces inside the oven : radiation exchange

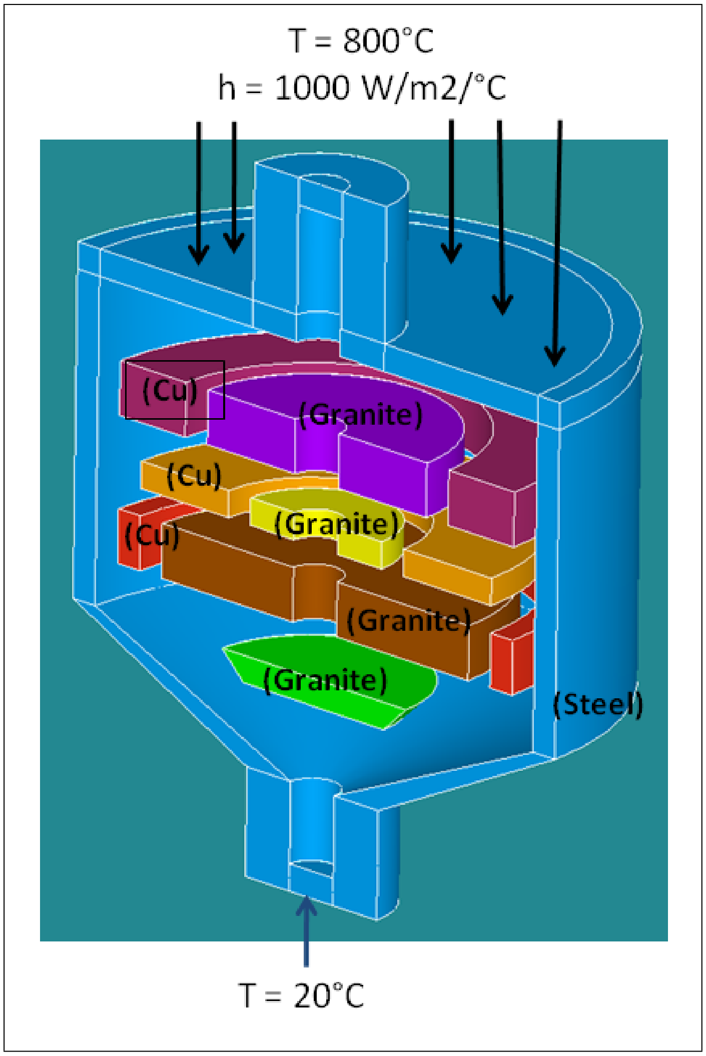

FIGURE 2.1 – Sketch of the problem

2.2 How to proceed ?

2.2.1 To organize the study

We propose in this section a possible organization for the different files of your study. This is only advice, and more expert users may do as they wish...

- create a new directory for your study : `mkdir plates3d`
- go inside : `cd plates3d`
- create a new directory for the creation of the mesh : `mkdir salome`

2.2.2 Creating the conduction mesh

We used Salome to define the geometry and create the mesh.

In the directory `mkdir plates3d/salome`, run Salome : `/.../runAppli` (the command is depending on your local installation of Salome. Your are ready to create your mesh. Save your Salome-study and export your mesh to MED format in this directory.

In order to define the different boundary conditions and material properties, groups of volumes and faces have been created.

Here, the conduction mesh counts 186024 elements (4-nodes tetrahedra) and 43152 nodes.
The radiation mesh counts 3440 faces (3-nodes triangles).

You can create your own mesh, but below, we describe the characteritics of the mesh provided in the syrthes distribution :
`/..../syrthes5.x/arch`/*your_ arch*/`share/syrthes/tests/2-cas_plates3d`.

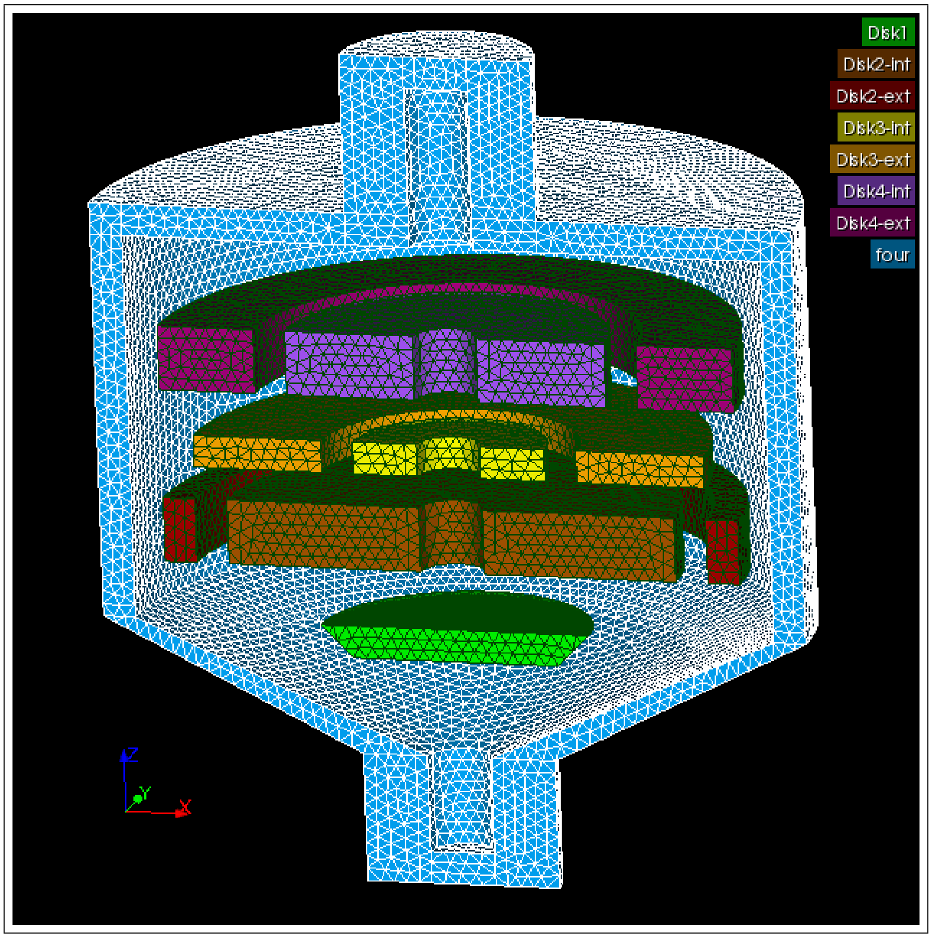

Figure 2.2 – Group names for volumes

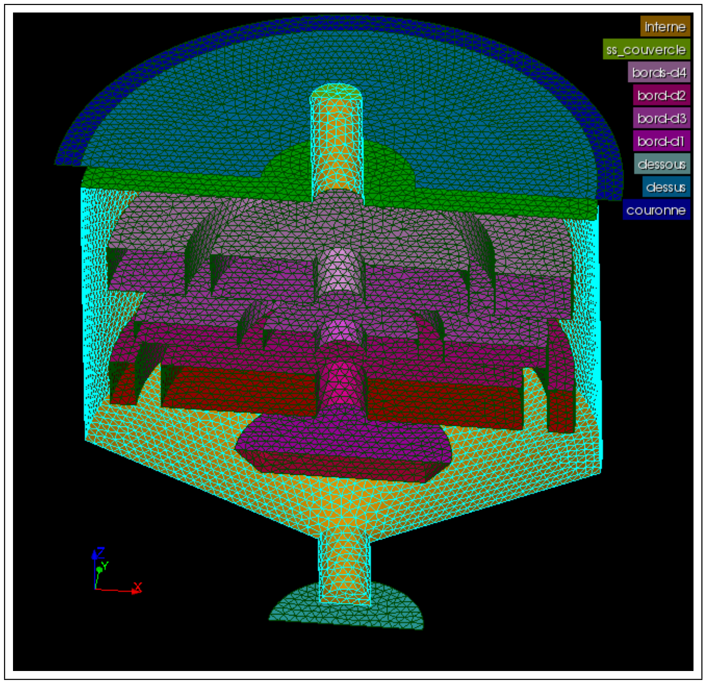

FIGURE 2.3 – Group names for surfaces

2.2.3 Creating the radiation mesh

The radiation mesh consists in inner surfaces of the oven and surfaces of all the disks and rings. We have created 2 groups of faces to separate the granite with an emissivity of 0.8 and steel and copper with an emissivity of 0.96.

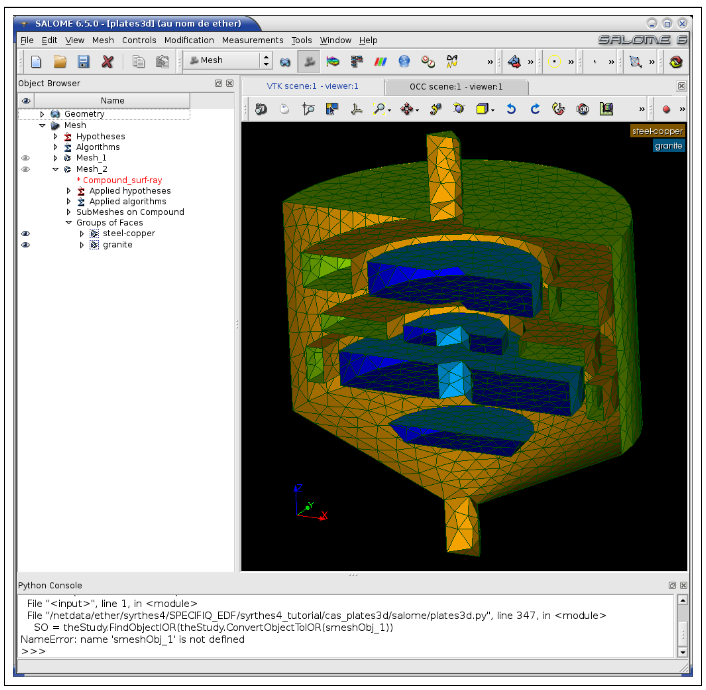

FIGURE 2.4 – Radiation mesh

2.2.4 Create your SYRTHES-study

Go back to the initial directory : `plates3d`
If not already done : source the SYRTHES environment (Linux only) :
`source /.../syrthes4.1/arch/myarch/bin/syrthes.profile`
Run the SYRTHES-gui : `syrthes.gui`

FIGURE 2.5 – SYRTHES Managing your cases

Create a new case : `syrthes` Now, all your calculation will be managed by the SYRTHES Graphic User Interface.

2.2.5 Main view

Give a title to your study. The dimension of the problem is set to 3D.
Click the "Thermal radiation" button to activate the radiation module.

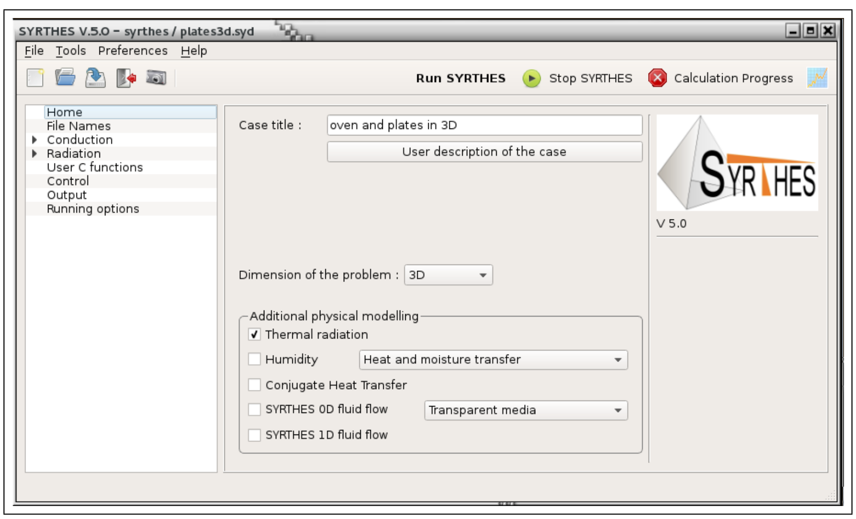

FIGURE 2.6 – SYRTHES Main View

Save your data file (either using the main menu or the icon) :

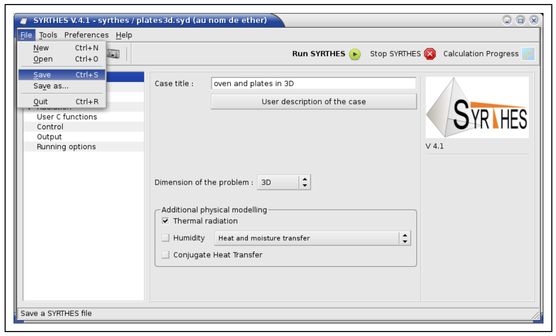

FIGURE 2.7 – SYRTHES Save your data file

2.2.6 File Names

- Click on the next item in the menu on the left : File Names
- Select your conduction mesh : `plates3d/salome/plates3d.med`
- Select your radiation mesh : `plates3d/salome/plates3d-rad.med`

For both files, a conversion of a proper format (here from MED to SYRTHES) is done automatically and the you should get the message :

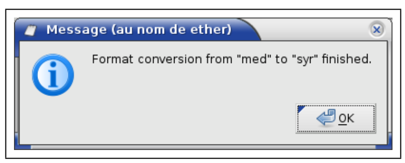

FIGURE 2.8 – SYRTHES File format conversion OK

Finally, provide a name for your results files (a name without extension ; SYRTHES will create different files with the same radical but different extensions depending on the type of files).
Now, the File Names window should look like :

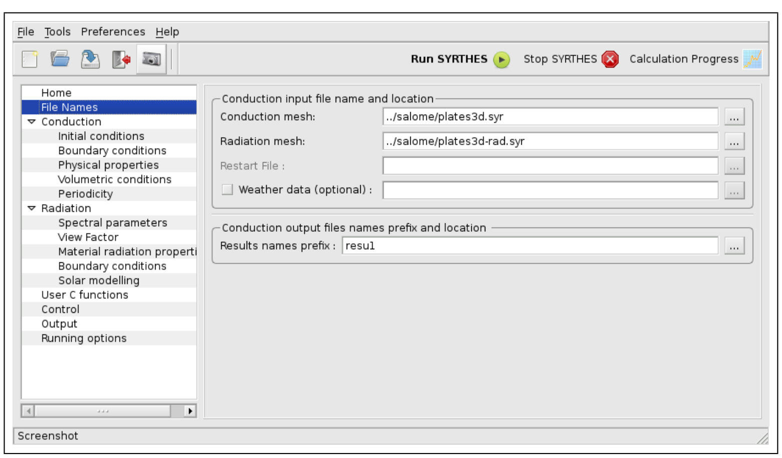

FIGURE 2.9 – SYRTHES File Names window

2.2.7 Input data for conduction computation

2.2.7.1 Group names and references numbers

While Salome is using group names to distinguish the different parts of the mesh, SYRTHES is using reference numbers. Group names and reference numbers are included in the mesh file `plates3d.med`. The links between both are given in an additional file `.syr_descr`. This file is automatically created (next to the .med file) when the convertion from Med file to SYRTHES file takes place. You can open this ASCII file in the SYRTHES GUI : menu “Tools” and “Open Desc”.

WARNING : depending on Salome version, pairs (group-name , number) could change. So make sure to check your description file and adapt numbers accordingly before proceeding.

Here is `plates3d.syr_desc` provided file :

```
group_of_faces    10    bord-d3
group_of_faces    11    bord-d1
group_of_faces    12    dessous
group_of_faces    13    dessus
group_of_faces    14    couronne
group_of_volumes    15    Disk1
group_of_volumes    16    Disk2-int
group_of_volumes    17    Disk2-ext
group_of_volumes    18    Disk3-int
group_of_volumes    19    Disk3-ext
group_of_volumes    20    Disk4-int
group_of_volumes    21    Disk4-ext
group_of_volumes    22    four
group_of_faces    6    interne
group_of_faces    7    interne_ss_couvercle
group_of_faces    8    bords-d4
group_of_faces    9    bord-d2
```

Here is the `plates3d-rad.syr_desc` file content :

```
group_of_faces    4    steel-copper
group_of_faces    5    granite
```

2.2.7.2 Initial conditions

Unroll the conduction menu by clicking on the arrow, and select the first item : Initial conditions
Set the initial temperature (here $20°C$), and the list of volumes considered (15 16 17 18 19 20 21 22). If all the volumes are concerned by the same initial condition, you may set “-1” instead of the explicit list. You can add a comment in the last column (optional).

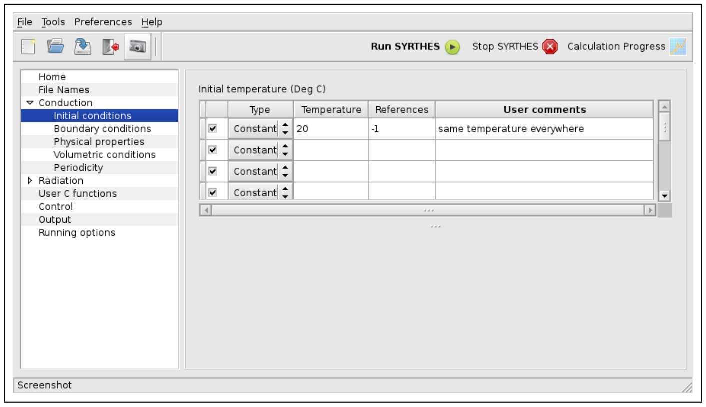

FIGURE 2.10 – SYRTHES Initial conditions

2.2.7.3 Boundary conditions

We want to set an heat exchange coefficient for the upper surface of the oven and an imposed temperature at the bottom.

Click on the "Heat Exchange" tab and set the external tempaerature and the heat exchange coefficient :

- External temperature $= 800°C$
- Heat exchange coefficient $= 1000\ W/m^2°C$
- References of the concerned faces : 13 14 (groups : "dessus" + "couronne")

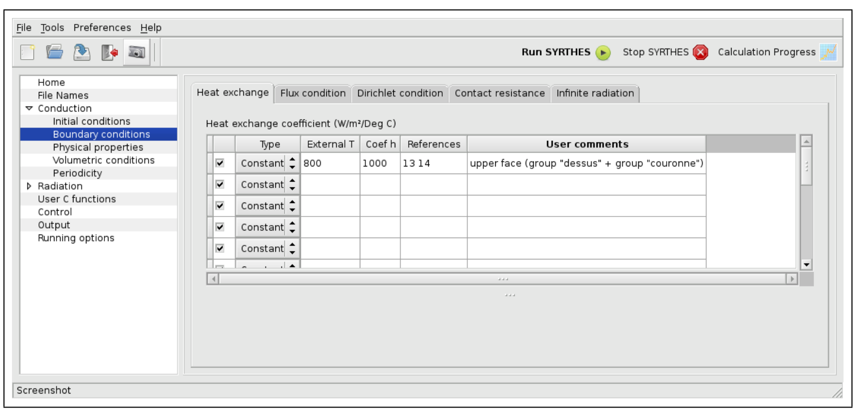

FIGURE 2.11 – SYRTHES Boundary conditions : heat exchange

Click on the “Diricihlet condition” tab and set the imposed temperature :

- Dirichlet T $= 20°C$
- References of the faces concerned : 12 (groups : “dessous”)

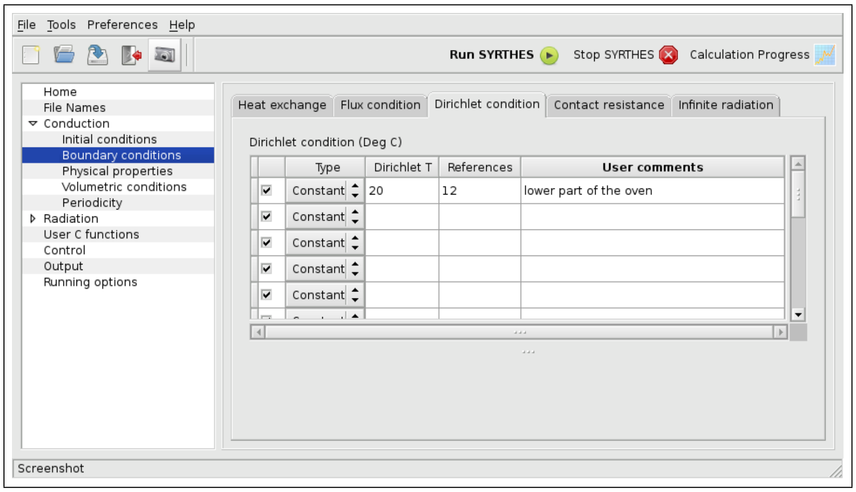

FIGURE 2.12 – SYRTHES Boundary conditions : T imposed

2.2.7.4 Physical properties

In this case, we have to consider 3 different materials, but all are assumed to be isotropic. Click “Physical properties” in the left menu and be sure to have the “Isotropic” tab selected. Set the values of the physical properties :

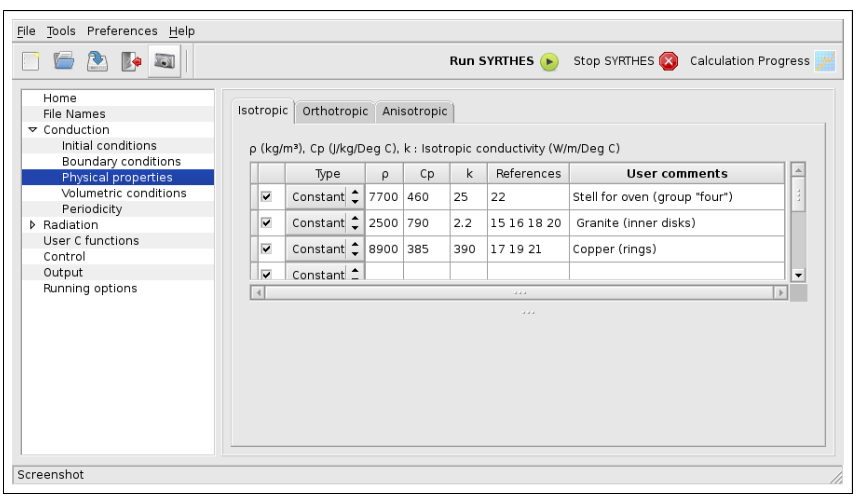

FIGURE 2.13 – SYRTHES Material properties

Now all the physical parameters are defined, you can jump directly to the radiation menu.

2.2.8 Radiation

In this part, you will define all the parameters related to the radiation solver.

2.2.8.1 Spectral parameters

By default, in this simple configuration, we consider only one spectral band. If required for further problems you will be abble to define several spectral bands here, depending on your material behaviour. Even, if only one material must be defined on two bands, all materials will have to be specified for these two bands.

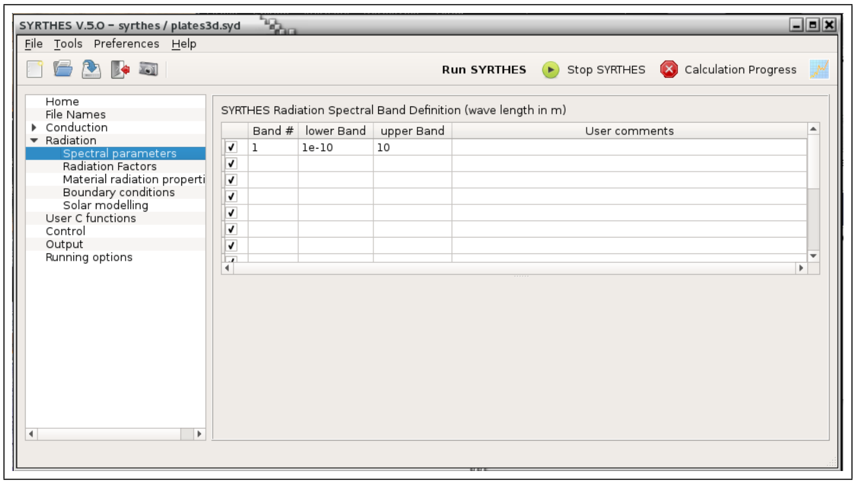

FIGURE 2.14 – SYRTHES Spectral bands

2.2.8.2 View factors

For a given surfacic mesh, it is impossible for SYRTHES to determine which part is inside or outside your computational domain (because many mesh generators don't orient properly the faces of a surfacic mesh). For example, if your radiation surfacic mesh is a sphere, do you want to calculate radiation inside the ball, or radiation through the space outside the ball ? To give an answer to this problem, it is compulsory to give an "interior point" of the radiation problem. If we come back to the example of the ball, and if you want to calculate the radiation inside the ball, the center of the ball is a good choice for this point. Select the "View Factor" tab and set the coordinates of an inner point : (0, 0.05, 0.2). For more complex cases with shadowing, it is a good idea to locate the inner point at a non-ambiguous position.

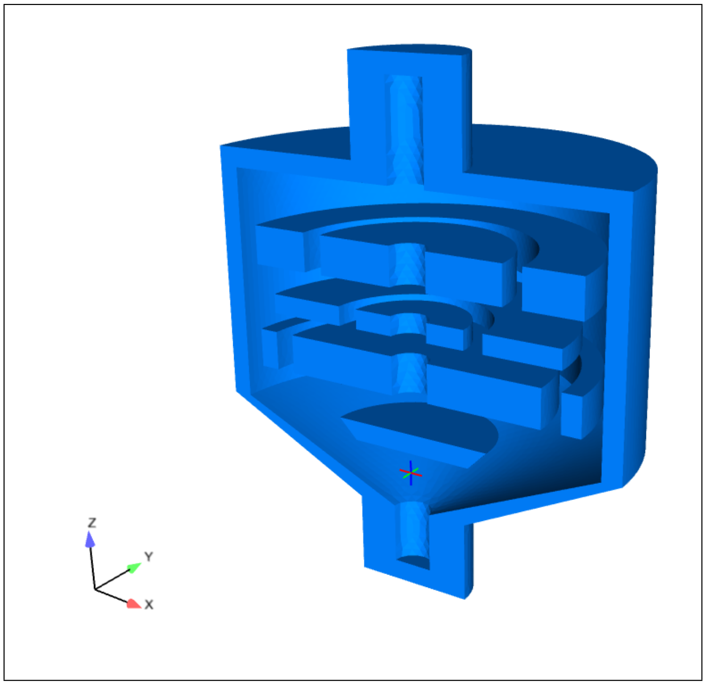

FIGURE 2.15 – SYRTHES Position of the inner point for view factors calculation

It is straightforward to take into account a symmetry for the conduction phenomena (by default a symmetry corresponds to an adiabatic condition for which no boundary condition is required), it is much more difficult for the radiation point of view. Fortunately for users, to handle symmetrical geometries as the present case, you just have to define the position of the symmetry plane.
Click the "symmetry/periodicity" tab and set the coefficients of symmetry plane equation (here $y = 0$).

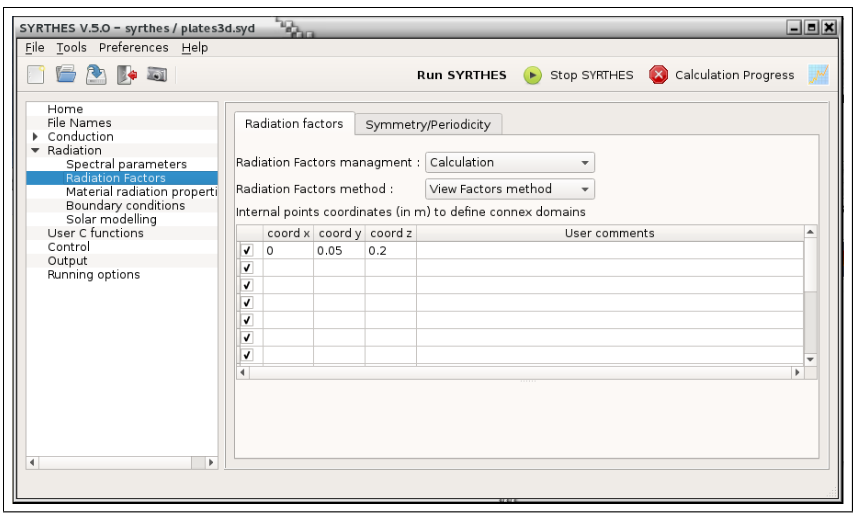

FIGURE 2.16 – SYRTHES View factors

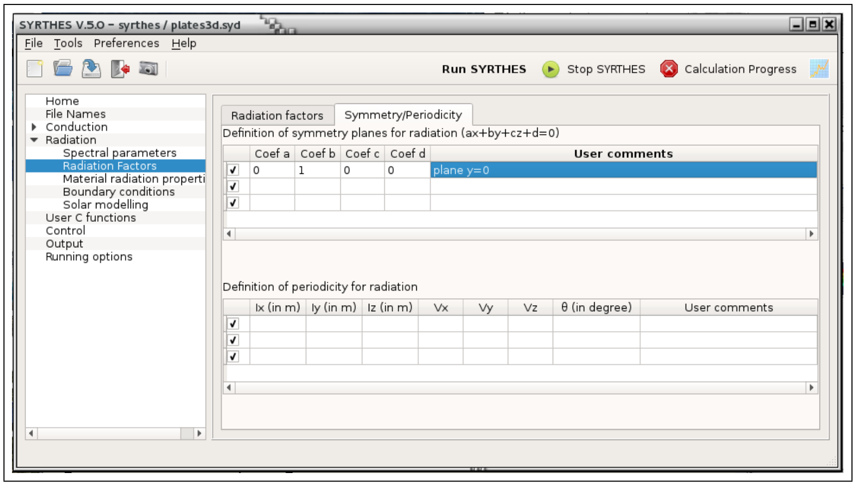

FIGURE 2.17 – SYRTHES View factors

2.2.8.3 Material radiation properties

In this case, only 1 spectral band is defined (grey material), and we consider 2 different emissivities : 0.8 for steel and copper, and 0.96 for granite.

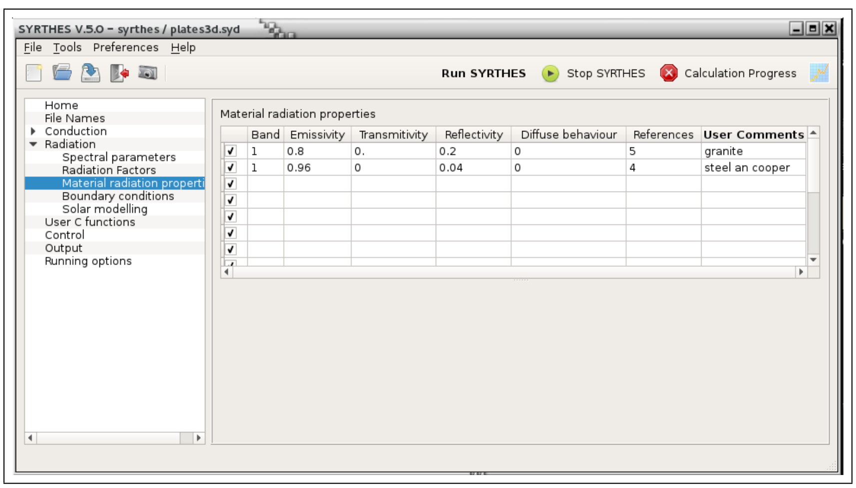

FIGURE 2.18 – SYRTHES Emissivity definition

2.2.8.4 Boundary conditions

In this section, you have to define how conduction mesh and radiation are coupled together. You must at least define :

- the references of the faces of the radiation mesh coupled with the conduction mesh. In this case, it's quite easy, because all faces are coupled, so we have all the references to set : 4 and 5 (groups "steel-copper" and "granite" in the radiation mesh)
- the references of the boundary faces of the conduction mesh which are coupled with the radiation mesh. In the present case, it corresponds to all boundary surfaces inside the oven : 11 9 10 8 6 7

Now, all physical and geometrical parameters are defined, you can jump to the Control section.

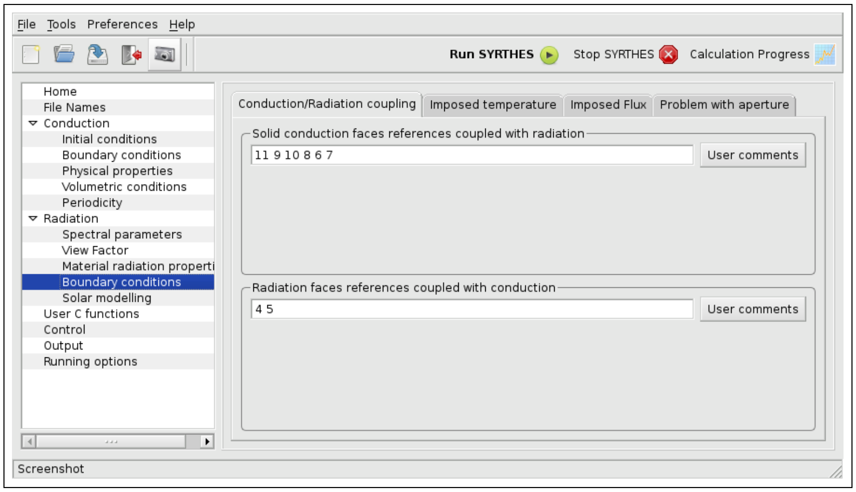

FIGURE 2.19 – SYRTHES Coupling conduction and radiation

2.2.9 Control

We want to reach a steady state. This one will be reached after a transient calculation, but since we are not interested by the transient, we can set a fairly large time step. For your first run, you can compute 230 time steps. An average time step for this case is around 300 seconds.

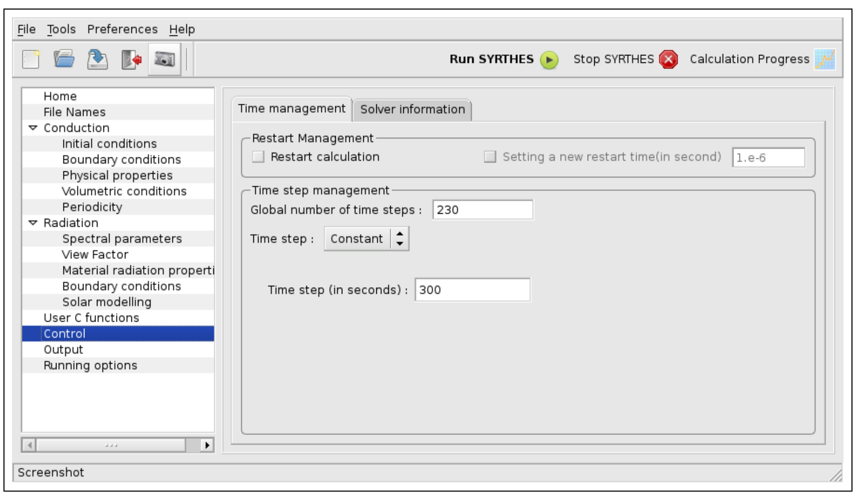

FIGURE 2.20 – SYRTHES Control window

Generally, there is no need to change the default values for the solver.

2.2.10 Output

In this section, we are going to define the type of results generated by SYRTHES.
Whatever the options you will have a result file with the temperature on all the nodes of the conduction mesh.
But, sometimes, it is also advisable to define some thermal probes : during the run, you will be able to follow during the calculation the temperature evolution at some strategic points of your domain.
Set the coordinates of points in your domain. Here we have selected 1 point per disk and ring. Then, define the frequency at which you want to record the temperature values of the probes on file : here, 1, which means "every time step".

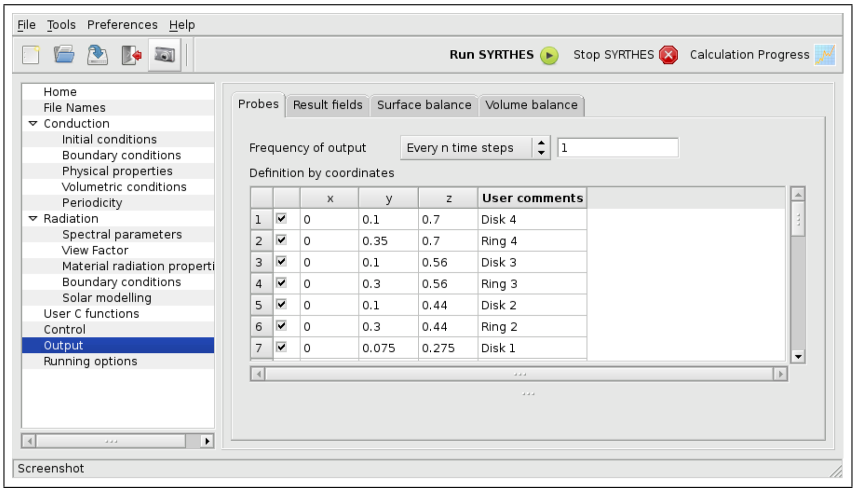

FIGURE 2.21 – SYRTHES Defining probes

Click on the next tab in the same window (Control) to select "result fields".
At the end of the calculation, you will get of course the temperature field corresponding to the last time step (unless you have specified otherwise). But, you can also get intermediate results with a chosen frequency. Click "Fields" and set the frequency to 50 : you will get the temperature every 50 time steps. These temperature fields will be recorded in the "`.rdt`" file and can be post-processed like the final result (".res" file).

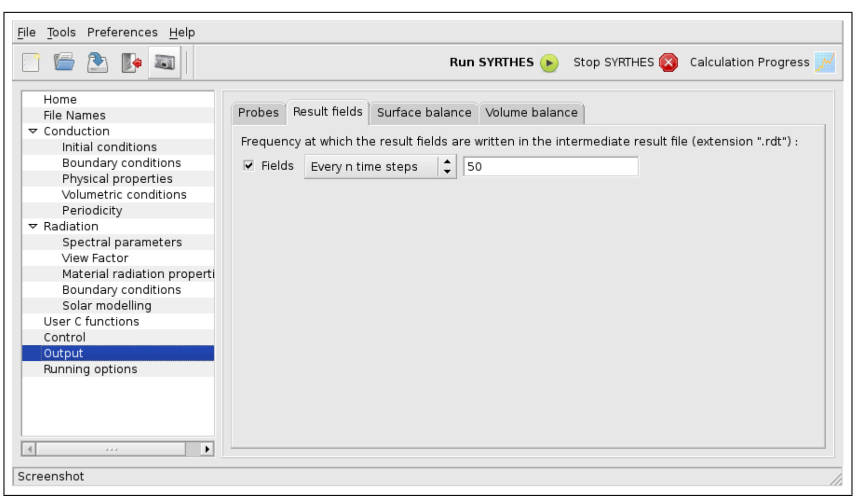

FIGURE 2.22 – SYRTHES Defining transient result file

2.2.11 Running options

This is the last step : give a name for the listing file. In that file, SYRTHES will give you some information about the calculation (option summary, solver convergence,...)
Furthermore, you may try a parallel computation, setting the number of processor to 3 or 4 (but, as the number of elements of this mesh is quite small, the benefit in term of CPU time will be low).

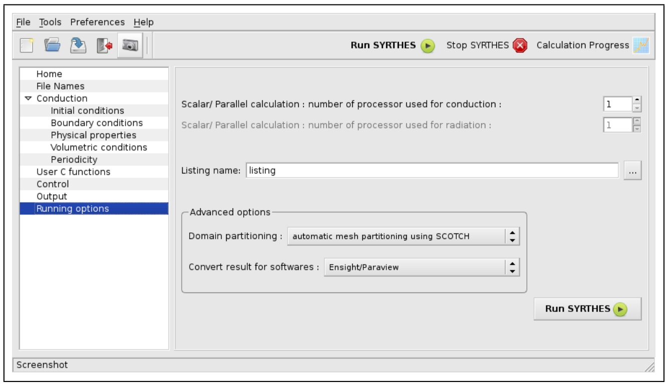

FIGURE 2.23 – SYRTHES Running options window

2.2.12 You're ready to run SYRTHES !

Click "Run SYRTHES", the "calculation progress" window appears and you can display the evolution of the temperature at the points previously defined.

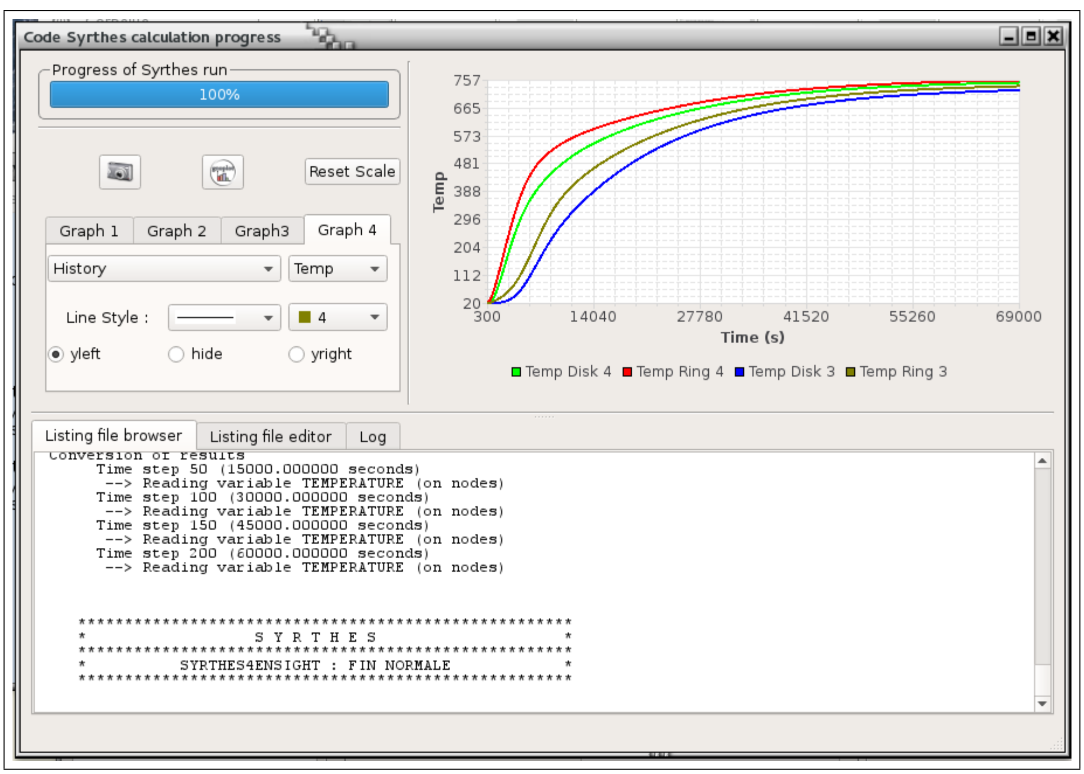

FIGURE 2.24 – SYRTHES Calculation progress window

The 3 tabs display :

- Listing file browser : displays the 100 last lines of the listings
- Listing file editor : whole SYRTHES listing file
- Log : system messages (usefull in case of error during user files compilation)

2.2.13 Analyzing the results

When SYRTHES calculation is finished, you can visualize the results using a post-processor. For this example, we are using the open source postprocessor Paraview (but you can also use Ensight or even your own post-processor provided that you have written a conversion tool between the SYRTHES format and your own).

2.2.13.1 Final temperature field

Run Paraview and open the file : `mkdir plates3d/syrthes/POST/resu1.ensight.case` and visualize the temperature field after 19.17 hours

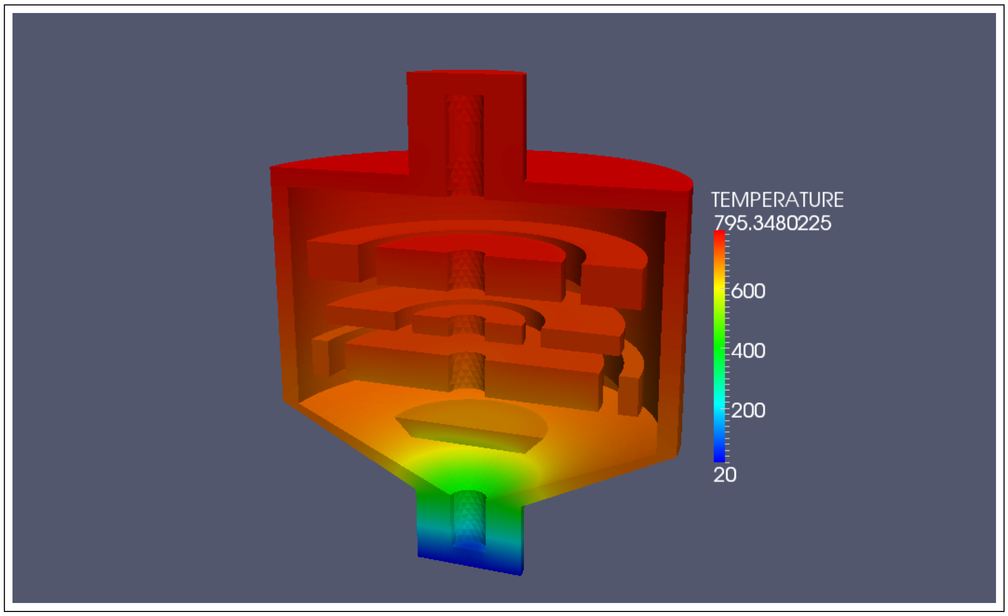

FIGURE 2.25 – Temperature field after 69000 s

2.2.13.2 Transient temperature field

As we selected the transient result file option (see figure 2.22), we can also post-process this file. Run Paraview and open the file : `mkdir plates3d/syrthes/POST/resu1_rdt.ensight.case`. Then the temperature field can be visualized at different times : 15000 s, 30000 s, 45000 s, and finally 60000 s.

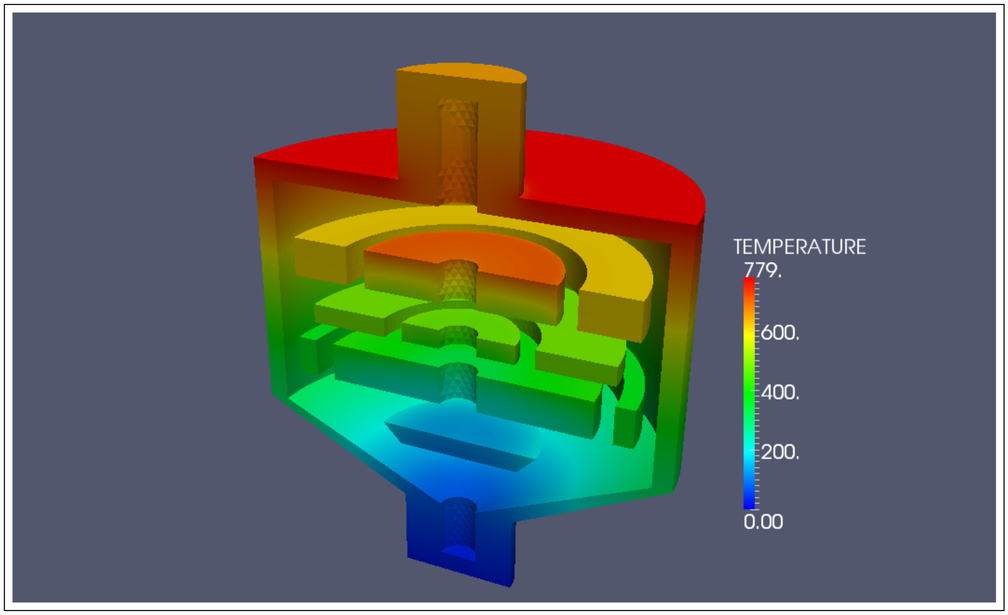

FIGURE 2.26 – Temperature field after 15000 s

2.2.13.3 Result files

At the end of the run, in the directory where the calculation has taken place, one should be able to find several files :

- `resu1.res` : final result file
  Temperature field at the end of the calculation. This file is automaticaly converted to a post-processor file format (Ensight/Paraview or SALOME-MED) in the `POST` directory.
- `resu1.rdt` : transient result file Temperature field during the transient. This is an optional file (depending on the output options defined). If existing, it is automaticaly convert to a post-processor file format (Ensight/Paraview or SALOME-MED) in the `POST` directory.
- `resu1.his` : temperature probes
  This file contains the temerature at the probes initially defined. This is an "in columns" file.
  — column 1 : time
  — column 2 : temperature
  — columns 3-5 : coordinates (only columns 3-4 in 2D)
- `resu1.mnx` : min-max values
  The min and max values of each variable of the calculation and the place where there are reached. This is an "in columns" file. Content depends on the type of calculation and is described at the beginning of the file.
  Each variable is corresponding to a column. This file can be post-process with a 1D plotter.
  To help users, a gnuplot command file can be also be automatically written thanks to an icon of the calculation progress window. Example :

```
# 1=temps  2=T_min    3=x   4=y   5=z   6=T_max     7=x   8=y   9=z
#         10=rho_min 11=x  12=y  13=z  14=rho_max  15=x  16=y  17=z }
```

  In this file, you will find :
  — column 1 : time
  — column 2 : minimum of temperature
  — columns 3-5 : coordinates where is located the minimum of temperature
  — columns 6 : maximum of temperature
  — columns 7-9 : coordinates where is located the maximum of temperature
  — column 10 : minimum of density

- columns 11-13 : coordinates where is located the minimum of density
- columns 14 : maximum of density
- columns 15-16 : coordinates where is located the maximum of density

- `resu1.add` : additionnal file This file is unused in this study. In other cases, you can calculate some specific entities and save them in this file for further graphical post-processing.

Chapter 3

Getting started with 1D model SYRTHES fluid model

pipes2_fluid1d

3.1 What is the problem?

We would like to compute the temperature inside a concrete block in which 2 water pipes are going through. Adiabatic conditions are put around the block. Inlet fluid temperature is $10°C$ for the first pipe and $40°C$ for the second one.

3.1.1 Geometrical description

The solid domain consists in a block whose dimensions are 0.5m x 0.3 m x 0.2 m. Diameter of the pipes is 0.1 m.
Distance between the pipes is 0.15 m.
Geometrical characteristics are shown on figure 5.1

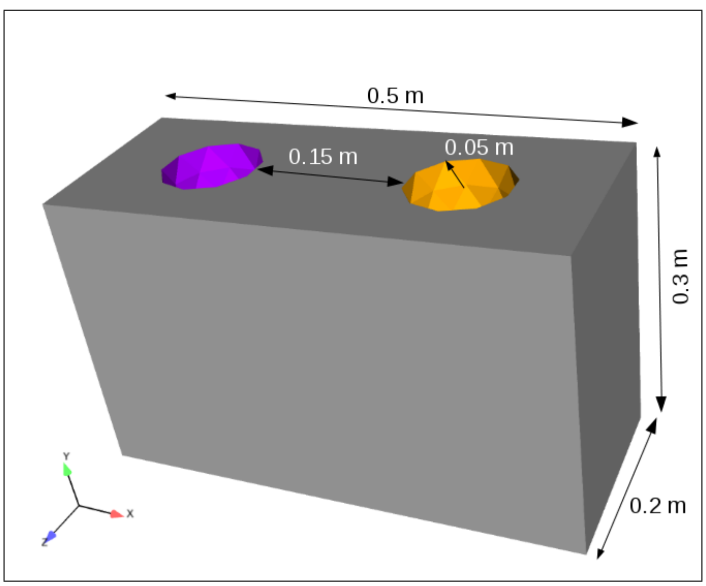

FIGURE 3.1 – Sketch of the problem

3.1.2 Physical description

Physical properties are :

- Density : $\rho = 2500\ kg/m^3$
- Heat capacity : $C_p = 1000\ J/kg°C$
- Conductivy $\lambda = 2\ W/m/°C$

3.1.3 Initial conditions and boundary conditions for the concrete block

The initial temperature is $20°C$. Boundary conditions on the external side of the block are adiabatic.

3.1.4 Description of the 1D fluif flow

- Fluid is water :
  — Density : $\rho = 1000\ kg/m^3$
  — Heat capacity : $C_p = 4185\ J/kg°C$
  — Dynamic vicosity : $\mu = 0.1\ Pa.s$
  — Conductivy $\lambda = 0.6\ W/m/°C$
- Flow rates at the inlet are set to 1 kg/s for both pipes.
- Initial and inlet temperatures are :
  — pipe #1 : $10°C$
  — pipe #2 : $40°C$

3.2 How to do that ?

3.2.1 To organize the study

We propose in this section a possible organisation for the different files of your study. This is only a piece of advice, but expert users can do as they wish...

- create a new directory for your study : `mkdir mycase_pipes2`
- go inside : `cd mycase_pipes2`
- create a new directory for the creation of the mesh : `mkdir mesh`

3.2.2 Creating meshes

Here, even if the geometry stays very simple and do not create any trouble, attention must however be paid to the references allowing to identify materials and boundary conditions.
We used Salome to define the geometry and create the mesh. Two meshes have to be built : a 3D mesh

with tetrahedra for the structure (concrete block), and one a 1D one (containing only edges) for the fluid flow. Meshes are independent and have to be saved in separated files.
In the directory `mkdir mycase_pipes2/mesh`, run Salome, create your geometry and mesh.
Save your Salome-study and export your mesh to MED format in this directory.

You can create your own mesh, but below, we describe the characteritics of the mesh provided in the syrthes distribution :
`/..../syrthes5.x/arch`/*your_ arch*`/share/syrthes/tests/3-cas_pipes2_fluid1d`.

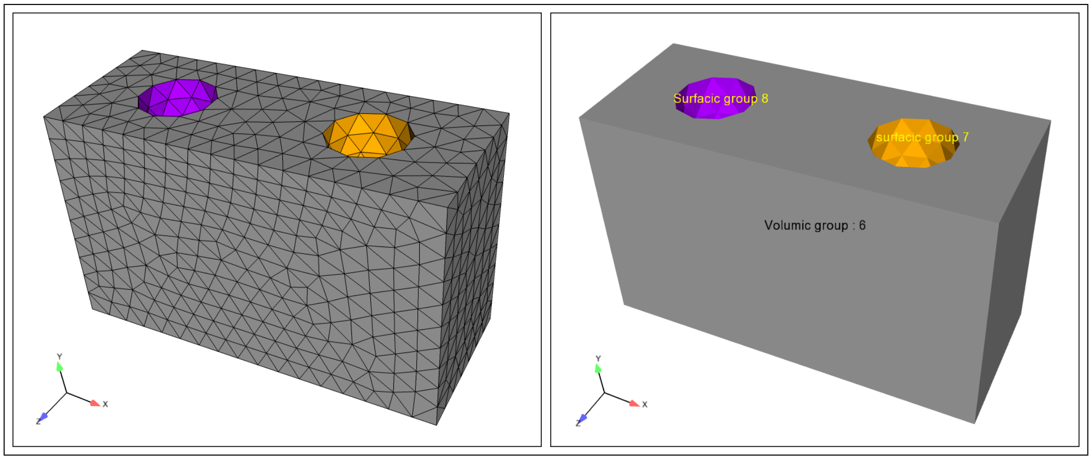

FIGURE 3.2 – 3D mesh and volumic/surfacic groups

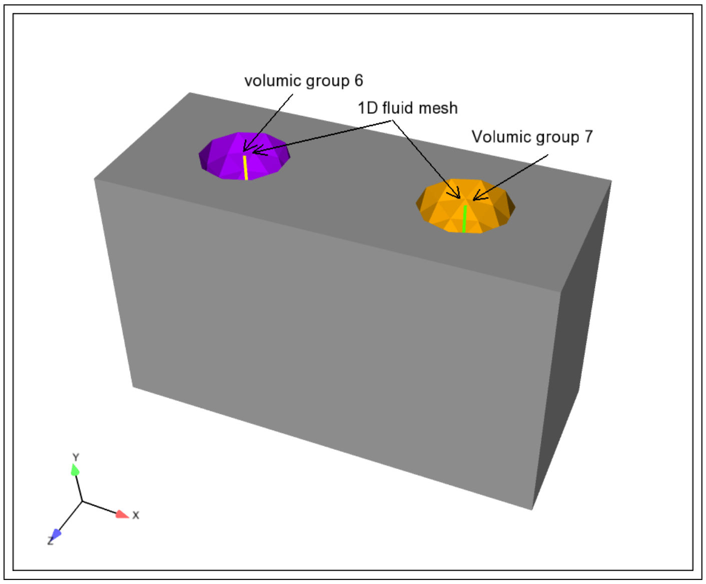

FIGURE 3.3 – 1D mesh and groups

3.2.3 Create your SYRTHES-study

Go back to the initial directory : `mycase_pipes2`
If not already done : source the SYRTHES environment (Linux only) :
`source /home/.../syrthes5.x/arch/myarch/bin/syrthes.profile`
Run the SYRTHES-gui : `syrthes.gui`

Create a new case : `syrthes`

Now, all your calculation will be managed by the SYRTHES Graphic User Interface.

FIGURE 3.4 – SYRTHES Managing your cases

3.2.4 Main view

Give a title to your study. The dimension of the problem is 3D.

Check the dimension of the problem : 3D

Select an additional physical modelling : “SYRTHES 1D fluid flow”

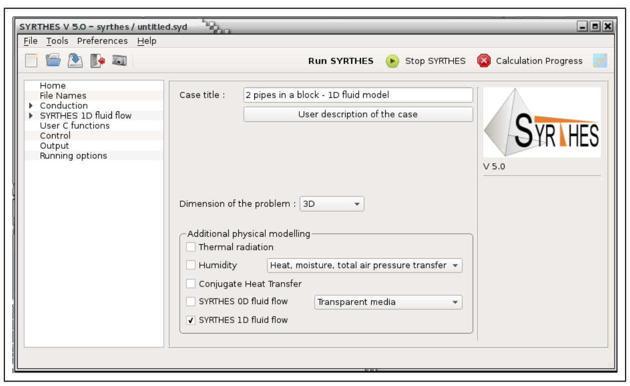

FIGURE 3.5 – SYRTHES Main View

Save your data file (“file”, “save as”) as `pipes2.syd`.

3.2.5 File Names

- Click on the next item in the menu on the left : File Names
- Select your conduction mesh : `mycase_pipes2/mesh/Structure.med`
- Select your 1D fluid mesh : `mycase_pipes2/mesh/Fluid.med`

A conversion of the file format is done automatically and you should get the following message for both files :

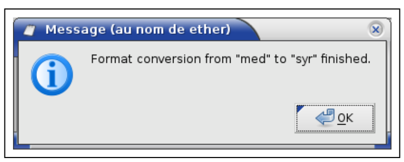

FIGURE 3.6 – SYRTHES File format conversion OK

Finally, give a name for your results files (a name without extension ; SYRTHES will automatically create different files with the same radical but different extensions depending on the type of files).

The file Names should look as shown below :

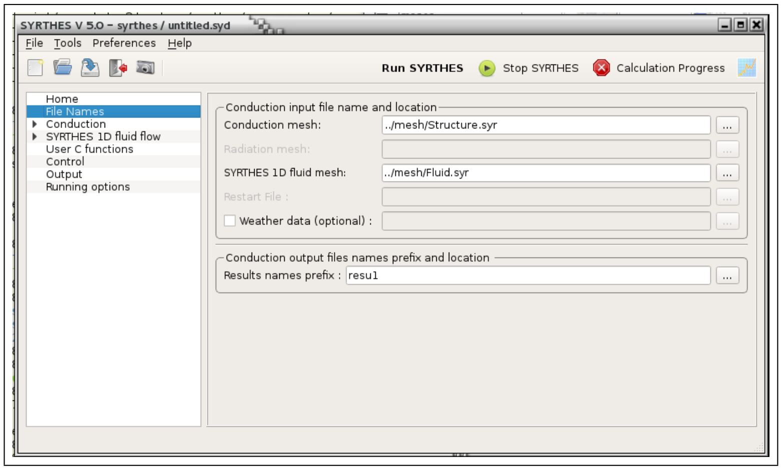

FIGURE 3.7 – SYRTHES File Names window

3.2.6 Input data for conduction computation

On the left, unroll the conduction menu by clicking on the arrow.

3.2.6.1 Group names and references numbers

While SALOME is using group names to distinguish the different parts of the mesh, SYRTHES is using reference numbers. Group names and reference numbers are included in the mesh file `Structure.med`. The links between both files are given in an additional file (.syr_descr) created while you have converted the MED file to the SYRTHES file. You can open it in the SYRTHES GUI : menu "Tools" and "Open Desc".

WARNING : depending on SALOME version, pairs (group-name , number) may change. So check the description file and adapt numbers accordingly before proceeding.

Here is `Structure.syr_desc` provided file :

```
group_of_faces     10    dessus-dessous
group_of_volumes   6     solid
group_of_faces     7     int-cyl1
group_of_faces     8     int-cyl2
group_of_faces     9     bordure
```

3.2.6.2 Initial conditions

Select the first item of the Conduction Menu : Initial conditions

Set the initial temperature (20°C), and the volume considered, here it is 6. Note that if all the volumes are affected by the same initial condition, you can put "-1" instead of the explicit list. You can add a comment in the last column (optional).

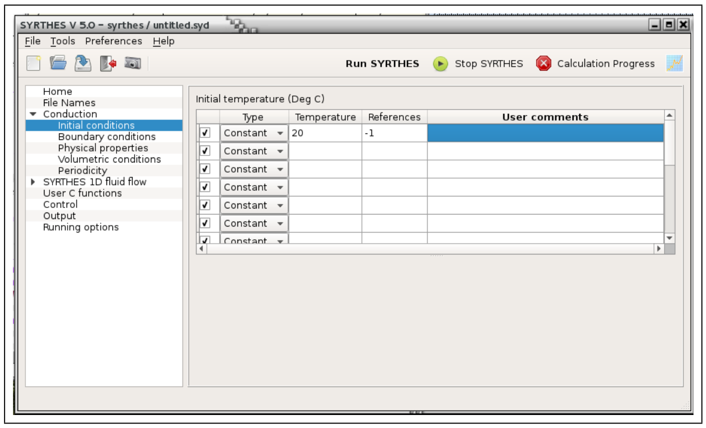

FIGURE 3.8 – SYRTHES - Initial conditions

3.2.6.3 Boundary conditions

No boundary conditions are set on the boundaries of the concrete block.

3.2.6.4 Physical properties

Put the values of density, heat capacity and conductivity.

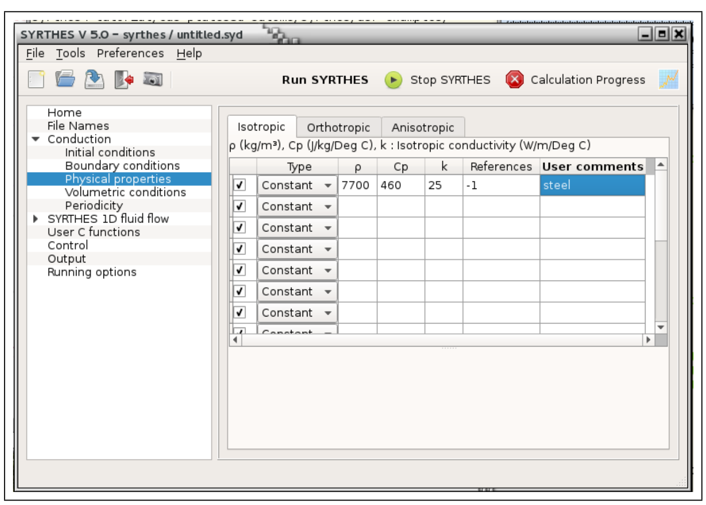

FIGURE 3.9 – SYRTHES - material properties

All is done regarding the 3D thermal problem. Now, the 1D fluid flow model has to be defined.

3.2.7 Input data for 1D fluid flow model

On the left side of the window, unroll the SYRTHES 1D fluid flow menu by clicking on the arrow.

3.2.7.1 1D fluid Geometry

Definition of the geometry of the pipes : for each part of the pipes, depending on the reference number of the 1D elements of the mesh, you have to give the hydraulic diameter in meter, the section in square meter, and possibly the roughness (0 for a smooth wall).
In the present case, we have only 2 references, one for each pipe.

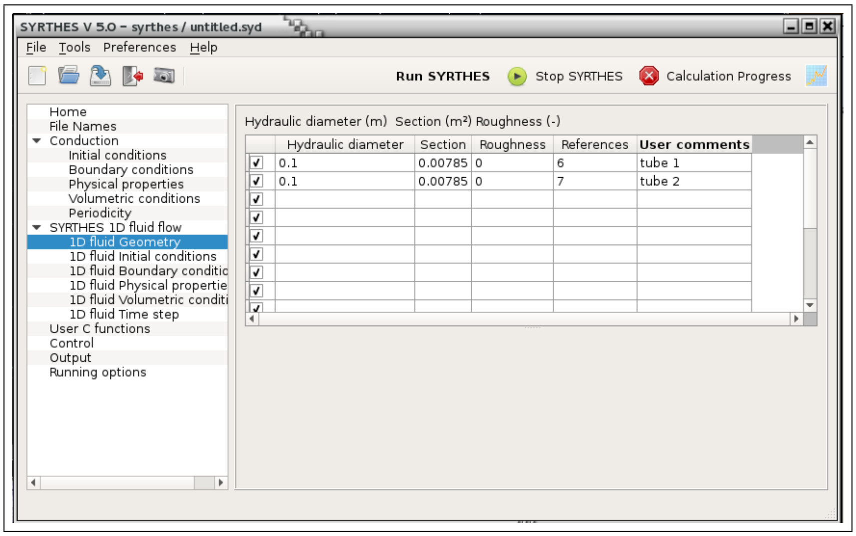

FIGURE 3.10 – SYRTHES - 1D fluid Geometry

3.2.7.2 1D fluid Initial conditions

For each pipe, we define the initial conditions of the fluid flow : temperature and velocity.

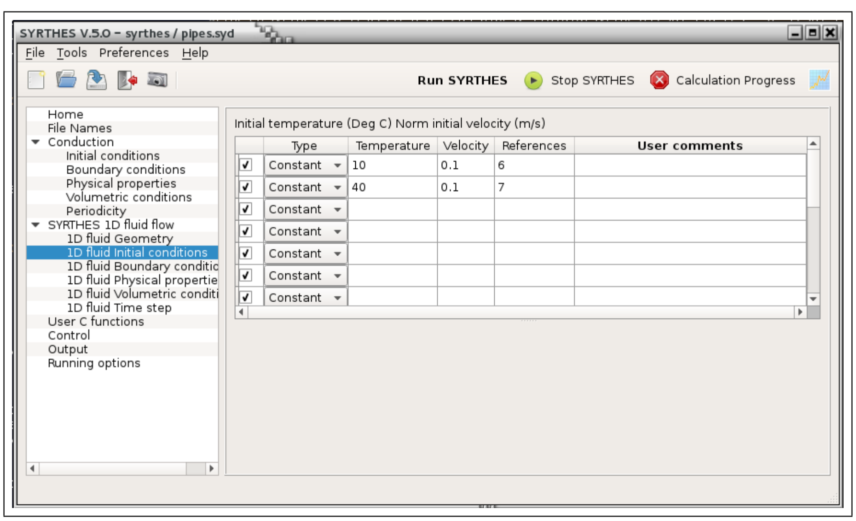

FIGURE 3.11 – SYRTHES - 1D fluid Geometry

3.2.7.3 1D fluid Boundary conditions

Click first on the “Conduction/1D fluid coupling” tab.
Here we define how fluid flow and thermal calculations are coupled.
In the table, we consider the 1D fluid mesh and we have to define the heat exchange coefficient between fluid and solid. In that case, we choose an automatic computation of the heat exchange computation using a Colburn correlation (based on the velocity, hydraulic diameter, and fluid properties). So, we only have to define the part of the pipes where this law has to be applied : all along the 2 pipes (ie : references 6 and 7).

The second step is to define which solid wall is coupled with the fluid flows.
We have to give the references on the structure mesh wich are the wall of the fluid flow : here faces references are 7 and 8.

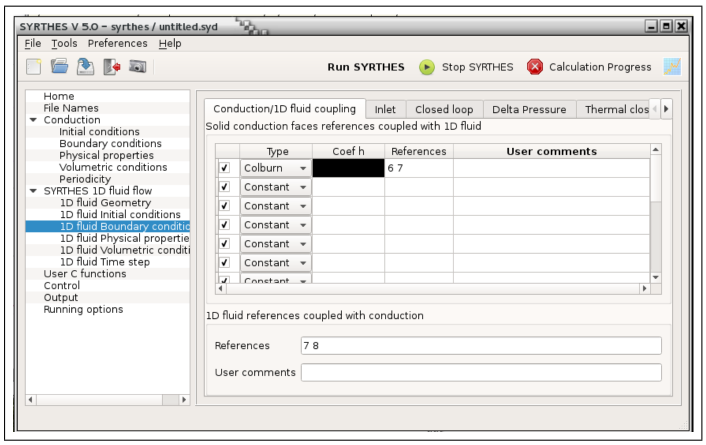

FIGURE 3.12 – SYRTHES - Conduction/1D fluid coupling

Now, we define how the fluid goes through the pipes : definition of the inlet.
In the same window, click on the “Inlet” tab.
Define the location of the inlet for each pipe : give a point close to the inlet of each tube. Points are not necessarily included in the mesh. Then, give the flow rate of each pipe (kg/s) and the inlet temperature ($^{\circ}C$)

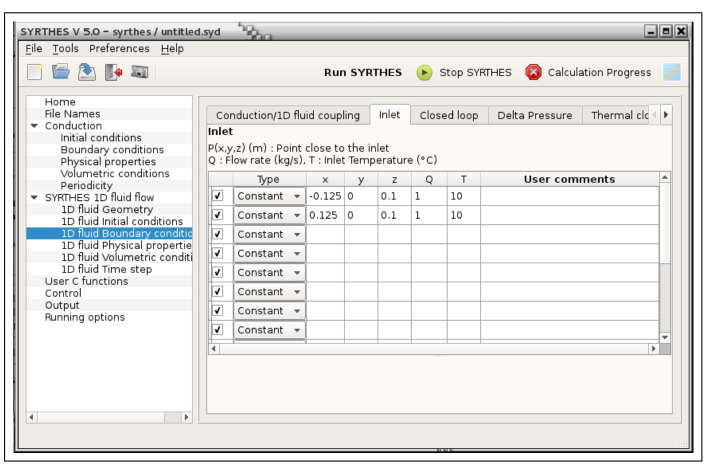

FIGURE 3.13 – SYRTHES - Inlet

3.2.7.4 1D fluid Physical properties

Definition of the physical properties of the fluid, here water. We have the same fluid in both pipes.

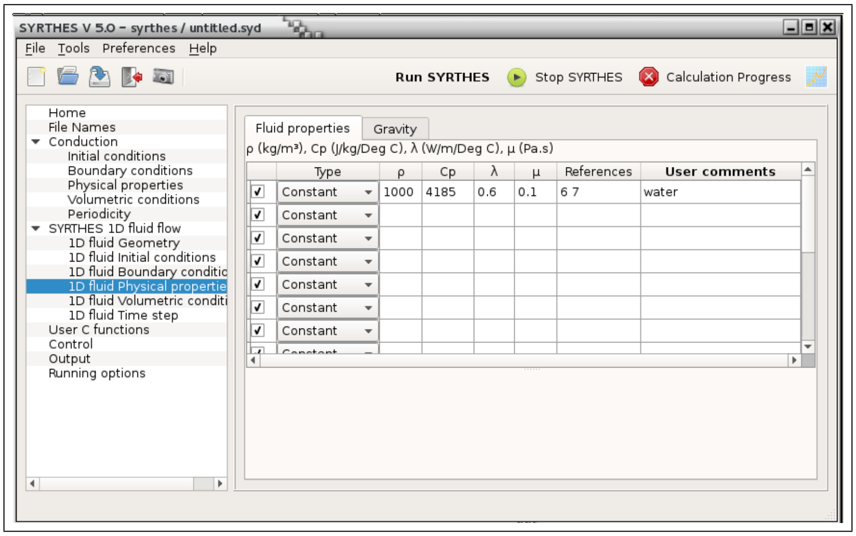

FIGURE 3.14 – SYRTHES - Physical properties

3.2.7.5 1D fluid Time step

As we are interested by a steady state, we choose to not have the same time step in fluid and solid to speed up the convergence : fluid time step is set to 60s.

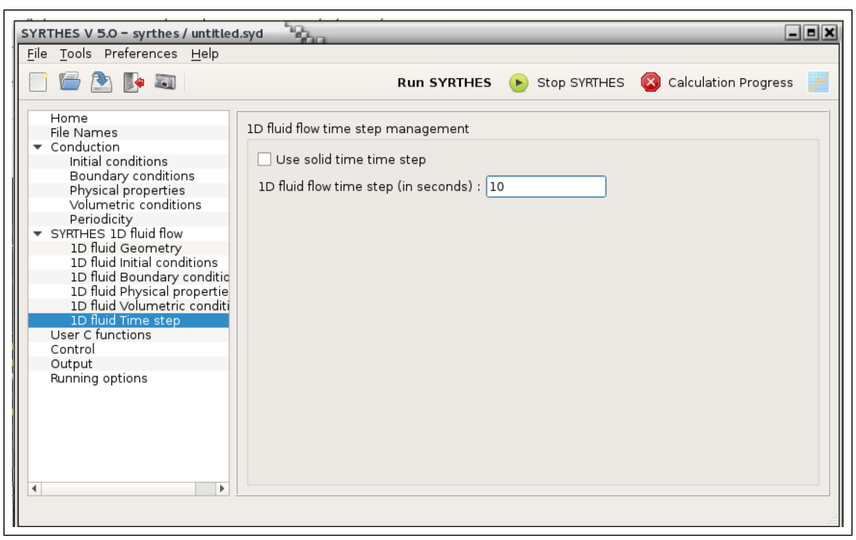

FIGURE 3.15 – SYRTHES - Time step

3.2.8 Control

To reach a steady state, we can use a quite large time step on the solid : 600 s and perform 500 time steps.

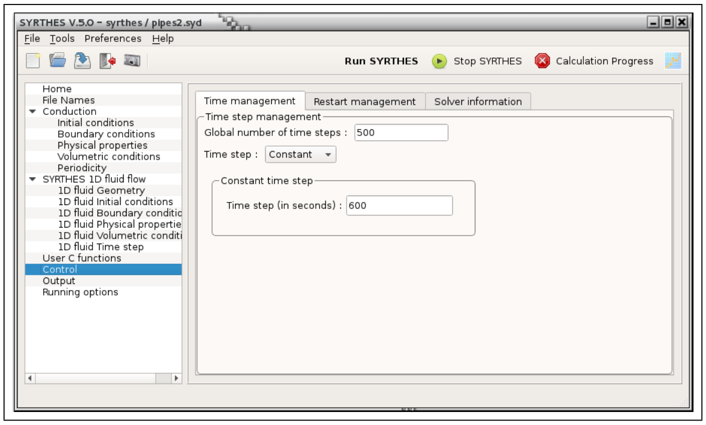

FIGURE 3.16 – SYRTHES - Control window

Generally, there is no need to change the default values provided for the solver.

3.2.9 Output

In this section, we are going to define the type of results to be generated by SYRTHES.
Whatever the options you will have a result file with the temperature on all the nodes of the mesh.
But, it is also advisable to define some thermal probes : during the run, it will allow to follow the temperature evolution at some strategic points of the domain.
Set the coordinates of points in the domain. Then, define the frequency at which you want to save the temperature values of the probes on file : here, 1, which means every time step.

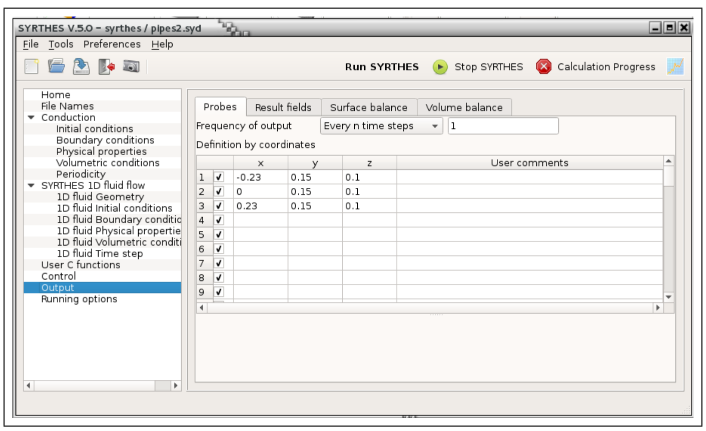

FIGURE 3.17 – SYRTHES - Defining thermal probes

Click on the next tab in the same window (Control) to select "result fields".
At the end of the calculation, you will get of course the temperature field corresponding to the last time step (unless you have specifically chosen not to obtain such a field). But, you can also get intermediate results with a choosen frequency. Click "Fields" and set the frequency to 100 : you will get the temperature every 100 time steps.
These temperature fields will be recorded in the ".rdt" file and will be post-processed in a way identical to the final result (".res" file).

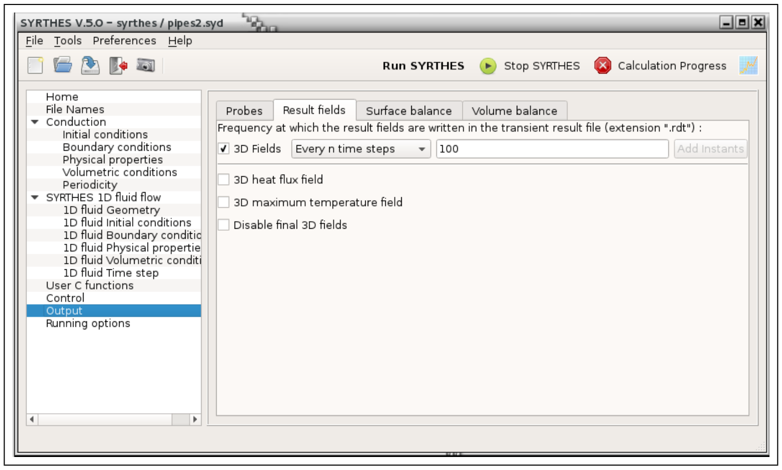

FIGURE 3.18 – SYRTHES - Defining transient results file

3.2.10 Running options

This is the last step : give a name for the listing file. SYRTHES will provide you with some information about the calculation (option summary, solver convergence,...)
For bigger cases, you could try a parallel computation, setting the number of processor to 3 or 4 or much more (but here, as the number of elements of this mesh is very small, the benefit in term of CPU time is likely to be very low).

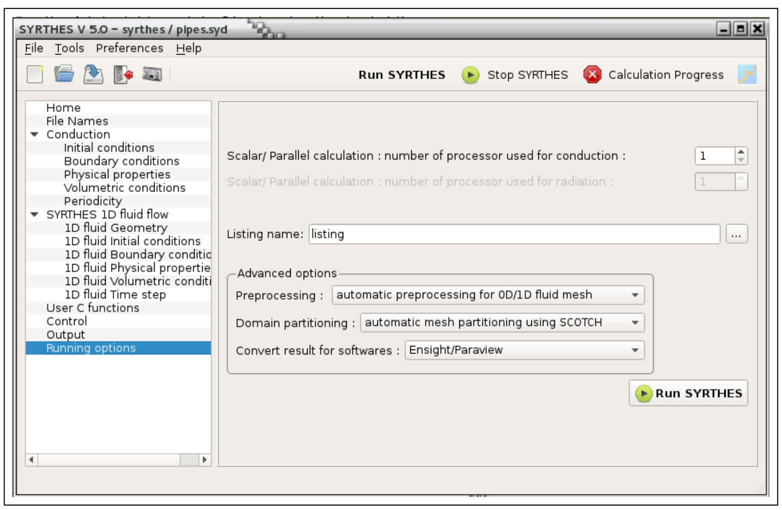

FIGURE 3.19 – SYRTHES - Running options window

3.2.11 You're ready to run SYRTHES !

Click "Run SYRTHES", the "calculation progress" window should appear and you can display the evolution of the temperature at the 3 locations defined previously.

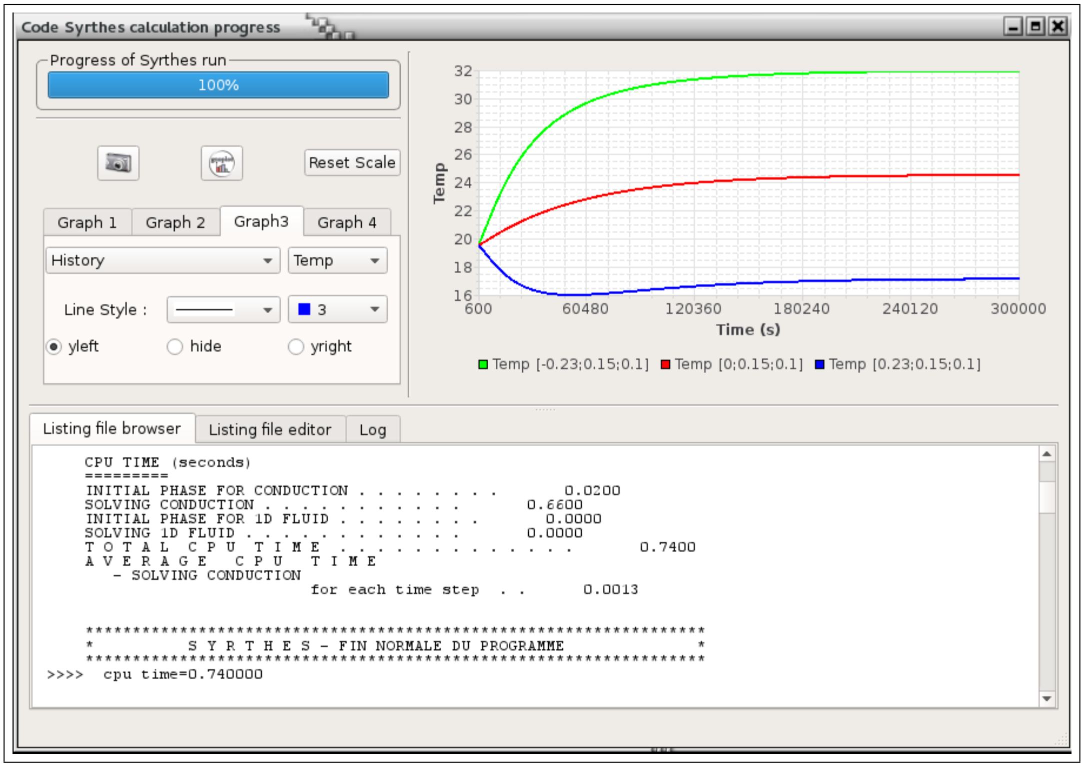

FIGURE 3.20 – SYRTHES Calculation progress window

The 3 tabs display :

- Listing file browser : displays the 200 last lines of the listing
- Listing file editor : whole SYRTHES listing file
- Log : system messages (usefull in case of error during user files compilation)

3.2.12 Analyzing the results

When SYRTHES calculation ends, you can visualize the results using a post-processor. For this example, we are using Ensight, but you can also use the open source post-processor Paraview.

3.2.12.1 Final temperature field

Run paraview and open the file : `mycase_pipes2/syrthes/POST/resu1.ensight.case`

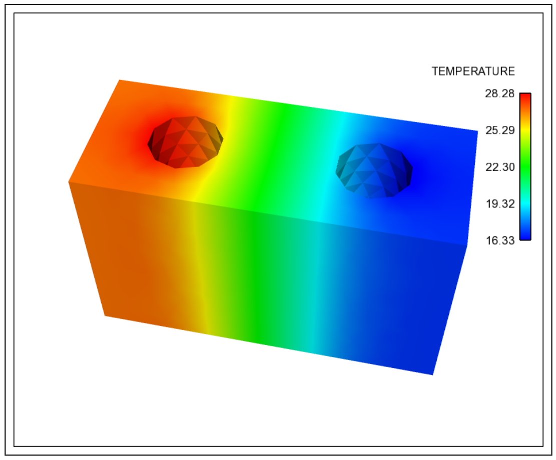

FIGURE 3.21 – Temperature field at steady state

3.2.12.2 Intermediate results

As we selected transient result file (see figure 4.20), we can also post-process this file.
Run the post-processor and open the file : `mkdir mycase_pipes2/syrthes/POST/resu1_rdt.ensight.case`.
Then the temperature field can be visualized at different time steps : 100, 200, 300, 400.
As time steps for fluid and structure computationare not the same, the intermediate time steps can't be considered as realistic fields, they are just intermediate states before reaching a steady state.

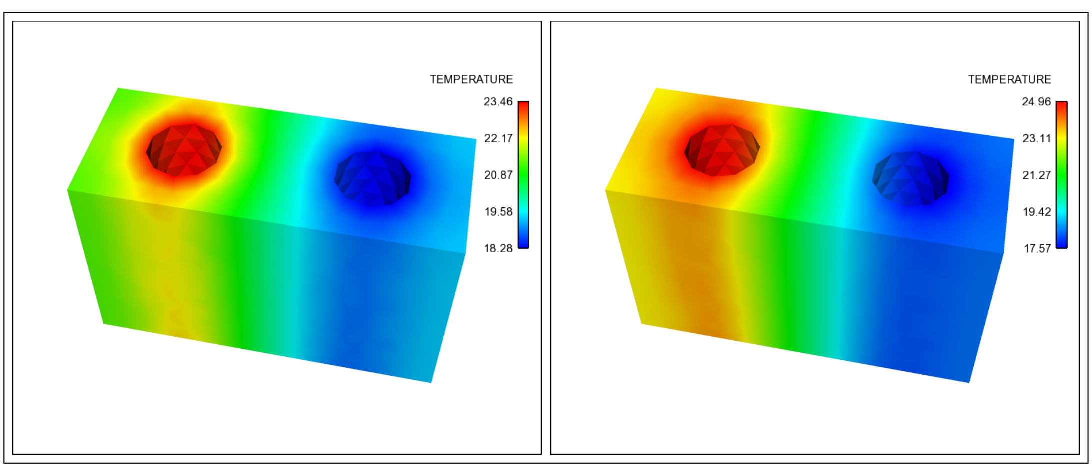

FIGURE 3.22 – Temperature fields after 100 and 200 time steps

3.2.12.3 Fluid flow results

Run the post-processor Paraview and open the file : `mycase_pipes2/syrthes/POST/resu1_f1d.ensight.case`

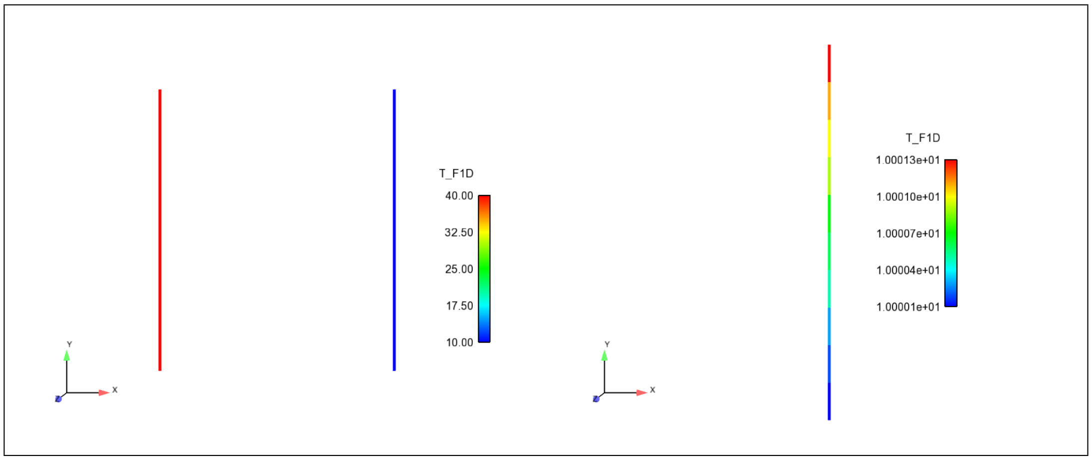

FIGURE 3.23 – Temperature field at the steady state and a zoom on the cold pipe

3.2.12.4 Result files

At the end of the run, in the directory where the SYRTHES calculation has taken place, one should be able to find several files :

- `resu1.res` : final result file
  Temperature field at the end of the calculation. This file is automaticaly converted to a post-processor file format (Ensight/Paraview or SALOME-MED) in the `POST` directory.
- `resu1.rdt` : transient result file Temperature field during the transient. This is an optional file (depending on the output options defined). If existing, it is automaticaly convert to a post-processor file format (Ensight/Paraview or SALOME-MED) in the `POST` directory.
- `resu1.his` : temperature probes
  This file contains the temperature at the probes initially defined. This is an “in columns” file.
  — column 1 : time
  — column 2 : temperature
  — columns 3-5 : coordinates (only columns 3-4 in 2D)
  To help users, it is also possible to save a gnuplot command file (in the calculation progress window).
- `resu1.mnx` : min-max values
  The min and max values of each variable of the calculation and the place where there are reached. This is an “in columns” file. Content depends on the type of calculation and is described at the beginning of the file.
  Each variable is corresponding to a column. This file can be post-processed with a 1D plotter.
  Example :

```
# 1=temps  2=T_min    3=x   4=y   5=z   6=T_max    7=x   8=y   9=z
#         10=rho_min 11=x  12=y  13=z  14=rho_max  15=x  16=y  17=z }
```

  In this file, you will find :
  — column 1 : time
  — column 2 : minimum of temperature
  — columns 3-5 : coordinates where is located the minimum of temperature
  — columns 6 : maximum of temperature
  — columns 7-9 : coordinates where is located the maximum of temperature

— column 10 : minimum of density
— columns 11-13 : coordinates where is located the minimum of density
— columns 14 : maximum of density
— columns 15-16 : coordinates where is located the maximum of density

- `resu1_f1d.res` and `resu1_f1d.rdt` : 1D fluid flow result with temperature, velocity and heat exchange coefficient. Results are provided at the end of the calculation and for intermediate time steps.
- `resu1_cplf1d.res` and `resu1_cplf1d.rdt` : results at the fluid/solid interface (on the structure mesh) : boundary fluid temperature and fluid heat exchange coefficent projected on the solid coupled wall

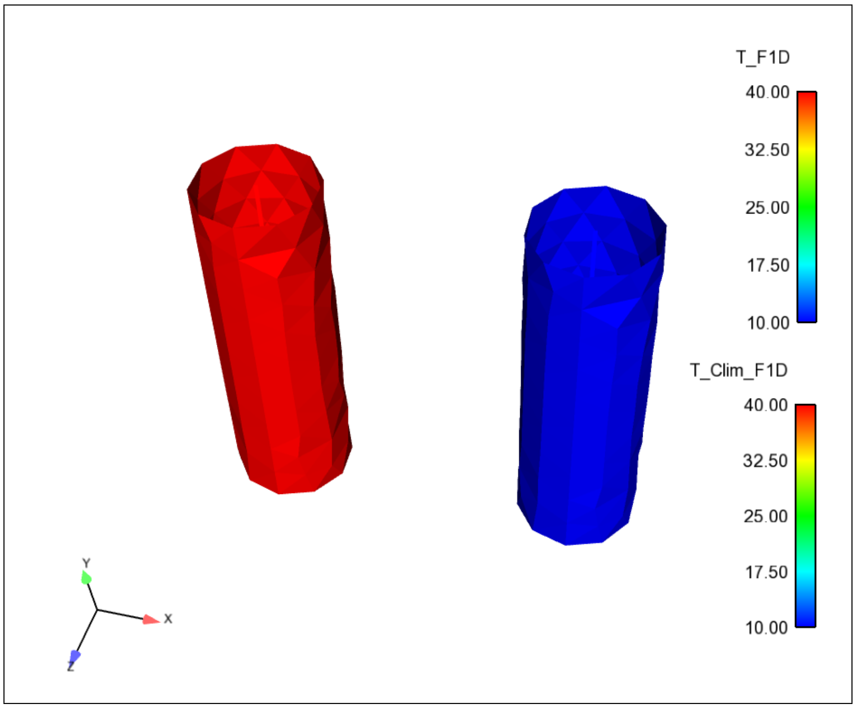

FIGURE 3.24 – Fluid boundary temerature on the solid mesh

- `resu1.add` : additionnal file This file is unused in this study. In other more complex cases, some specific entities can be calculated by users and saved them in this file for further graphical post-processing.

Chapter 4

Getting started with 0D fluid model

spheres_fluid0d

4.1 What is the problem ?

We consider 2 concentric spheres with an annular space filled with air. The external side of half of the sphere is heated. We study conduction transfer inside the outer sphere, radiation exchanges between the spheres, conduction transfer in the inside sphere, and the temperature of the air between the spheres.

4.1.1 Geometrical description

The solid domain consists in 2 concentric spheres with a radius of 0.1 m and 0.2 m. The thickness of the hollow outer sphere is 0.05m.

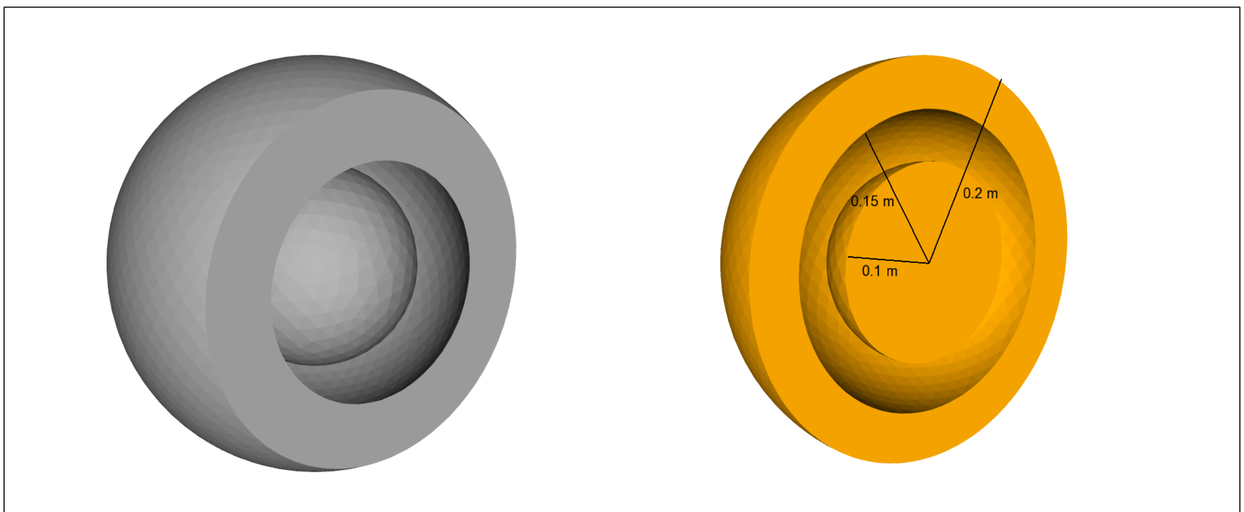

Figure 4.1 – Sketch of the problem

4.1.2 Physical description

Material is steel for both spheres and physical properties are :

- Density : $\rho = 7700\ kg/m^3$
- Heat capacity : $C_p = 460\ J/kg°C$
- Conductivy $\lambda = 25\ W/m/°C$

4.1.3 Initial conditions and boundary conditions for the concrete block

The initial temperature is $20°C$.

Boundary conditions : temperature and heat exchange coefficient on the half top of the outer sphere : T = $50°C$, h = 1000 $W/m^2/°C$

4.1.4 Air in the cavity between the two spheres

- Fluid is air :
  — Density : $\rho = 1\ kg/m^3$
  — Heat capacity : $C_p = 1017\ J/kg°C$

4.2 How to do that ?

4.2.1 To organize the study

We propose in this section a possible organization of the different files of your study. This is only a piece of advice, but advanced users can do as they wish...

- create a new directory for your study : `mkdir mycase_sphere`
- go inside : `cd mycase_sphere`
- create a new directory for the creation of the mesh : `mkdir mesh`

4.2.2 Creating meshes

If the geometry stays very simple and do not create any trouble, attention must however be paid to the references allowing to identify the different materials and boundary conditions.
SALOME is used to define the geometry and create the mesh.

Two meshes have to be built : a 3D mesh with tetrahedra for the structure (the two spheres), and one surfacic 2D mesh with triangles for the radiative solver. This surfacic mesh corresponds to the 2 spherical surfaces facing each other.
Meshes are independent and have to be saved in two separate files.

In the directory `mkdir mycase_sphere/mesh`, run SALOME, create your geometry and mesh.
Save your SALOME-study and export your mesh to MED format in this directory.

You can create your own mesh, but below, we describe the characteritics of the mesh provided in the SYRTHES distribution :
`/..../syrthes5.x/arch`*your_ arch*`/share/syrthes/tests/4-cas_spheres_fluid0d`.

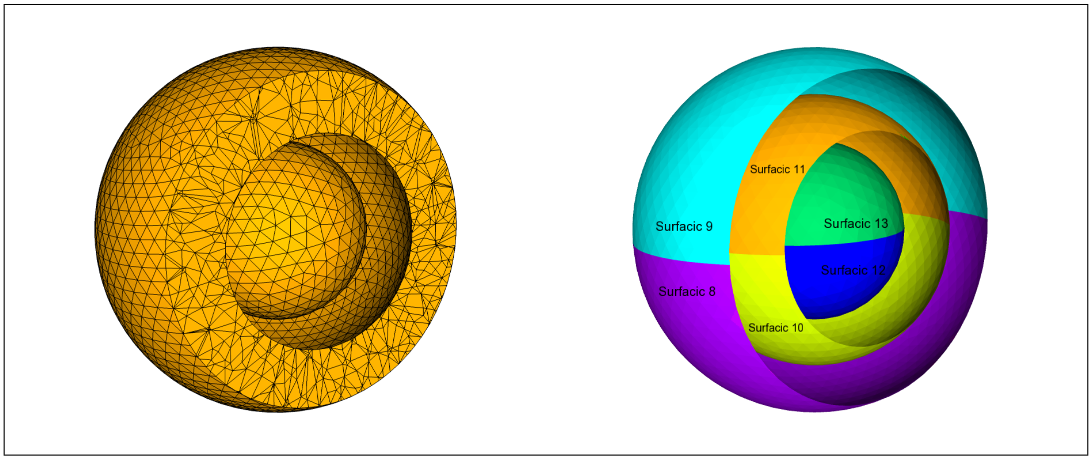

FIGURE 4.2 – 3D mesh and volumic/surfacic groups for conduction

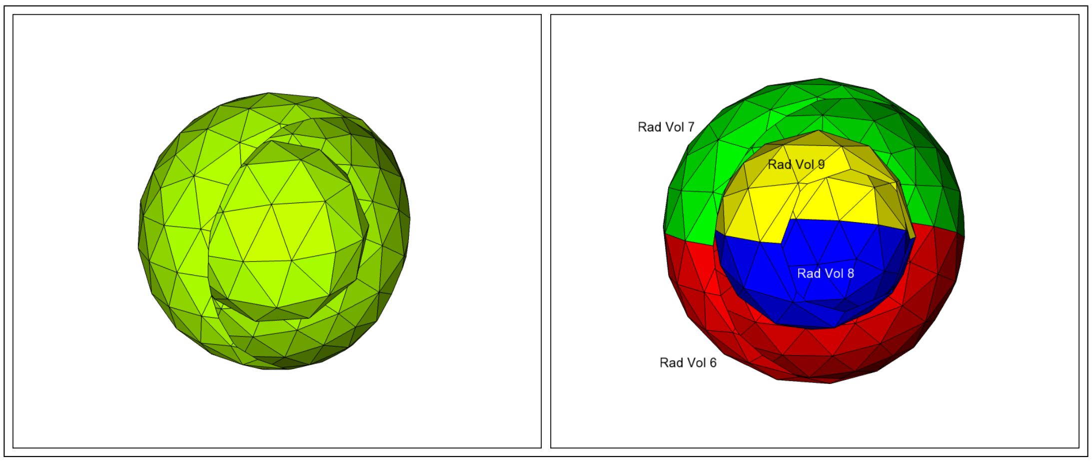

FIGURE 4.3 – 2D mesh and volumic groups for radiation

4.2.3 Create your SYRTHES-study

Go back to the initial directory : `mycase_sphere`
If not already done : source the SYRTHES environment (Linux only) :
`source /home/.../syrthes5.x/arch/myarch/bin/syrthes.profile`
Run the SYRTHES-gui : `syrthes.gui`

Create a new case : `syrthes`

Now, all your calculation will be managed by the SYRTHES Graphic User Interface.

FIGURE 4.4 – SYRTHES Managing your cases

4.2.4 Main view

Give a title to your study. The dimension of the problem is 3D.

Check the dimension of the problem : 3D

Select an additional physical modelling : "SYRTHES 1D fluid flow"

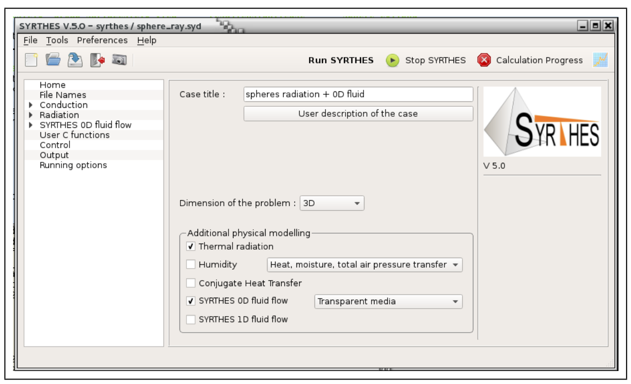

FIGURE 4.5 – SYRTHES Main View

Save your data file ("file", "save as") as `sphere.syd`.

4.2.5 File Names

- Click on the next item in the menu on the left : File Names
- Select your conduction mesh : `mycase_sphere/mesh/solid.med`
- Select your radiation mesh : `mycase_sphere/mesh/ray.med`

A conversion of the file format is done automatically and then you should get the following message for both files :

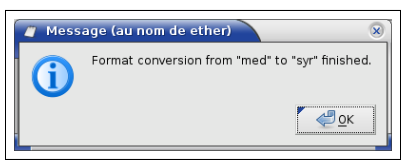

FIGURE 4.6 – SYRTHES File format conversion OK

Finally, give a name for your results files (a name without extension ; SYRTHES will create different files with the same radical but different extensions depending on the type of files).

The file Names window looks as shown below :

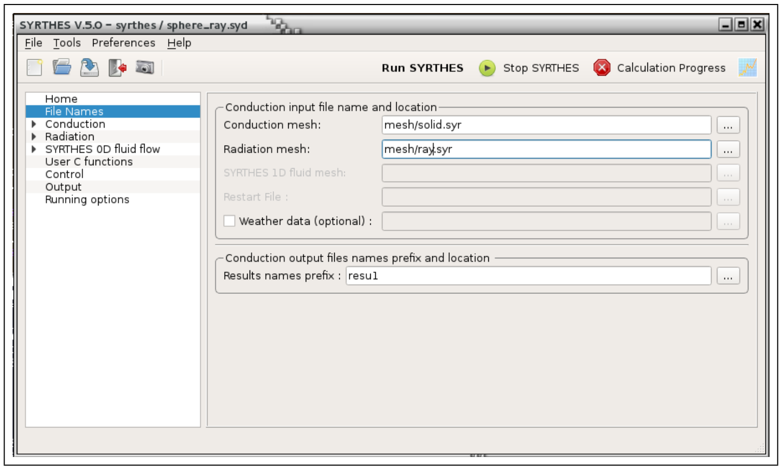

FIGURE 4.7 – SYRTHES File Names window

4.2.6 Input data for conduction computation

On the left part of the window, unroll the conduction menu by clicking on the arrow.

4.2.6.1 Group names and references numbers

While SALOME is using group names to distinguish the different parts of the mesh, SYRTHES is using reference numbers. Group names and reference numbers are included in the mesh files `solid.med` and `ray.med`. The links between both are given in the additional files (.syr_descr) created while you have converted the MED file to the SYRTHES file. You can open this file thanks to the SYRTHES GUI : menu “Tools” and “Open Desc”.

You can create your own mesh, but below, we describe the characteristics of the mesh provided in the SYRTHES distribution : `/..../syrthes5.x/arch/`*your_ arch*`/share/syrthes/tests/4-cas_spheres_fluid0d`.

Here is `solid.syr_desc` provided file :

```
group_of_faces     10    anneau_int_bas
group_of_faces     11    anneau_int_haut
group_of_faces     12    sphere_int_bas
group_of_faces     13    sphere_int_haut
group_of_volumes    6     sphere_int
group_of_volumes    7     sphere_ext
group_of_faces     8    anneau_ext_bas
group_of_faces     9    anneau_ext_haut
```

Here is `ray.syr_desc` provided file :

```
group_of_faces     6    peau_ext_haut
group_of_faces     7    peau_ext_bas
group_of_faces     8    peau_int_haut
group_of_faces     9    peau_int_bas
```

4.2.6.2 Initial conditions

Select the first item of the Conduction Menu : Initial conditions

Set the initial temperature (20°C). As the initial temperature is uniform (same values for all volumes), we can put "-1" instead of the explicit list of references. You can add a comment in the last column (optional but often usefull).

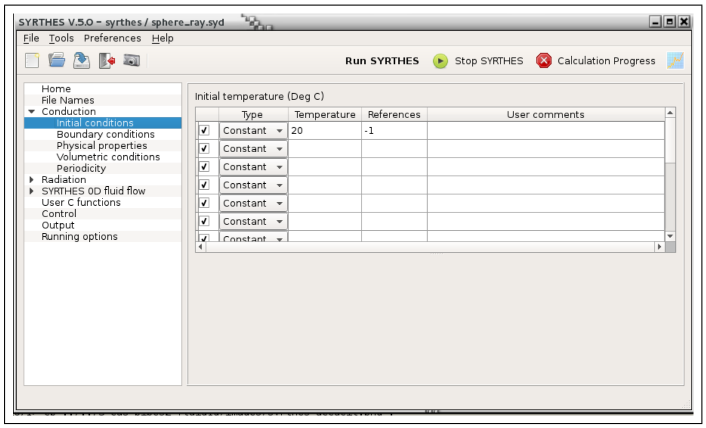

FIGURE 4.8 – SYRTHES - Initial conditions

4.2.6.3 Boundary conditions

A heat exchange boundary condition is set on the top half of the outer sphere :

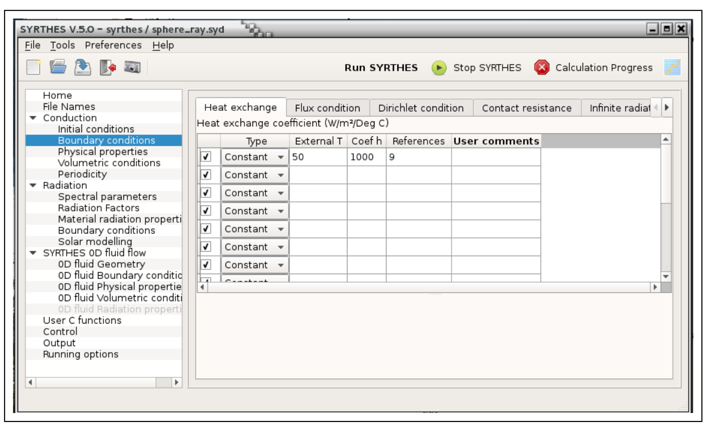

FIGURE 4.9 – SYRTHES - Initial conditions

4.2.6.4 Physical properties

Put the values of density, heat capacity and conductivity (here steel has been retained) :

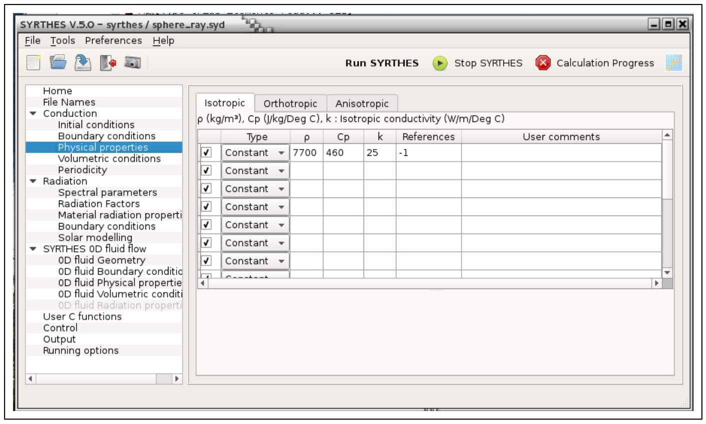

FIGURE 4.10 – SYRTHES - material properties

That's all for the 3D conduction problem. Now, we have to define the radiation model.

4.2.7 Input data for the radiation model

On the left part of the window, unroll the SYRTHES radiation menu by clicking on the arrow.

4.2.7.1 Spectral parameters

By default, in this simple configuration, we consider only one spectral band (grey material physical property). If required for more complex problems you will be able to define several spectral bands here, depending on your material behaviour (even if it turns out to be quite difficult to have access to such properties).

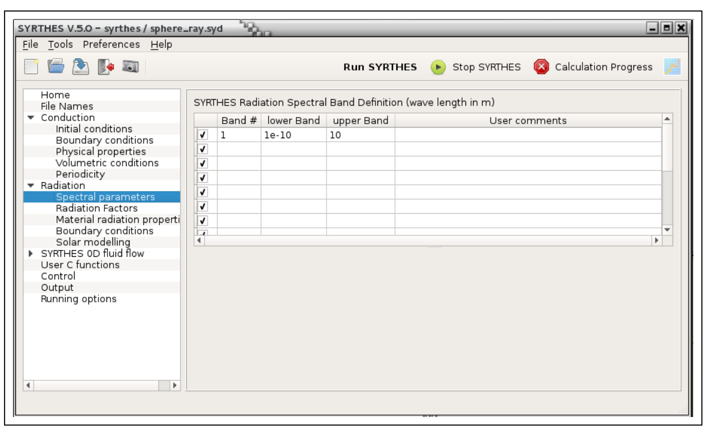

FIGURE 4.11 – SYRTHES Spectral bands

4.2.7.2 View factors

For a given surfacic mesh, SYRTHES can't determine what is inside or outside your computational domain (because mesh generators generally don't orient properly the surfacic faces of the mesh). In our case, SYRTHES needs to know if you want to calculate radiation inside or outside the balls. To give an answer to this problem, it is compulsory to give an "interior point" for the radiation problem. It will be a point located inside the radiation cavity, and for our case, a point in the air cavity located between the 2 spheres. Select the "View Factor" tab and set the coordinates of an inner point : (0.12, 0., 0.)

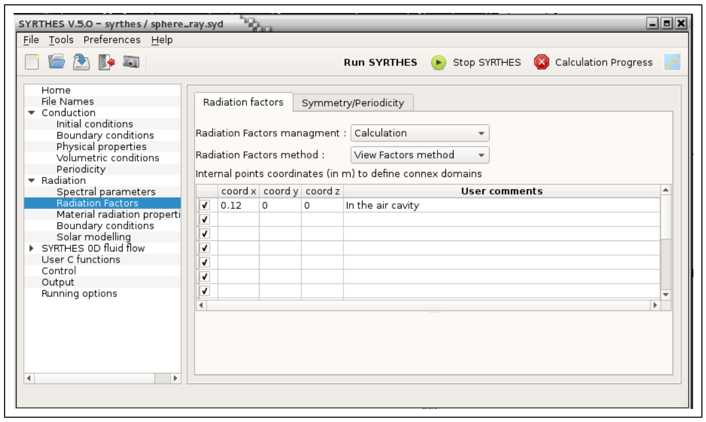

FIGURE 4.12 – SYRTHES Position of the inner point for view factors calculation

4.2.7.3 Material radiation properties

In this case, only 1 spectral band is defined (grey material), and we condider an emissivity of 0.9 for the steel.

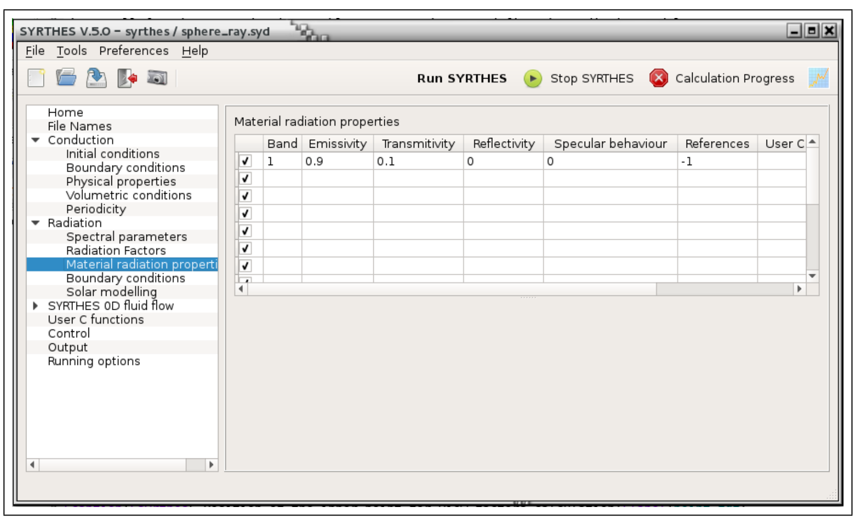

FIGURE 4.13 – SYRTHES Emissivity definition

4.2.7.4 Boundary conditions

In this section, you have to define how conduction mesh and radiation are coupled. You must at least define :

- the references of the **boundary faces of the conduction mesh** which are coupled with the radiation mesh. In the present case, it corresponds to surface of the inner sphere and the inner surface of the outer sphere 10 11 12 13
- the references of the faces of the **radiation mesh** which are coupled with the conduction mesh. In this case, it's easy, since all faces are to be coupled, so all references have to be set : 6 7 8 9

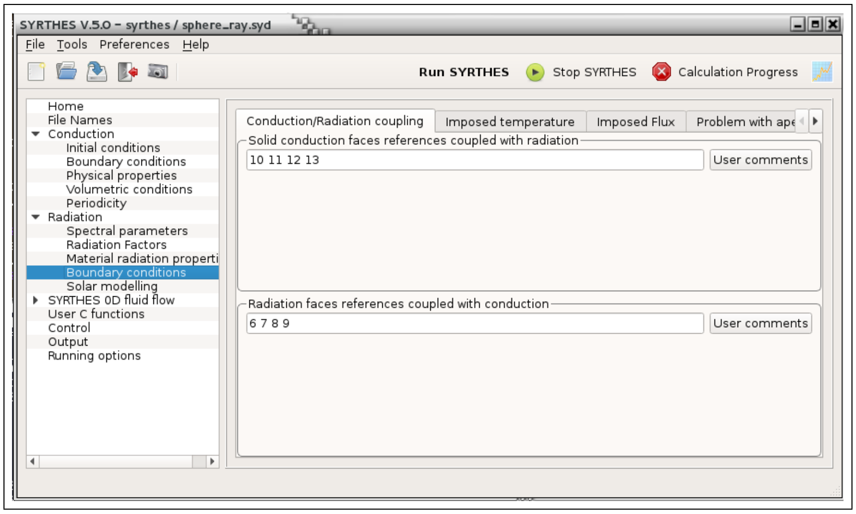

FIGURE 4.14 – SYRTHES Coupling conduction and radiation

4.2.8 Input data for the 0D fluid model

Let's define the last model : the 0D fluid model to take into account the air located between the 2 spheres. On the left of the main view, unroll the SYRTHES 0D fluid flow menu by clicking on the arrow.

4.2.8.1 0D fluid Geometry

First, define the characteristics of the fluid cavity.
Give a number to your cavity (indeed a complex case may contain several cavities), its volume, and the surfaces surrounding each cavity (on the conduction mesh).

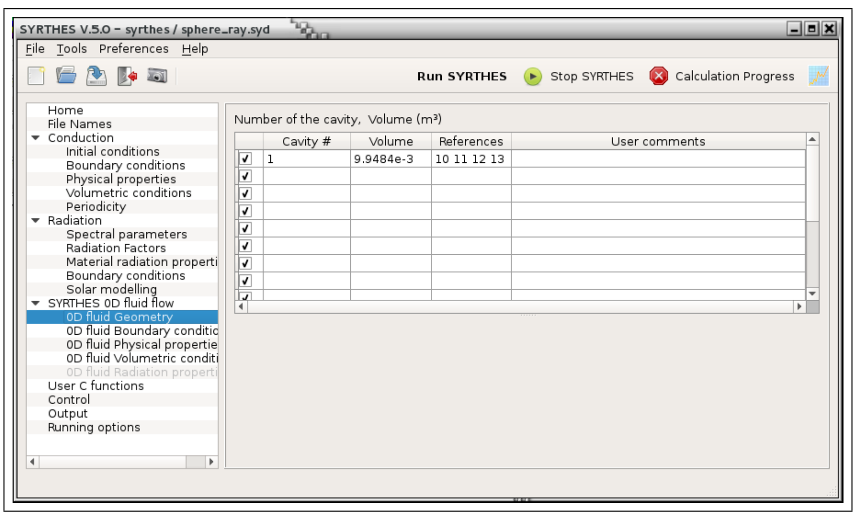

FIGURE 4.15 – SYRTHES - 0D fluid Geometry

4.2.8.2 0D fluid Boundary conditions

We define how fluid and solid thermal calculations are coupled by defining an heat exchange coefficient : in our case, a value of 9 $W/m^2{}^\circ C$ for all the surfaces.

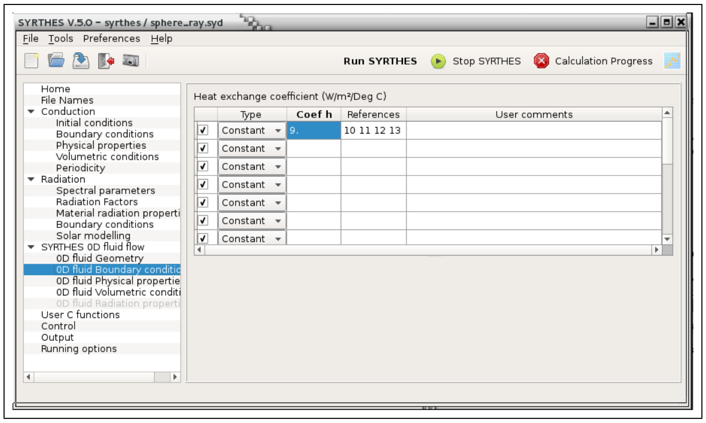

FIGURE 4.16 – SYRTHES - Conduction/0D fluid coupling

4.2.8.3 0D fluid Physical properties

For the present cavity, give the fluid physical properties and the initial temperature.

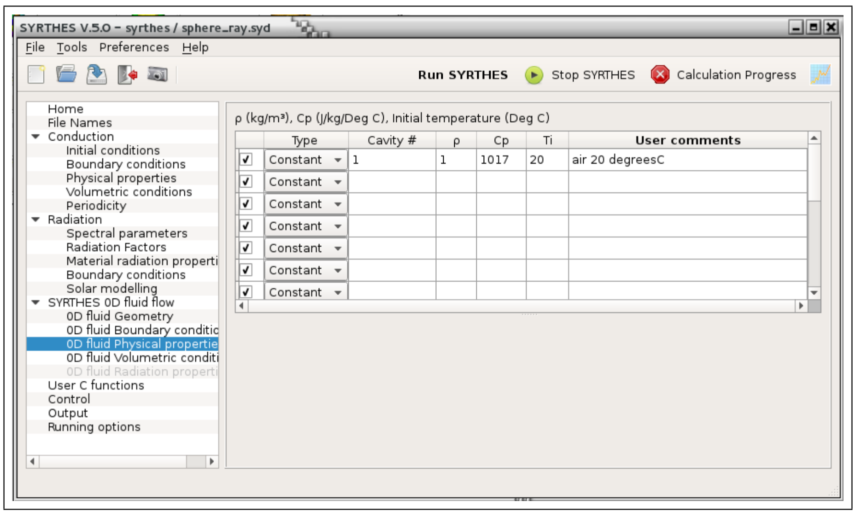

FIGURE 4.17 – SYRTHES - 0D Physical properties

Now, all physical and geometrical parameters are defined, you can jump to the Control section.

4.2.9 Control

We are going to compute the temperature during a transient lasting 1 hour.

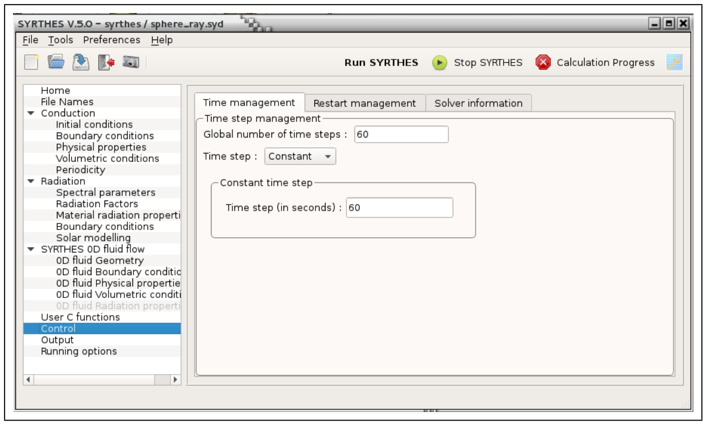

FIGURE 4.18 – SYRTHES - Control window

Generally, there is no need to change the default values provided for the solver.

4.2.10 Output

In this section, we are going to define the type of results to be generated by SYRTHES.
Whatever the options you will have a result file with the temperature on all the nodes of the mesh.
But, it is also advisable to define some thermal probes : during the run, you will be able to follow the temperature evolution at some strategic locations of the domain.
Set the coordinates of points in your domain. Then, define the frequency at which you want to record the values of the probes on file : here, 1 , which means every time step.

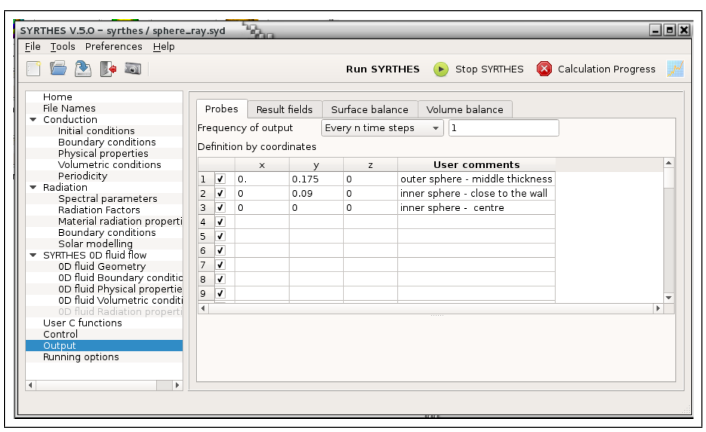

FIGURE 4.19 – SYRTHES - Defining thermal probes

Click on the next tab in the same window (Control) to select "result fields".
At the end of the calculation, you will get of course the temperature field corresponding to the last time step. But, you can also get intermediate results with a choosen frequency. Click "Fields" and set the frequency to 15 : you will get the temperature every 15 time steps.
These temperature fields will be recorded in the ".rdt" file and will be post-processed like the final result (".res" file).

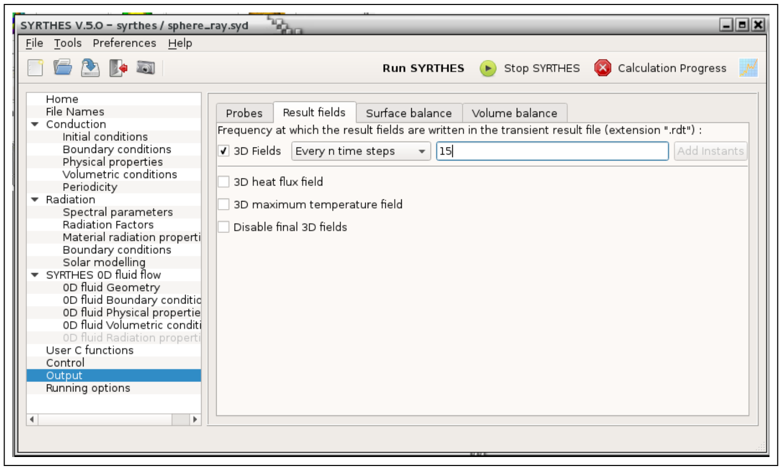

FIGURE 4.20 – SYRTHES - Defining transient results file

4.2.11 Running options

This is the last step : give a name for the listing file. SYRTHES will rpovide you with some information about the calculation (option summary, solver convergence,...)
For bigger cases, you could try a parallel computation, setting the number of processor to 3 or 4 or much more (but here, as the element number of this mesh is very low, there is little gain to be gained).

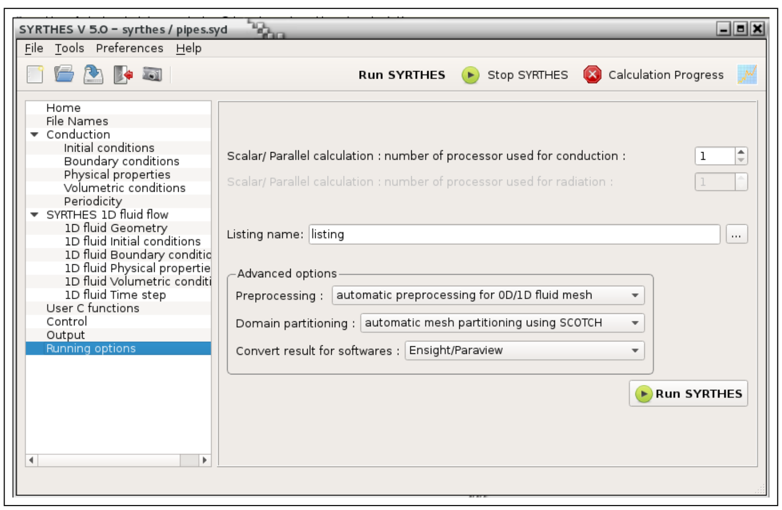

FIGURE 4.21 – SYRTHES - Running options window

4.2.12 You're ready to run SYRTHES !

Click "Run SYRTHES", the "calculation progress" window appears and you can display the evolution of the temperature at the 3 locations defined previously.

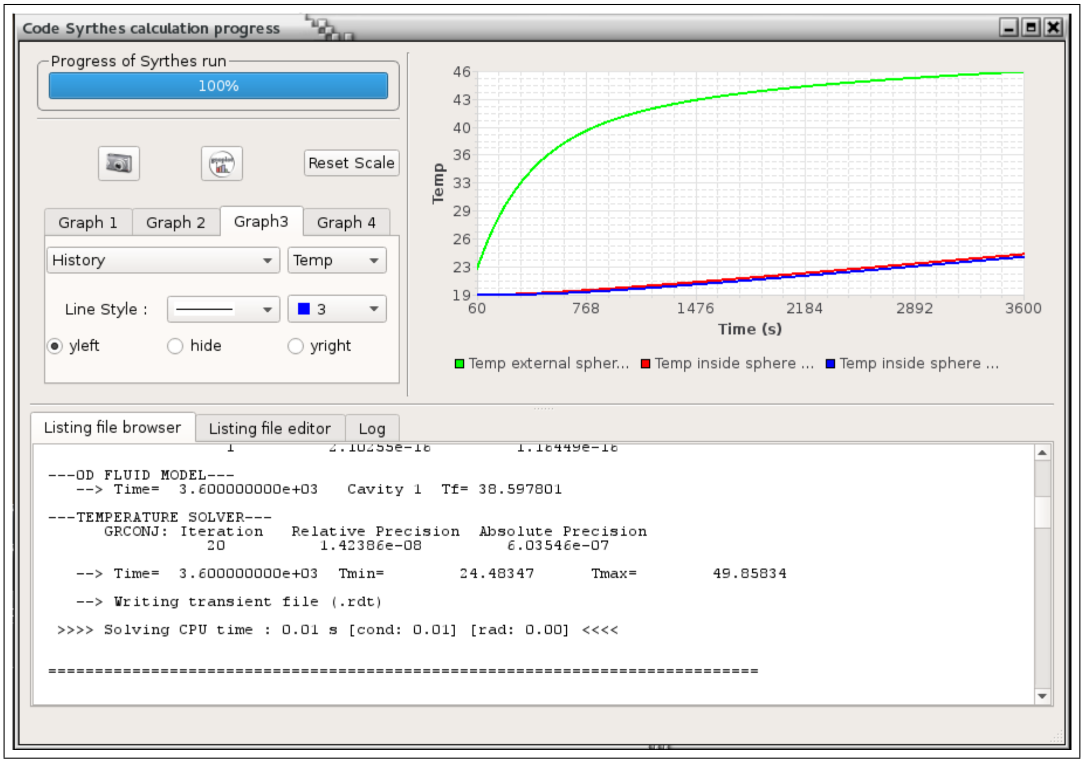

FIGURE 4.22 – SYRTHES Calculation progress window

The 3 tabs display :

- Listing file browser : displays the 200 last lines of the listings
- Listing file editor : whole SYRTHES listing file
- Log : system messages (usefull in case of error during user files compilation)

4.2.13 Analyzing the results

When SYRTHES calculation ends, you can visualize the results using a post-processor. For this example, we are using Ensight, but you can also use the open source post-processor Paraview.

4.2.13.1 Final temperature field

Run the post-processor Paraview and open the file : `mycase_sphere/syrthes/POST/resu1.ensight.case`

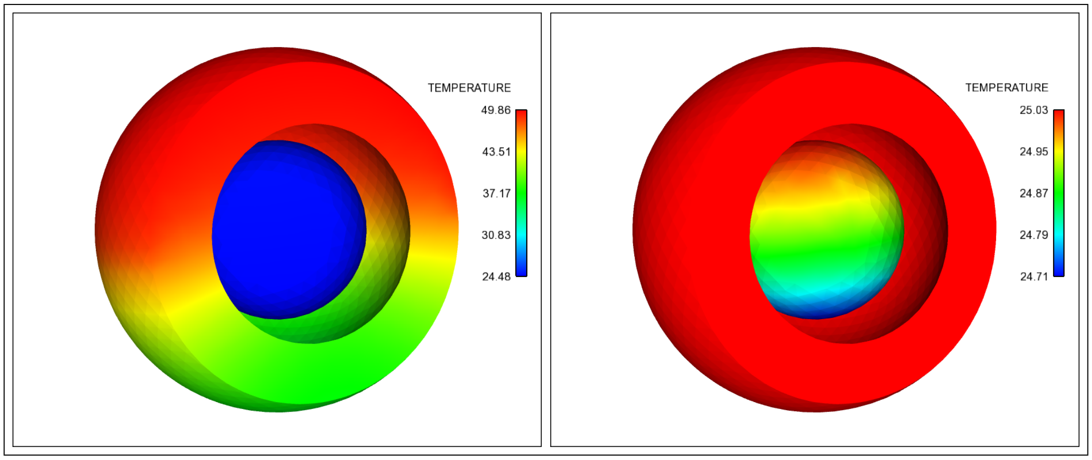

FIGURE 4.23 – Temperature after 1 hour (using 2 different scales)

4.2.13.2 Intermediate results

As we selected transient result file (see figure 4.20), we can also post-process this file.
Run the post-processor and open the file : `mkdir mycase_sphere/syrthes/POST/resu1_rdt.ensight.case`.
Then the temperature field can be visualized at different time step : 25,50,75,100, 200, 300, 400.
As time steps for the fluid and the solid structure computation are not identical, the intermediate time steps can't be considered as realistic fields, they are just intermediate states before reaching the steady state.

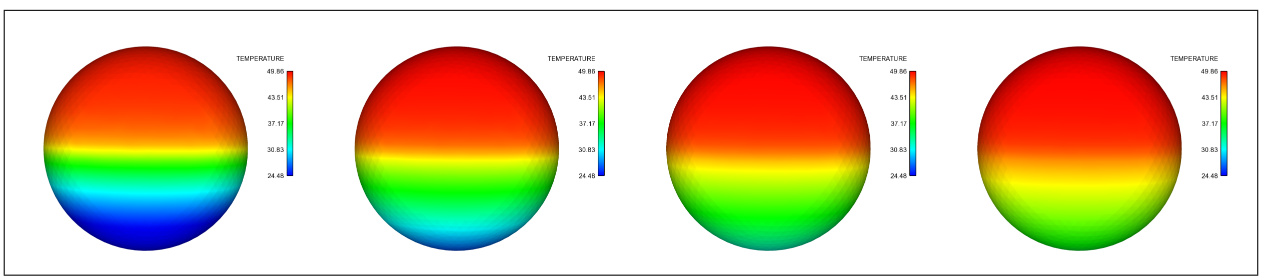

FIGURE 4.24 – Temperature fields after 100 and 200 time steps

4.2.13.3 Fluid results

Fluid results are saved into the file `resu1_f0d.res` (last value only) and `resu1_f0d.rdt` (values during the transient). You can plot the values with a 1D plotter like *gnuplot, excel,...).*

Run Paraview and open the file : `mycase_sphere/syrthes/POST/resu1_f1d.ensight.case`

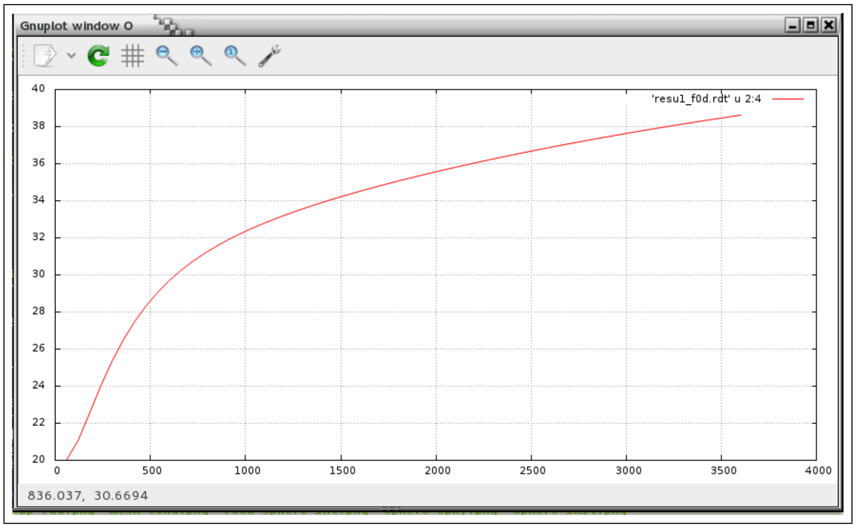

FIGURE 4.25 – Evolution of the fluid temperature

4.2.13.4 Result files

At the end of the run, in the directory where the calculation has taken place, one should be able to find several files :

- `resu1.res` : final result file
  Temperature field at the end of the calculation. This file is automaticaly converted to a post-processor file format (Ensight/Paraview or SALOME-MED) in the `POST` directory.
- `resu1.rdt` : transient result file Temperature field during the transient. This is an optional file (depending on the output options defined). If existing, it is automaticaly converted to a post-processor file format (Ensight/Paraview or SALOME-MED) in the `POST` directory.
- `resu1.his` : temperature probes
  This file contains the temperature at the probes initially defined. This is an "in columns" ASCII file.
  — column 1 : time
  — column 2 : temperature
  — columns 3-5 : coordinates (only columns 3-4 in 2D)
- `resu1.mnx` : min-max values
  The min and max values of each variable of the calculation and the place where there are reached. This is an "in columns" file. Content depends on the type of calculation and is described at the beginning of the file.
  Each variable is corresponding to a column. This file can be post-process with a 1D plotter.
  Example :

```
# 1=temps   2=T_min    3=x    4=y    5=z    6=T_max     7=x    8=y    9=z
#          10=rho_min 11=x   12=y   13=z   14=rho_max  15=x   16=y   17=z }
```

  In this file, you will find :
  — column 1 : time
  — column 2 : minimum of temperature
  — columns 3-5 : coordinates where is located the minimum of temperature

- columns 6 : maximum of temperature
- columns 7-9 : coordinates where is located the maximum of temperature
- column 10 : minimum of density
- columns 11-13 : coordinates where is located the minimum of density
- columns 14 : maximum of density
- columns 15-16 : coordinates where is located the maximum of density

- `resu1_f0d.res` and `resu1_f0d.rdt` : temperature of the 0D fluid flow model. Results are provided at the end of the calculation and for intermediate time steps.
- `resu1.add` : additionnal file This file is unused in this simple study. In other more complex cases, user may calculate some specific entities and save them in this file for further graphical post-processing.

Chapter 5

Working by yourself

Lost flux

5.1 What is the problem ?

We consider 3 underground pipes. Hot water at a temperature of $90°C$ flows through the pipes. We want to compute the flux transmitted by conduction through the ground and the energy lost at the surface.

5.1.1 Geometrical description

The solid domain (in 2D) consists of a rectangular portion of ground with the section of 3 pipes represented here by three holes.
Internal radius of each hole is 0.05 m and centers of each pipe are 0.2 m apart.

Geometrical characteristics are shown on figure 5.1

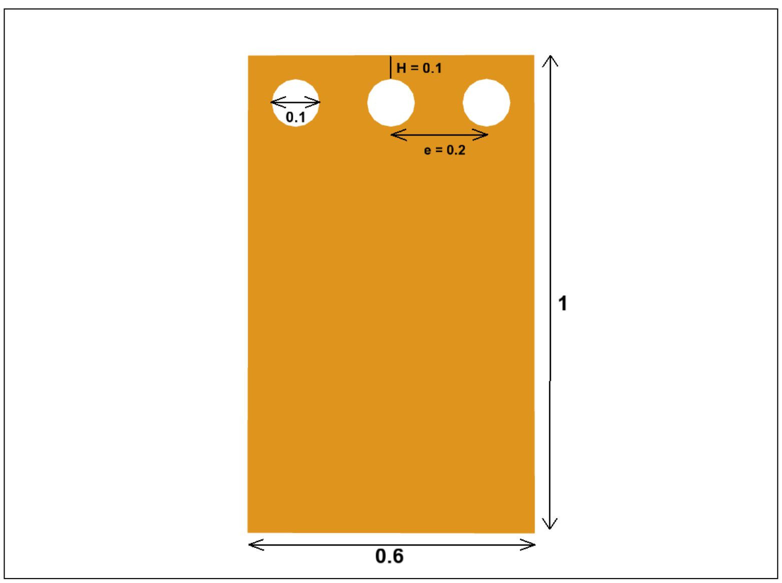

FIGURE 5.1 – Sketch of the problem

5.1.2 Physical description

Physical properties of the ground are :

- $\rho = 1250\ kg/m^3$,
- $C_p = 600\ J/kg°C$
- $k = 1\ W/m/°C$

5.1.3 Initial conditions

The initial temperature is $20°C$.

5.1.4 Boundary conditions

Boundary conditions are :

- pipes boundaries : $T_p = 90°C$, $h_p = 10000\ W/m^2/°C$
- surface : $T_{ext} = 20°C$, $h_{ext} = 5\ W/m^2/°C$
- other surfaces : adiabatic

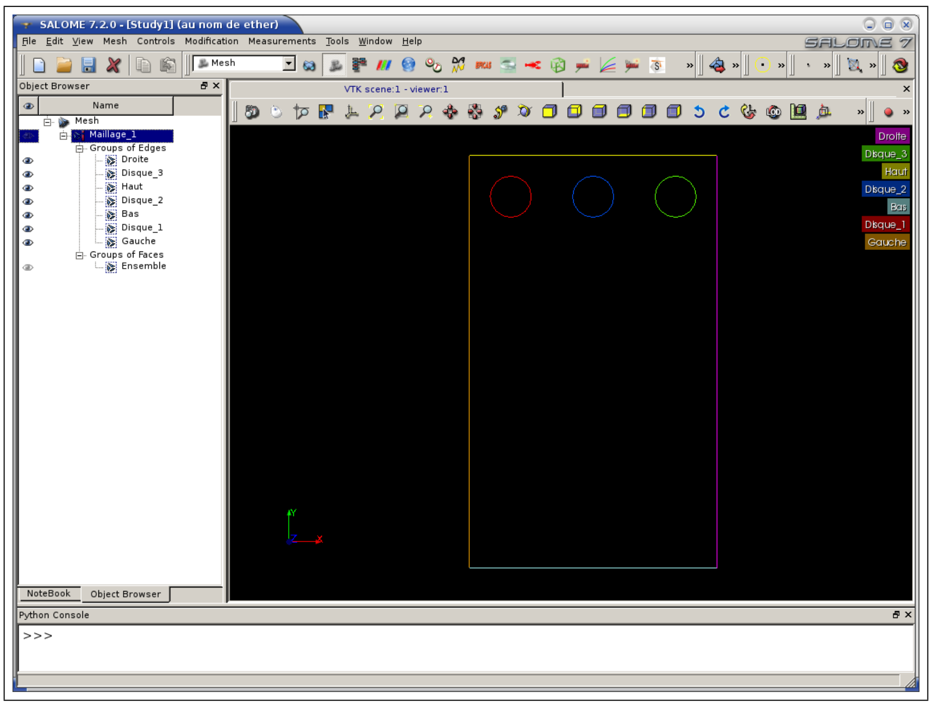

FIGURE 5.2 – Group names for volumes and boundary conditions

5.2 Mesh provided

Mesh can be found in the SYRTHES distribution :
`/..../syrthes5.x/arch/`*your_ arch*`/share/syrthes/tests/5-cas_flux`. The 2D mesh counts 6822 triangles. Groups have been created for material and boundary conditions.

WARNING : depending on SALOME version, pairs (group-name , number) could defered. So have a look on your description file and adapt numbers when going on.

Here is `flux.syr_desc` provided file :

```
group_of_edges    4    Gauche
group_of_edges    5    Disque_1
group_of_edges    6    Bas
group_of_edges    7    Disque_2
group_of_edges    8    Haut
group_of_edges    9    Disque_3
group_of_edges    10    Droite
group_of_faces    11    Ensemble
```

5.3 Approached theorical solution

For a row of pipes of lenght L and radius r, spaced by e and at a depth H, an approximation of the flux between surfaces T_p and T_{ext} is given by :

$$\phi = \frac{T_p - T_{ext}}{\frac{e}{2\pi\lambda}\ ln\left(\frac{e}{\pi r}\ sh\left(\frac{2\pi H}{e}\right)\right) + \frac{1}{h_{ext}}}$$

Warning : this formula supposes that conductivity and temperatures at the surface are constants (and generally, it's not the case !).

5.4 Results

Figures 5.3 and 5.4 shows the temperature in the ground at convergence.

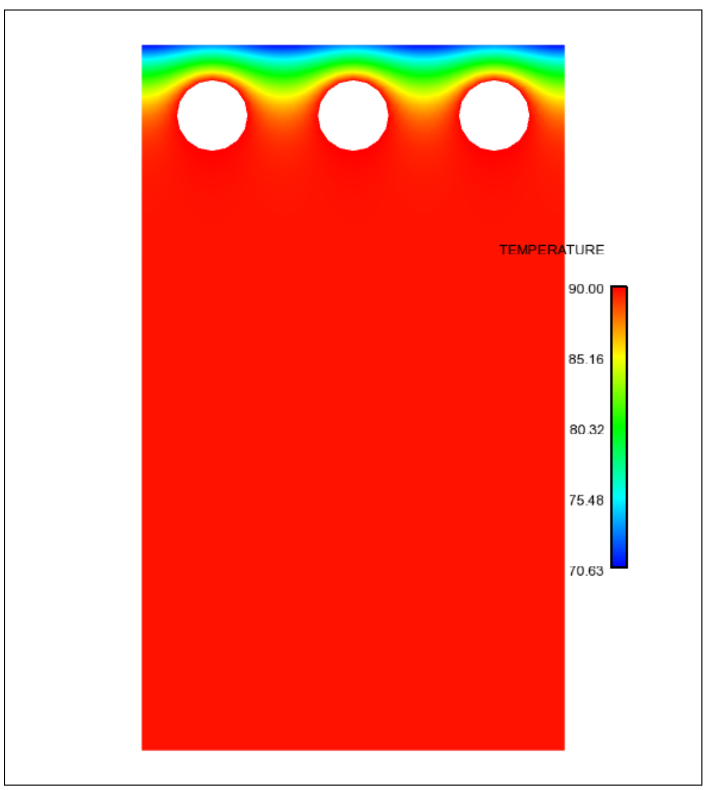

FIGURE 5.3 – Ground temperature

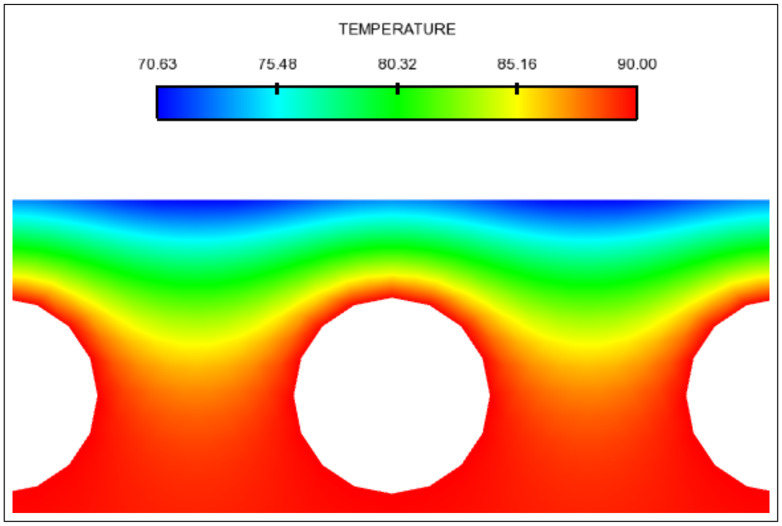

FIGURE 5.4 – Ground temperature - zoom around the surface

Using the approached theorical formula : $\phi_T = 245.12\ W/m^2$. Then, the surface temperature can be evaluated :

$$T_{surface} = \frac{\phi}{h_{ext}} + T_{ext} = \frac{245.12}{5} + 20 = 69.02$$

SYRTHES simulation gives : $\phi_S = 155.922/0.6 = 259.87 W/m^2$ The computed value is likely to be more precise because, as shown in figure 5.5 the temperature isn't constant along the upper surface (as it was approximated in the formula).

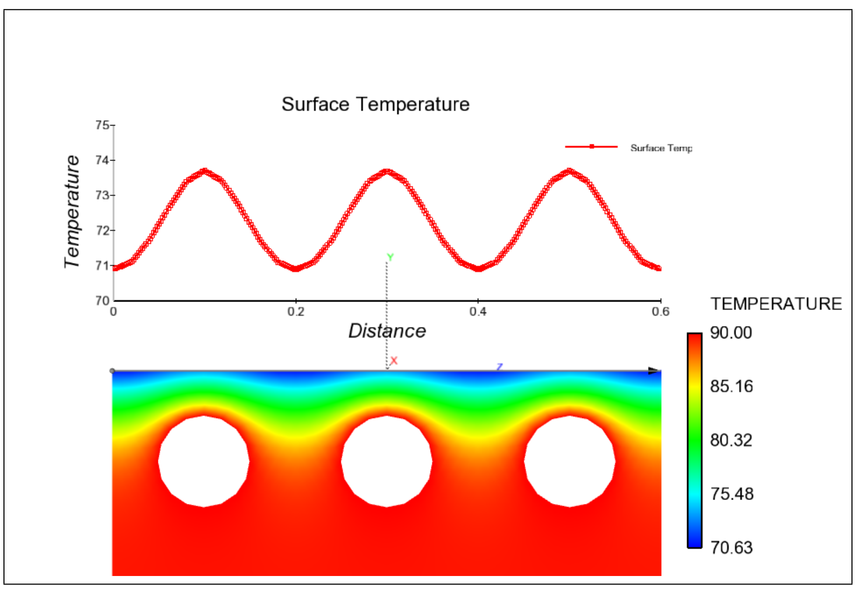

FIGURE 5.5 – Surface temperature