SYRTHES 4.2
Tutorial

I. Rupp, C. Peniguel

2014

AVERTISSEMENT / CAUTION

Contents

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 anisitropic.

1.1.1 Geometrical description

The solid domain constists in three separated disks with an hole in the center.
Internal radius of the hole 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

The density and the 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, the conductivity is isotropic : 25 $W/m/°C$
- For the disk 2, the conductivity is orthotropic : 25 $W/m/°C$ along the direction x, 5 $W/m/°C$ along the direction y, and 25 $W/m°C$ along the direction z along the axis y.
- For the disk 3, the 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 $W/m^2/°C$

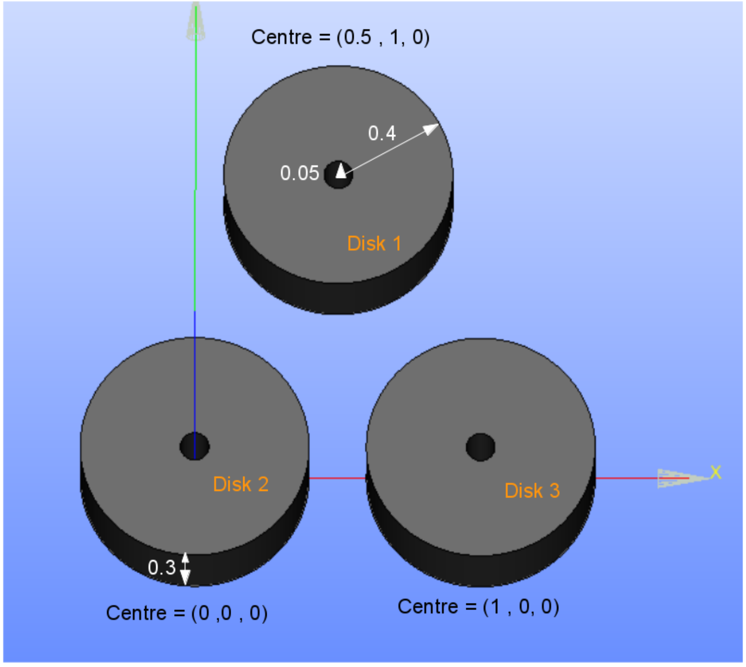

Figure 1.1: Sketch of the problem

- other surfaces : adiabatic

1.2 How to do that ?

1.2.1 To organize the study

We propose in this section an arrangment of the different files of your study. This is only advice, and for further use, you could do as you 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 creat 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.

The conduction mesh counts 23667 nodes and 120722 elements (4-nodes tetrahedra).

You can create your own mesh, but bellow, we describe the characteritics of the mesh provided with this tutorial.

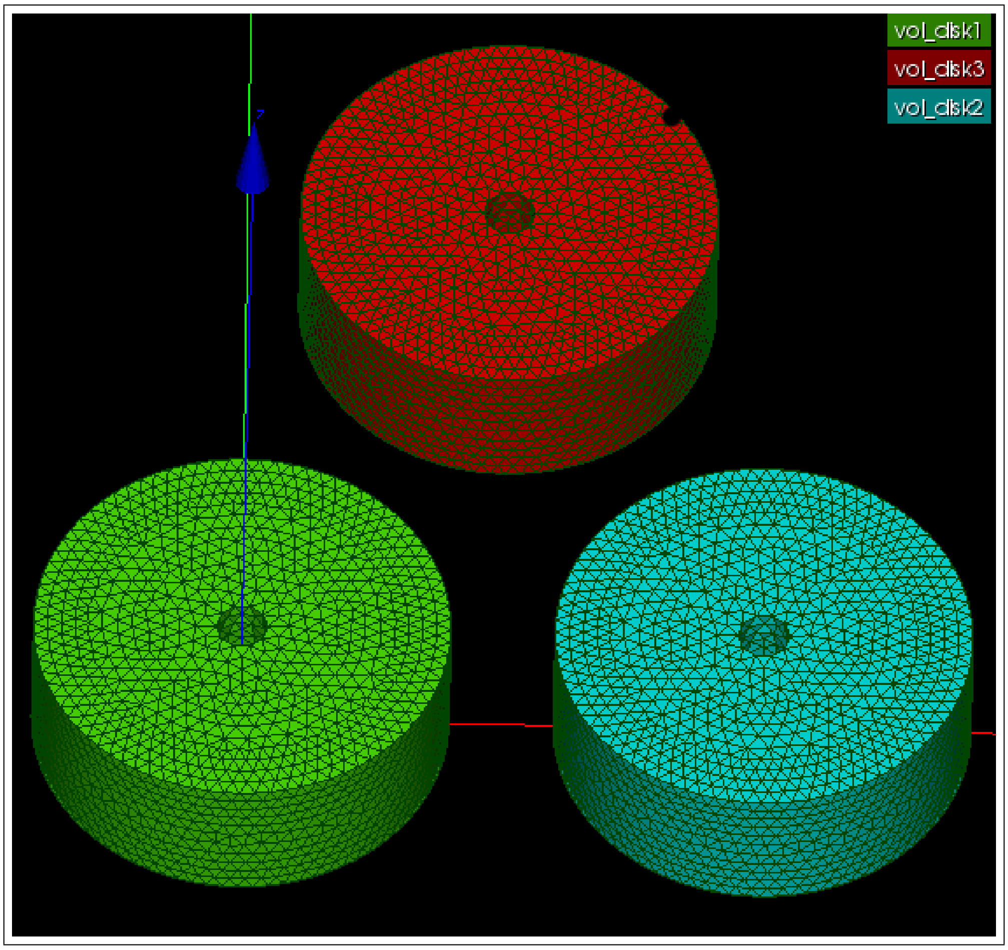

Figure 1.2: Group names for volumes

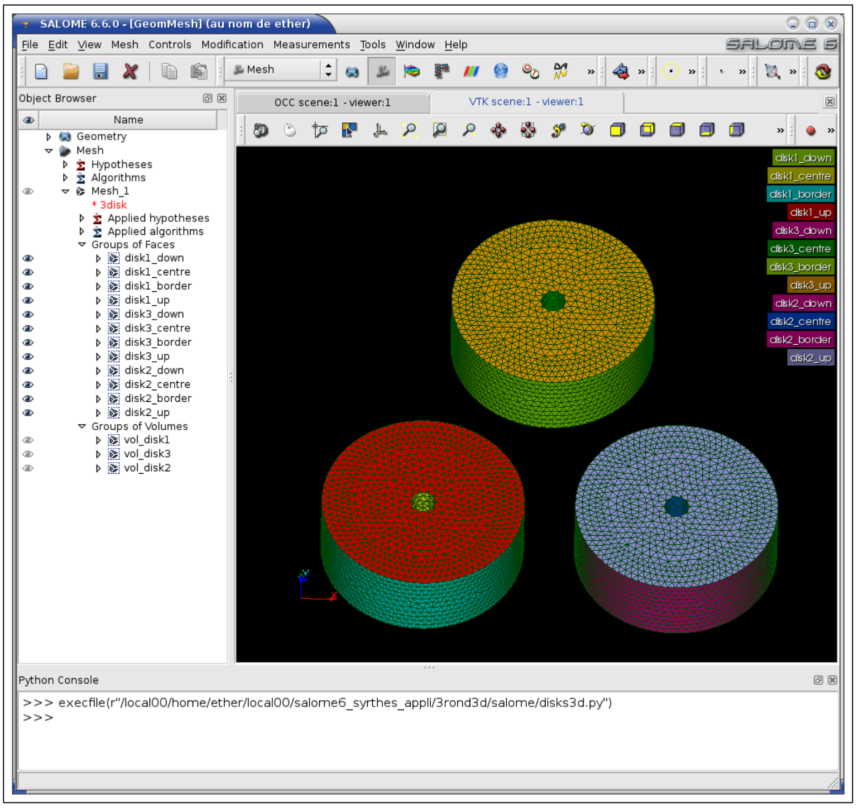

Figure 1.3: Group names for surfaces

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 /.../syrthes4.1/arch/myarch/bin/syrthes.profile`
Run the SYRTHES-gui : `syrthes.gui`

Figure 1.4: SYRTHES Managing your cases

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.

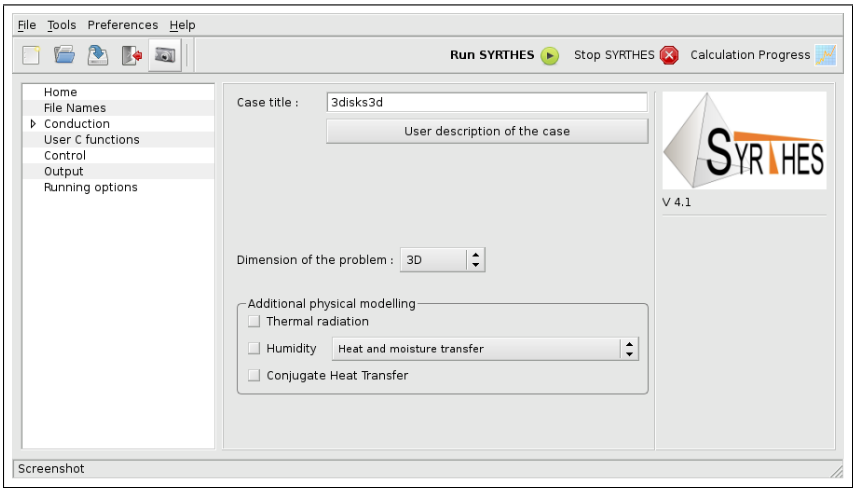

Figure 1.5: SYRTHES Main View

Save your data file :

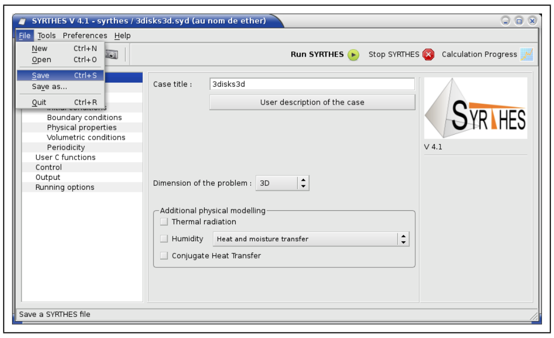

Figure 1.6: SYRTHES Save your data file

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 the you get the message :

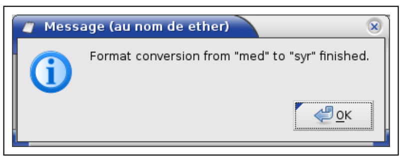

Figure 1.7: SYRTHES File format conversion OK

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 :

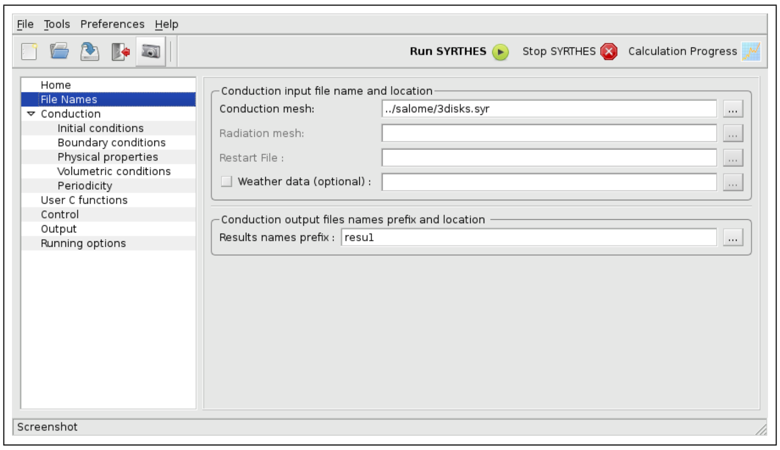

Figure 1.8: SYRTHES File Names window

1.2.6 Input data for conduction computation

1.2.6.1 Group names and references numbers

As 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 `3disks.med`. The links between both are given in an additional file (.syr_descr) created while you have converted the MED file to the SYRTHES file.

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 my `3disks.syr_desc` 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 concerned by the same initial condition, you can put "-1" instead of the list. You can add a comment in the last column (optional).

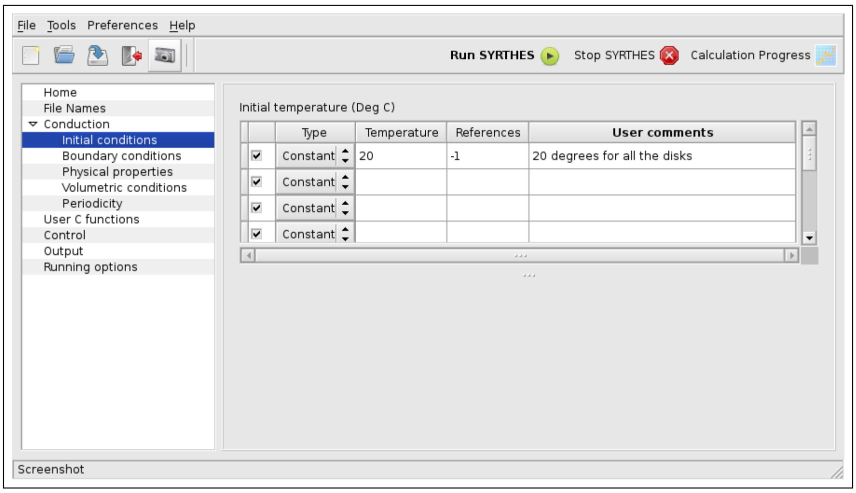

Figure 1.9: SYRTHES - Initial conditions

1.2.6.3 Boundary conditions

We would like 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, 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,)

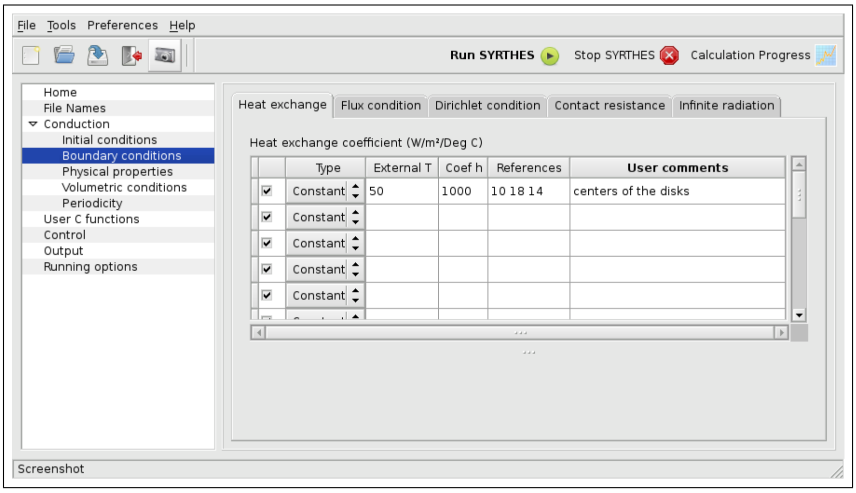

Figure 1.10: SYRTHES - Boundary conditions

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.

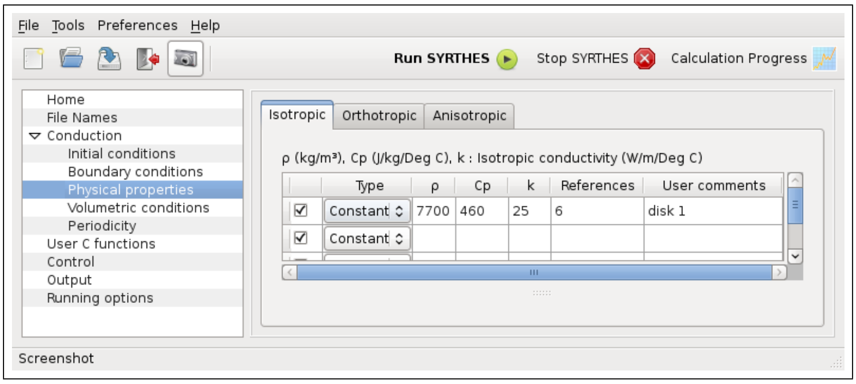

Figure 1.11: SYRTHES - Isotropic conductivity

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.

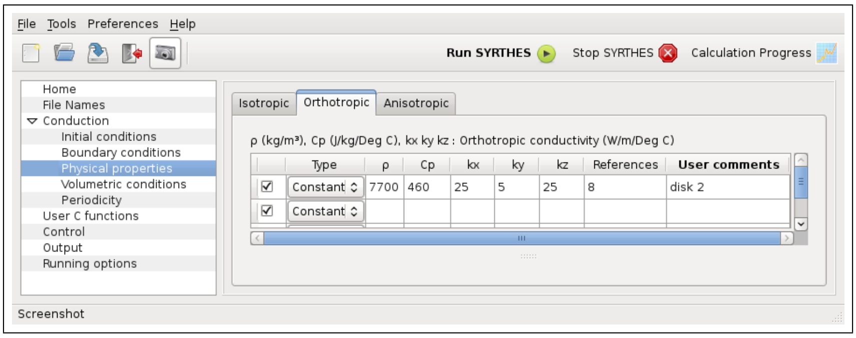

Figure 1.12: SYRTHES - Orthotropic conductivity

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

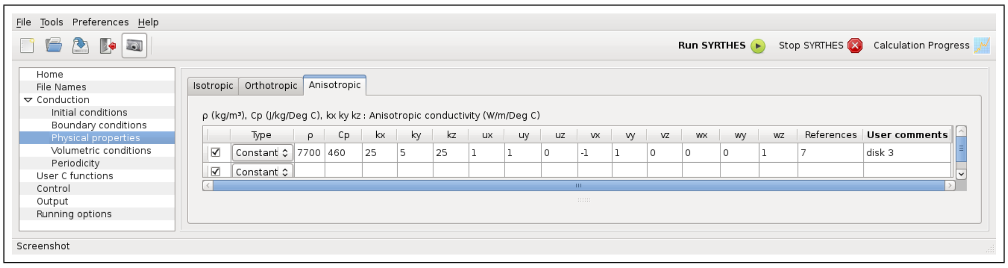

Figure 1.13: SYRTHES - Anisotropic conductivity

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

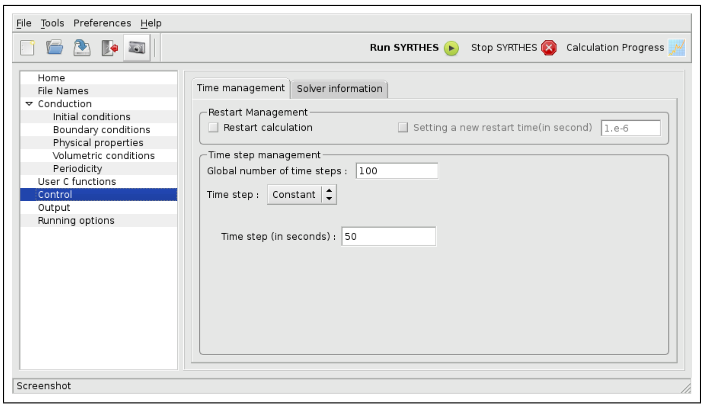

Figure 1.14: SYRTHES - Control window

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 the 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 1 point per disk.
Then, define the frequency you want to save the values of the probes on file : here, 1 = every time step.

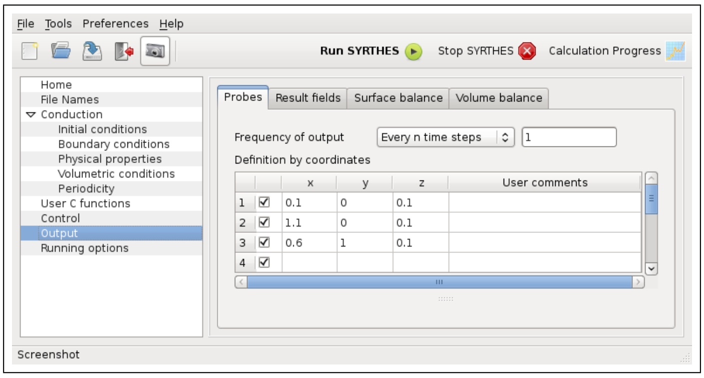

Figure 1.15: SYRTHES - Defining thermal probes

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).

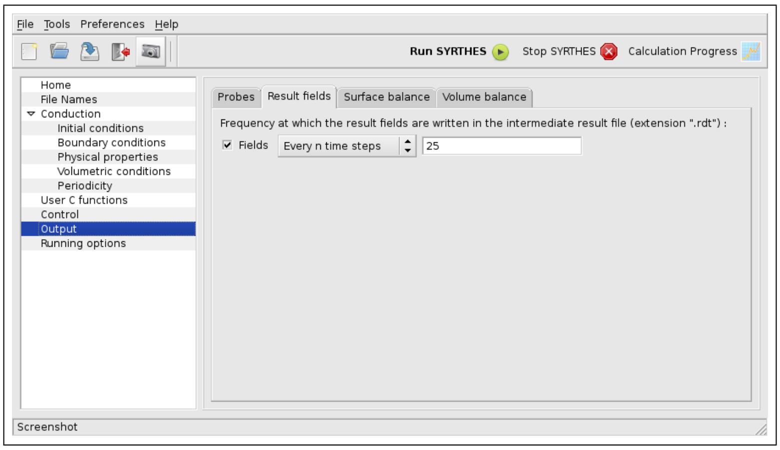

Figure 1.16: SYRTHES - Defining transient result file

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,...)
Further, you could try a parallel computation, setting the number of processor 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).

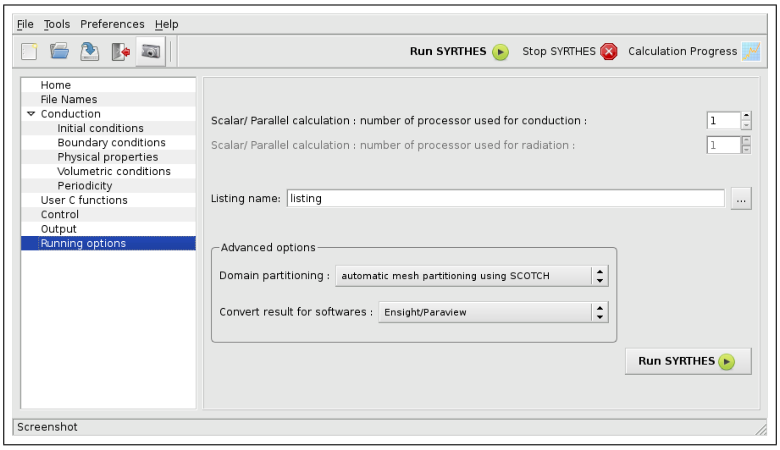

Figure 1.17: SYRTHES - Running options window

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.

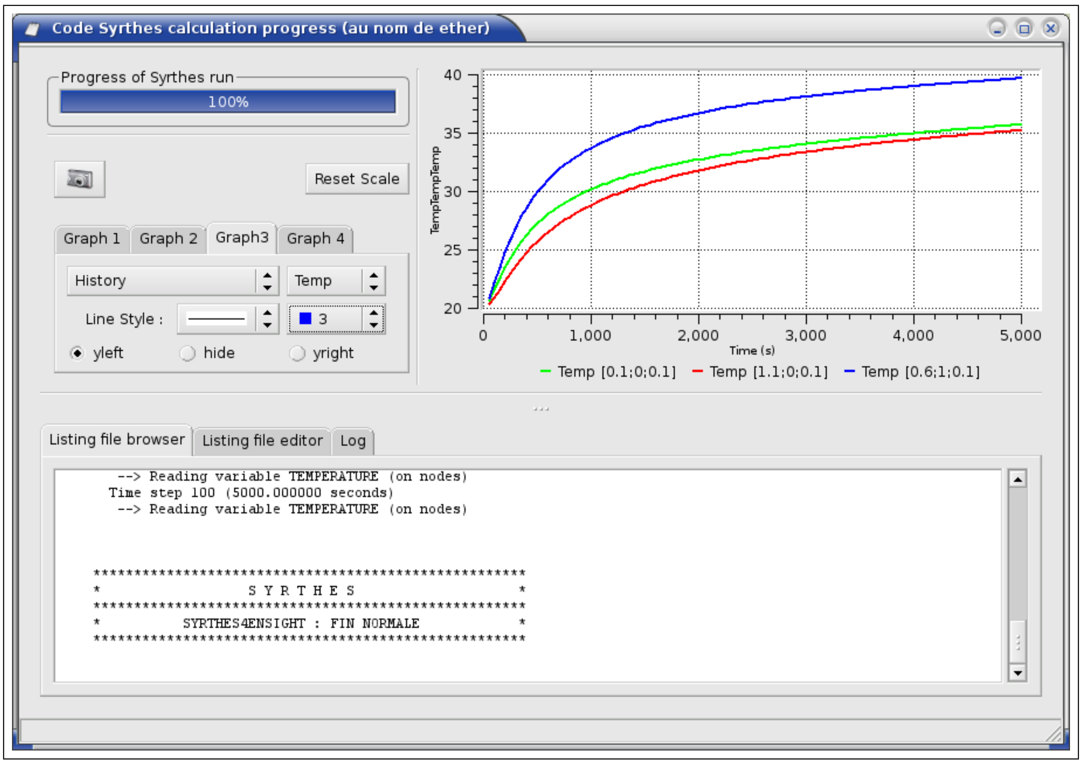

Figure 1.18: SYRTHES Calculation progress window

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)

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 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 in-

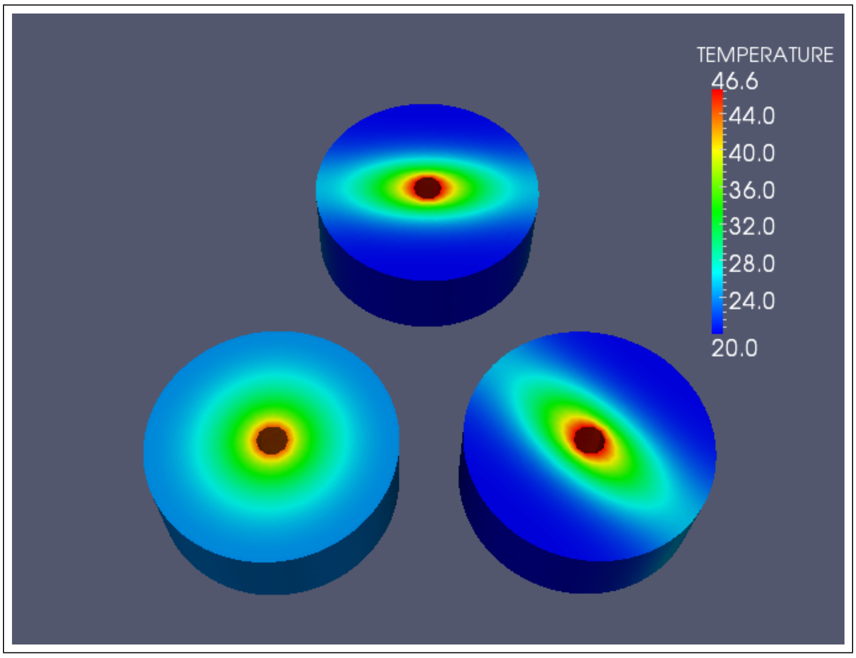

Figure 1.19: Temperature field after 5000 s

teresting to note the behaviour difference between the different materials. The isothermals of the disk 1 stay concentric, while for the others, ellipses appears before being affected by the outer boundary condition. 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 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 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)
  - columns 6-7 : unused
  - columns 8 : element number where the probe is located
- `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.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 and 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 $W/m^2/°C$

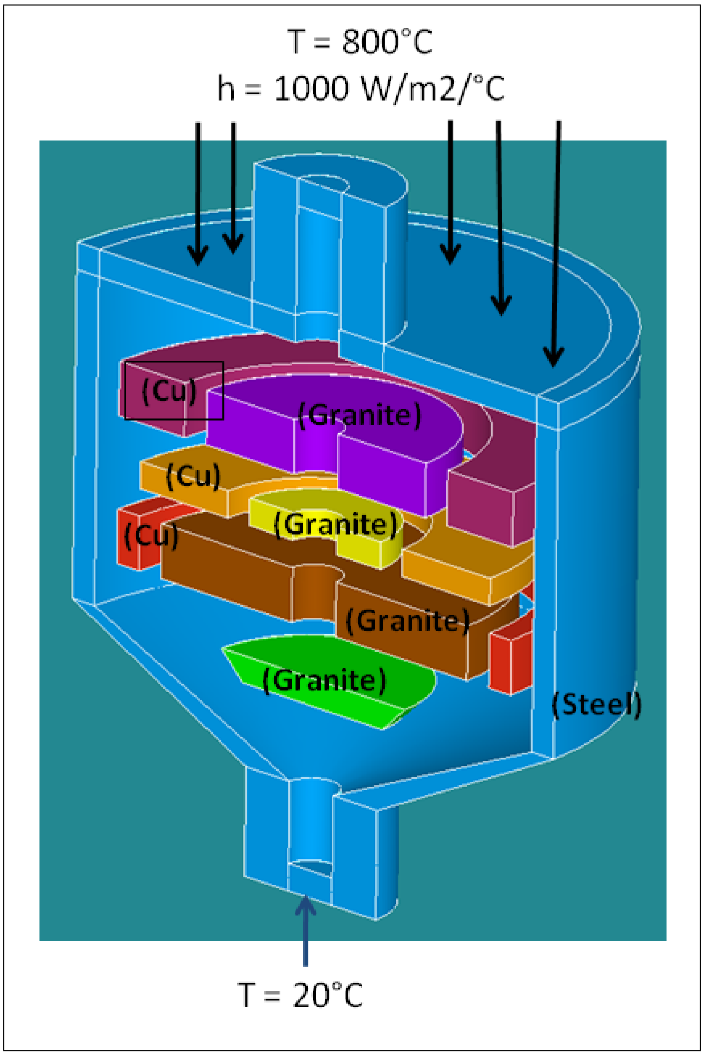

Figure 2.1: Sketch of the problem

- other extern surfaces of the oven : adiabatic
- all the surfaces inside the oven : radiation exchange

2.2 How to do that ?

2.2.1 To organize the study

We propose in this section an arrangment of the different files of your study. This is only advice, and for further use, you may do as you 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.
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 with this tutorial.

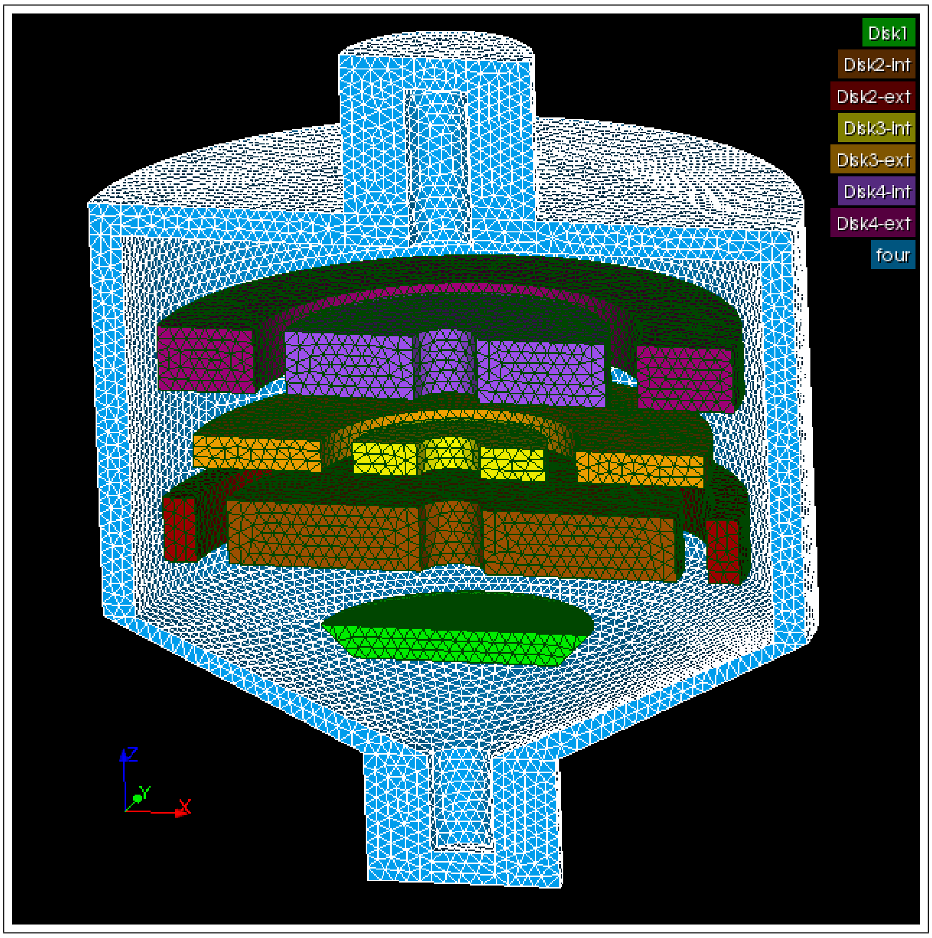

Figure 2.2: Group names for volumes

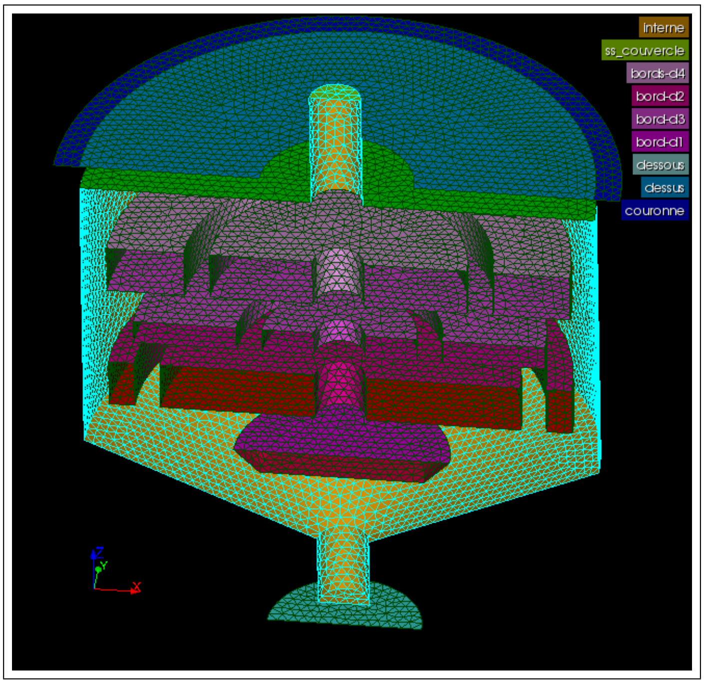

Figure 2.3: Group names for surfaces

2.2.3 Creating the radiation mesh

The radiation mesh consists of the inner surfaces of the oven and the 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.

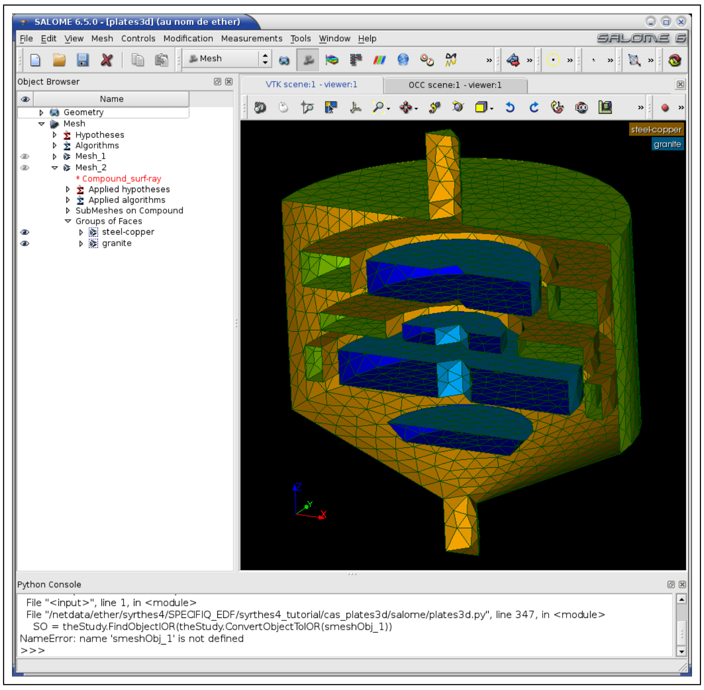

Figure 2.4: Radiation mesh

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.

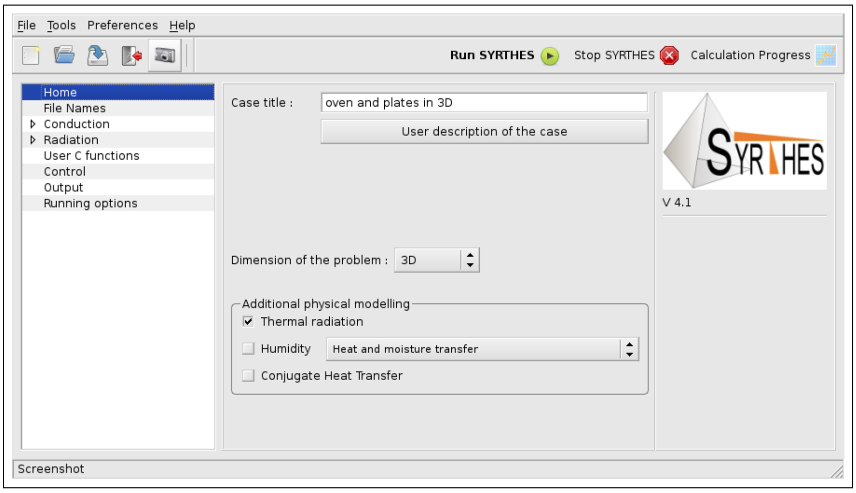

Figure 2.6: SYRTHES Main View

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

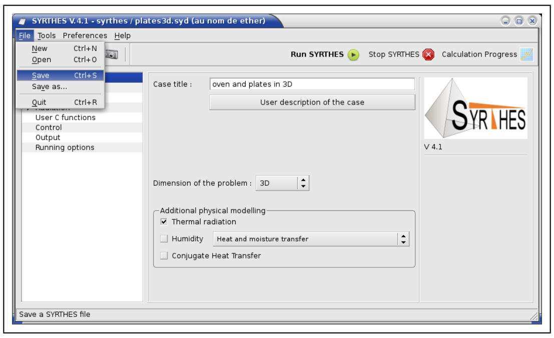

Figure 2.7: SYRTHES Save your data file

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 the format (here from MED to SYRTHES) is done automatically and the you get the message :

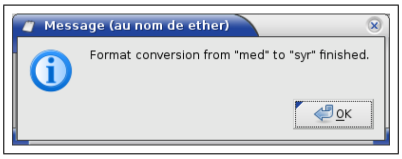

Figure 2.8: SYRTHES File format conversion OK

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 looks like :

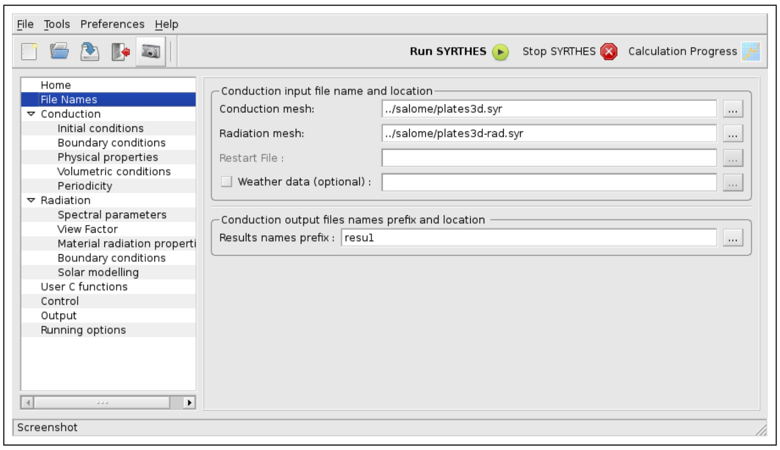

Figure 2.9: SYRTHES File Names window

2.2.7 Input data for conduction computation

2.2.7.1 Group names and references numbers

As 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.

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 my `plates3d.syr_desc` 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 can may set "-1" instead of the list. You can add a comment in the last column (optional).

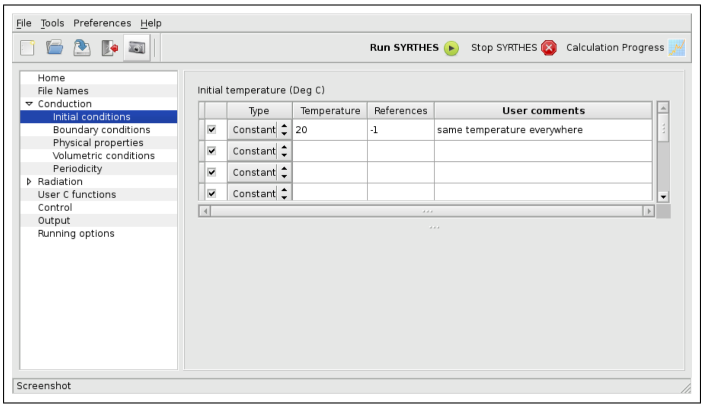

Figure 2.10: SYRTHES Initial conditions

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

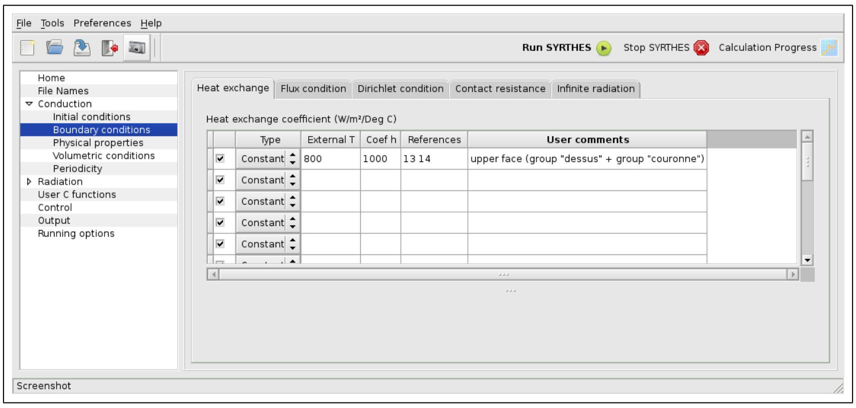

Figure 2.11: SYRTHES Boundary conditions : heat exchange

Click on the "Diricihlet condition" tab and set the imposed temperature :

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

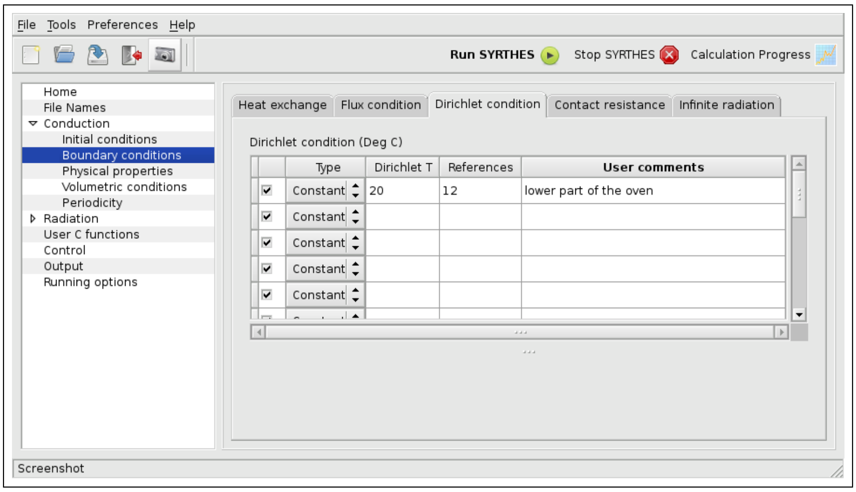

Figure 2.12: SYRTHES Boundary conditions : T imposed

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 :

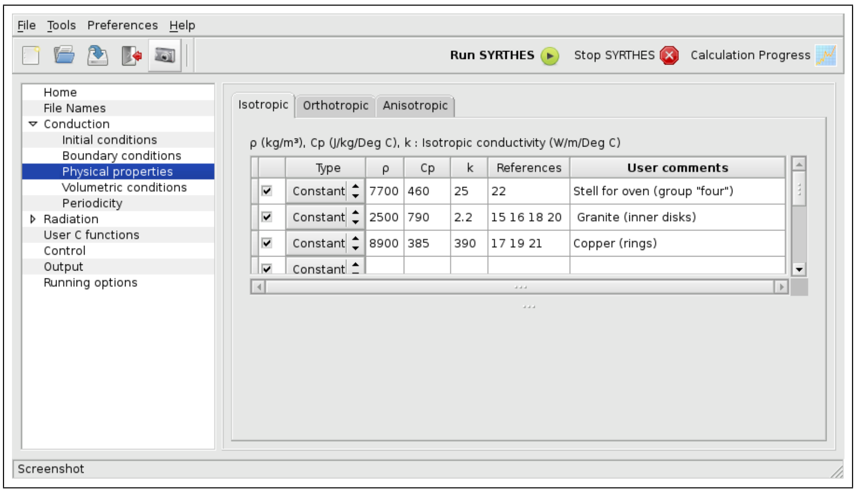

Figure 2.13: SYRTHES Material properties

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.

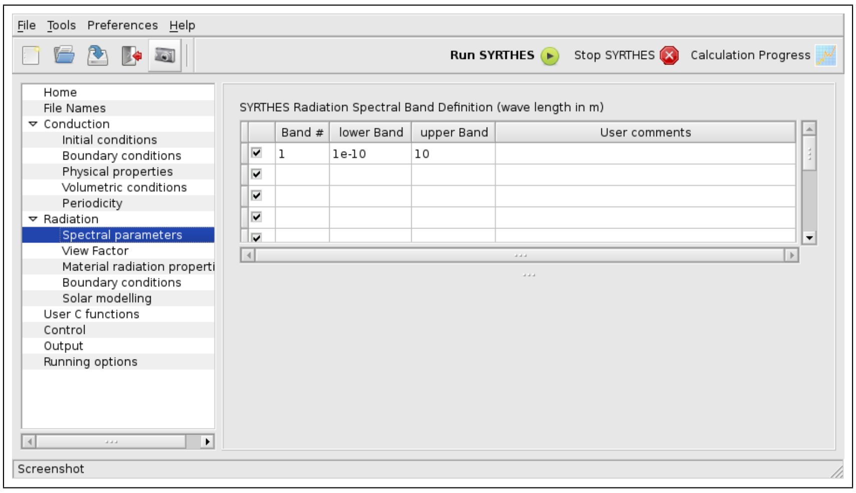

Figure 2.14: SYRTHES Spectral bands

2.2.8.2 View factors

For a given surfacic mesh, SYRTHES can't determine what is inside or outside your computational domain (because mesh generators don't orient the surfacic faces of the mesh). For example, if your radiation mesh is a ball, 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)

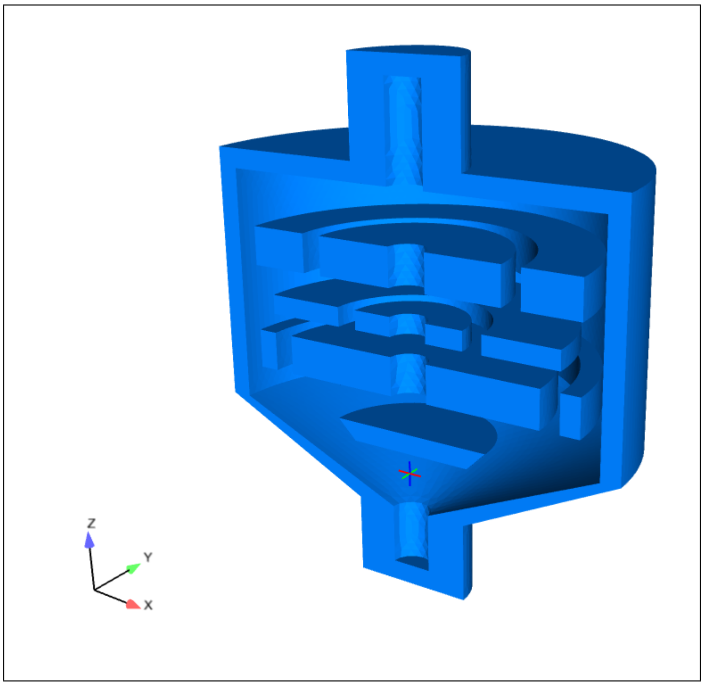

Figure 2.15: SYRTHES Position of the inner point for view factors calculation

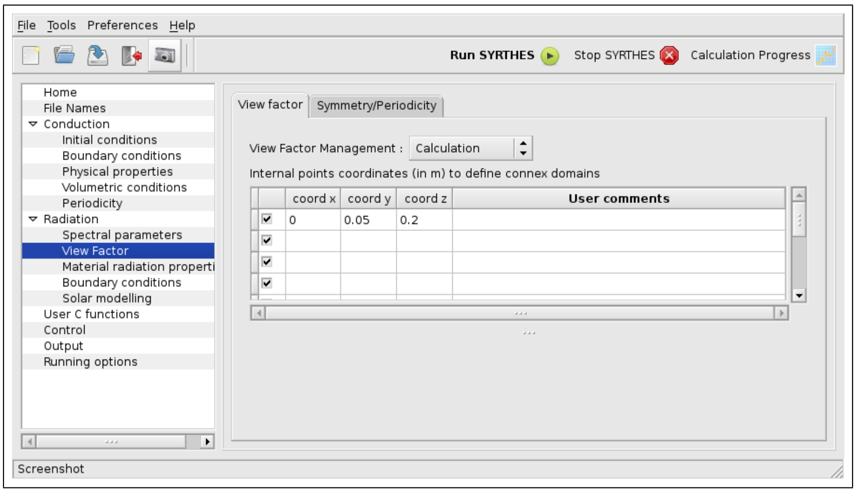

Figure 2.16: SYRTHES View factors

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. SYRTHES can handle symmetrical geometries as our case, and you just have to define the position of the symmetry plane.
Click the "symmetry/periodicity" tab and set the coefficient of symmetry plane equation ($y = 0$).

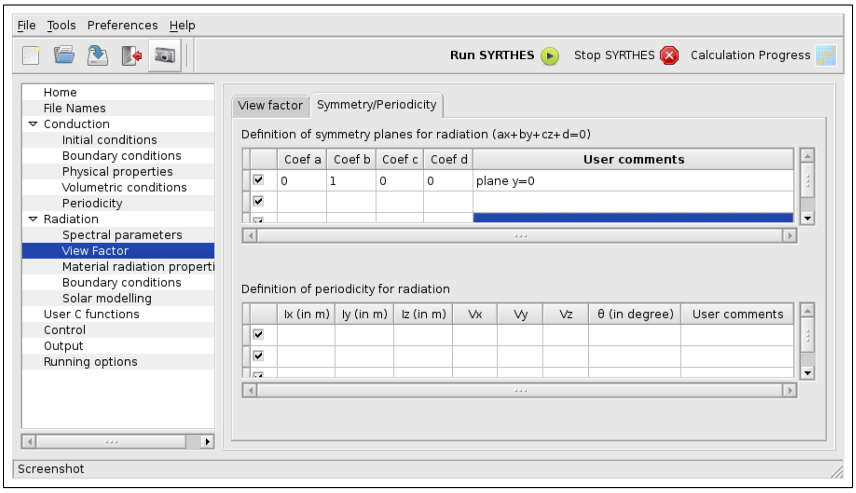

Figure 2.17: SYRTHES View factors

2.2.8.3 Material radiation properties

In this case, only 1 spectral band is defined, and we condider 2 different emissivities : 0.8 for steel and copper, and 0.96 for granite.

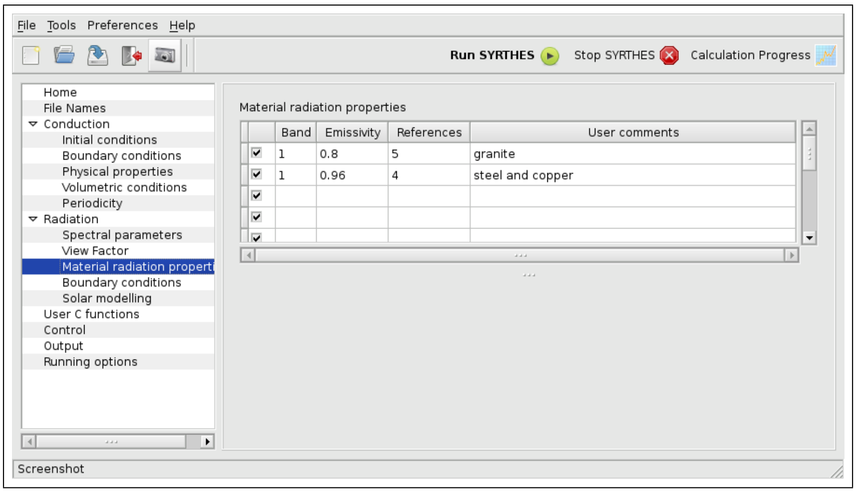

Figure 2.18: SYRTHES Emissivity definition

2.2.8.4 Boundary conditions

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

- the references of the faces of the radiation mesh which are coupled with the conduction mesh. In this case, it's easy, all the 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 border faces of the conduction mesh which are coupled with the radiation mesh. In the present case, it corresponds to all border surfaces inside the oven : 11 9 10 8 6 7

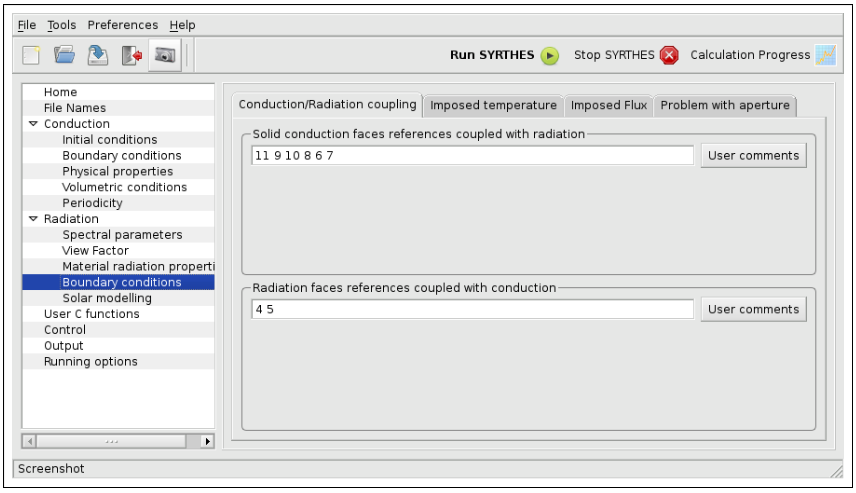

Figure 2.19: SYRTHES Coupling conduction and radiation

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

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.

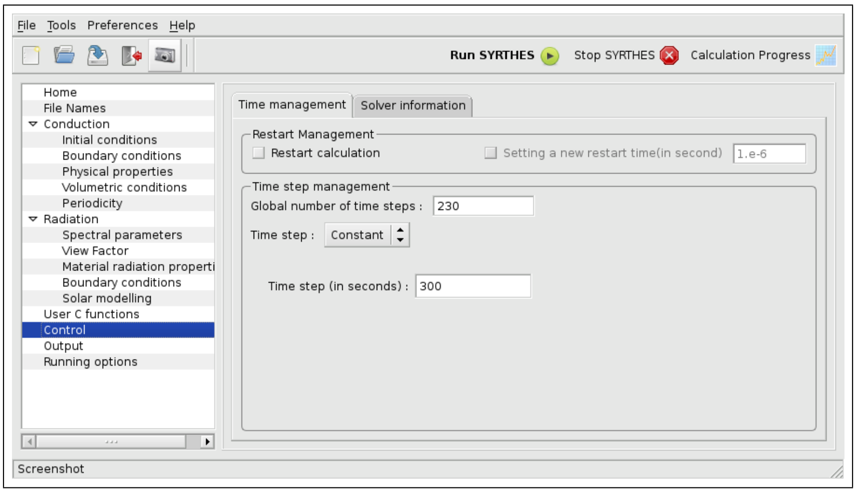

Figure 2.20: SYRTHES Control window

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 mesh.
But, sometimes, it is also advisable to define some thermal probes : during the run, you will be abble to follow 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 you want to save the values of the probes on file : here, 1 = every time step.

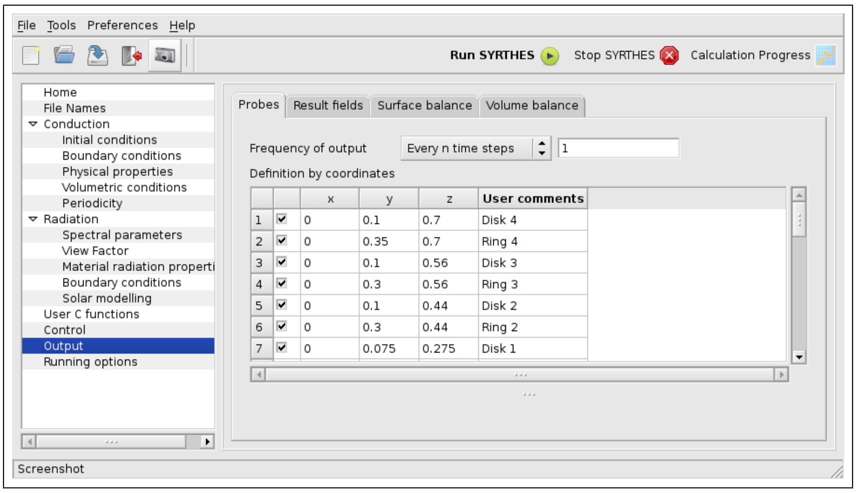

Figure 2.21: SYRTHES Defining probes

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. 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).

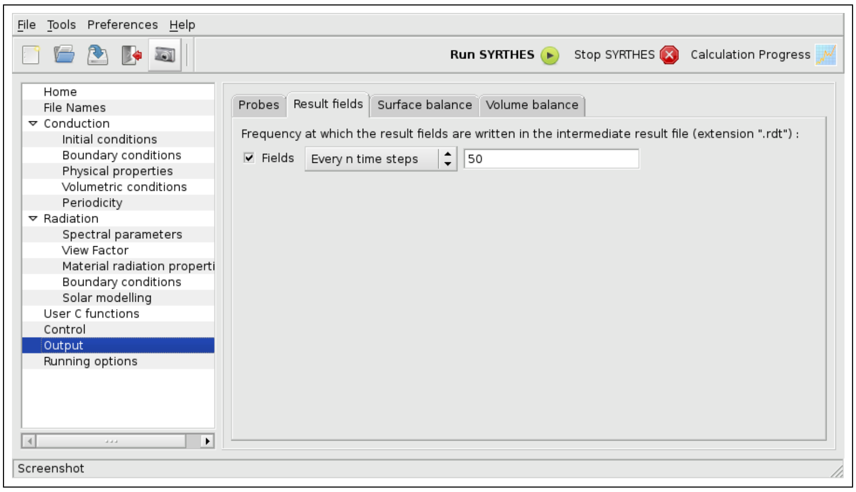

Figure 2.22: SYRTHES Defining transient result file

2.2.11 Running options

This is the last step : give a name for the listing file. 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 element number of this mesh is quite very low, the benefit in term of CPU time will be low as well).

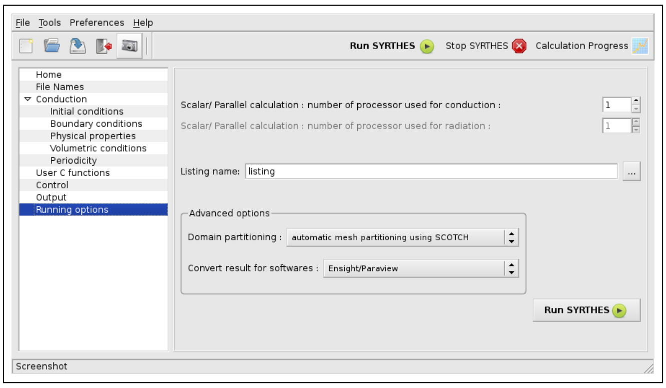

Figure 2.23: SYRTHES Running options window

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.

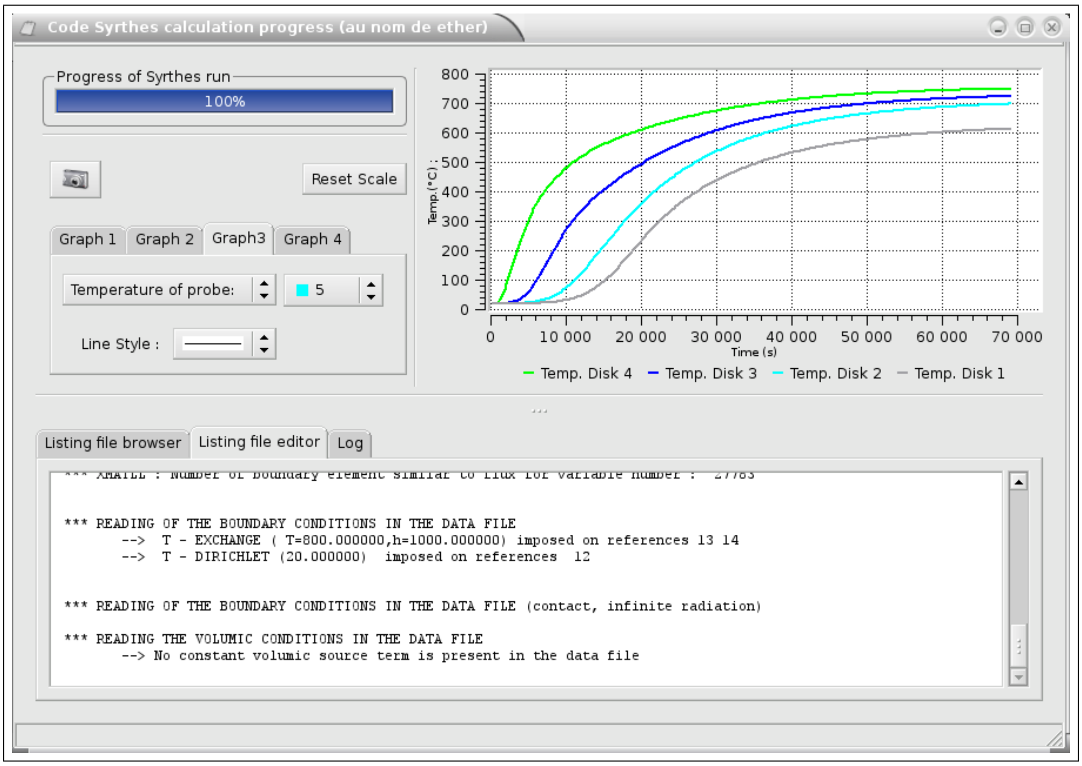

Figure 2.24: SYRTHES Calculation progress window

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 Paraview (but you can also use Ensight or even your own post-processor provided that you have written the 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

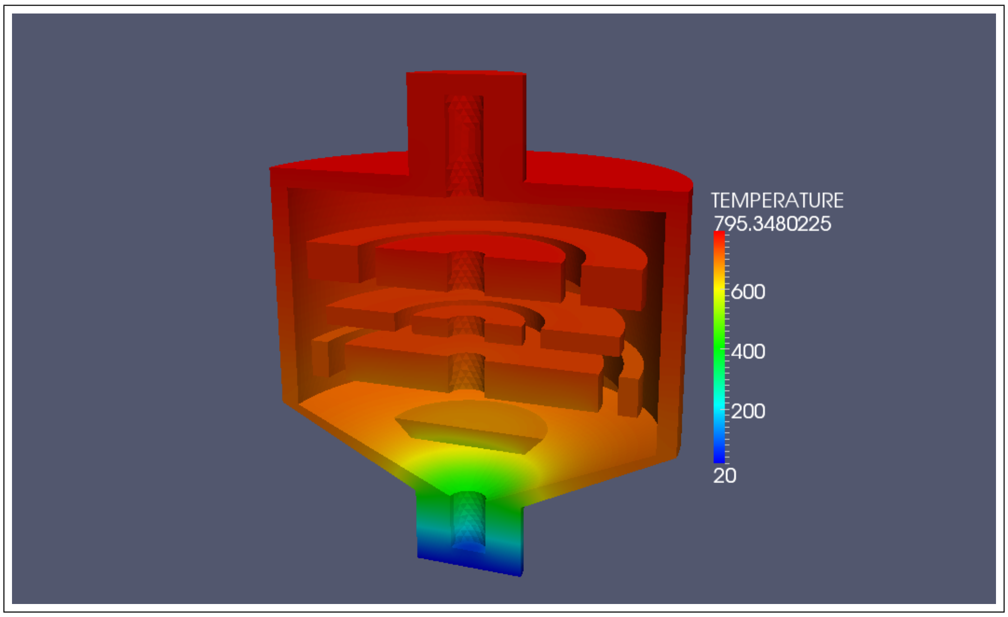

Figure 2.25: Temperature field after 69000 s

2.2.13.2 Transient temperature field

As we selected transient result file (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.

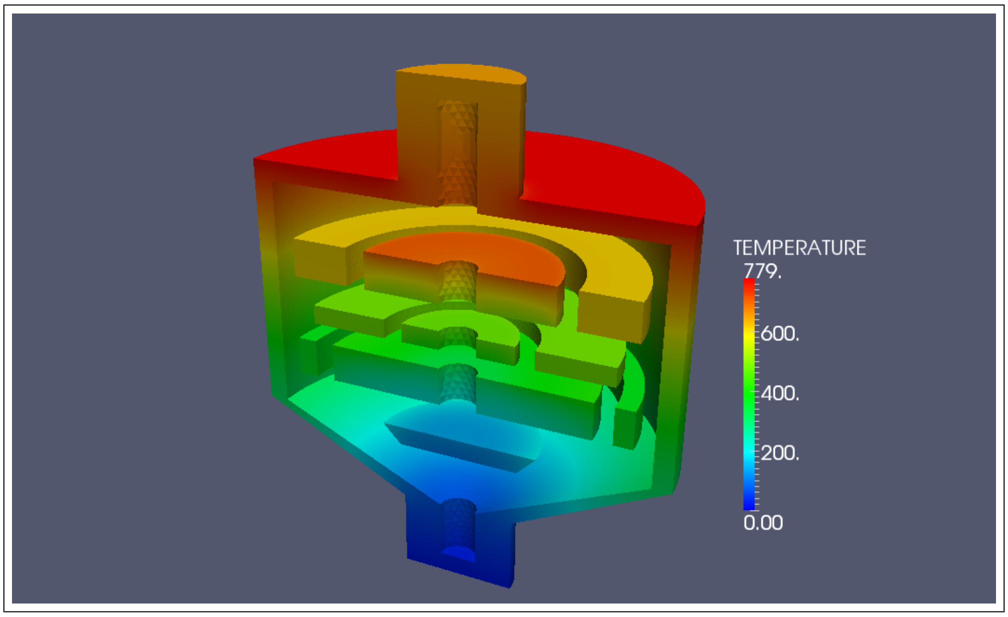

Figure 2.26: Temperature field after 15000 s

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)
  - columns 6-7 : unused
  - columns 8 : element number where the probe is located

- `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.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

Conduction and radiation in an oven SYRTHES in the SALOME-platform

plates3d

3.1 What is the problem ?

This test-case is exactly the same as in the previous chapter. But here, you are going to use SYRTHES using the SALOME environment.

Here is just a short description of the problem, all the parameters are provided in the previous chapter.

We want to determine the temperature field inside an oven. 7 disks or rings are located in the oven. The oven is supposed to be heated at the top, and the bottom is maintained at 20°C.

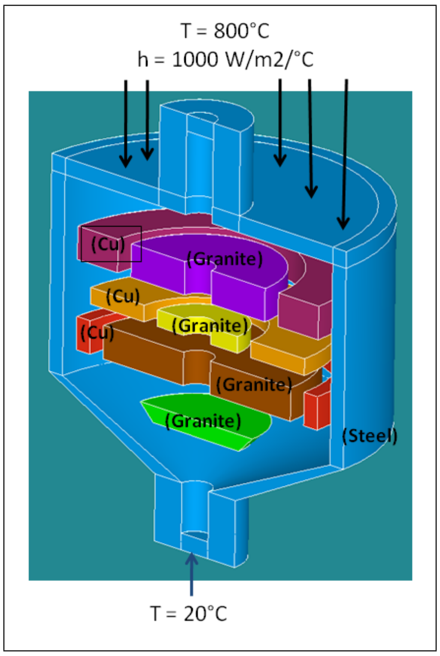

Figure 3.1: Sketch of the problem

3.1.1 Physical description

The physical characteristics of the materials are :

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

3.1.2 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 = $1000\ W/m^2/K$
- other extern surfaces of the oven : adiabatic
- all the surfaces inside the oven : radiation exchange

3.2 How to do that with SALOME-SYRTHES?

Run SALOME, then open a new study and load the python script file `plates3d.py`. Now, you can activate the GEOM or SMESH modules to look at the geometry or meshes provided.

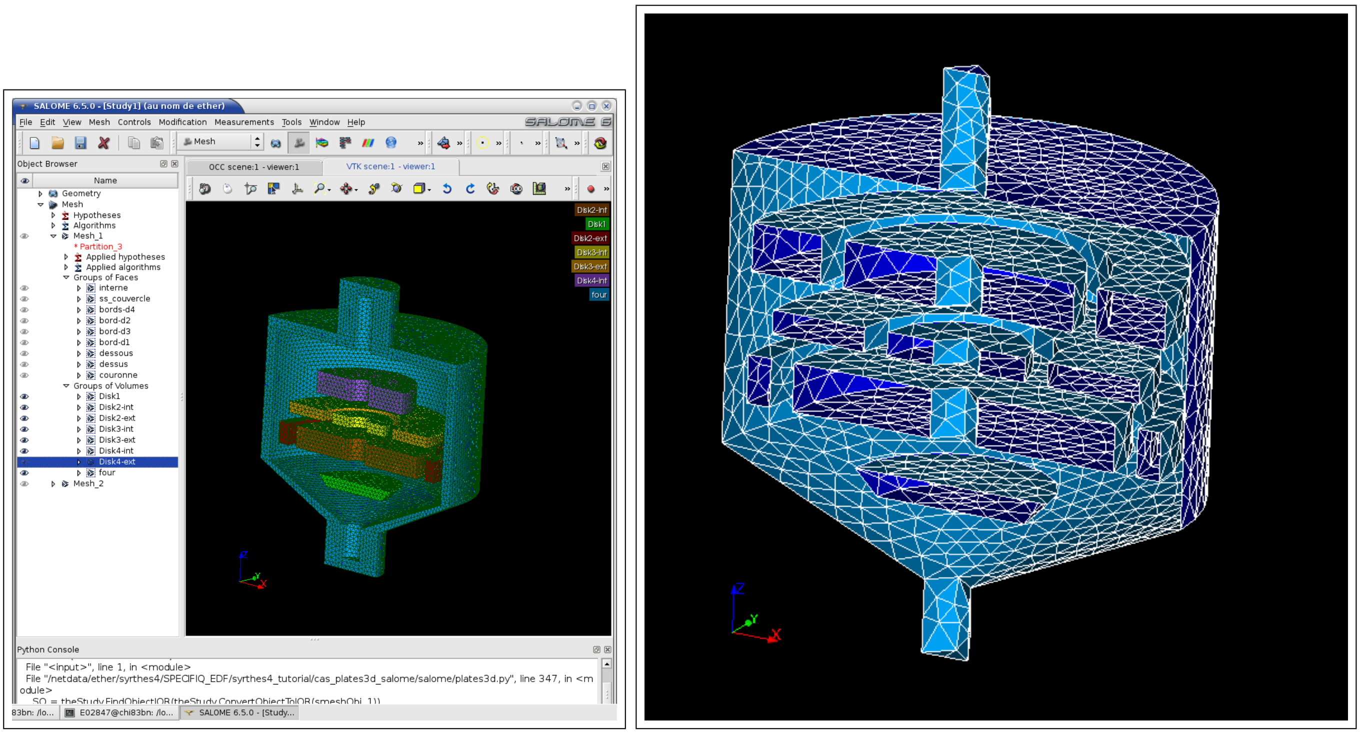

Figure 3.2: Conduction and radiation meshes of the oven and plates

3.2.1 SYRTHES module

Activate the SYRTHES module, create a SYRTHES study, and create a new SYRTHES case (as in the stand alone use described earlier).

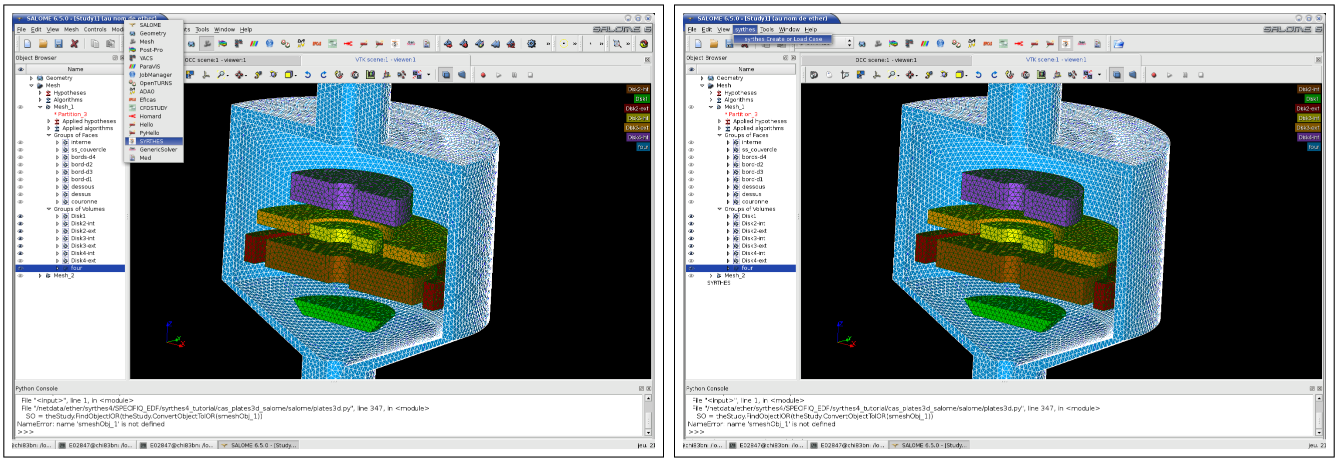

Figure 3.3: Activating the SYRTHES module and creating a SYRTHES study in SALOME

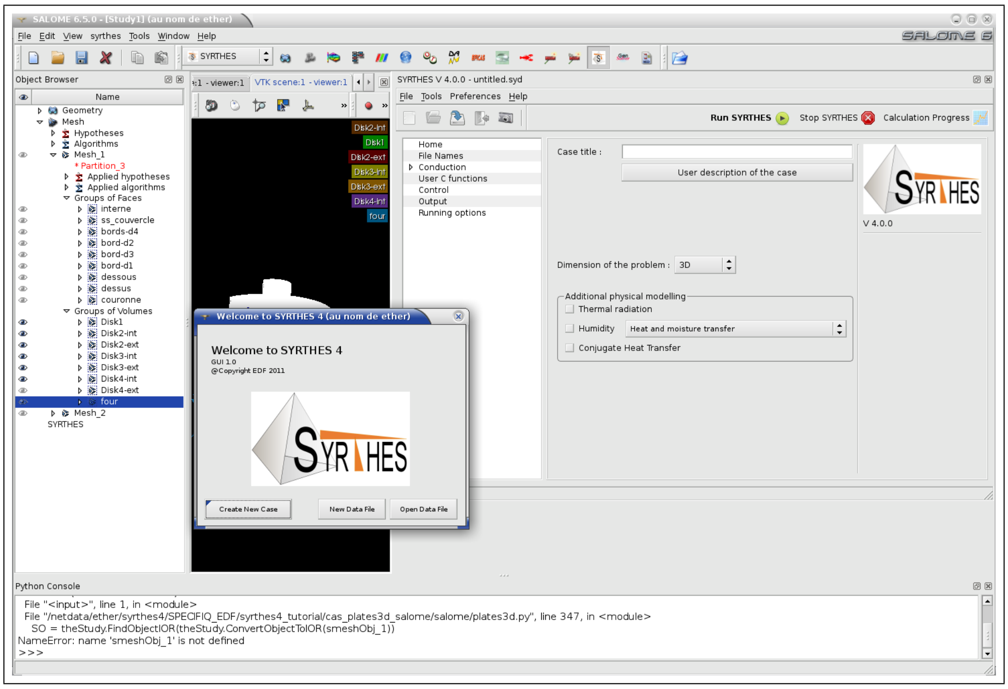

Figure 3.4: Creating a new SYRTHES case

In the SYRTHES main view, give a title to your study, and activate the "radiation" module.

Now go to the next window : "File names"
Select the conduction mesh file and the radiation mesh file in the `salome` directory.

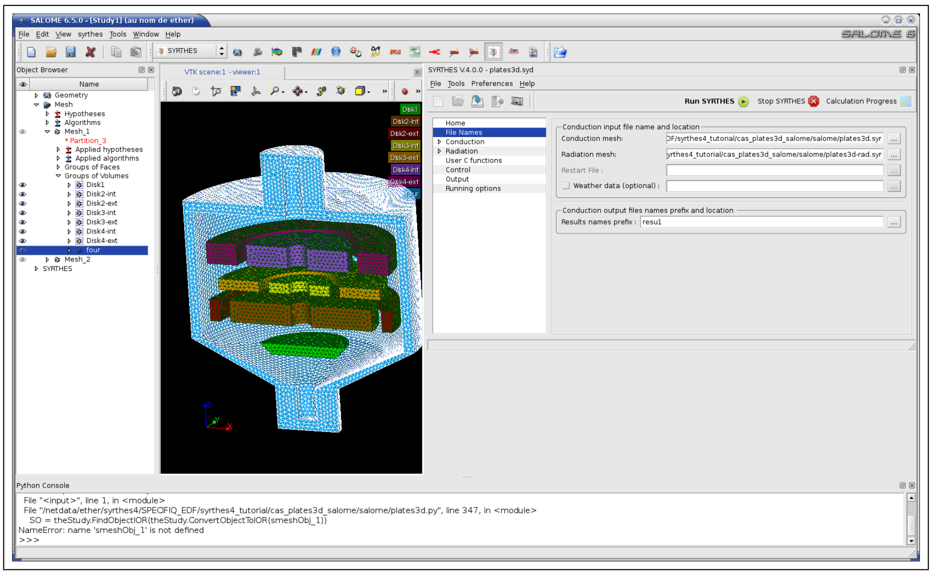

Figure 3.5: SYRTHES file names

At this stage, it is important to **save** your SYRTHES data file.

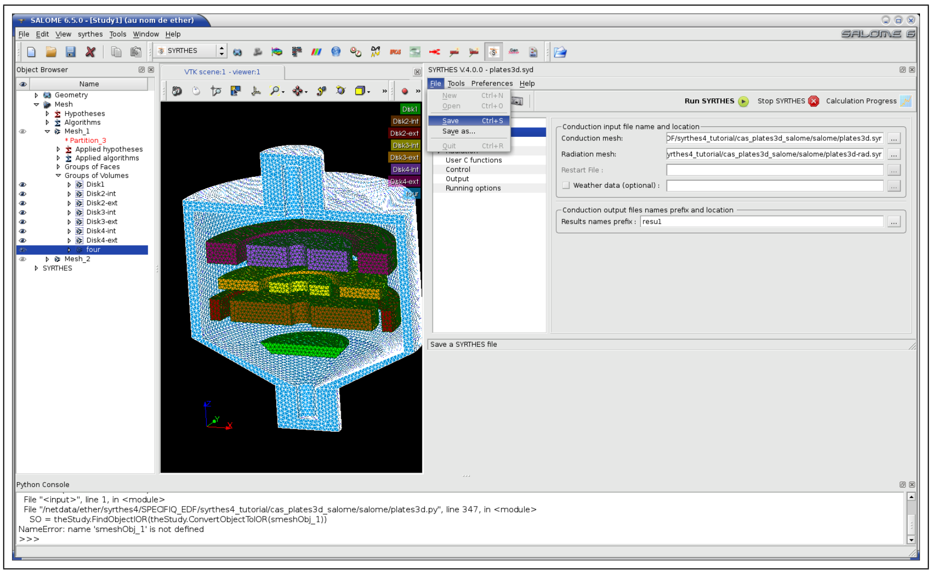

Figure 3.6: Saving the SYRTHES data file

After saving your SYRTHES data file, you can open the SYRTHES menu in the SALOME object browser.

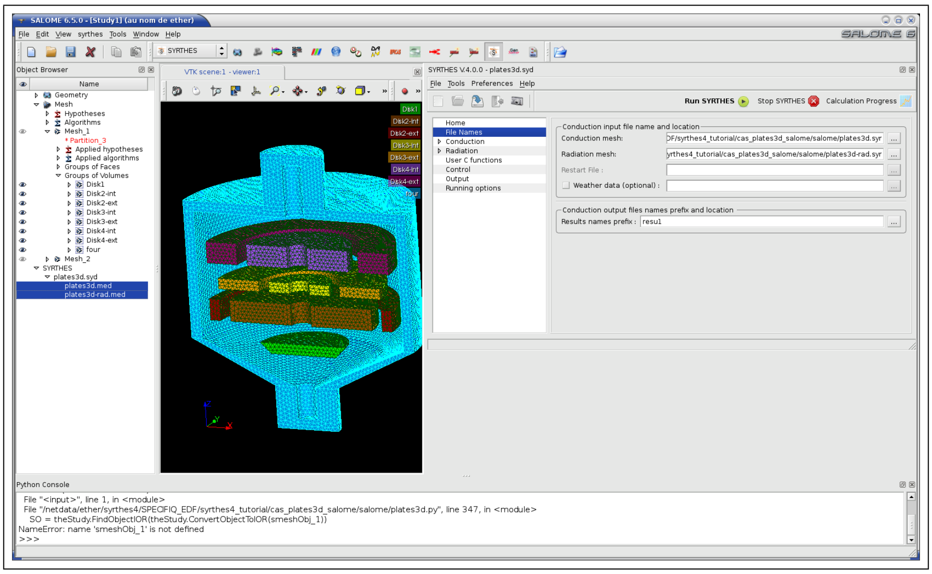

Figure 3.7: SALOME Object Browser

As it is explained in the previous chapter, SALOME is using group names and SYRTHESis using references to distinguished groups of elements or faces of the meshes.
So, a quick operation is required to create links between SALOME groups and SYRTHES references.
To do that click right on the med file (`plates3d.med` in the object browser and choose "Export in smesh".
Do the same with the radiation mesh.

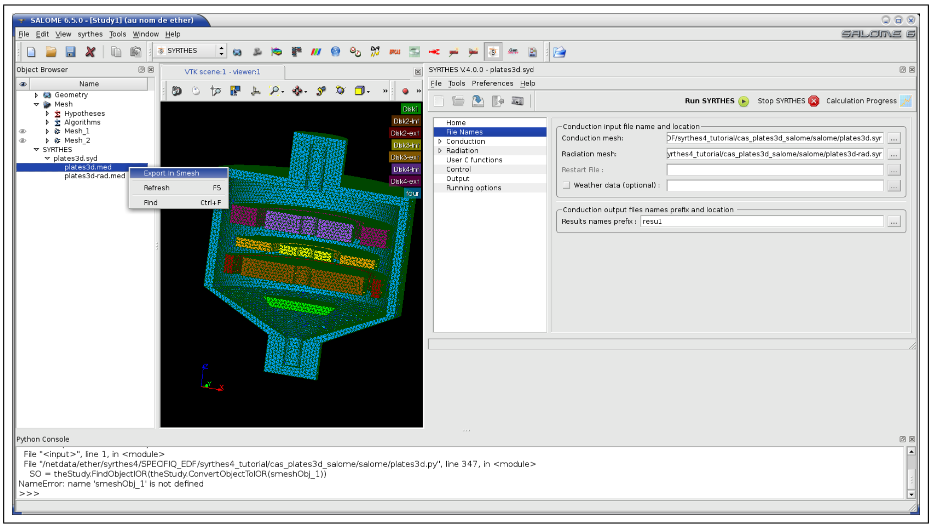

Figure 3.8: Export SYRTHES med file to SMESH module Object Browser

Go back to the SMESH module (activate the SMESH module). Hide Mesh_1 and Mesh_2 and now show Mesh_1 and the different groups of faces and volumes.
Go back to the SYRTHES module. You are ready to copy and paste groups from SMESH to SYRTHES.
In the SYRTHES-gui, select the "initial conditions" to set a temperature of 20 degrees everywhere.

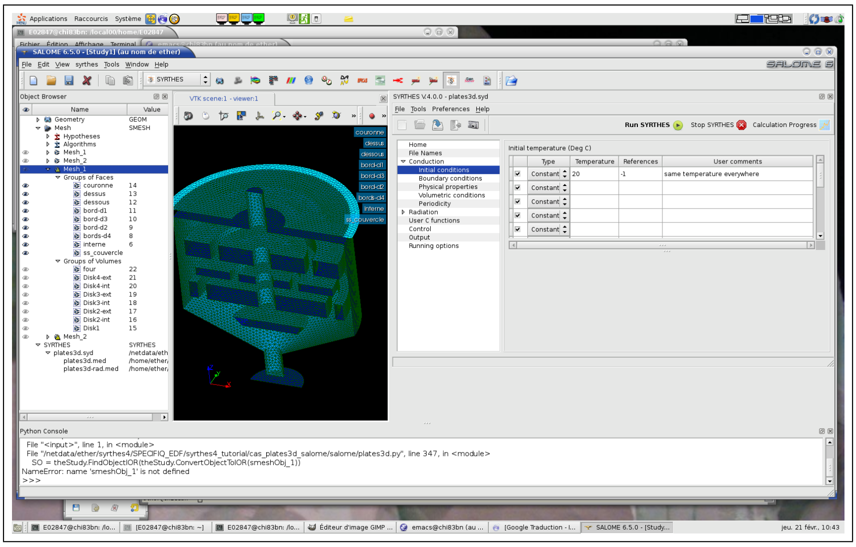

Figure 3.9: Uniform initial temperature

3.2.1.1 Boundary conditions

Select the "boundary conditions" item.
In the SALOME object browser, select the groups "dessus" and "couronne" (or select them in the graphical window).
In the SYRTHES-gui, set the temperature and heat exchange coefficient values. Then, do a right click in the References case and choose "Paste SALOME references". The references are set automatically and the name of the groups appears in the "user comments" space.

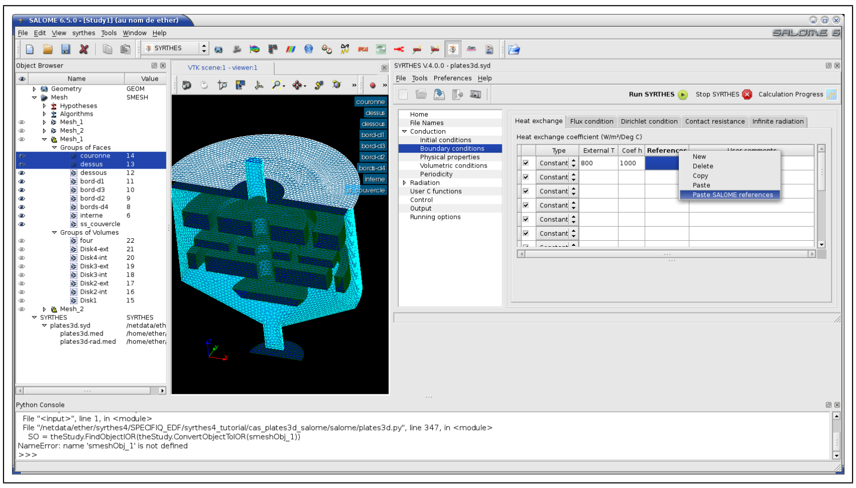

Figure 3.10: Copy and paste references from SMESH to SYRTHES

3.2.1.2 Physical properties

Select the "Physical properties" item.
In the SALOME object browser, select the groups "Disk4-ext", "Disk3-ext" and "Disk2-ext" (or select them in the graphical window).
In the SYRTHES-gui, set the density, specific heat and conductivity values. Then do a right click in the References case and choose "Paste SALOME references". The references are set automatically and the name of the groups appears in the "user comments" space.

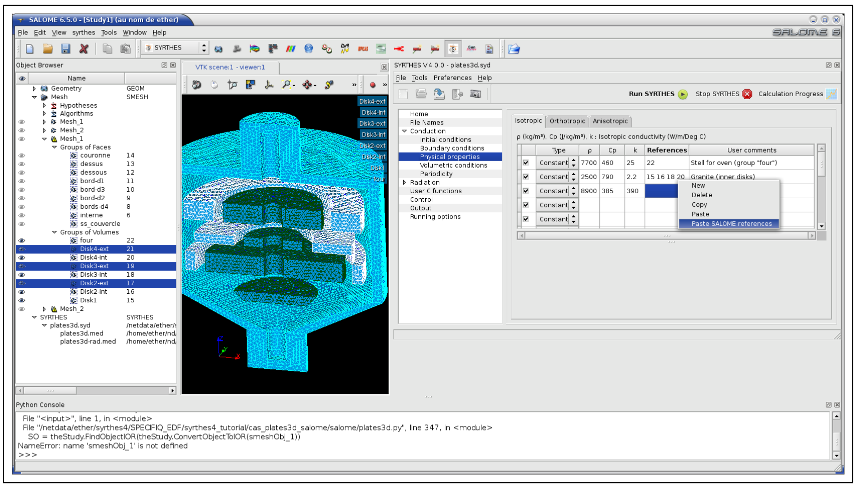

Figure 3.11: Copy and paste references from SMESH to SYRTHES for the volumes

Do the same for the 2 other materials.

3.2.1.3 Other parameters

Follow instructions of the previous chapter to define and set all the physical and numerical parameters of your study.

3.2.2 Running SYRTHES

Same instructions as in the previous chapter.

3.2.3 Analysing results

Results are available exactly in the same way as described in the previous chapter. All the files are gather in the SYRTHES directory.
To visualize the results, activate the "Paravis" module. Select "File" menu and then "Open Paraview File..."

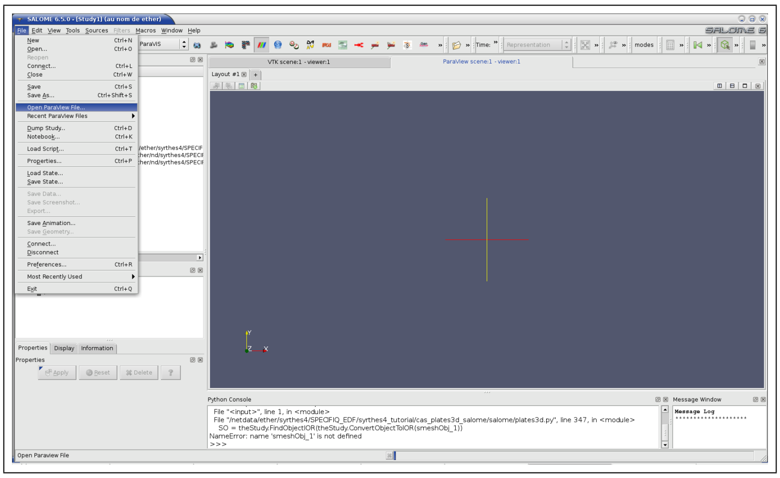

Figure 3.12: Open SYRTHES result files for Paravis

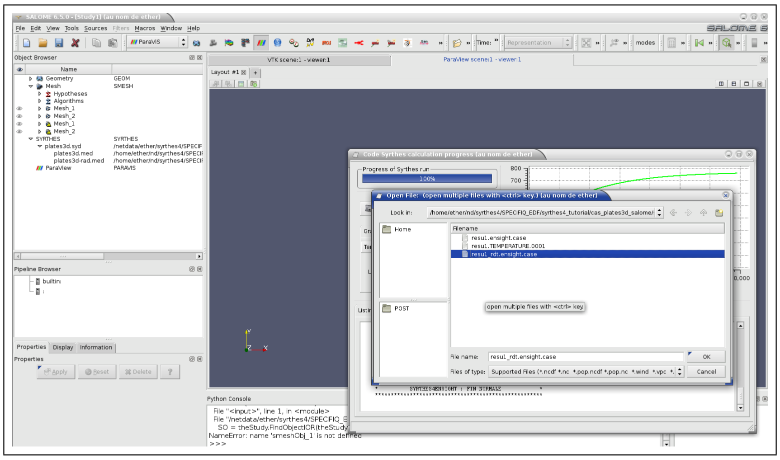

Figure 3.13: Select the result files for Paravis

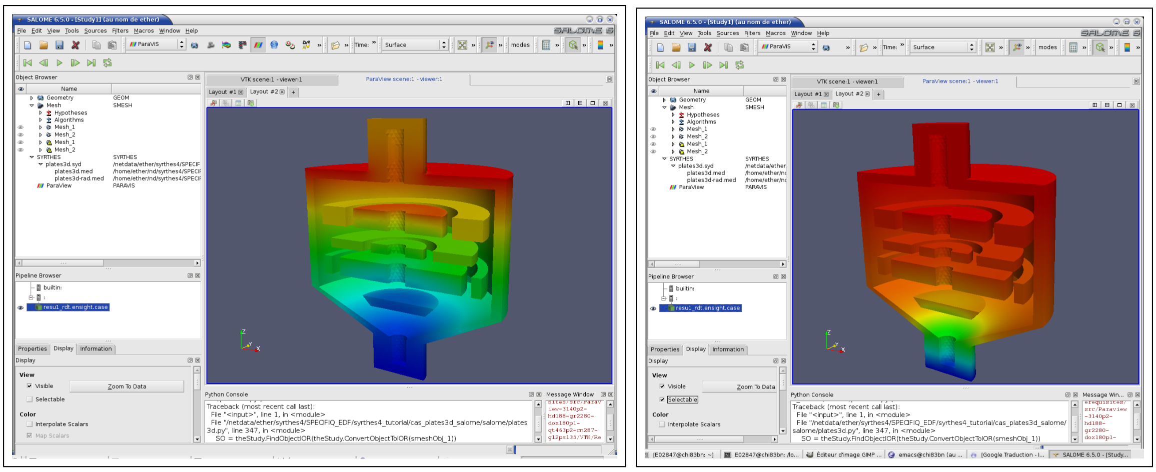

Figure 3.14: Temperature at 15000 s and 60000 s

Chapter 4

Parametrical simulation with SYRTHES and SALOME-YACS

Heat exhanger

4.1 What is the problem ?

In a metal structure, heated by a volumic source, and cooled by severals ducts, we want to estimate the best heat exchange coefficient between coolant water in the duct and metal structure that ensure that the metal temperature never exceeds $100°C$.

4.1.1 Geometrical and physical description

The solid domain consists of a block (1 m large) bored by 9 cooling ducts (0.1 m radius).

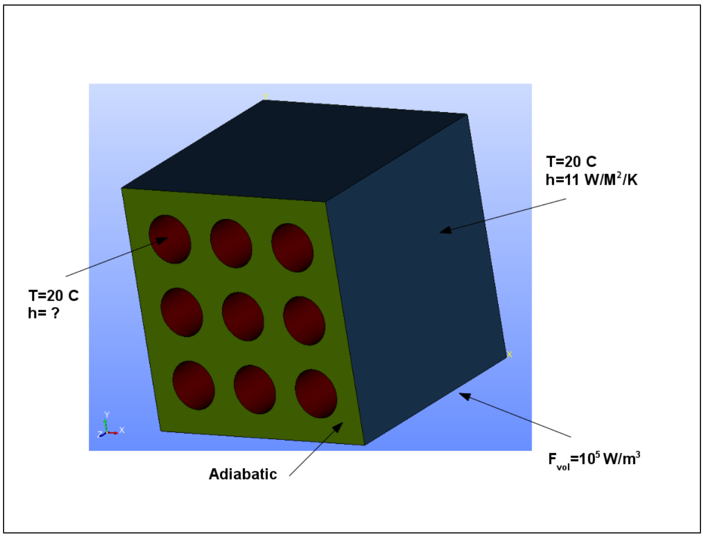

Figure 4.1: Sketch of the problem

Density, specific heat and conductivity of steel are set to : $\rho = 7700\ kg/m^3$, $C_p = 460\ J/kg/°C$ and 25 $W/m/°C$.

4.1.2 Initial conditions, boundary conditions and source term

The initial temperature is $20°C$.

The boundary conditions are :

- faces of the block without holes : T=$20°C$, h=11 $W/m^2/°C$
- faces of the block with holes : adiabatic
- faces of ducts T=$20°C$, h_{ducts} = ?? (the aim of the study is to evaluate this value)

We consider a uniform source term in the metal : 10^5 W/m^3

4.2 Initial step : to update paths for files

SALOME works with whole paths for files. So, before beginning, you have to update the paths according your system.

- To edit `cas_param/yacs/CalculationLoop.xml` and update all paths like `/home/.../` (4 occurences)
- To edit `cas_param/syrthes_template/box_holes_template.syd` and update `MAILLAGE CONDUCTION= /home/.../` (1 occurence)

4.3 First step : build a study with an initial value for h_{ducts}

In the directory `cas_param` you will find the folders :

- `salome` : script file for meshing and mesh file (MED and SYRTHES form)
- `syrthes` : the original SYRTHES case
- `syrthes_template` : same as `syrthes` but with a parametric SYRTHES data file (`box_holes_template.syd`)
- `yacs` : YACS schema and associated python script
- `scratch` : working directory for parametric calculations

4.3.1 Mesh generation

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

In the directory `mkdir cas_param/salome`, run SALOME : `/.../runAppli` (the command is depending on your local installation of SALOME. You can load the MED file SALOME/`box_hole.med`.

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 my `box_hole.syr_desc` file :

```
group_of_volumes    6    Group_vol
group_of_faces     7    Group_faces_box
group_of_faces     8    Group_faces_holes
group_of_faces     9    in_out
```

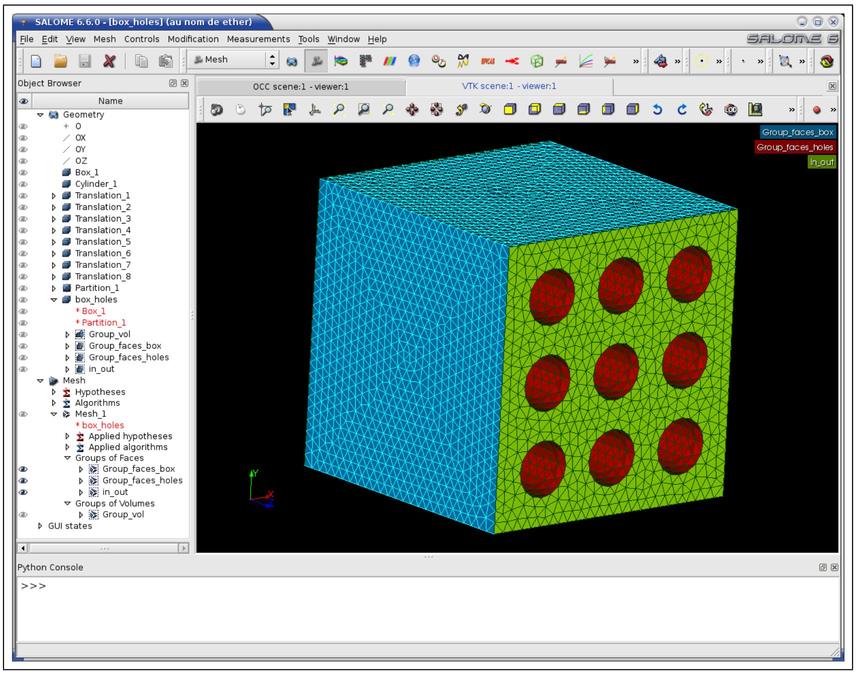

Figure 4.2: Mesh

4.3.2 SYRTHES-study

Go back to the initial directory : `cas_param` and run SYRTHES-GUI and choose the "Open Data File" option. Choose the SYRTHES data file : `cas_param/syrthes/box_holes.syd`

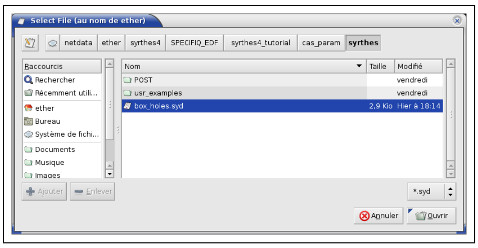

Figure 4.3: Open SYRTHES data file

Have a look to the boundary condition window : we have set an initial value of 150 $W/m^2/°C$ for the heat exchange coefficient inside the ducts.

This case is ready-to-run. Click on the "Run SYRTHES" button.

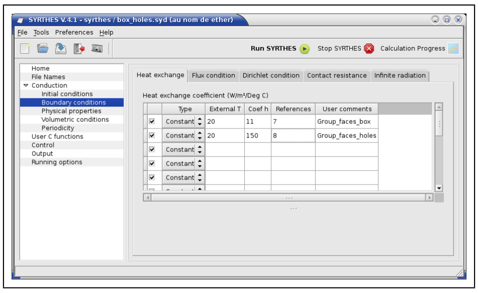

Figure 4.4: SYRTHES data file : first calculation with $h = 150\ W/m^2/°C$

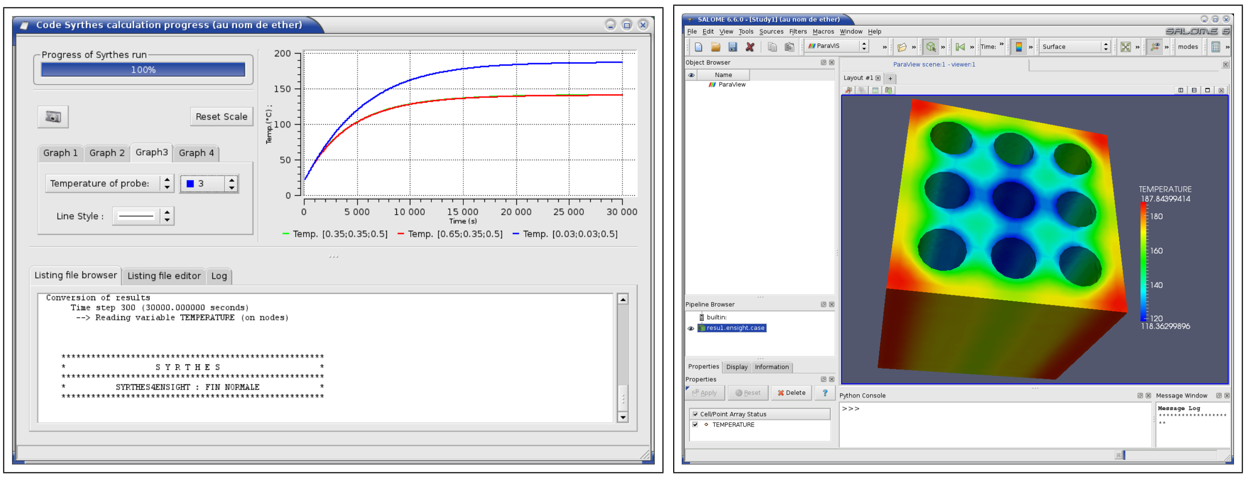

Figure 4.5: Calculation window and temperature field after calculation

4.4 Second step : build a SYRTHES parametric study

The `cas_param/syrthes_templpate` is a copy of the initial case `cas_param/syrthes`. The only change is the introduction of a parametric value for the heat exchange coefficient.

Again go back to the initial directory : `cas_param` and run SYRTHES-GUI and choose the "Open Data File" option. But, this time, choose the SYRTHES data file : `cas_param/syrthes_template/box_holes_template.syd`

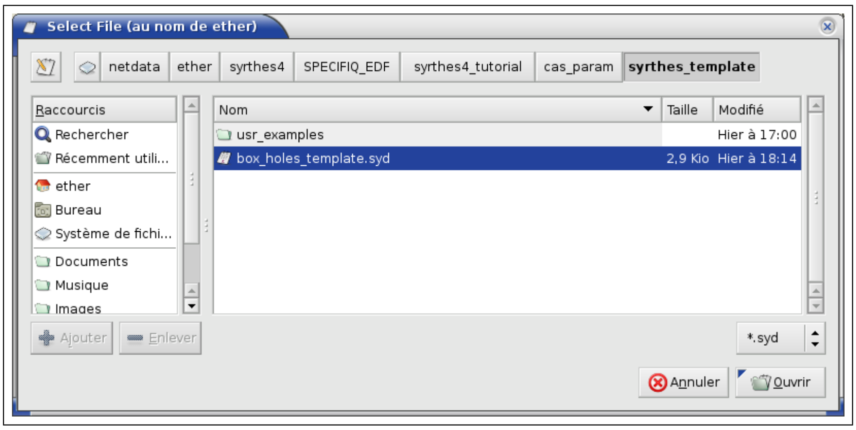

Figure 4.6: Open SYRTHES data file in the `syrthes_template` case

Now, have a look again to the boundary condition window : the initial value of h is replaced by a variable $h. For the next runs, this variable will have the different values tested.

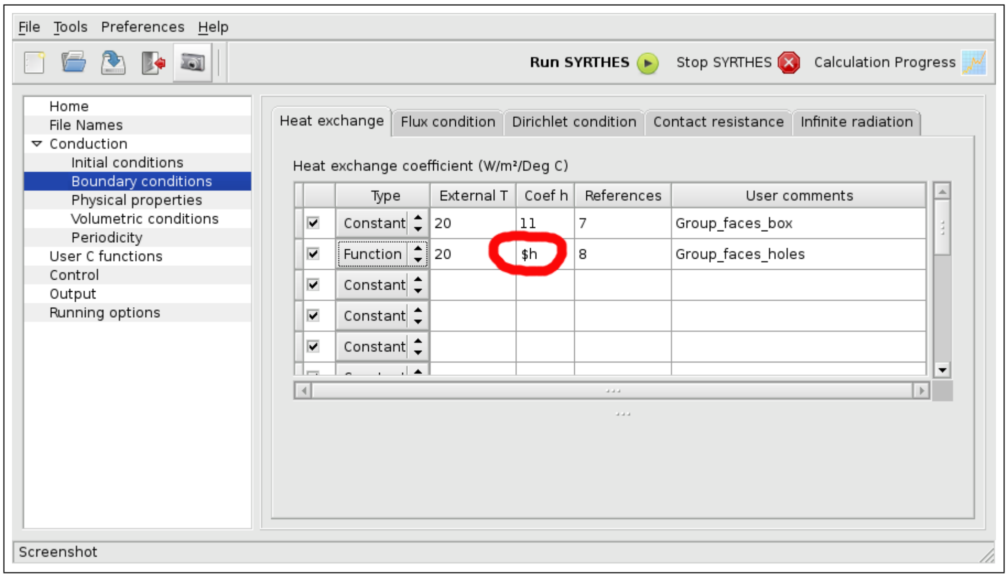

Figure 4.7: `syrthes_template case` : boundary conditions with a parametric value for h

4.5 Third step : YACS schema

Run SALOME and activate the YACS module :

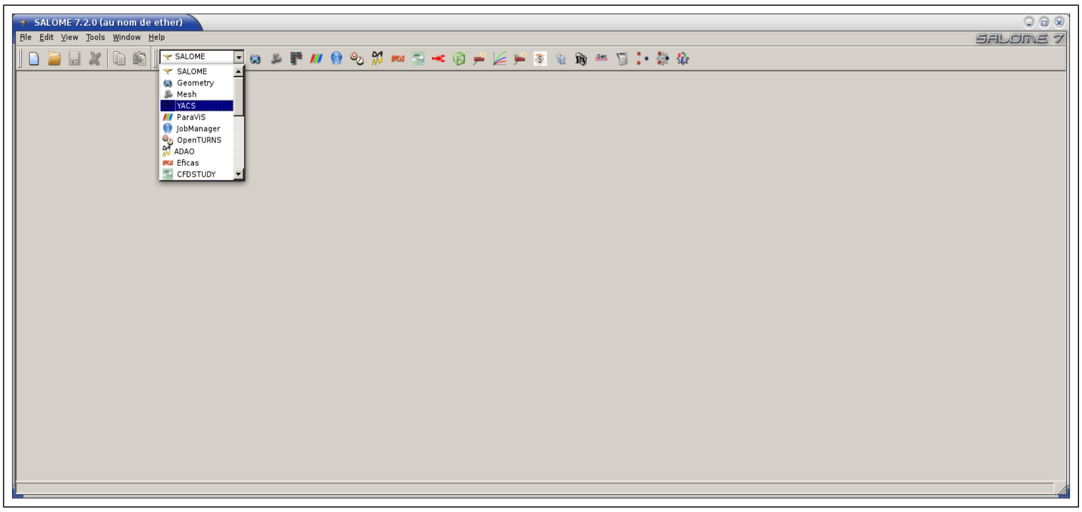

Figure 4.8: SALOME - Activating YACS module

Choose a new study :

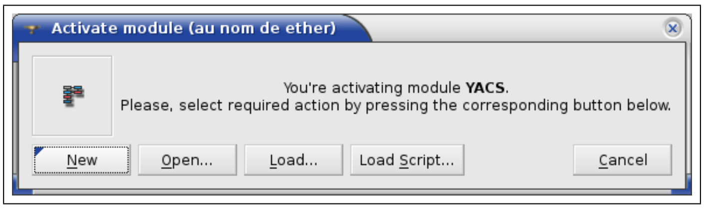

Figure 4.9: SALOME-YACS- New study

And import the YACS schema `cas_param/yacs/CalculationLoop.xml` :

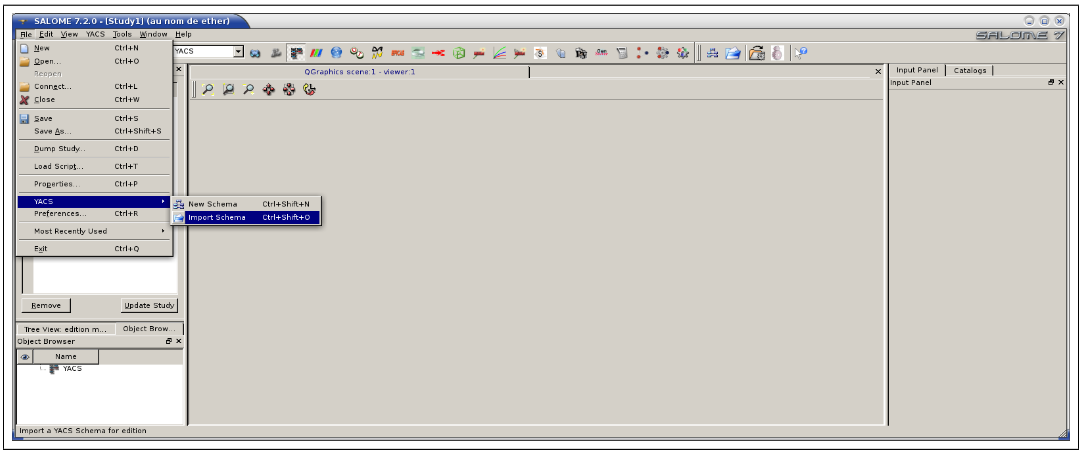

Figure 4.10: SALOME-YACS- Import a YACS schema

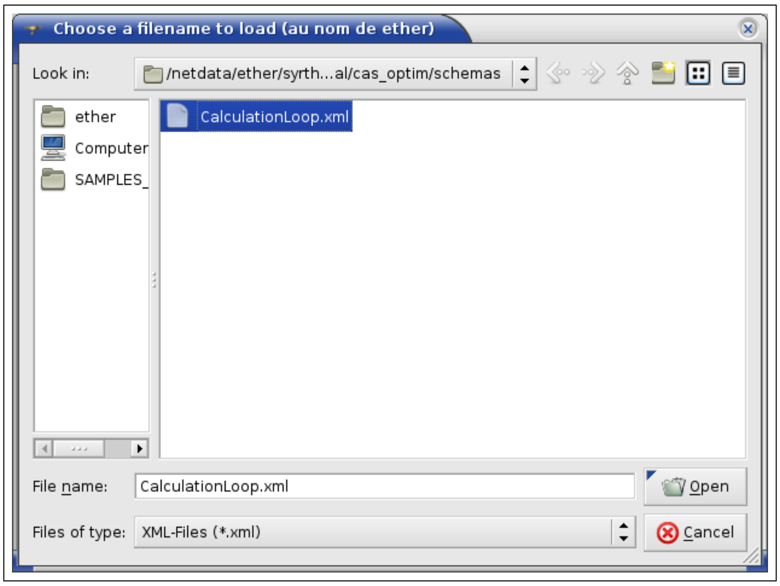

Figure 4.11: SALOME-YACS- Choose the schema

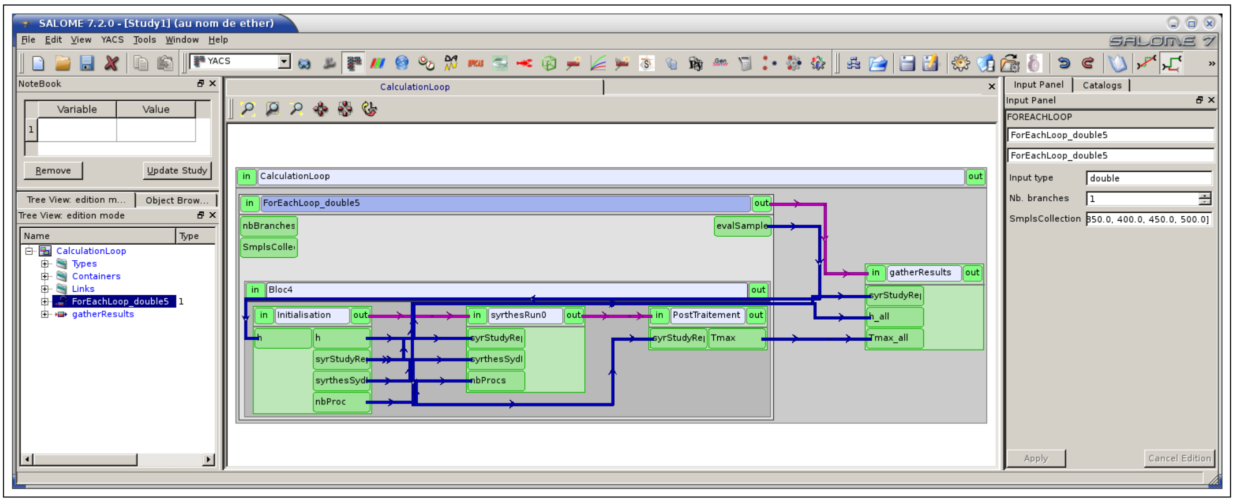

Figure 4.12: SALOME-YACS- Calculation scheme

The next step is to prepare the YACS schema to run :

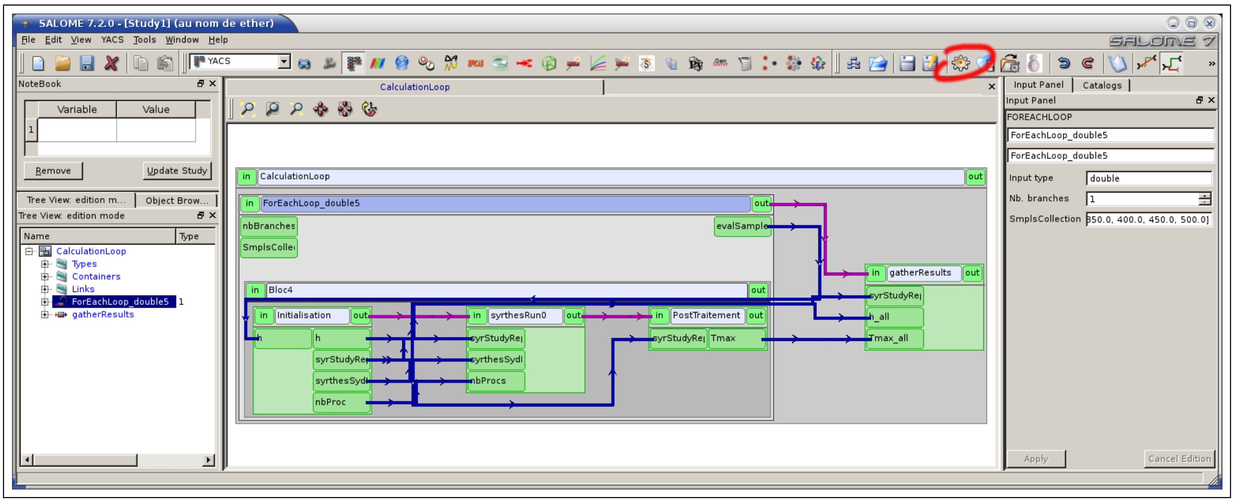

Figure 4.13: SALOME-YACS- Prepare the YACS schema

And finally, you can run the schema

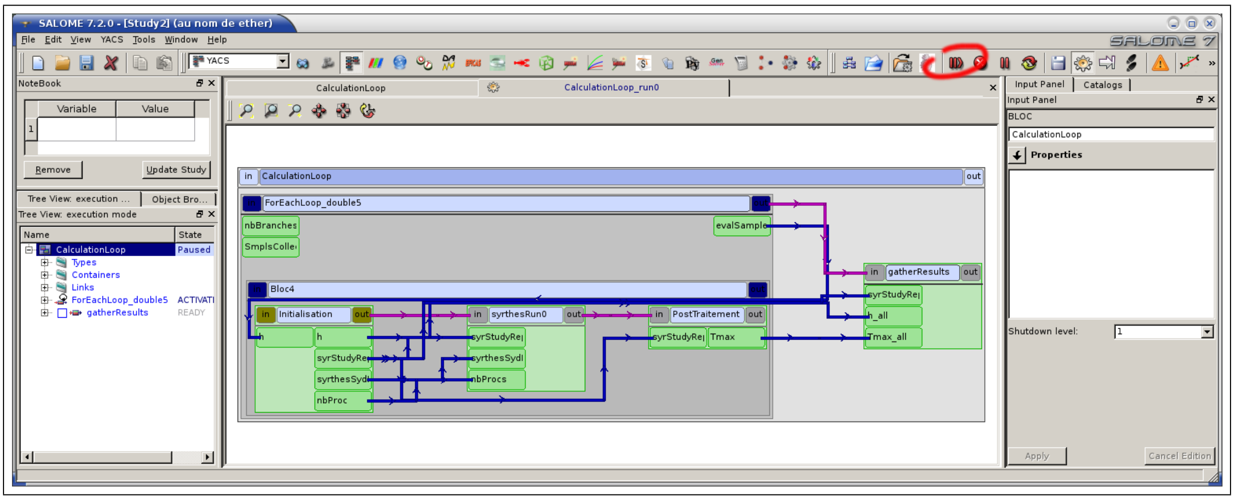

Figure 4.14: SALOME-YACS- Run the YACS schema

The next figure shows the result of the run : we get the maximum of temperature reached for each case :

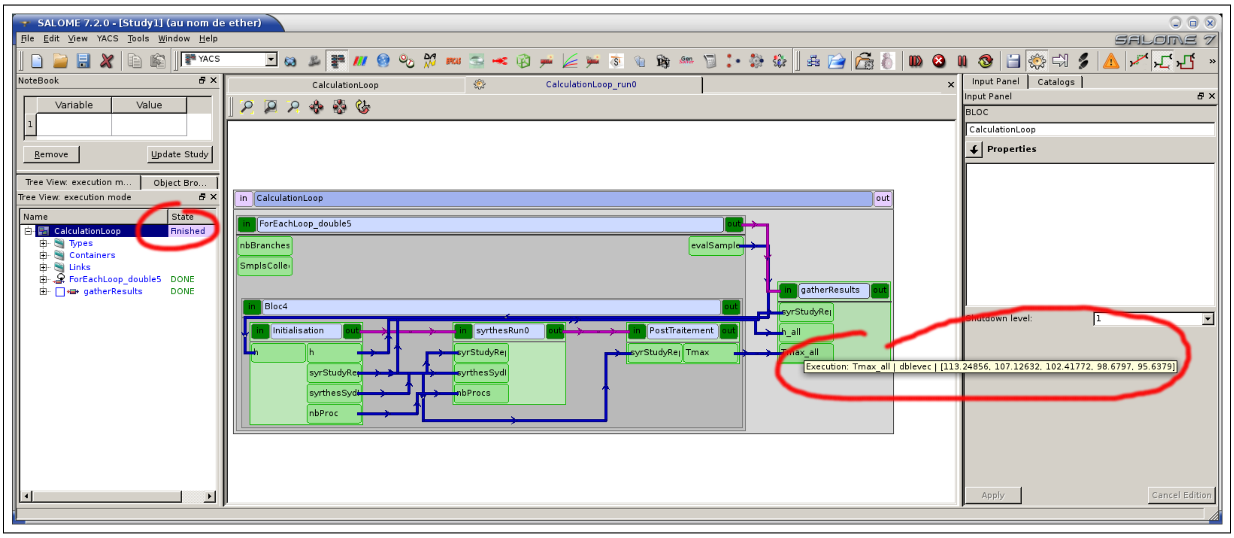

Figure 4.15: SALOME-YACS- Results of the 5 SYRTHES calculations

The results are presented in the table :

Heat exchange coefficient	300	350	400	450	500
Maximum of temperature ($°C$)	113.24	107.12	102.4	98.67	95.63

For this schema, we have chosen to test 5 values of the heat exchange coefficient : 300,350,400,450 and 500, but it would be easy to test many more. The corresponding SYRTHES calculations show that the coefficient must be greater than 450 $W/m^2/°C$ to ensure that the maximum of temperature stays under 100$°C$.

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 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

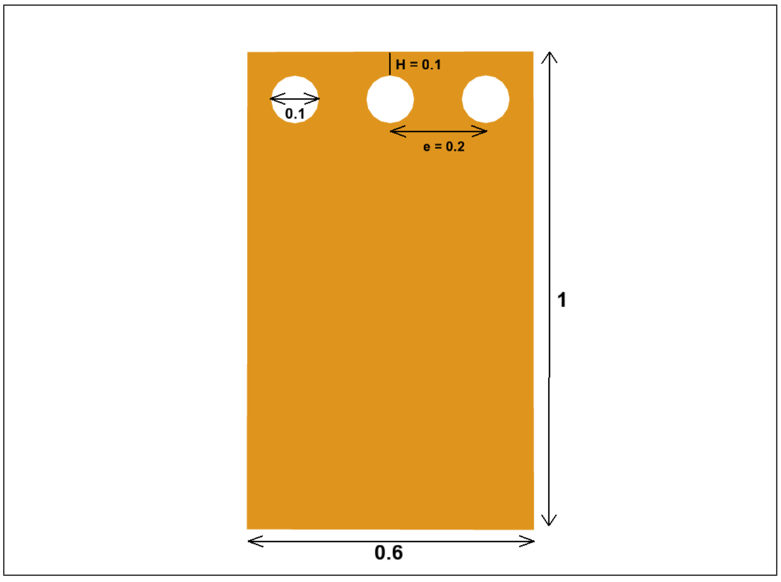

Figure 5.1: Sketch of the problem

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

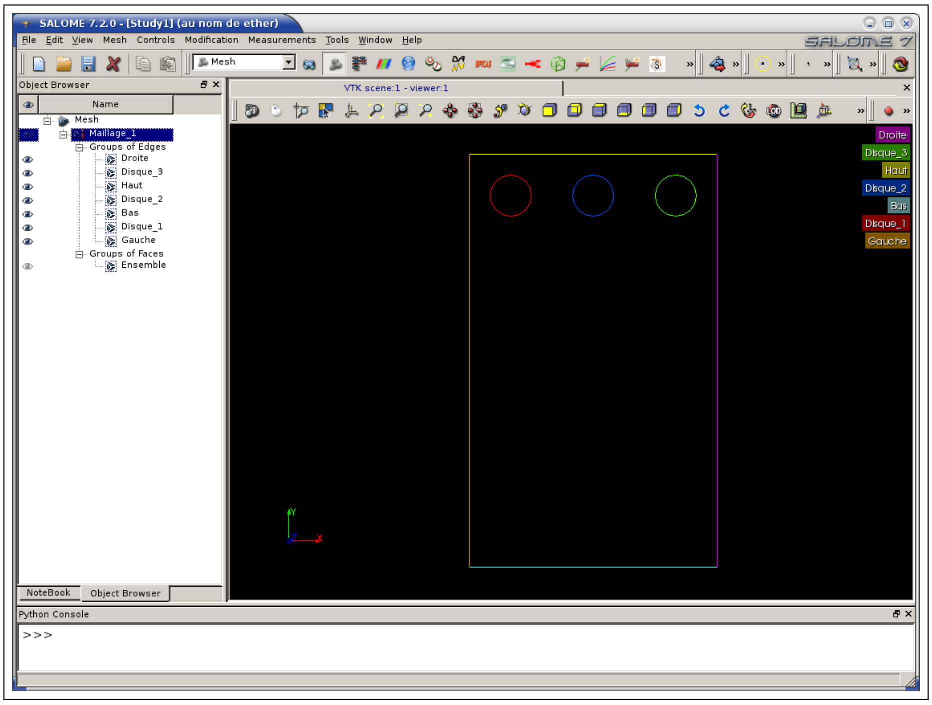

Figure 5.2: Group names for volumes and boundary conditions

5.2 Mesh provided

Mesh 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 my `flux.syr_desc` 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, the flux between surfaces T_p and T_{ext} is

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

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

5.4 Results

Figure 5.4 shows the temperature in the ground at convergence.

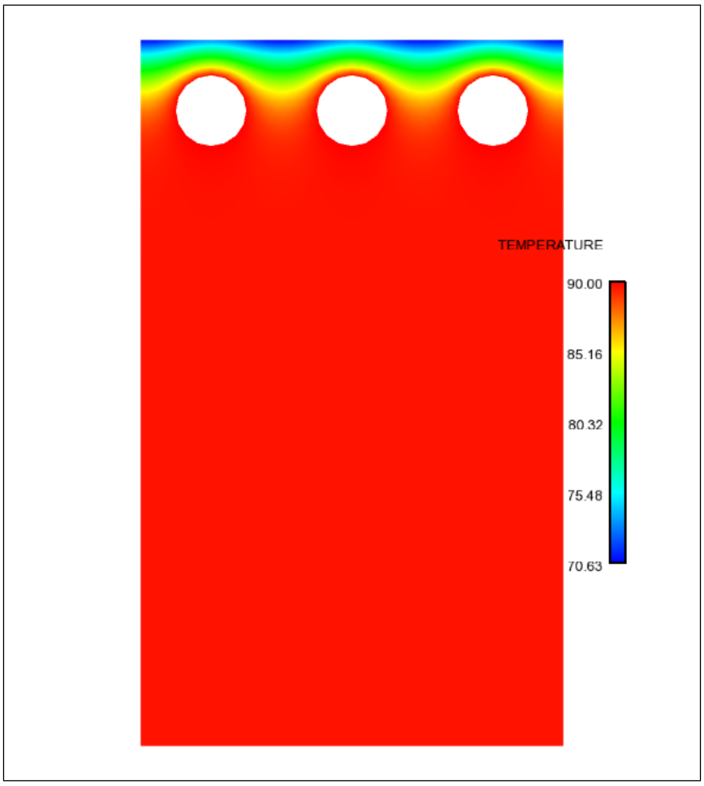

Figure 5.3: Ground temperature

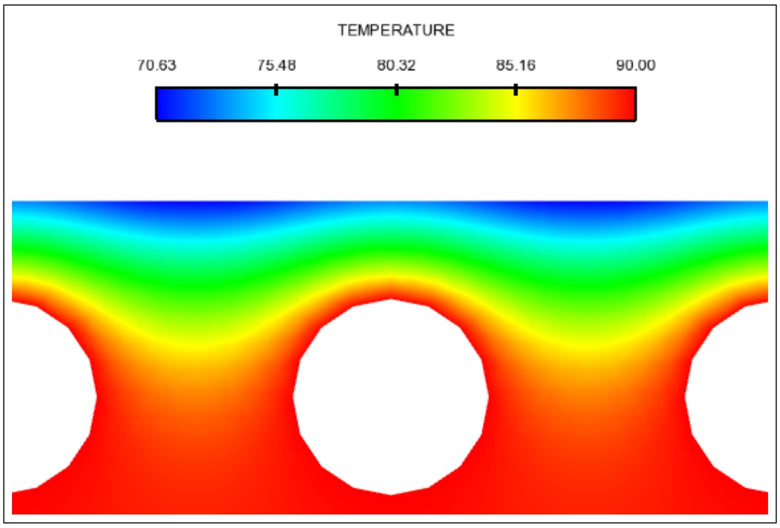

Figure 5.4: Ground temperature - zoom around the surface

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

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

SYRTHES simulation gives : $\varphi_S = 155.922/0.6 = 259.87 W/m^2$ The computed value is more precise because, as shown in figure 5.5 the temperature isn't constant along the upper surface.

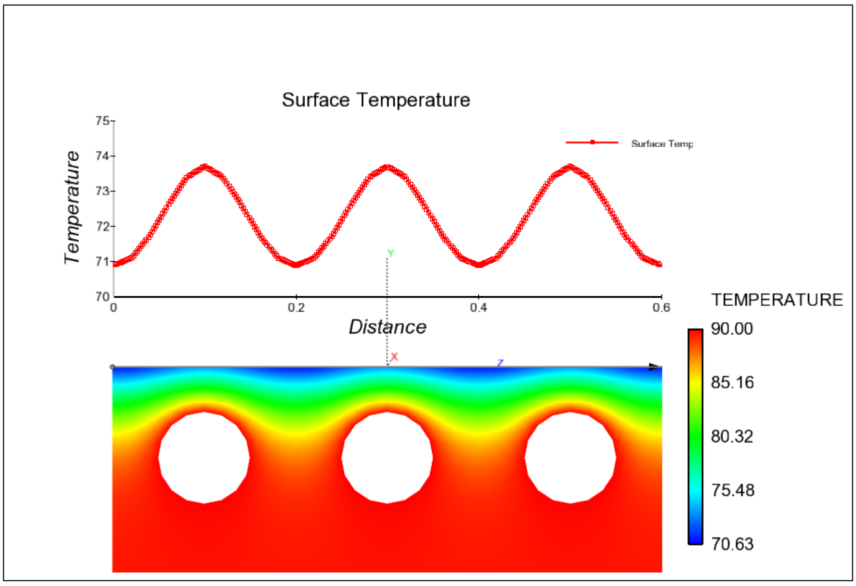

Figure 5.5: Surface temperature