EDF R&D

Fluid Dynamics, Power Generation and Environment Department
Single Phase Thermal-Hydraulics Group

6, quai Watier
F-78401 Chatou Cedex

Tel: 33 1 30 87 75 40
Fax: 33 1 30 87 79 16

SEPTEMBER 2017

Code_Saturne documentation

Code_Saturne version 5.0 tutorial: three 2D disks

contact: saturne-support@edf.fr

TABLE OF CONTENTS

Part I

Introduction

1 Introduction

Code_Saturne is a system designed to solve the Navier-Stokes equations in the cases of 2D, 2D axisymmetric or 3D flows. Its main module is designed for the simulation of flows which may be steady or unsteady, laminar or turbulent, incompressible or potentially dilatable, isothermal or not. Scalars and turbulent fluctuations of scalars can be taken into account. The code includes specific modules, referred to as "specific physics", for the treatment of lagrangian particle tracking, semi-transparent radiative transfer, gas, pulverized coal and heavy fuel oil combustion, electricity effects (Joule effect and electric arcs) and compressible flows. *Code_Saturne* relies on a finite volume discretization and allows the use of various mesh types which may be hybrid (containing several kinds of elements) and may have structural non-conformities (hanging nodes).

The present document is a tutorial for *Code_Saturne* version 5.0. It presents five simple test cases and guides the future *Code_Saturne* user step by step into the preparation and the computation of the cases.

The test case directories, containing the necessary meshes and data are available in the `examples` directory.

This tutorial focuses on the procedure and the preparation of the *Code_Saturne* computations. For more elements on the structure of the code and the definition of the different variables, it is higly recommended to refer to the user manual.

Part II

Three 2D disks

1 General description

1.1 Objective

The aim of this case is to train the *Code_Saturne* coupling with a thermal conduction and radiation code SYRTHES on a simplified 2D problem. It corresponds to a natural convection inside a sheath with different electric wires.

We can see with this test-case the conjugate heat transfer phenomenon between the solid and fluid domains.

1.2 Remarks

• **Remark - 1**: Create the `3disks2D` study directory, two subdirectories `fluid` and `solid` as below:

```
$ code_saturne create -s 3disks2D -c fluid --syrthes solid
```

• **Remark - 2**: The fluid mesh must be copied in the directory `MESH`. The solid mesh must be copied in the subdirectory `solid`.

• **Remark - 3**: Launch the SYRTHES Graphical User Interface (Gui) (`$ syrthes.gui &`) inside the subdirectory `solid` for the first solid computation alone.

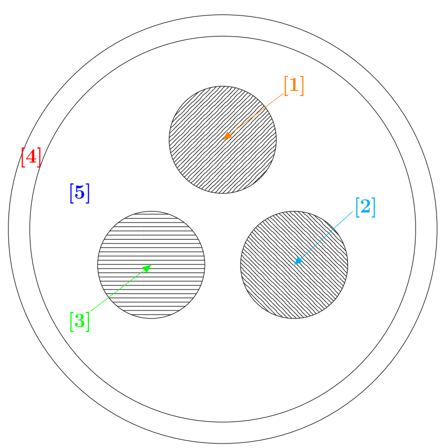
• **Remark - 4**: Launch the *Code_Saturne* Graphic User Interface (GUI) inside the subdirectory `fluid` for the fluid computation alone.

• **Remark - 5**: Launch the *Code_Saturne*-SYRTHES coupling computation with the top-level `runcase` script.

1.3 Description of the configuration

The 2D configuration represents a simplification of the real 3D geometry of the wires inside an electric sheath. As we can see, we have 3 different wires represented as 3 different disks inside a bigger disk for the sheath. We assume that the 3 disks are in contact with an air flow inside the electric sheath.

The geometry is shown on figure II.1.



$$K_{[1,4]} = \begin{pmatrix} 25 & 0 \\ 0 & 25 \end{pmatrix}$$

$$K_{[3]} = \begin{pmatrix} 25 & 0 \\ 0 & 5 \end{pmatrix}$$

$$K_{[2]} = \begin{pmatrix} 25 & 0 \\ 0 & 25 \end{pmatrix}$$
$$\theta = 45^o$$

Figure II.1: Geometry of the test-case with [1,2,3,4] the solid domain and [5] the fluid domain. The 4 disk physical properties are specified for the solid domain.

For the fluid domain, there are two symmetry conditions and walls conditions imposed to the faces coupling with the solid domain. We have no velocity imposed to create movement inside the fluid area and gravity force is taken into account.

Nevertheless, we define a density which is variable in function of the temperature for the air flow. The 3 disks, which are warmer than the air flow, generate a temperature difference creating a fluid movement. The warmer air flow is moving to the top and the colder air flow to the bottom of the fluid domain.

With this test-case, we can easily observe the effect of the solid disks on the air flow contained in the electric sheath.

1.4 Characteristics

- **Solid domain:**

The initial and boundary conditions to choose without conjugate heat transfer for the solid domain are defined hereafter:

Initial conditions	
Temperature condition	$T_{ini,s} = 20°\text{C}$

Boundary conditions	**Value**	**Surface reference**
Heat exchange conditions ($q_{w,ext}$)	$T_{ext} = 90°\text{C}.\ ;\ h_{ext} = 1000(W/m^2.K)$	color 2 or 5 or 8

Characteristics of the solid domain with the 4 different disks (1 to 3 for the electric wires and 4 for the disk for the electric sheath):

	Conductivity type	**Values** (W/m/°C)	**Volume reference**
Disk 1	Isotropic	$k_{11} = 25$	color 1
Disk 2	Orthotropic	$k_{11} = 25\ ;\ k_{22} = 5$	color 2
Disk 3	Anisotropic	$k_{11} = 25\ ;\ k_{22} = 5\quad \alpha = 45^o$	color 3
Disk 4	Isotropic	$k_{11} = 25$	color 4

Physical properties	**Values**
Density [ρ]	7700 (kg/m^2)
Specific heat [C_p]	460 $(J/kg/m^3)$

- **Fluid domain:**

The characteristics of the air flow inside the fluid domain are defined as following:

Thermophysical models	**Choosen type**
Time step	constant in time and uniform in space
Turbulence model	$k - \varepsilon$
Scalar	Temperature (°C)

The initial and boundary conditions to choose without conjugate heat transfer for the solid domain are defined below:

Initial conditions	
Temperature condition	$T_{ini,f} = 20°\text{C}.$

Boundary conditions	Values	Surface reference
Walls (Heat exchange $q_{w,ext}$)	$T_{ext} = 30°\text{C}$; $h_{ext} = 10(W/m^2.K)$	color 1
Symmetry		color 2 or 3

In this case, the fluid density is function of the temperature, the following ideal gas law is specified in the Graphical User Interface (GUI):

$$\rho = \frac{p_0}{R_g\ (T + 273.15)} \tag{II.1}$$

where ρ is the density, T is the temperature (°C), ideal gas constant $R_g = 287\ (m^2.s^{-2}.K^{-1})$, $p_0 = 101325\ (Pa)$ the reference pressure choosen as $p \approxeq p_{atmos}$.

1.5 Mesh characteristics

- **Description of the solid mesh:**

The solid mesh used in the conduction problem contains 11688 nodes (P_1 discretization) and 5688 elements. We have to take care of the references allowing to identify materials properties and boundary conditions which are specified in this solid mesh by reference colors.

Type: unstructured mesh
Mesh generator used: SIMAIL
Color definition: see figure II.3.

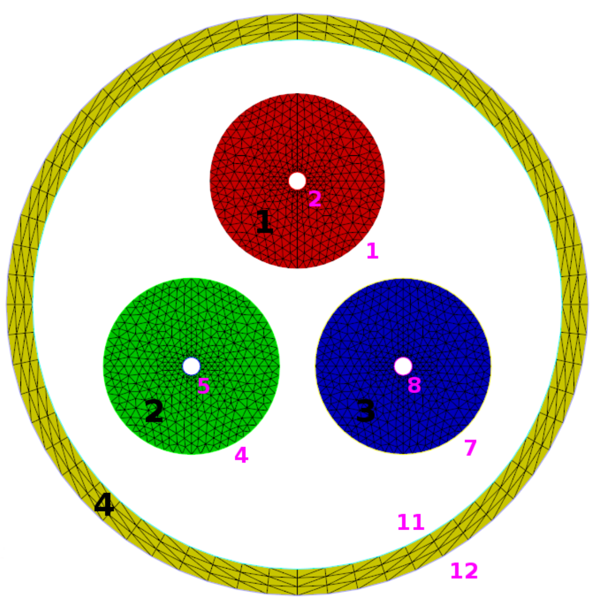


Figure II.2: Colors of the solid zones and boundary faces

- **Description of the fluid mesh:**

The fluid mesh contains 3866 nodes. We have to use the **Mesh Quality Criteria** run type in the *Code_Saturne* Graphical User Interface (**Calculation management**, **Pepare batch calculation** section) to check the quality criteria and identify the reference colors associated to the boundary conditions (color 1 is used for all sides, 2 for the bottom surface, and 3 for the top surface.

Type: unstructured mesh
Mesh generator used: SIMAIL
Color definition: see figure II.3.

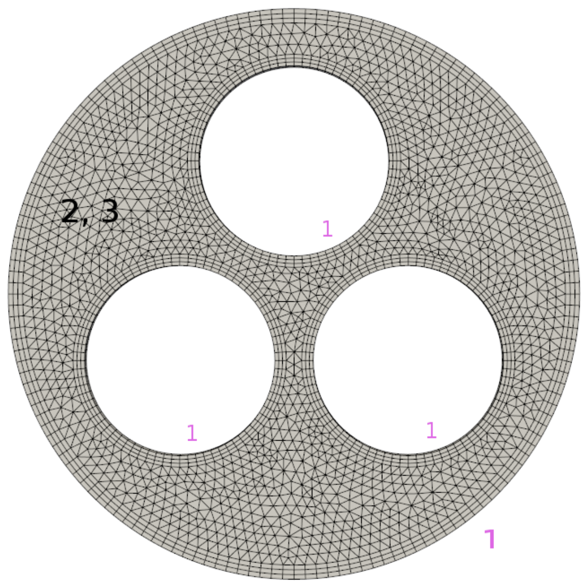


Figure II.3: Colors of the fluid boundary faces

2 CASE 6: 3 2D disks

The post-processing containing the **temperature** field will be post-processed on a sub-mesh with ParaView. A 2D clip plane will also be extracted along the symmetry plane of the fluid domain and temperature will be written on it.

2.1 Parameters

All the parameters necessary to this study can be defined through the *Code_Saturne* (GUI) and SYRTHES (GUI) respectively, as below:

Numerical parameters of solid computation	
Reference time step	10 (s)
Number of iterations	100
Numerical parameters of fluid computation	
Reference time step	0.1 (s)
Number of iterations	100

These numerical time steps and iterations number have been defined to run the fluid and solid computations independently one from each other. Thus, we can test the setting data for the fluid computation with *Code_Saturne* and the solid conduction computation with SYRTHES. After that we will be able to run the coupling computation with the computation option **Conjugate heat transfer** activated on both data settings.

2.2 Output management

The standard options for output management will be used. Only one monitoring point will be created for the solid conduction computation at the following coordinates:

Probe	x (m)	y (m)
1	0.003	-1.2

For this probing, we choose to save the temperature value every 10 time steps and the temperature field every 25 time steps.

2.3 Coupling computation

The numerical parameters used for the coupling computation must be modified to be sure to see the conjugate heat transfer phenomenon between the solid and fluid domains. For this reason, we increase the iterations number and the time step for the fluid and solid data setting.

By default, the smaller iterations number will be used to drive the coupling computation. If we choose an iterations number of 10000 for the fluid domain and 5000 for the solid domain, the coupling computation will be stopped after 5000 instead of 10000.

Numerical parameters of solid computation	
Reference time step	0.5 (s)
Number of iterations	600
Numerical parameters of fluid computation	
Reference time step	0.5 (s)
Number of iterations	600

The **Improved pressure interpolation in stratified flow** algorithm will be used.

2.4 Results

Figure II.4 shows the evolution of the temperature in the solid domain without **Conjugate heat transfer** with the fluid domain. We have represented in Figure II.5 the evolution of the temperature in the fluid domain without coupling with SYRTHES.

Figure II.6 shows the evolution of the temperature in the solid and fluid area with the **conjugate heat transfer activated**. The natural convection in the fluid domain due to the temperature difference imposed by the solid disks is clearly visible with the velocity field and vector.

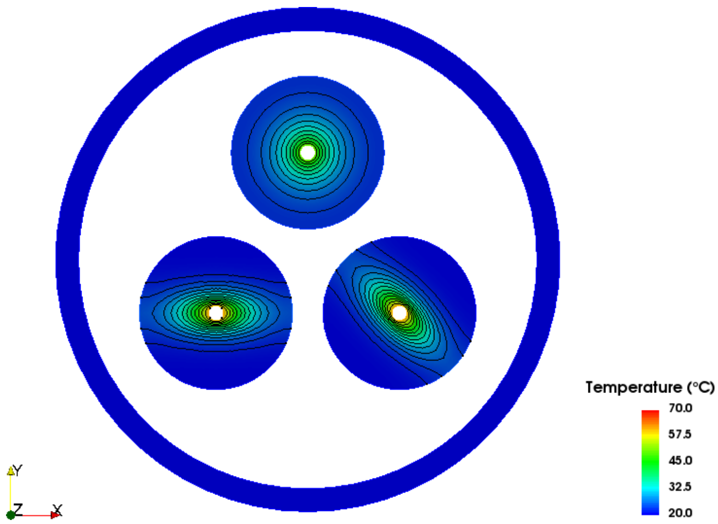


Figure II.4: The temperature evolution in the **solid domain without coupling method**

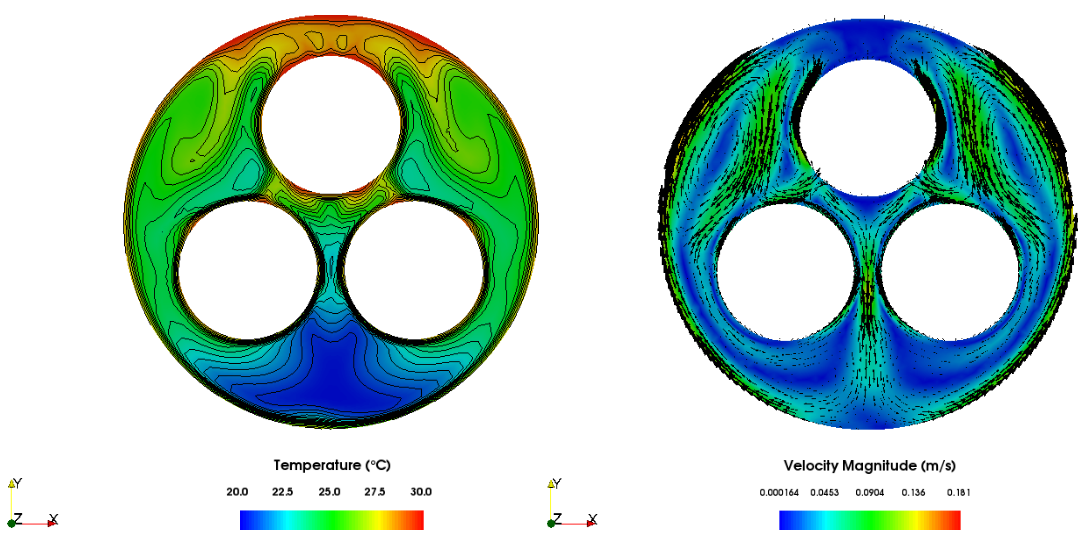
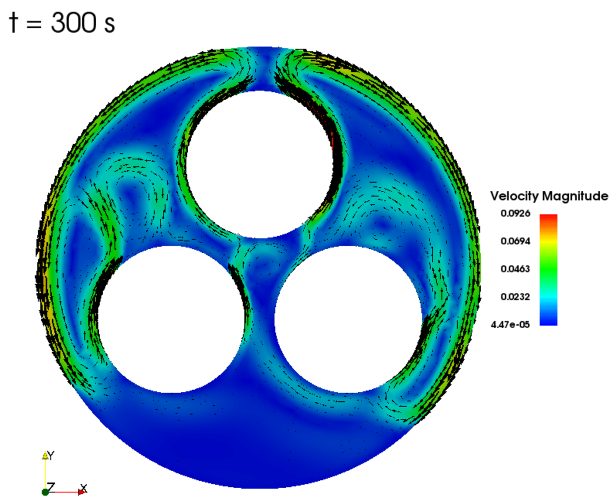


Figure II.5: The temperature evolution in the **fluid domain without coupling method**



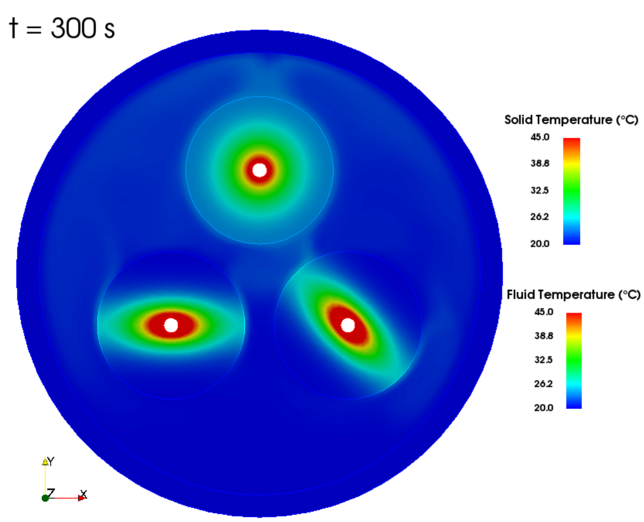


Figure II.6: Evolution of temperature and velocity magnitude

Part III

Step by step solution

1 Solution for case1

• **Step 1**: check the post-install required for coupling *Code_Saturne* with SYRTHES.
The first step is to check the post-install required for coupling with SYRTHES and verify if the SYRTHES `PATH` is correctly known in the system environment. We just need to edit the batch file[1] name `code_saturne.cfg` as below:

```
$ vim <install-prefix>/etc/code_saturne.cfg
>### Set the location to the SYRTHES installation directory.
> syrthes = <install-prefix-syrthes>
```

• **Step 2**: source the `syrthes.profile` file in your user environment.
Before using SYRTHES alone, you have to copy and source this file to define SYRTHES environment variables (like `$SYRTHES4_HOME`) in your terminal, as follows:

```
$ cp <install-prefix-syrthes>/bin/syrthes.profile .
$ source syrthes.profile
$ echo $SYRTHES4_HOME (to check the SYRTHES PATH in your environment)
```

After having defined correctly your environment, to be able to launch a coupling computation *Code_Saturne*-SYRTHES or a SYRTHES computation alone, you just have to create the coupling study directory.

• **Step 3**: create the 🗀 `3disks2D` study directory, and the two case subdirectories 🗀 `fluid` and 🗀 `solid`.

This is done using the standard command:

```
$ code_saturne create -s 3disks2D -c fluid --syrthes solid
> code_saturne 3.0 study/case generation
> o Creating study '3disk2D' ...
> o Creating case 'fluid' ...
> SYRTHES4 home directory:  <install-prefix-syrthes>
> MPI home directory:  /usr
>
>**************************************************
> solid :  creating SYRTHES case ...
> <install-prefix-syrthes>
> OK !
>**************************************************
```

• **Remark**: The fluid mesh must be copied in the directory 🗀 `MESH`. The solid mesh must be copied in the subdirectory 🗀 `solid`.

[1]see the installation guide, name `install.pdf`, in `<install-prefix>/share/doc/code_saturne/` directory.

1.1 Launching the SYRTHES computation alone

The preparation of the computation for `case5` is defined below:

- **Step 1**: `Launch the SYRTHES Graphical User Interface (syrthes.gui),`
- **Step 2**: `Create a` **New Data File**`,`
- **Step 3**: `Check the name of the mesh and convert this one in .syr format,`
- **Step 4**: `Define the initial and boundary conditions for the conduction problem,`
- **Step 5**: `Define the physical properties of each disk {1, 2, 3 and 4},`
- **Step 6**: `Running the SYRTHES computation alone.`

• **Step 1**: launch the SYRTHES Graphical User Interface (GUI).
The SYRTHES Graphical User Interface is launched by the following command lines in the solid sub-directory:

```
$ cd 3disks2D/solid/
$ syrthes.gui &
```

• **Step 2**: choose `New Data File` inside the (GUI).

Figure III.1: Running the SYRTHES IHM with `syrthes.gui`

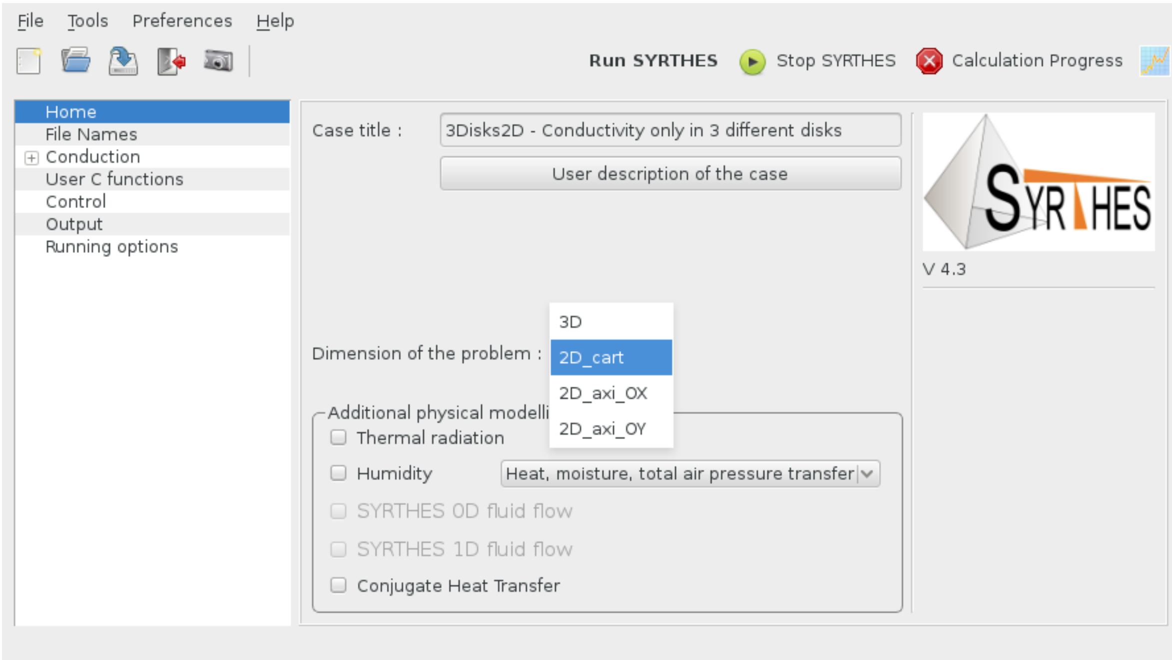


Figure III.2: Define the dimension and physical modelling of the treated problem

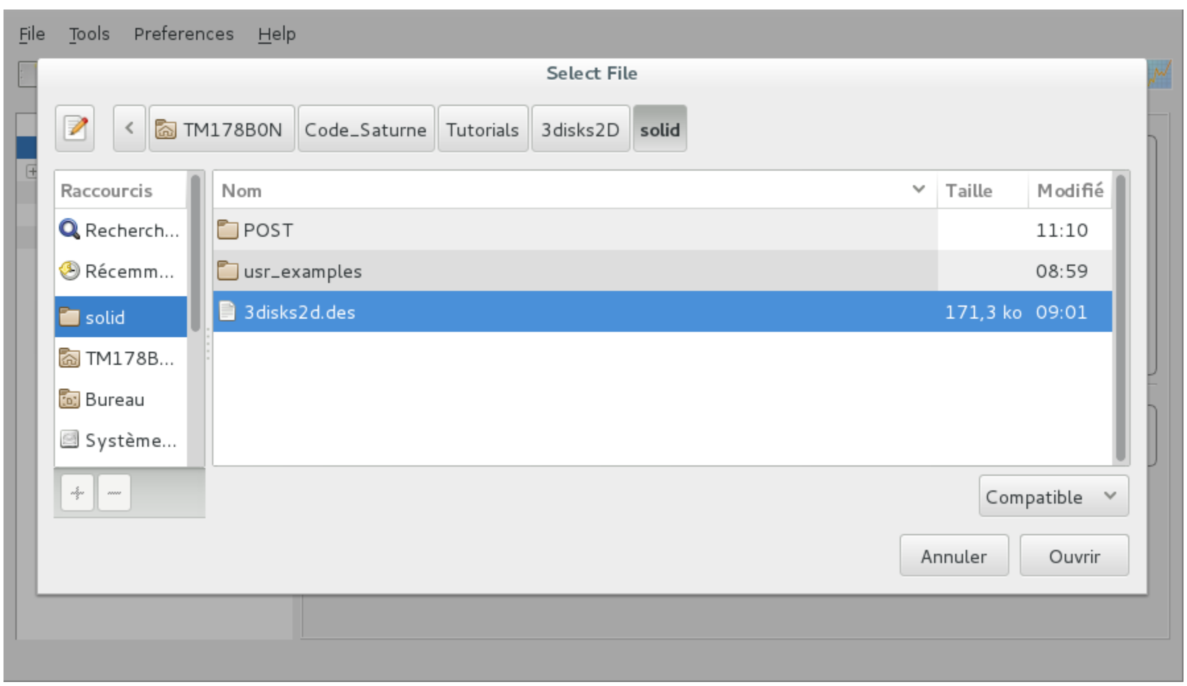


Figure III.3: Choose the 2D solid mesh file with the format `.des`.

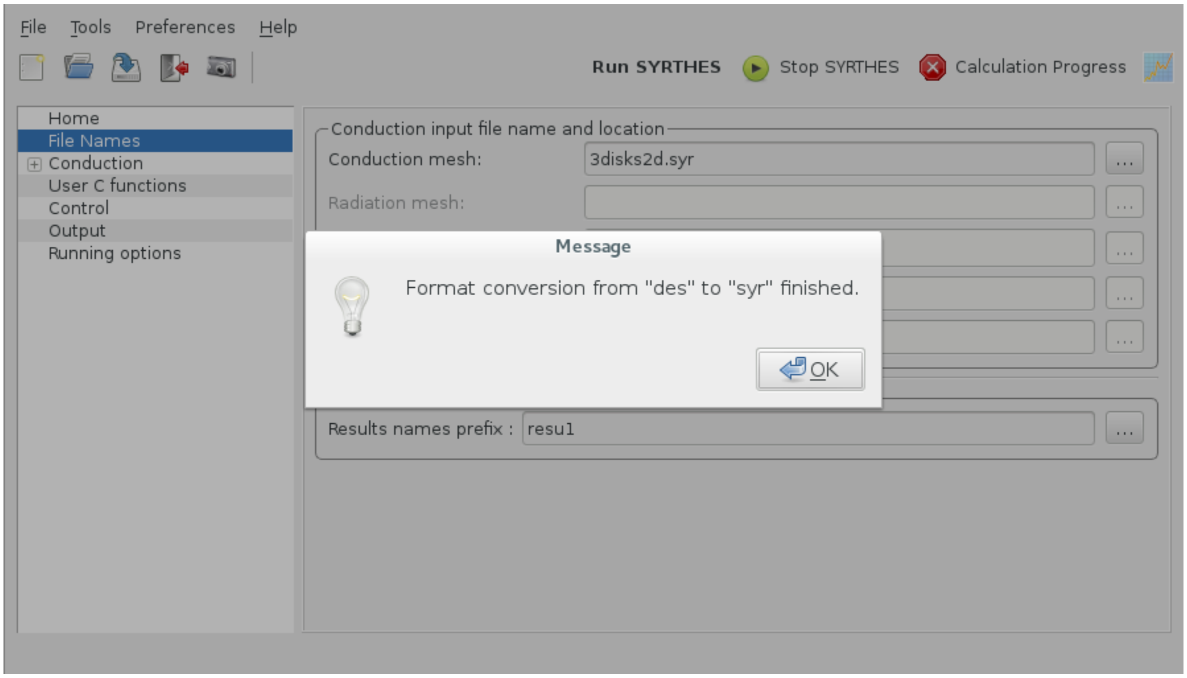


Figure III.4: The SYRTHES (GUI) directly converts the `.des` to the `.syr` format.

• **Remark**: Inside the SYRTHES Graphical User Interface (GUI), we can load the SIMAIL format `*.des` for the solid mesh. This one will be automatically transformed to the `*.syr` format.
It can also be done with the following command line:

```
$ convert2syrthes4 -m 3rond2d.des
```

• **Remark**: You can convert the `*.syr` format into a `*.med` format. Like that, you can load the `*.med` file inside SALOME, after having used this command line below:

```
$ syrthes4med30 -m 3rond2d.syr -o 3rond2d.med
```

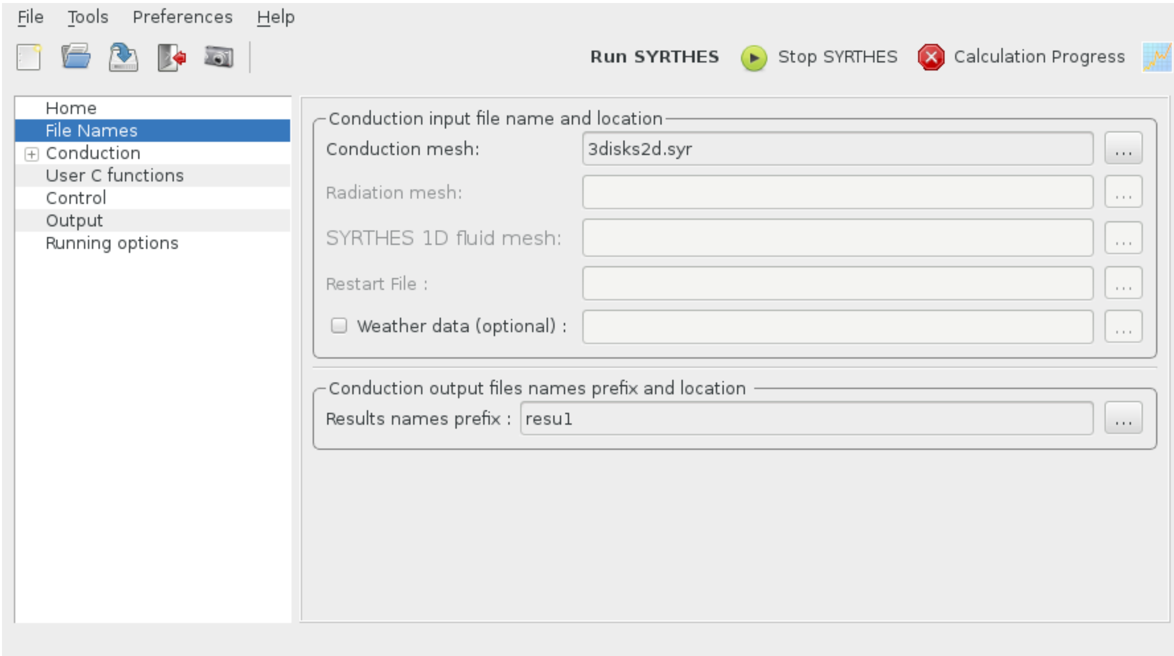


Figure III.5: Choose a name for the results files `.res`, `.his` and `.rdt`

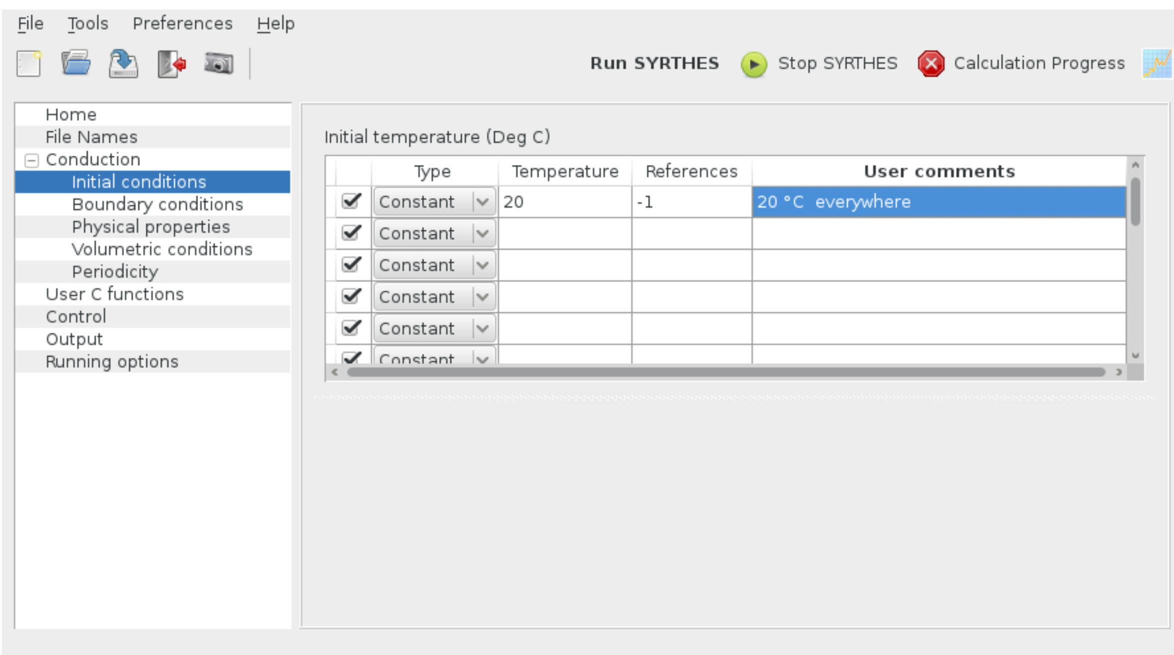


Figure III.6: Define the initial temperature conditions inside the different disks.

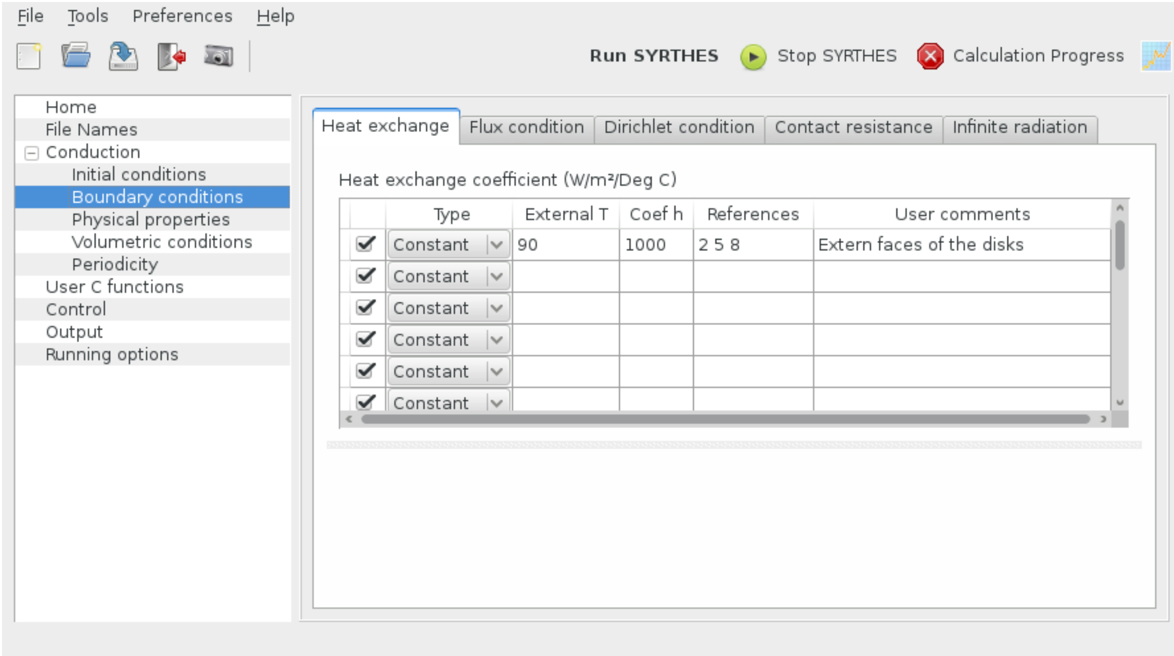


Figure III.7: Define the temperature boundary conditions for the extern faces of the three disks.

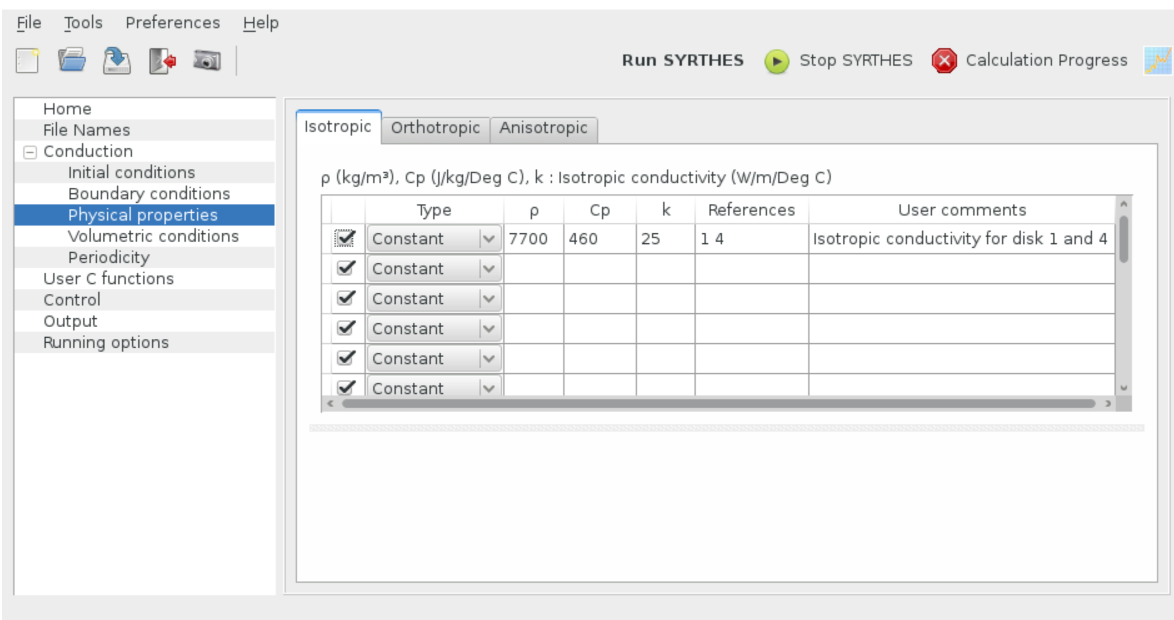


Figure III.8: Define the physical properties for the disk 1 and 4 with isotropic conductivity.

• **Remark**: To correctly identify the volume references associated to a specific physical property, we can check the mesh regions directly inside ParaView after having used following command line:

```
$ syrthes4ensight -m 3rond2d.syr -o mesh_3rond2d
******************************************************
--> geometry file name :  mesh_3rond2d.ensight.geom
--> case file name :  mesh_3rond2d.ensight.case
```

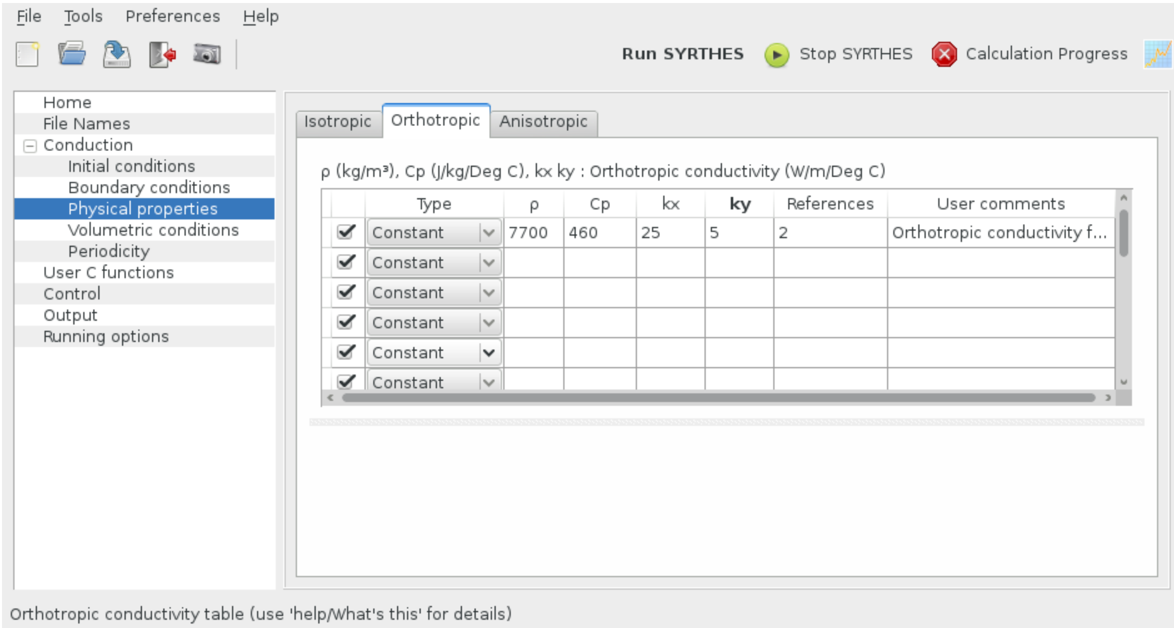


Figure III.9: Define the physical properties for the disk 2 with isotropic conductivity.

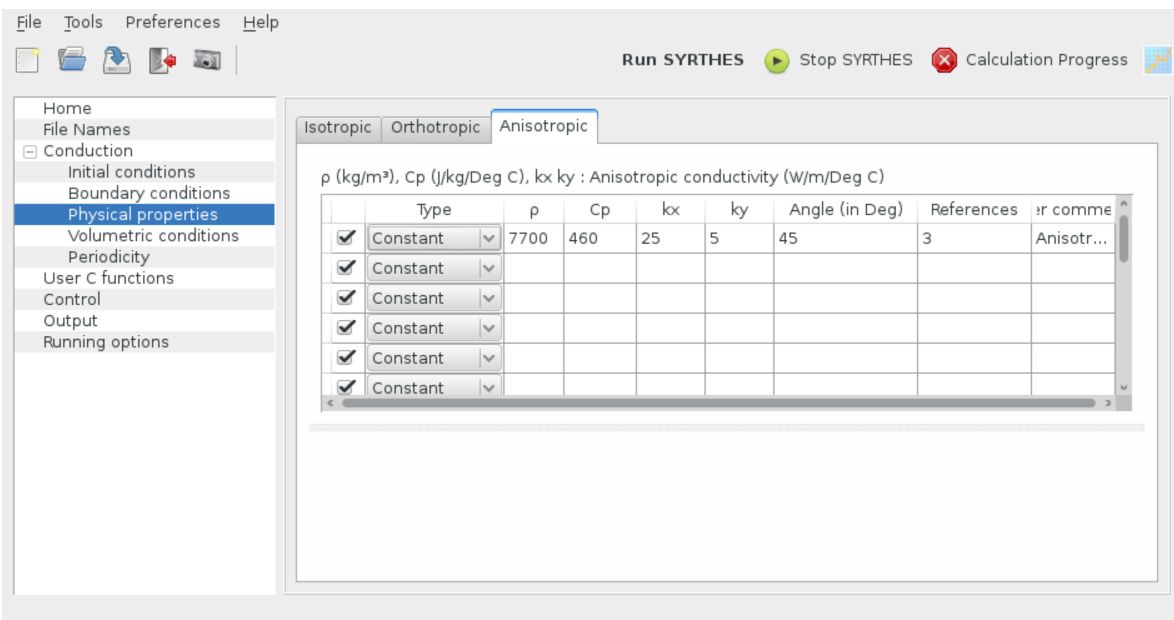


Figure III.10: Define the Physical properties for the disk 3 with anisotropic conductivity.

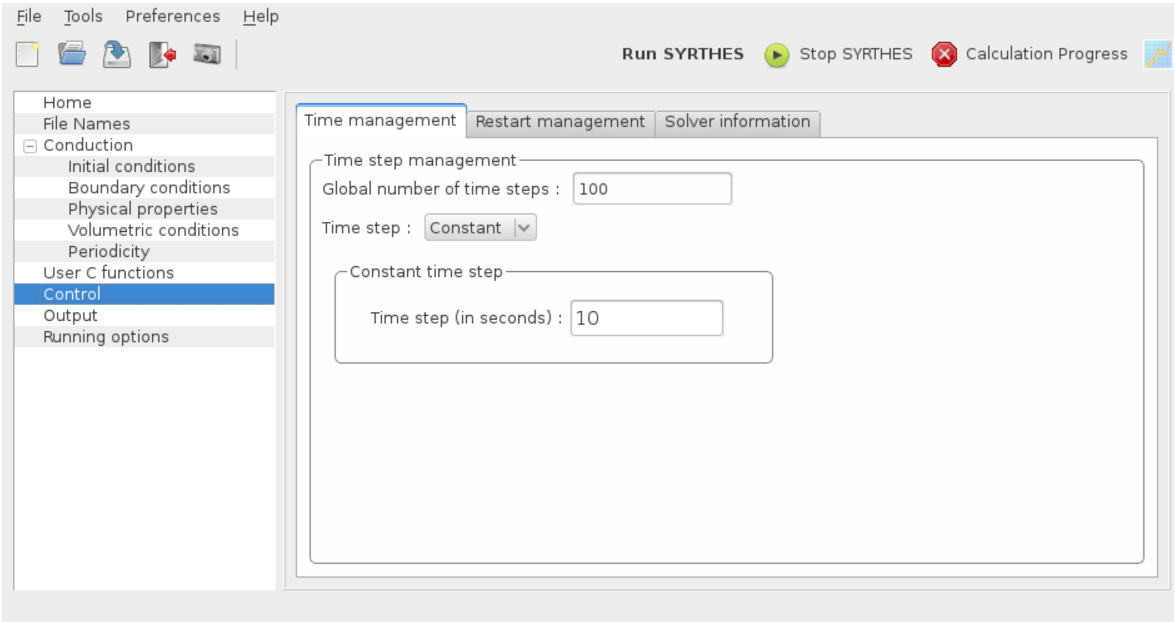


Figure III.11: Define the global number of time steps and the time step for the 2D solid conduction computation.

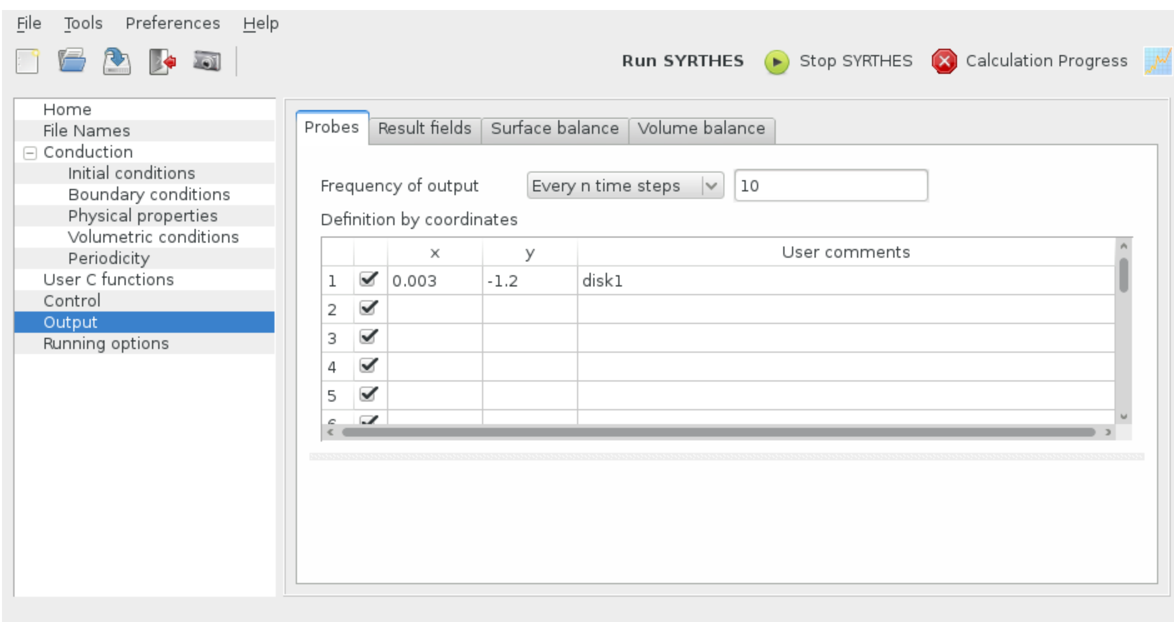


Figure III.12: Define the probe coordinates for output management.

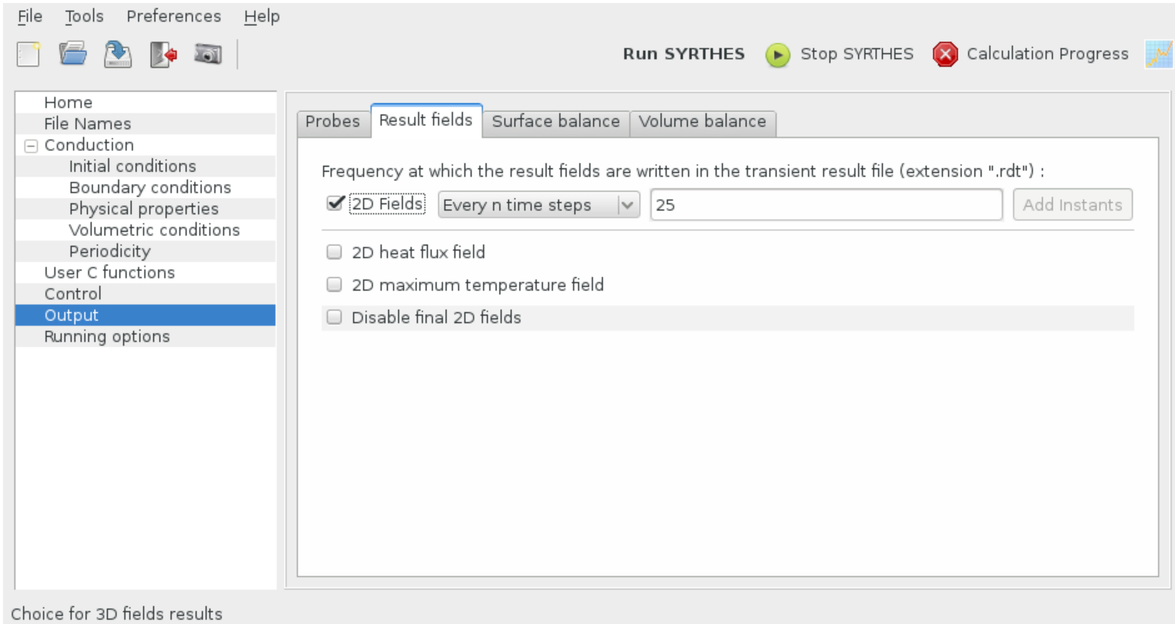


Figure III.13: Define the frequency at which the results fields are written

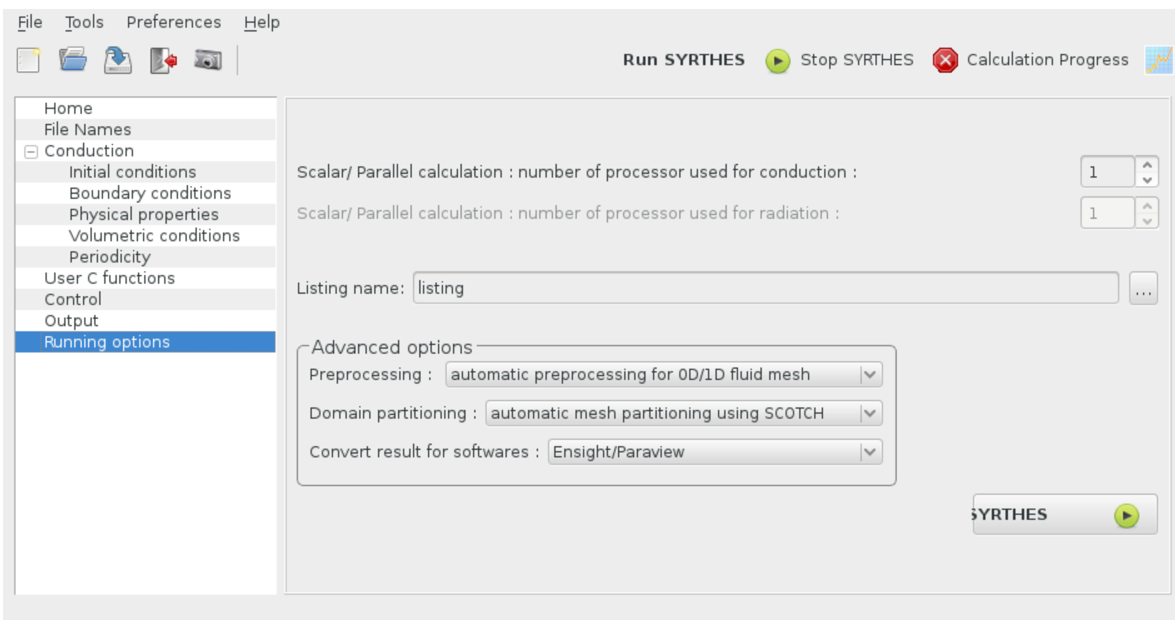


Figure III.14: Define the file name of the SYRTHES listing and the number of processors used.

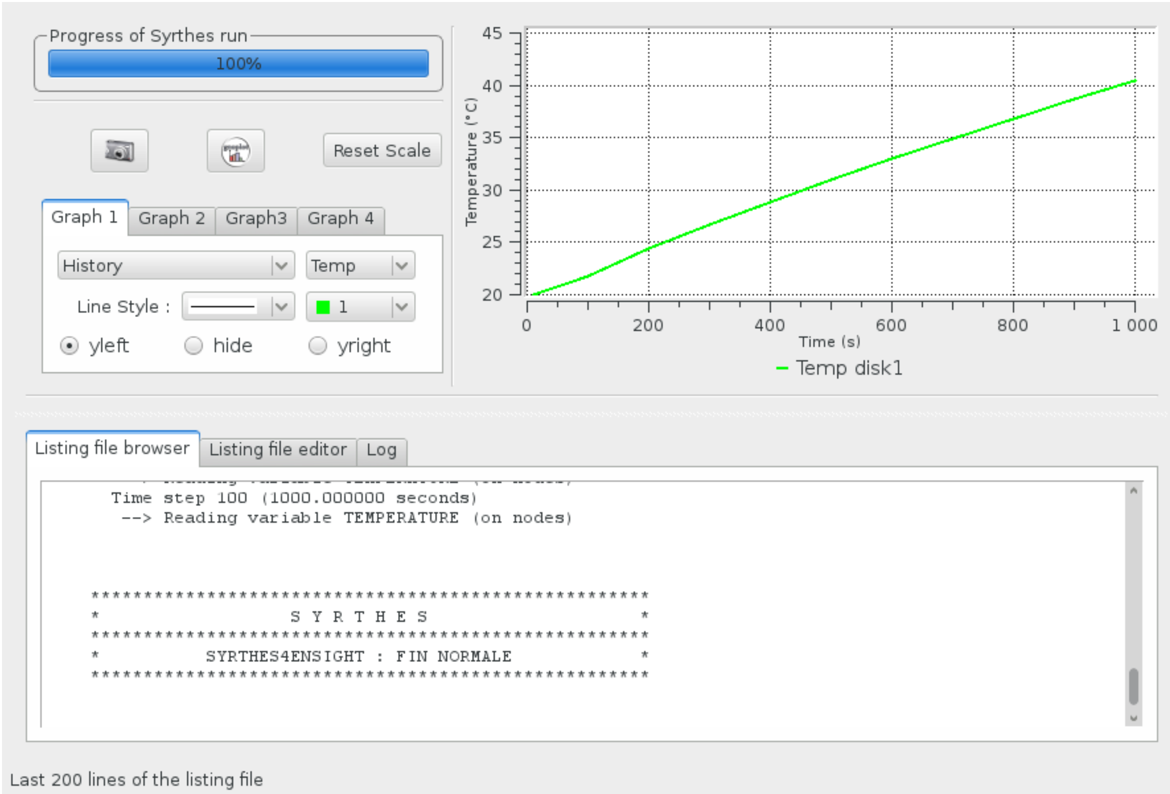


Figure III.15: Screenshot of the computation progress window.

• **Remark**: We can visualize the temperature results fields by applying the following command line to the results file `resu1.res` or `resu1.rdt` (for the results saved at the last time step or the results saved at each time step):

```
$ syrthes4ensight -m 3rond2d.syr -r resu1.res -o Results_Temp
$ syrthes4ensight -m 3rond2d.syr -r resu1.rdt -o Chrono_Temp
```

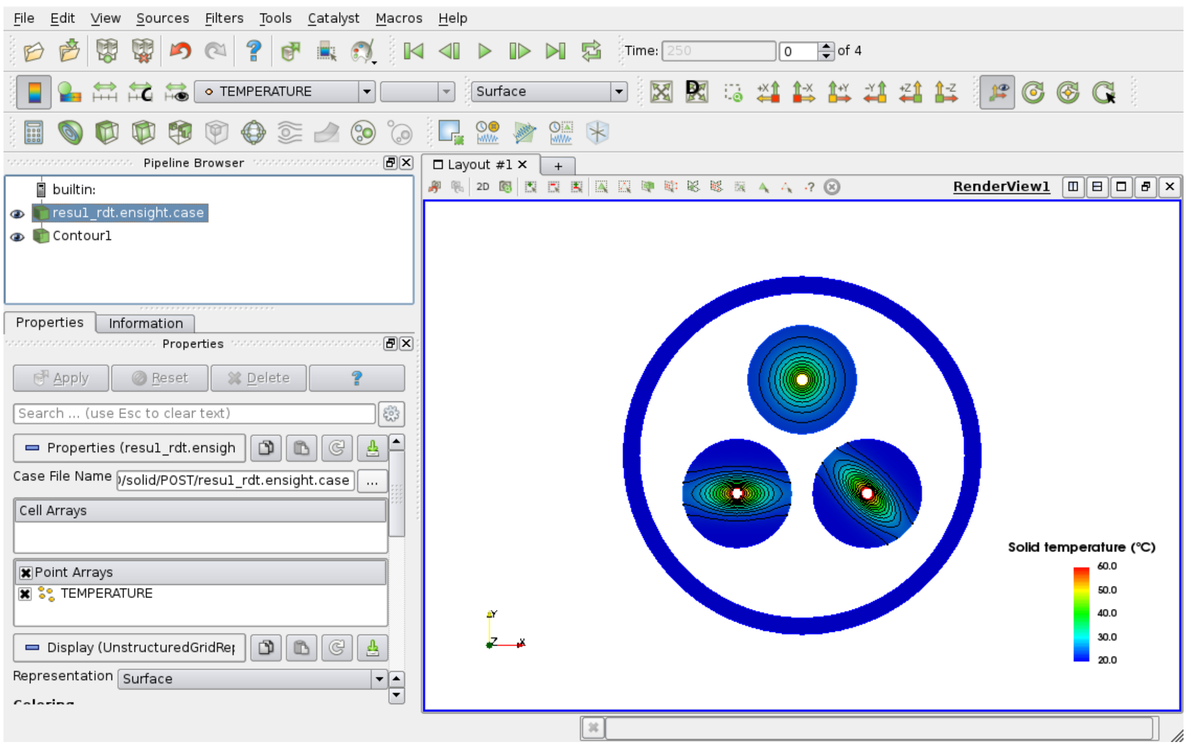


Figure III.16: Screenshot of the 2D solid temperature Field.

1.2 Launching the *Code_Saturne* computation alone

The main steps of the preparation of the fluid computation alone for the `fluid` case is defined below:
- **Step 1**: Launch the *Code_Saturne* Graphical User Interface (`./SaturneGUI`),
- **Step 2**: Create a **New case**,
- **Step 3**: Check the quality of the fluid mesh by running a **Mesh quality criteria** calculation,
- **Step 4**: Define the physical properties of the disk for the air flow,
- **Step 5**: Define the initial and boundary conditions for the air flow problem,
- **Step 6**: Running the *Code_Saturne* computation alone.

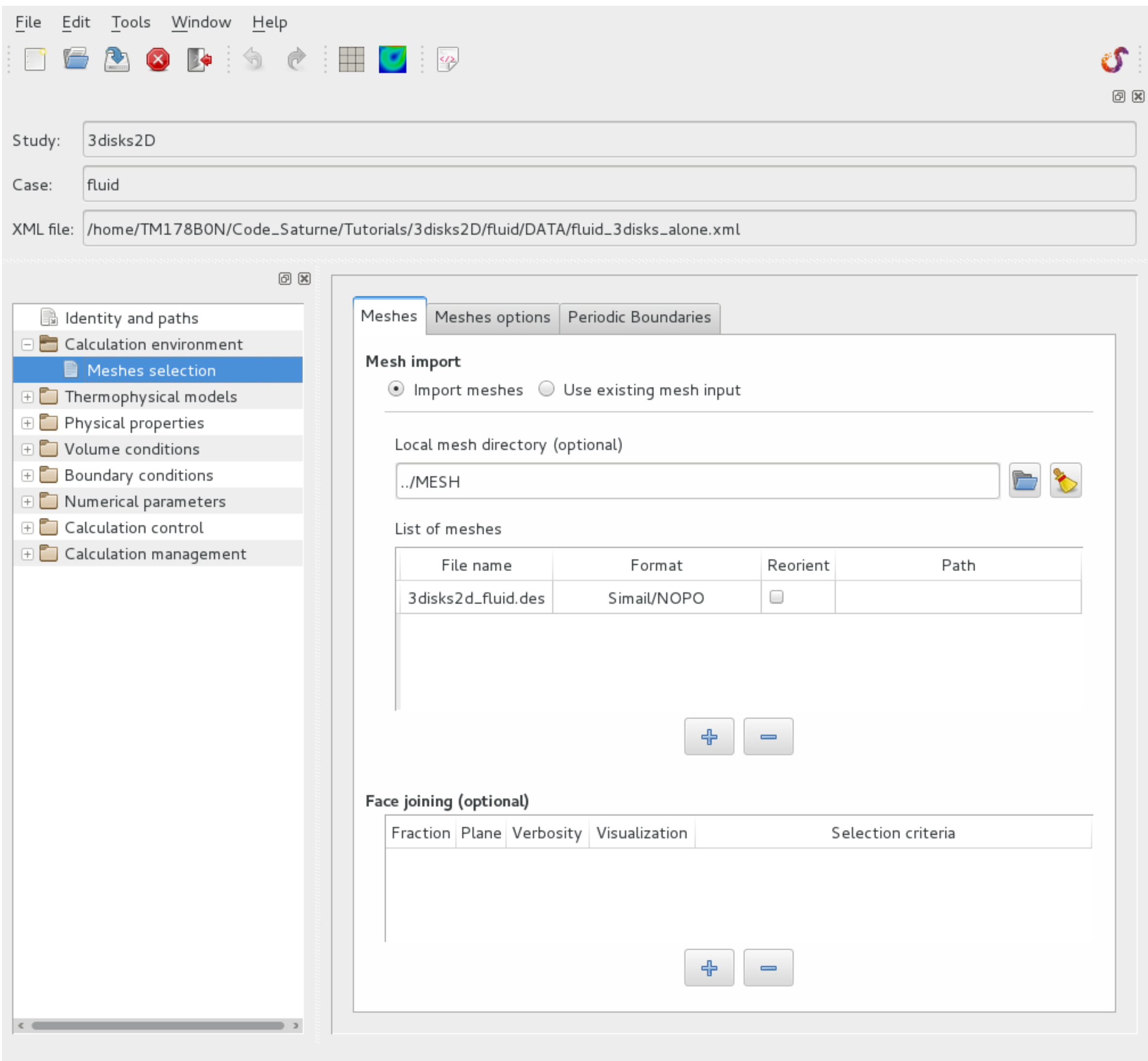


Figure III.17: Choose the fluid mesh with *Code_Saturne* (GUI)

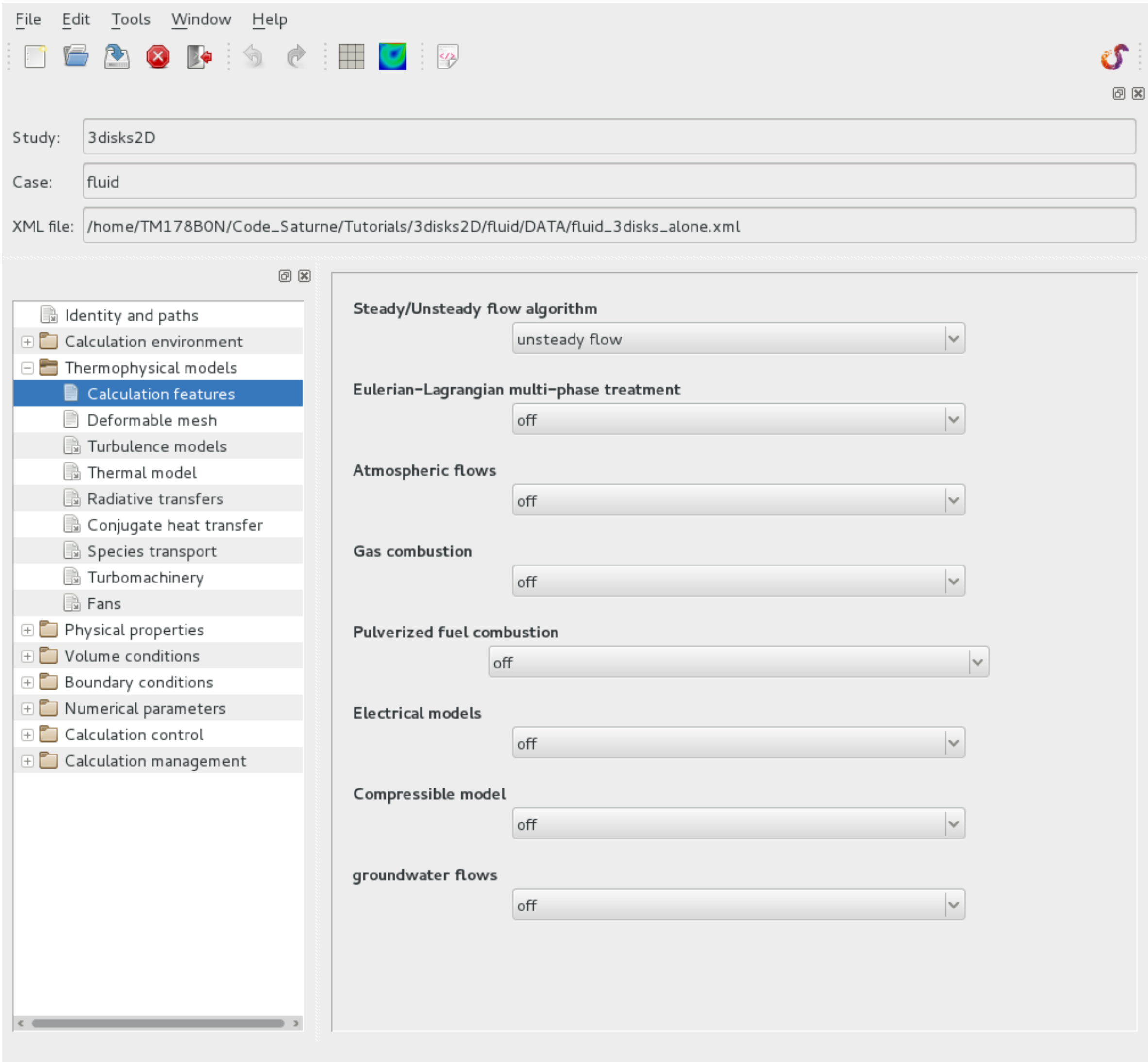


Figure III.18: Define the physical modelling associated to the air flow inside the fluid domain.

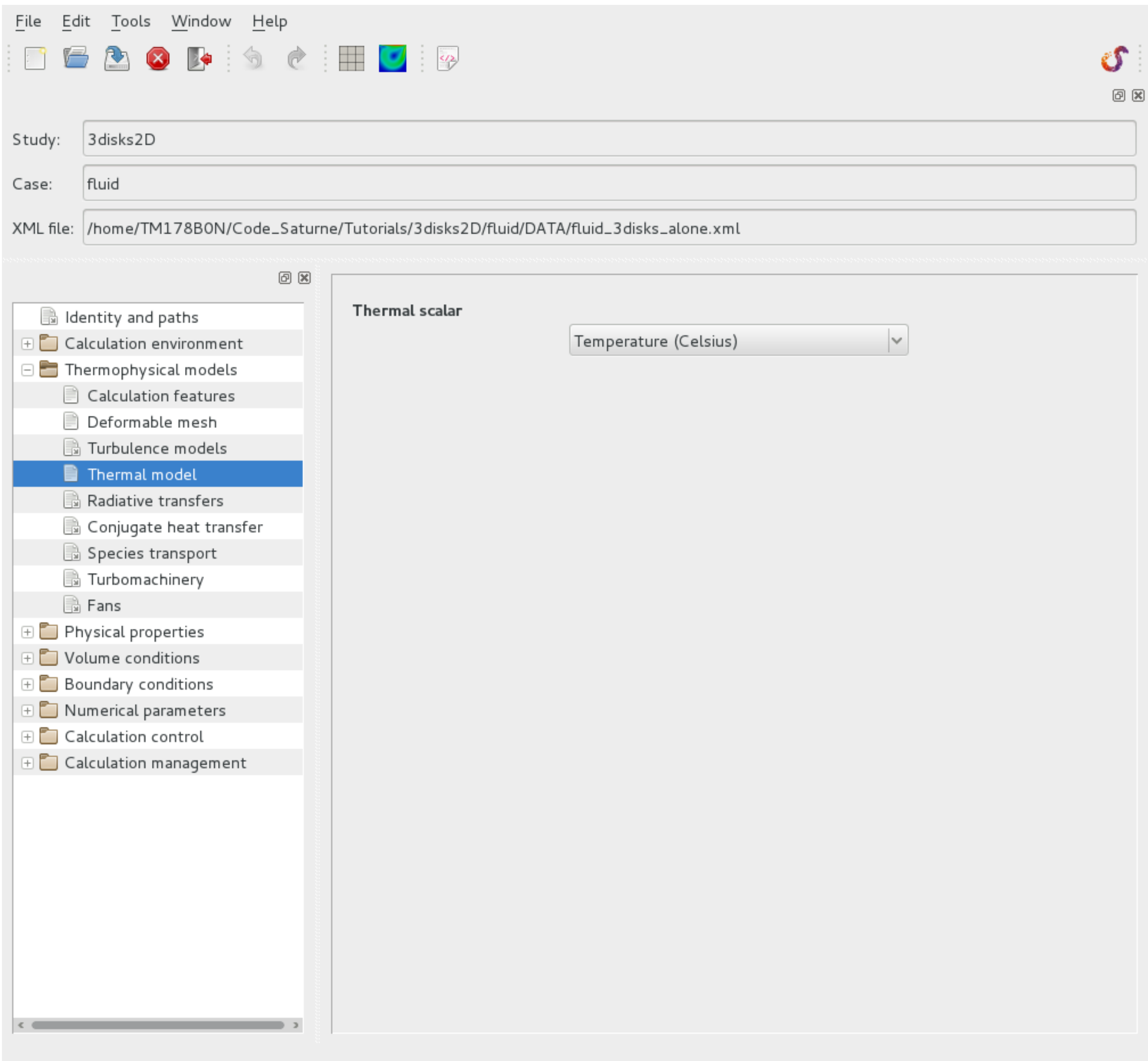


Figure III.19: Choose the Temperature scalar.

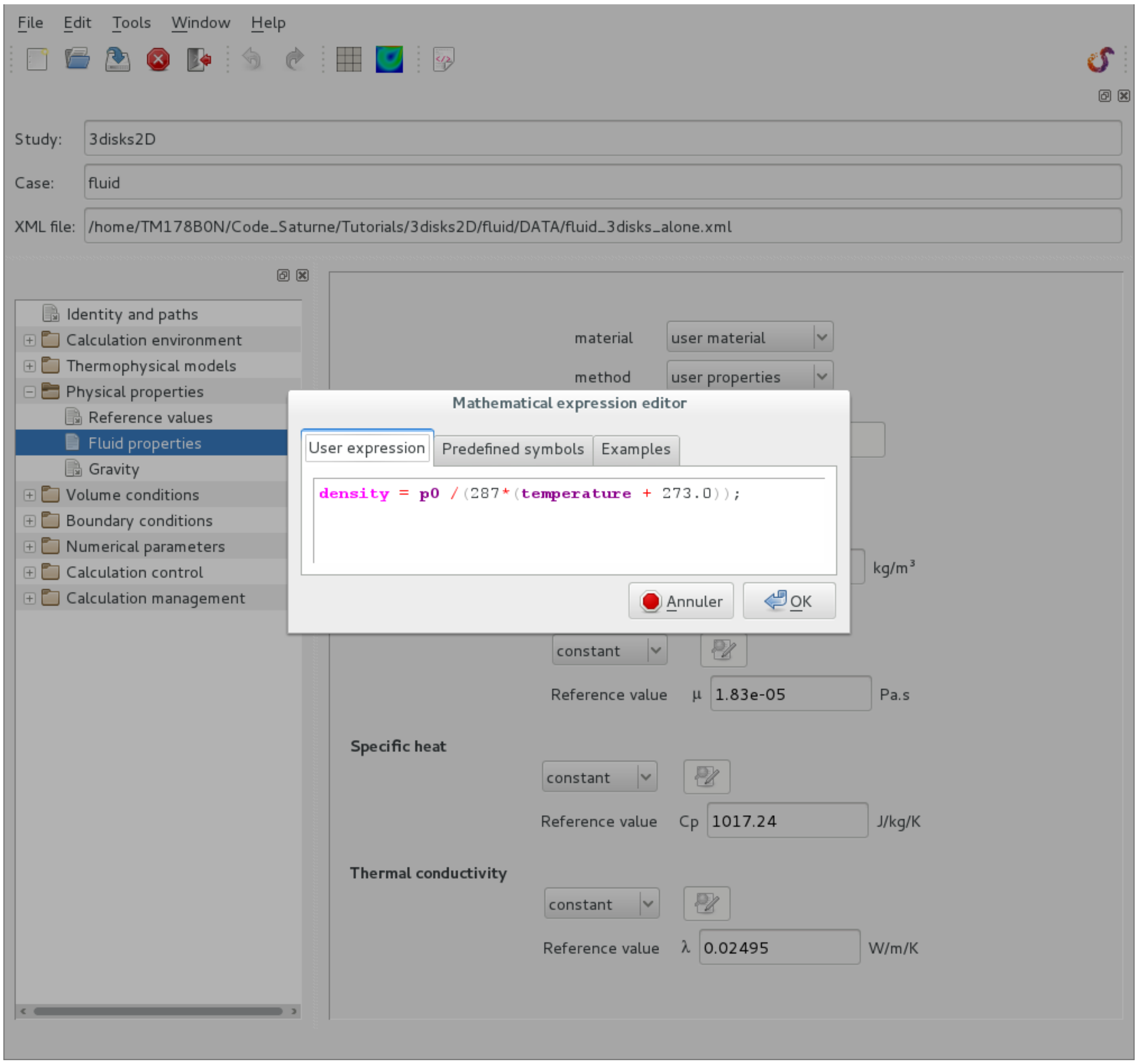


Figure III.20: Define the variable density with an ideal gas law inside the *Code_Saturne* (GUI).

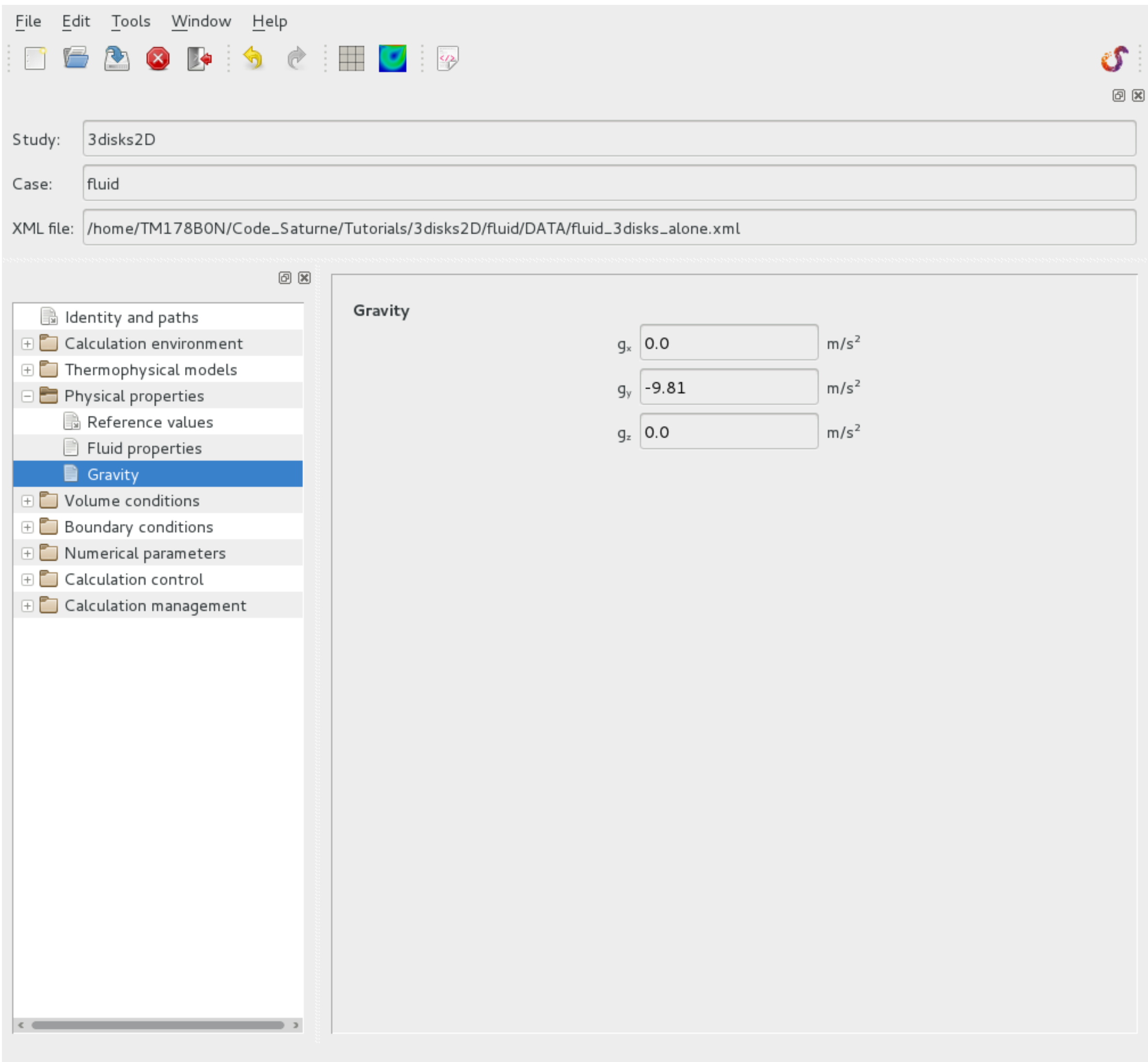


Figure III.21: Define the gravity

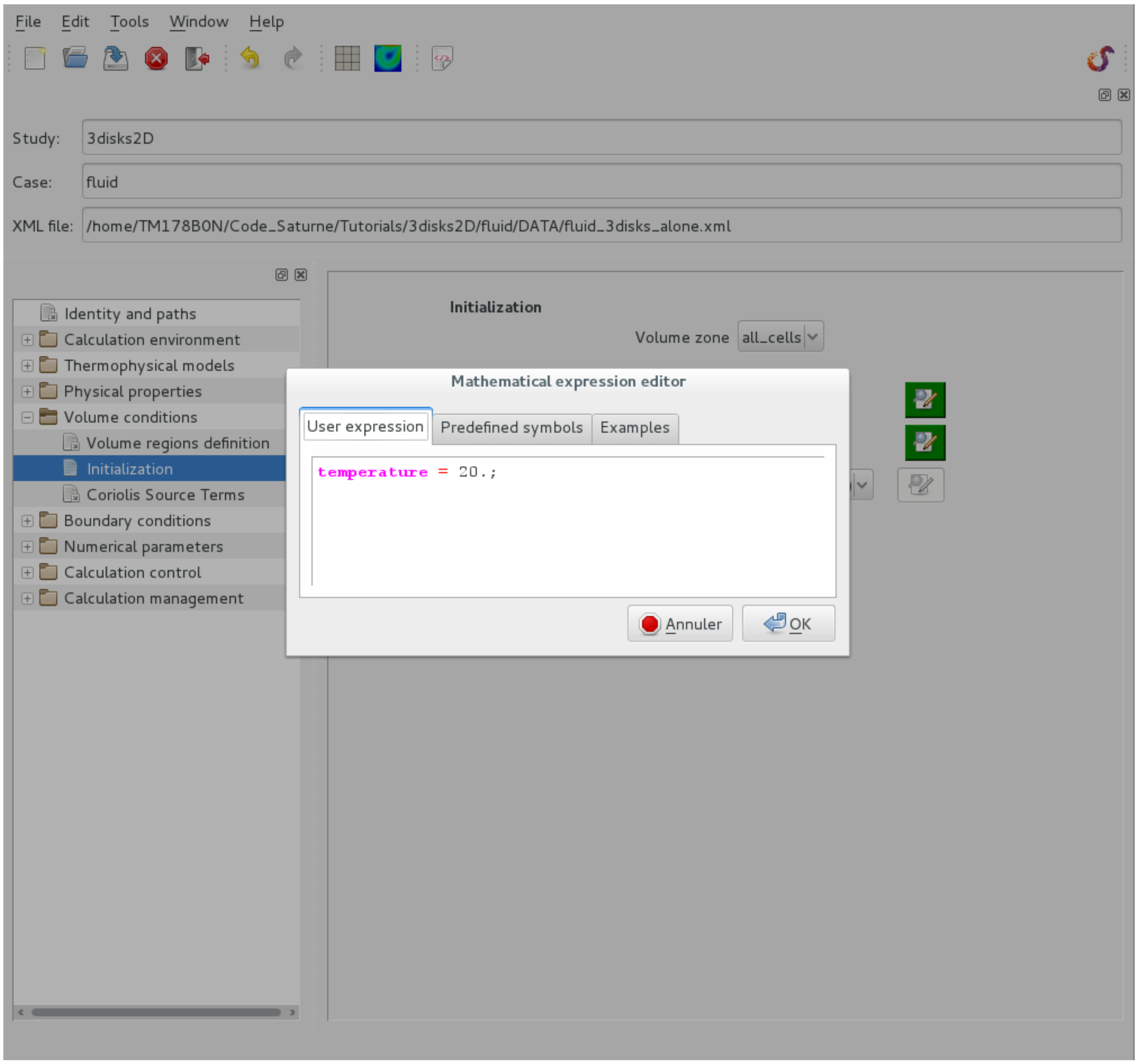


Figure III.22: Initalization of the velocity components and temperature variables.

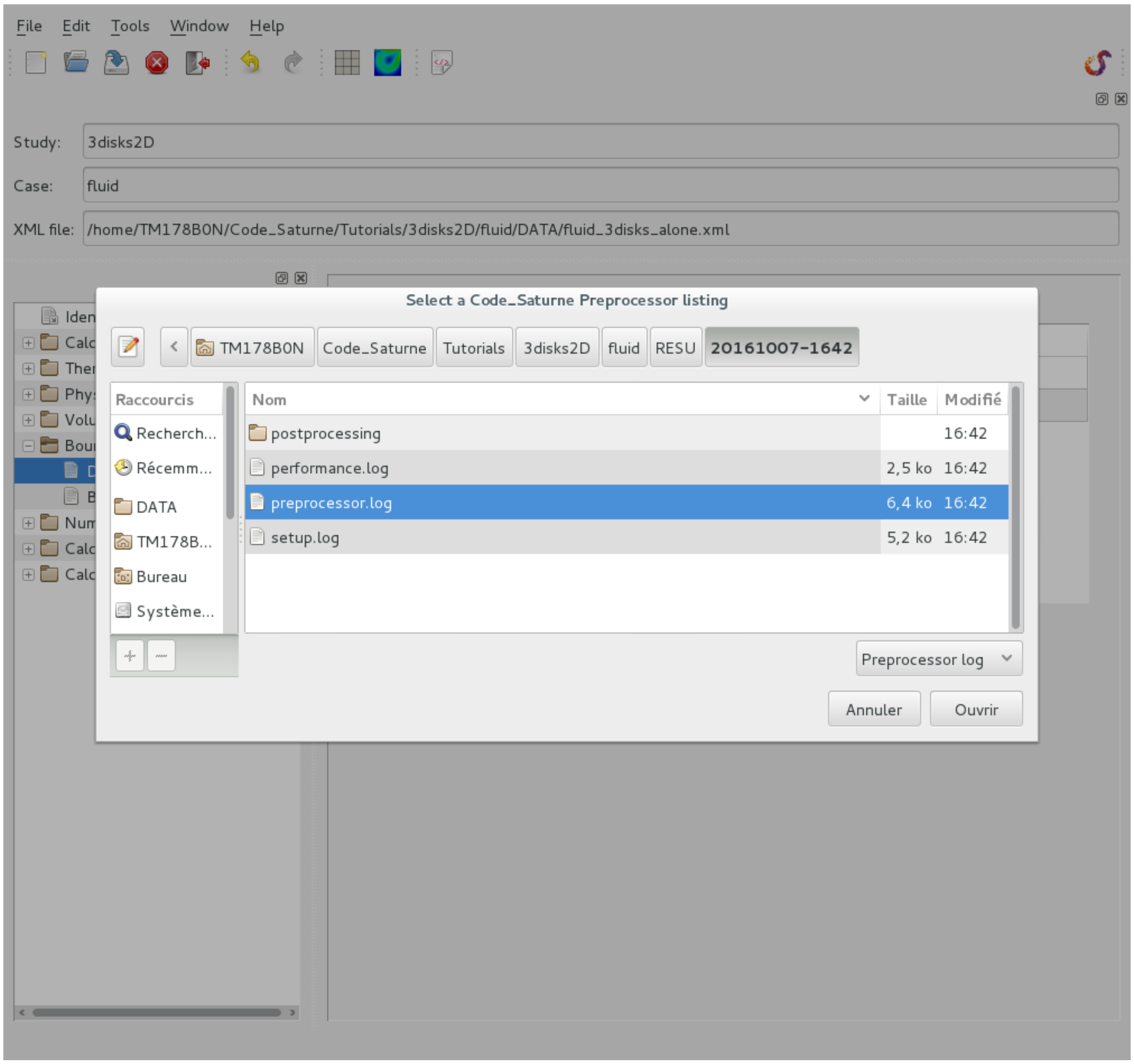


Figure III.23: Load the `preprocessor.log` file inside the *Code_Saturne* (GUI).

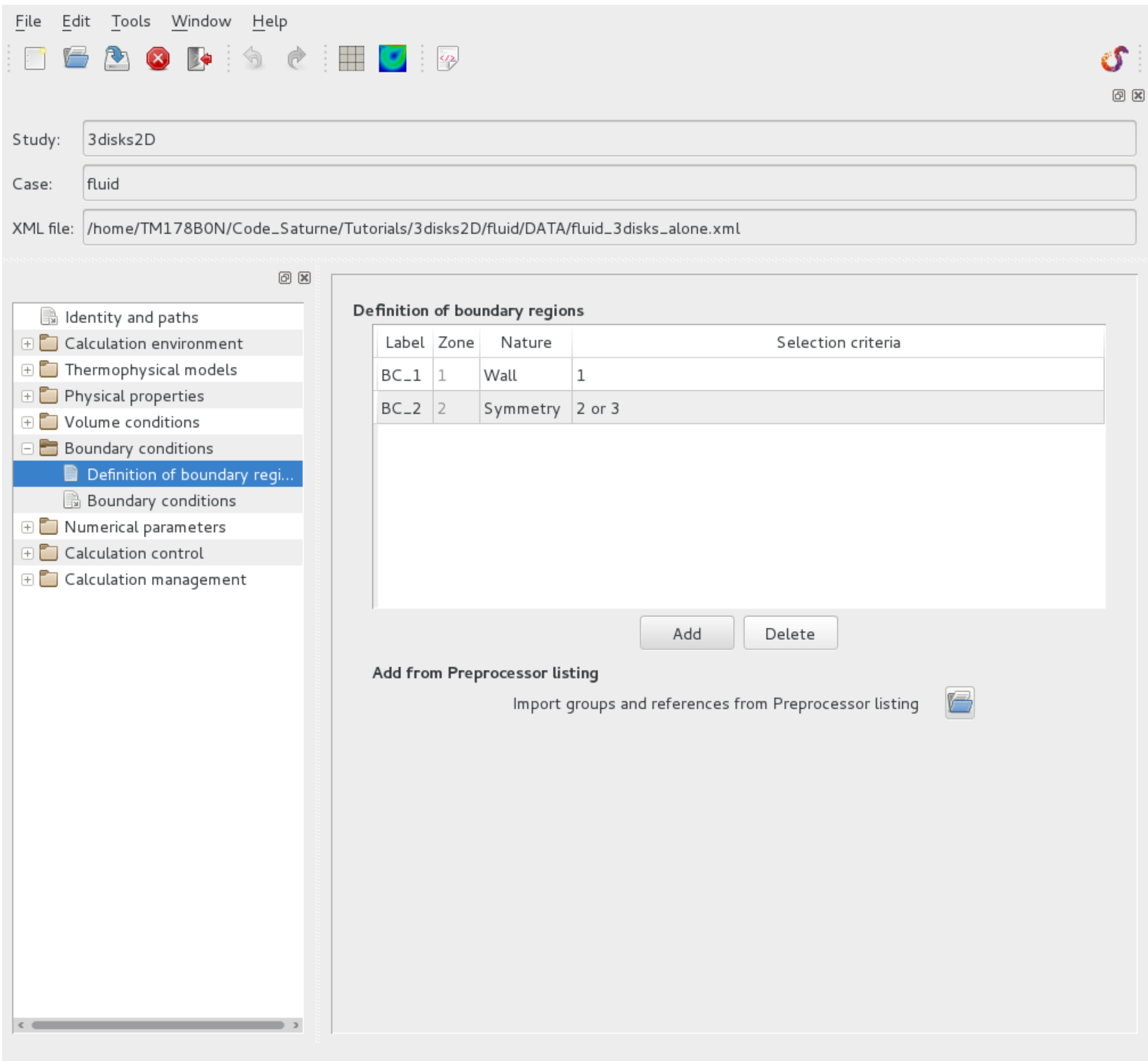


Figure III.24: Once the boundary regions automatically loaded, define the boundary conditions.

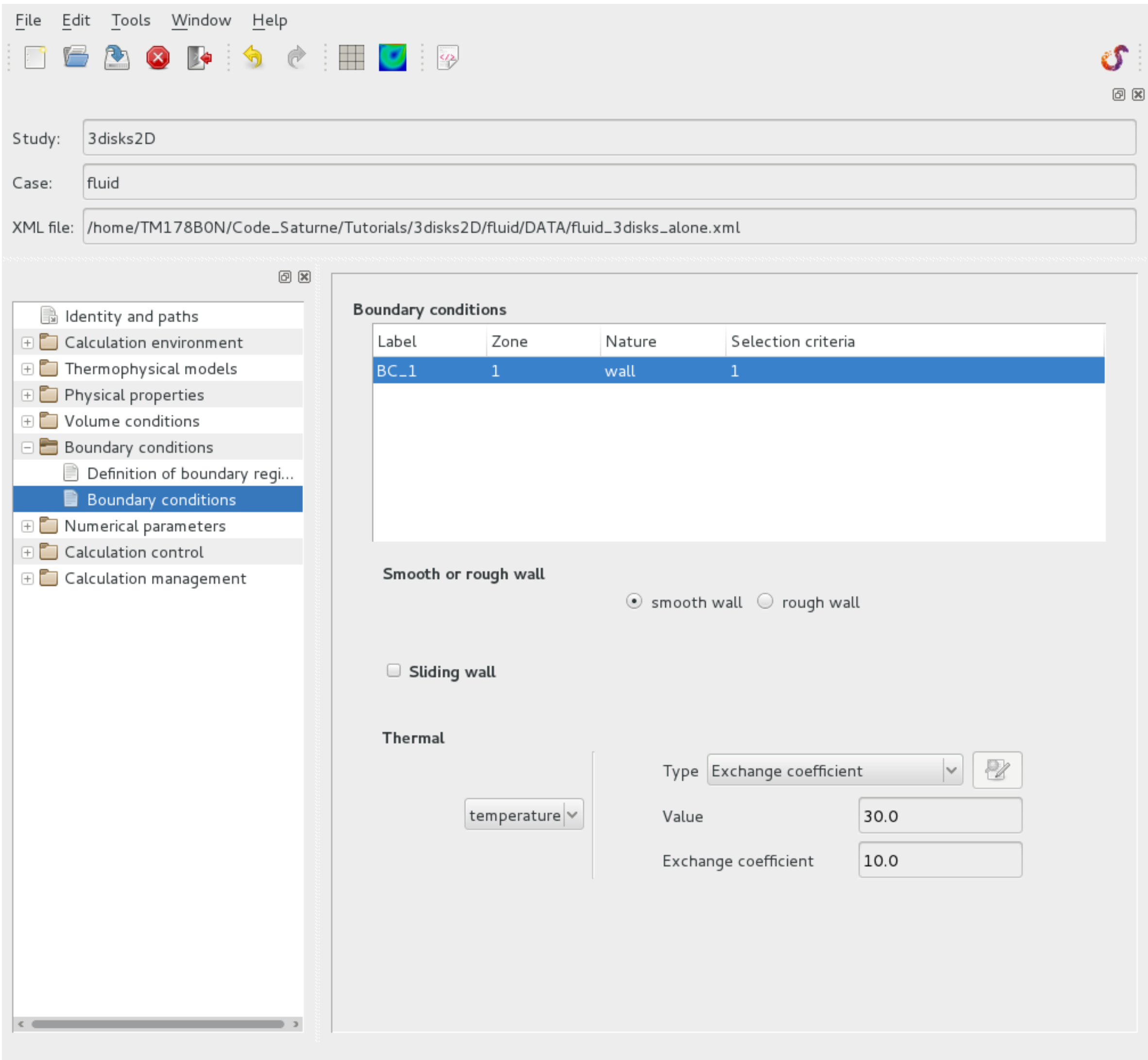


Figure III.25: Define a thermal transfer condition as wall boundary condition with a extern wall temperature $\mathrm{T}_{\mathrm{ext}} = 30°\mathrm{C}$ and a exchange coefficient $\mathrm{q}_{\mathrm{ext}} = 10$ $(W/m^2.K)$.

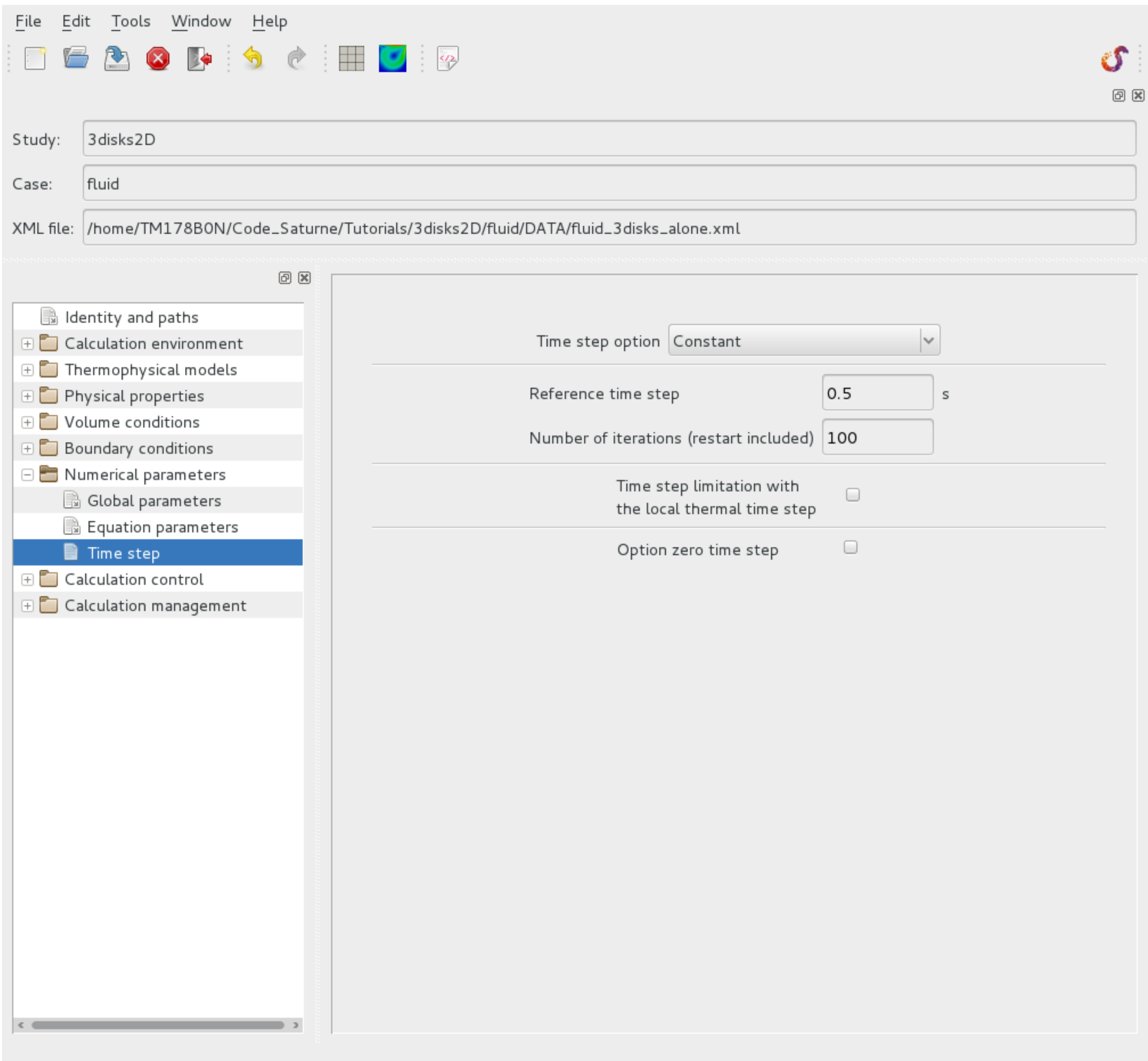


Figure III.26: Define the iterations number and time step.

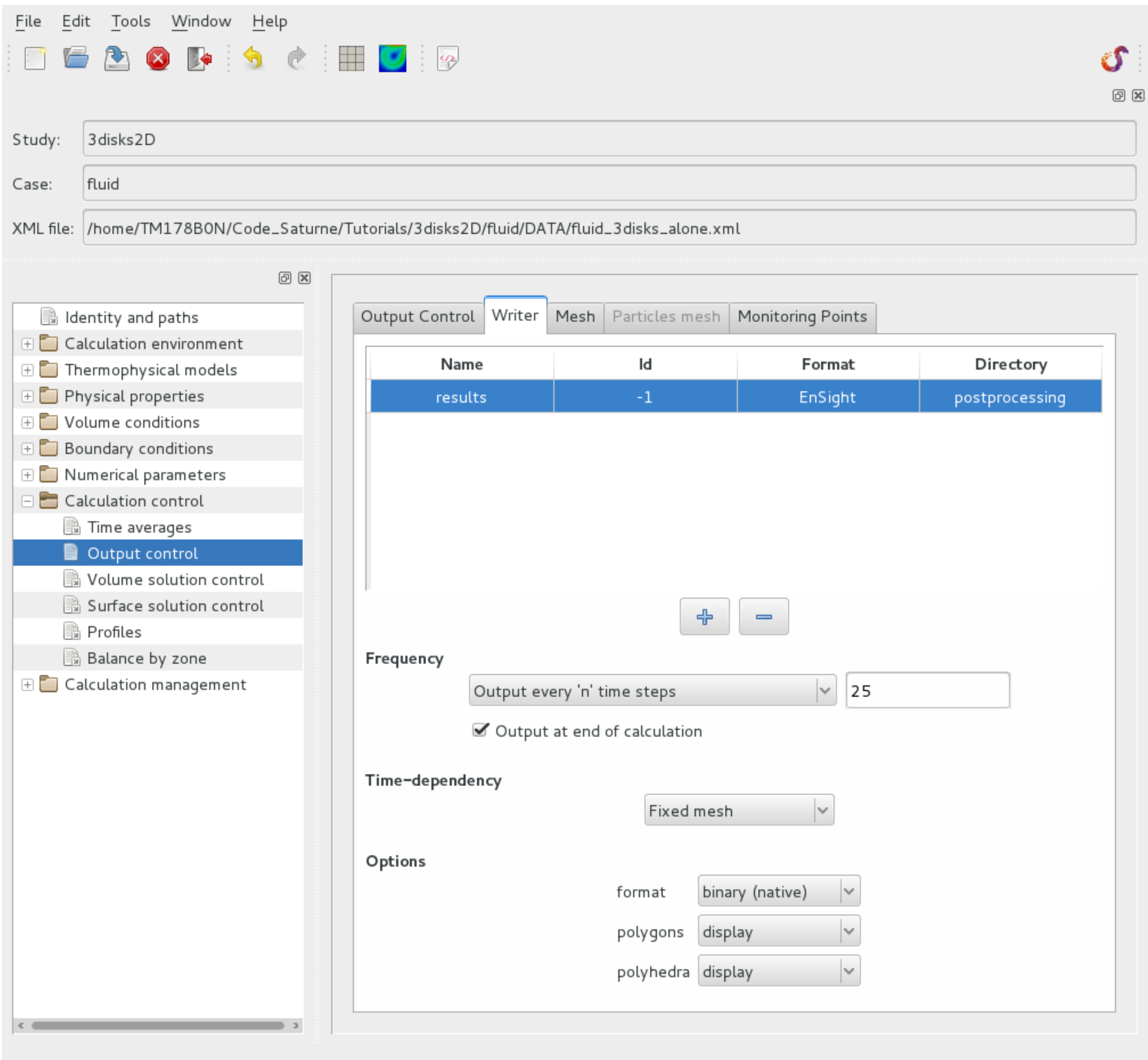


Figure III.27: Define the writer and frequency output inside the *Code_Saturne* (GUI).

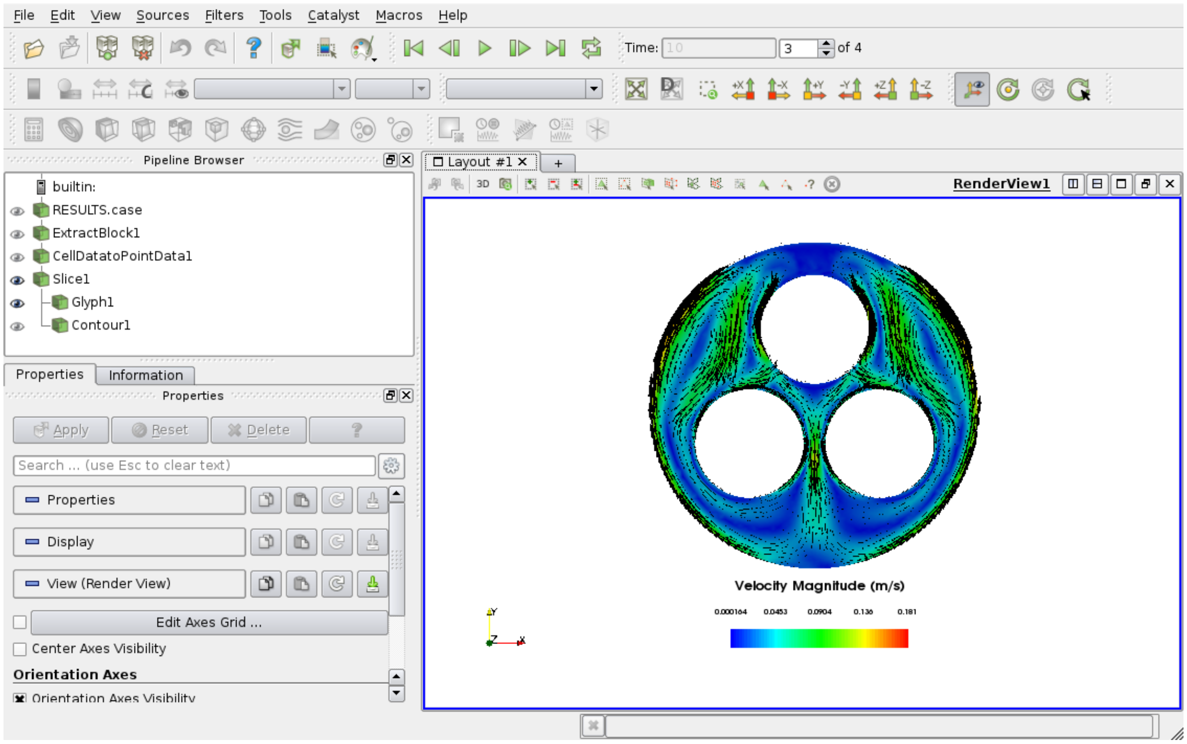


Figure III.28: Visualization of the 2D fluid velocity field

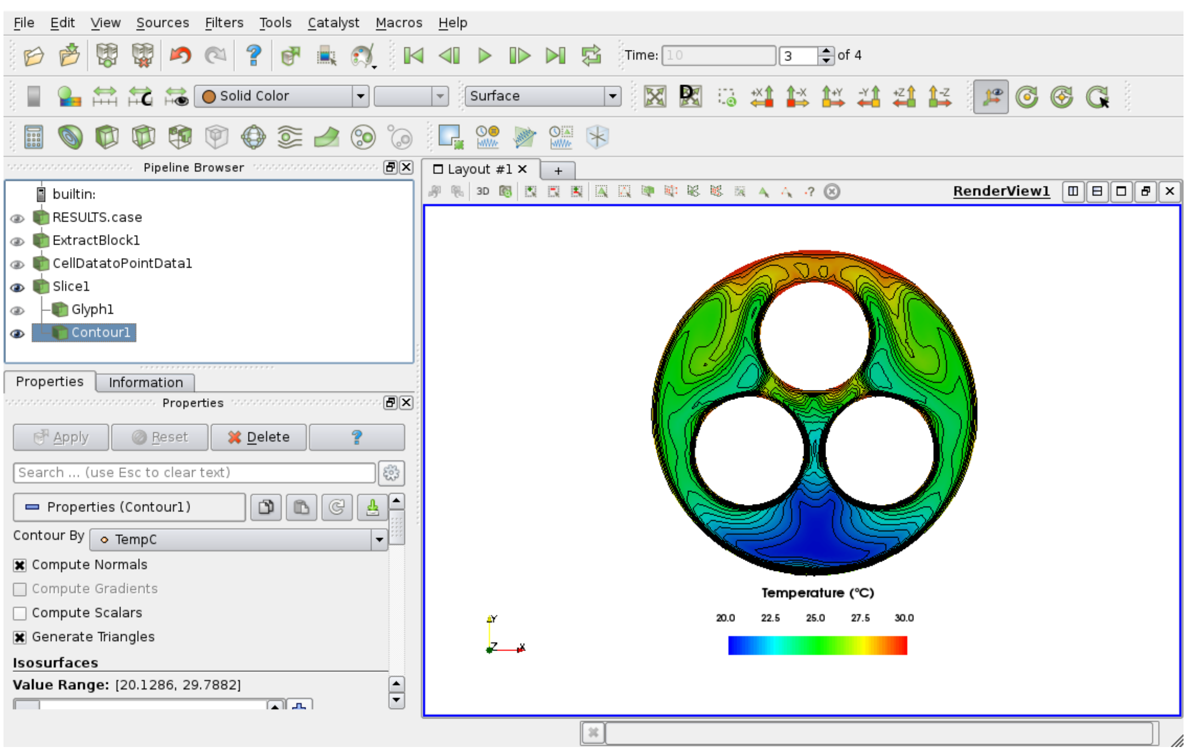


Figure III.29: Visualization of the 2D fluid temperature field

1.3 Launching the *Code_Saturne*-SYRTHES coupling computation

The last modification to prepare the coupling computation are given below:
- **Step 1**: Activate the conjugate heat transfer in the SYRTHES GUI,
- **Step 2**: Activate the conjugate heat transfer in the *Code_Saturne* GUI,
- **Step 3**: Give identical iterations number and time step for both codes,
- **Step 4**: Check the `coupling_parameters.py` python script and launch the calculation by executing the `runcase`.

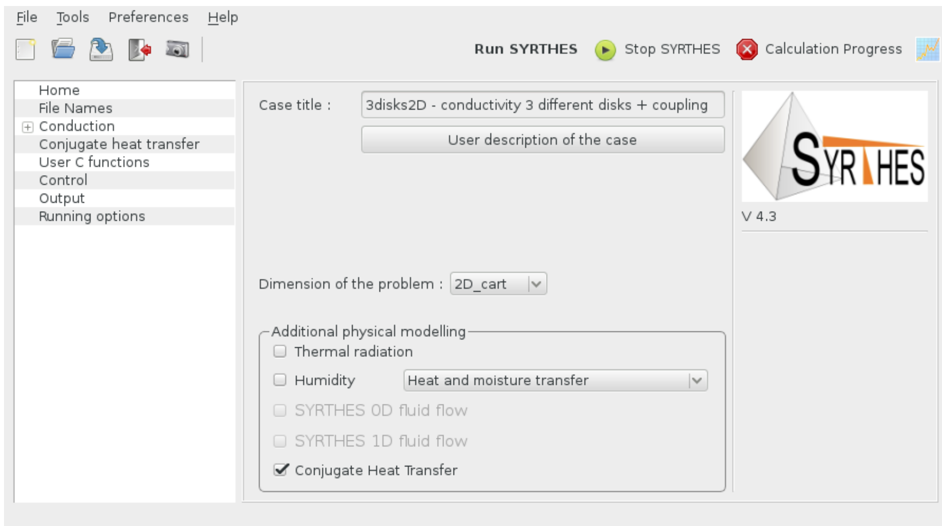


Figure III.30: Activate the conjugate heat transfer for the solid domain.

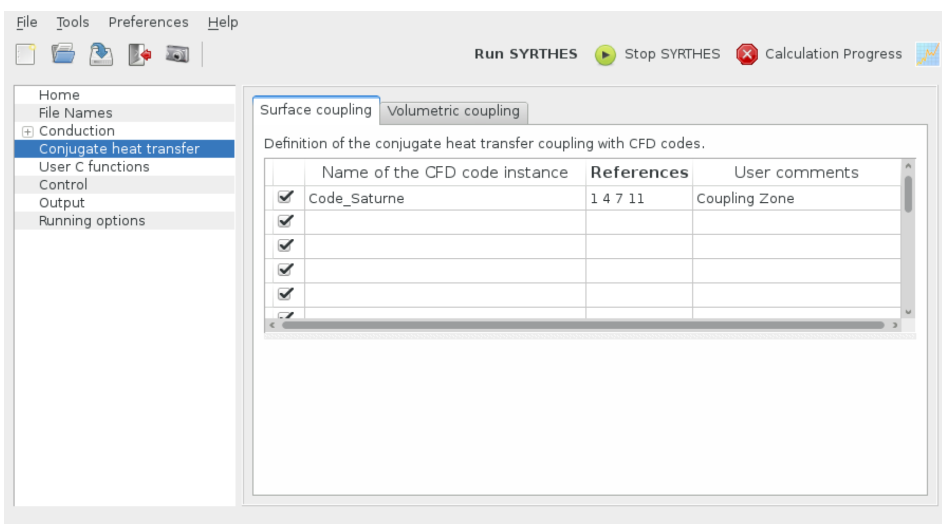


Figure III.31: Specify the reference zone for the coupling surfaces with *Code_Saturne*.

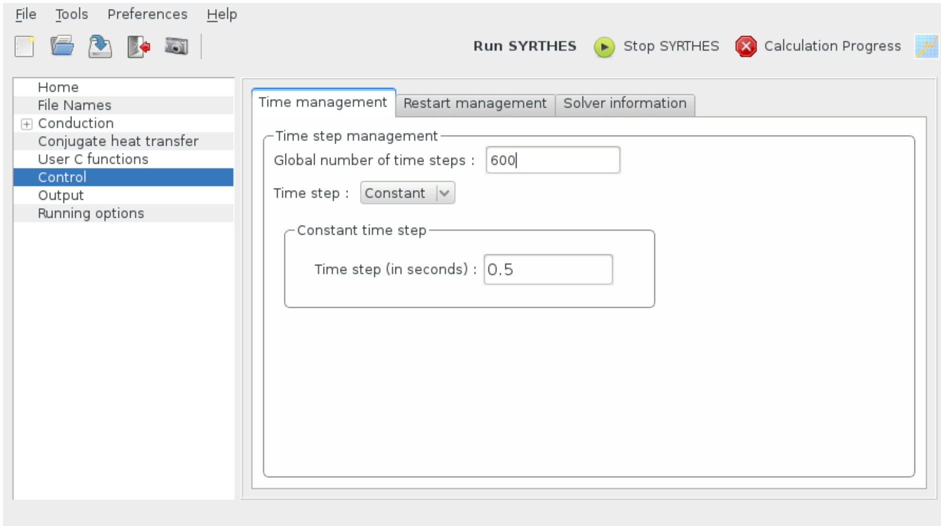


Figure III.32: Change the iterations number and time step for the solid domain.

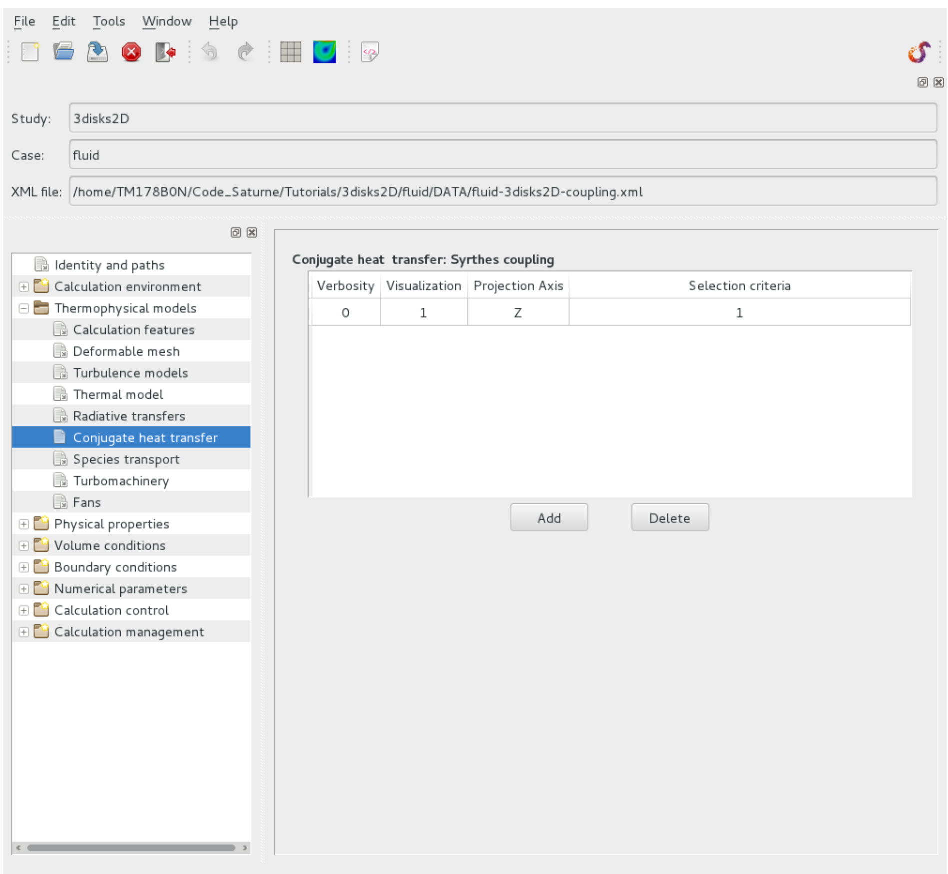


Figure III.33: Activate the conjugate heat transfer for the fluid domain.

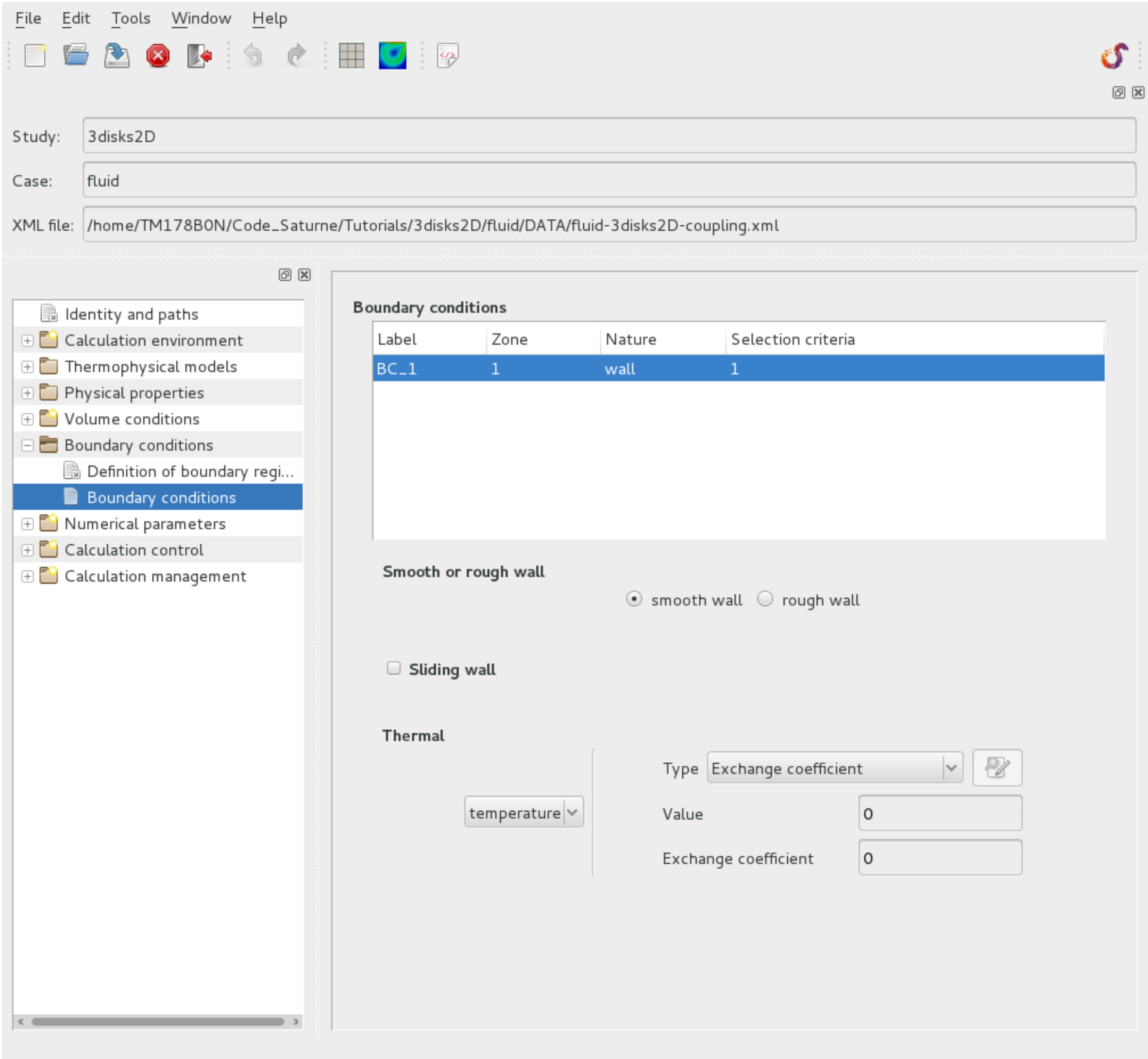


Figure III.34: Change the boundary conditions for the wall temperature.

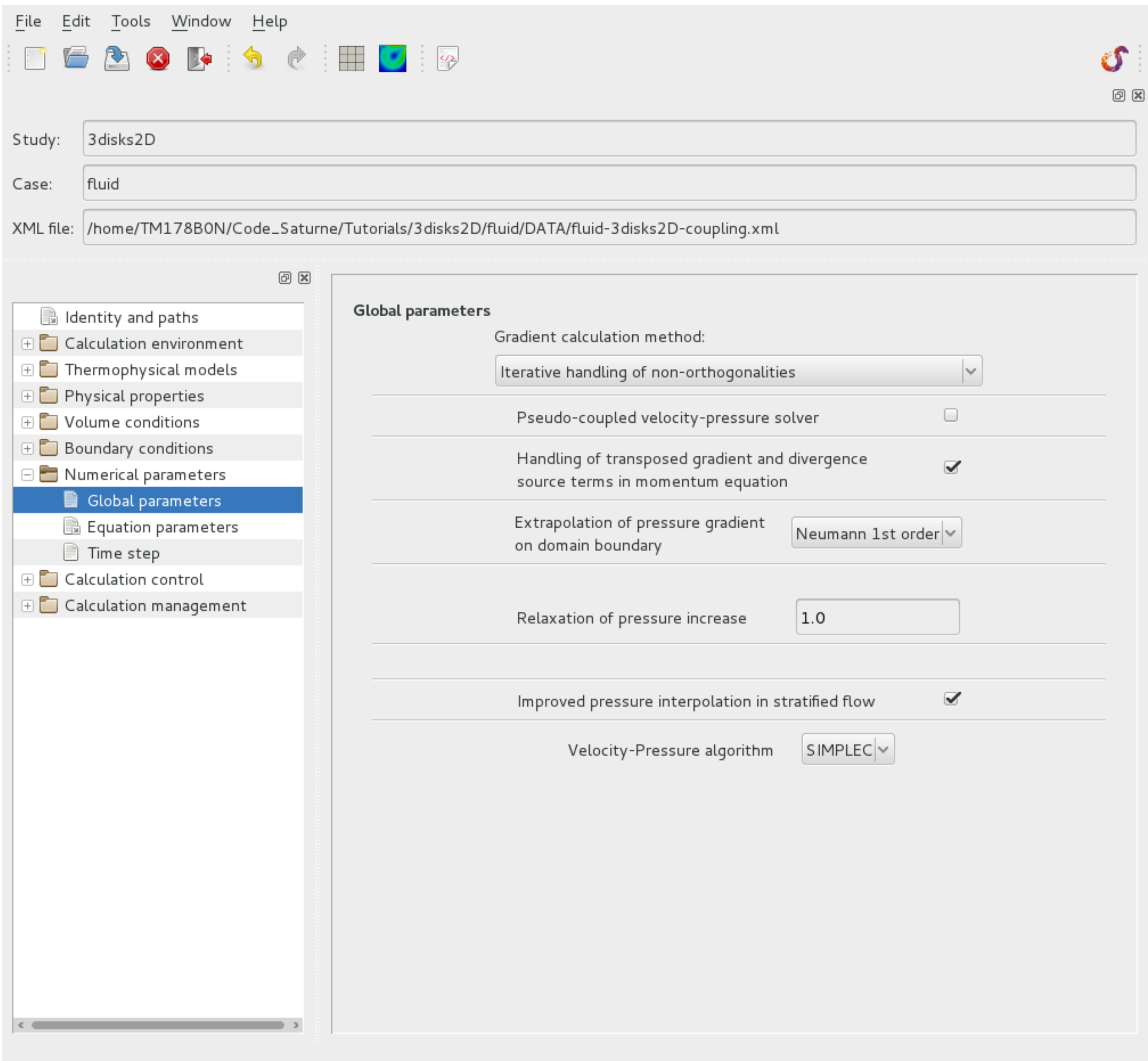


Figure III.35: Activate the **Improved pressure interpolation in stratified flow** algorithm.

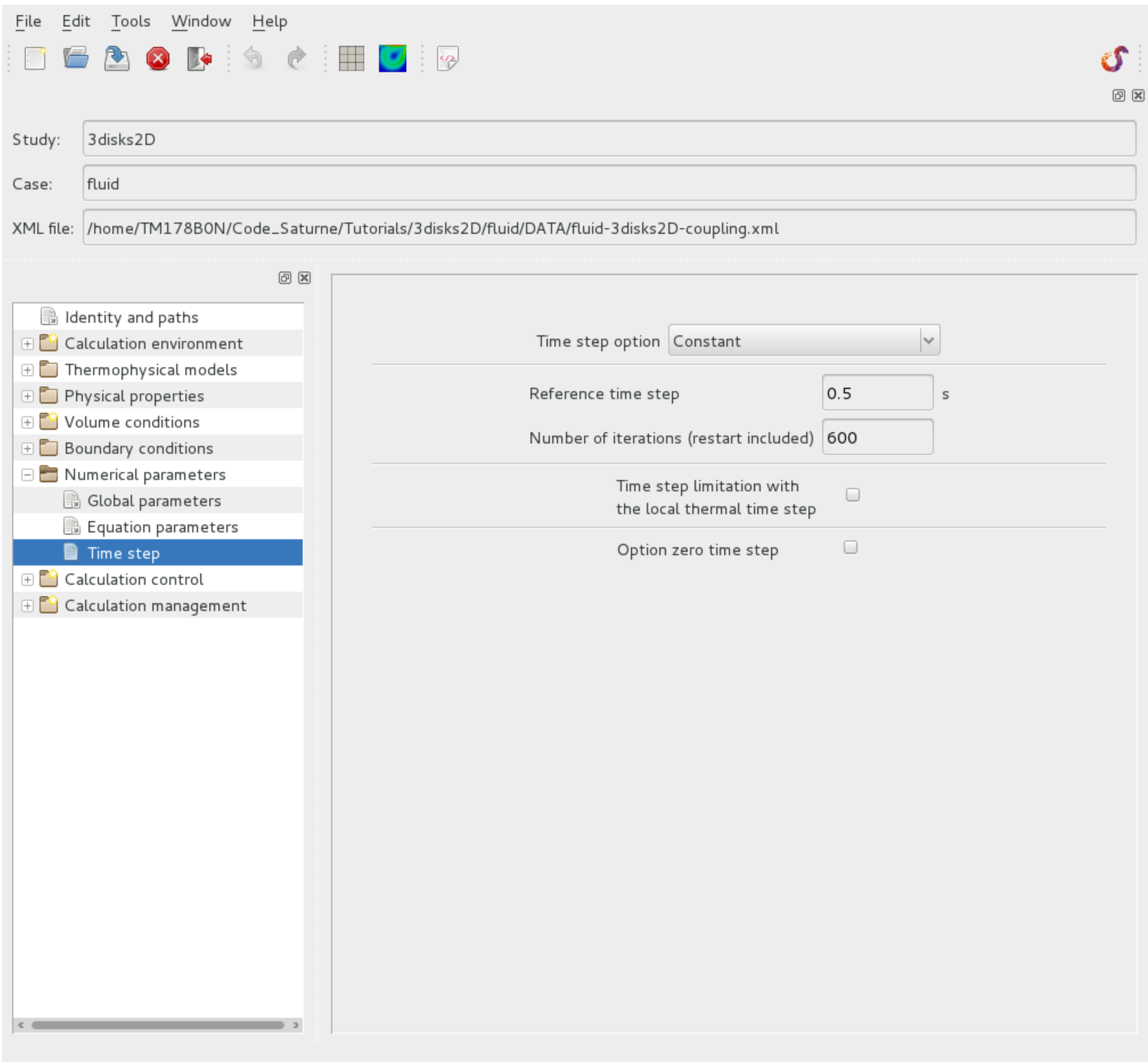


Figure III.36: Change the iterations number and time step for the fluid computation.

• **Remark**: After having modified the data setting for the fluid and solid domains to activate the conjugate heat transfer on both sides, one just has to increase the iterations number and check the `coupling_parameters.py` script.

One just needs to edit the `coupling_parameters.py` script and give the name of your SYRTHES script saved in the SYRTHES GUI as below:

```
$ vim coupling_parameters.py
> domains = [
>
> 'solver':  'Code_Saturne',
> 'domain':  'fluid',
> 'script':  'runcase',
> 'n_procs_weight':  None,
> 'n_procs_min':  4,
> 'n_procs_max':  4
>
> 'solver':  'SYRTHES',
> 'domain':  'solid',
> 'script':  'solid-coupling.syd',
> 'n_procs_weight':  None,
> 'n_procs_min':  2,
> 'n_procs_max':  2,
> 'opt' :  '-v ens'
>
> ]
```

Finally, one just has to launch the `runcase` present in the study directory (named in our case 🗀 `3disks2D`) and run the coupling computation, as follows:

```
$ runcase
```

• **Remarks**: in the `coupling_parameters.py`, the number of processors can be specified for each code (as this example with 4 processors for *Code_Saturne* and 2 processors for SYRTHES). It can be either both codes in parallel, one in parallel and the other one in sequential, or both in sequential.

One can specify the ouput results format for SYRTHES with an option (`opt`) which takes the value `-v ens` for a 3D fields output with a EnSight format or `-v med` for a 3D fields output with a SALOME format).