

*Code\_Saturne* documentation

***Code\_Saturne* version 2.1.3 practical user's guide**

contact: [saturne-support@edf.fr](mailto:saturne-support@edf.fr)



EDF R&D	<b><i>Code_Saturne</i> version 2.1.3 practical user's guide</b>	<i>Code_Saturne</i> documentation Page 2/ <a href="#">205</a>
---------	---	---

## ABSTRACT

*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 dilatible, 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 combustion, pulverised coal combustion, electricity effects (Joule effect and electric arcs) and compressible flows. *Code\_Saturne* relies on a finite volume discretisation 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 practical user's guide for *Code\_Saturne* version 2.1.3. It is the result of the joint effort of all the members in the development team. It presents all the necessary elements to run a calculation with *Code\_Saturne* version 2.1.3. It then lists all the variables of the code which may be useful for more advanced utilisation. The user subroutines of all the modules within the code are then documented. Eventually, for each key word and user-modifiable parameter in the code, their definition, allowed values, default values and conditions for use are given. These key words and parameters are grouped under headings based on their function. An alphabetical index list is also given at the end of the document for easier consultation.

*Code\_Saturne* is free software; you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version. *Code\_Saturne* is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details.

## TABLE OF CONTENTS

<b>1</b>	<b>Introduction</b>	9
<b>2</b>	<b>Quick start</b>	10
2.1	RUNNING A CALCULATION	10
2.2	TROUBLESHOOTING	11
<b>3</b>	<b>Practical information about Code_Saturne</b>	11
3.1	SYSTEM ENVIRONMENT FOR Code_Saturne	11
3.1.1	Preliminary settings	11
3.1.2	Standard directory hierarchy	12
3.1.3	Code_Saturne Kernel library files	13
3.2	SETTING UP AND RUNNING A CALCULATION	14
3.2.1	Step by step calculation	14
3.2.2	Temporary execution directory	16
3.2.3	Execution modes	16
3.2.4	Interactive modification of the target time step	17
3.3	CASE PREPARER	17
3.4	SUPPORTED MESH AND POST-PROCESSING OUTPUT FORMATS	18
3.4.1	Formats supported for input	18
3.4.2	Formats supported for input or output	22
3.4.3	Meshing tools and associated formats	24
3.4.4	Meshing remarks	24
3.5	PREPROCESSOR COMMAND LINE OPTIONS	25
3.6	KERNEL COMMAND LINE OPTIONS	25
3.7	LAUNCH SCRIPTS	26
3.8	GRAPHICAL USER INTERFACE	27
3.9	USER SUBROUTINES	28
3.9.1	Preliminary comments	28
3.9.2	Main variables	29
3.9.3	Using selection criteria in user subroutines	47
3.10	FACE AND CELL MESH-DEFINED PROPERTIES AND SELECTION	48
<b>4</b>	<b>Importing and Preprocessing Meshes</b>	51
4.1	PREPROCESSOR OPTIONS	51
4.1.1	Mesh selection	51
4.1.2	Post-processing output	52
4.1.3	Element orientation correction	52
4.2	ENVIRONMENT VARIABLES	52

4.2.1	<i>System environment variables</i>	52
4.3	OPTIONAL FUNCTIONALITY	53
4.4	GENERAL REMARKS	53
4.5	FILES PASSED TO THE KERNEL	53
4.6	MESH PREPROCESSING	53
4.6.1	<i>Joining of non-conforming meshes</i>	53
4.6.2	<i>Periodicity</i>	55
4.6.3	<i>Parameters for conforming or non-conforming mesh joinings</i>	55
4.6.4	<i>Parameters for the periodicity</i>	57
4.6.5	<i>Modification of the mesh geometry</i>	59
5	<b>Partitioning for parallel runs</b>	59
5.1	OPTIONS	60
5.1.1	<i>Ignore periodicity</i>	60
5.1.2	<i>Partitioner choice</i>	60
5.1.3	<i>Simulation mode</i>	60
5.1.4	<i>Environment variables</i>	60
6	<b>Basic modelling setup</b>	60
6.1	INITIALISATION OF THE MAIN PARAMETERS	60
6.2	SELECTION OF MESH INPUTS: <code>CS_USER_MESH_INPUT</code>	66
6.3	NON-DEFAULT VARIABLES INITIALISATION	67
6.4	MANAGE BOUNDARY CONDITIONS	68
6.4.1	<i>Coding of standard boundary conditions</i>	70
6.4.2	<i>Coding of non-standard boundary conditions</i>	72
6.4.3	<i>Checking of the boundary conditions</i>	73
6.4.4	<i>Sorting of the boundary faces</i>	74
6.4.5	<i>Boundary conditions with LES</i>	74
6.5	MANAGE THE VARIABLE PHYSICAL PROPERTIES	76
6.5.1	<i>Basic variable physical properties</i>	76
6.5.2	<i>Modification of the turbulent viscosity</i>	78
6.5.3	<i>Modification of the variable <math>C</math> of the dynamic LES model</i>	78
6.6	USER SOURCE TERMS	79
6.6.1	<i>In Navier-Stokes</i>	80
6.6.2	<i>For <math>k</math> and <math>\varepsilon</math></i>	81
6.6.3	<i>For <math>R_{ij}</math> and <math>\varepsilon</math></i>	81
6.6.4	<i>For <math>\varphi</math> and <math>\bar{f}</math></i>	81
6.6.5	<i>For <math>k</math> and <math>\omega</math></i>	81
6.6.6	<i>For user scalars</i>	82

6.7	PRESSURE DROPS (HEAD LOSSES)	82
6.8	MANAGEMENT OF THE MASS SOURCES	82
<b>7</b>	<b>Results analysis</b>	<b>85</b>
7.1	MANAGEMENT OF THE POST-PROCESSING INTERMEDIATE OUTPUTS	85
7.2	DEFINITION OF POST-PROCESSING AND MESH ZONES	86
7.3	MODIFICATION OF THE MESH ZONES TO POST-PROCESS	88
7.4	DEFINITION OF THE VARIABLES TO POST-PROCESS	88
7.5	MODIFICATION OF THE VARIABLES AT THE END OF A TIME STEP	89
7.6	NON-STANDARD MANAGEMENT OF THE CHRONOLOGICAL RECORD FILES	90
<b>8</b>	<b>Advanced modelling setup</b>	<b>90</b>
8.1	USE OF A SPECIFIC PHYSICS	90
8.2	PULVERISED COAL AND GAS COMBUSTION MODULE	97
8.2.1	Boundary conditions	99
8.2.2	Initialisation of the options of the variables	102
8.3	HEAVY FUEL OIL COMBUSTION MODULE	105
8.3.1	Initialisation of transported variables	105
8.3.2	Boundary conditions	105
8.3.3	Initialisation of the options of the variables	105
8.4	RADIATIVE THERMAL TRANSFERS IN SEMI-TRANSPARENT GRAY MEDIA	106
8.4.1	Initialisation of the radiation main parameters	106
8.4.2	Radiative transfers boundary conditions	107
8.4.3	Absorption coefficient of the medium, boundary conditions for the luminance and calculation of the net radiative flux	110
8.4.4	Encapsulation of the temperature-enthalpy conversion	110
8.4.5	Input of radiative transfer parameters	111
8.5	CONJUGATE HEAT TRANSFERS	111
8.5.1	Thermal module in a 1D wall	111
8.5.2	Fluid-Thermal coupling with SYRTHES	112
8.6	LAGRANGIAN MODELING OF MULTIPHASE FLOWS WITH DIPERSED INCLUSIONS	113
8.6.1	Initialisation of the Lagrangian modeling parameters	113
8.6.2	Management of the boundary conditions related to the particles	115
8.6.3	Treatment of the particle/boundary interaction	121
8.6.4	Option for particle cloning/merging	122
8.6.5	Manipulation of particulate variables at the end of an iteration and user volumetric statistics	122
8.6.6	User stochastic differential equations	123
8.6.7	Particle relaxation time	123

8.6.8	<i>Particle thermal characteristic time</i>	124
8.7	COMPRESSIBLE MODULE	124
8.7.1	<i>Initialisation of the options of the variables</i>	124
8.7.2	<i>Management of the boundary conditions</i>	124
8.7.3	<i>Initialisation of the variables</i>	125
8.7.4	<i>Thermodynamics</i>	125
8.7.5	<i>Management of variable physical properties</i>	125
8.8	MANAGEMENT OF THE ELECTRIC ARC MODULE	125
8.8.1	<i>Initialisation of the variables</i>	125
8.8.2	<i>Variable physical properties</i>	126
8.8.3	<i>Boundary Conditions</i>	126
8.8.4	<i>Initialisation of the variable options</i>	127
8.8.5	<i>EnSight output</i>	128
8.9	Code_Saturne-Code_Saturne COUPLING	129
8.10	FLUID-STRUCTURE EXTERNAL COUPLING	129
8.11	ALE MODULE	130
8.11.1	<i>Initialisation of the options</i>	130
8.11.2	<i>Boundary conditions of velocity mesh</i>	131
8.11.3	<i>Modification of the viscosity</i>	132
8.11.4	<i>Fluid - Structure internal coupling</i>	132
8.12	MANAGEMENT OF THE STRUCTURE PROPERTY	133
8.13	MANAGEMENT OF THE ATMOSPHERIC MODULE	133
8.13.1	<i>Initialisation of the variables</i>	133
8.13.2	<i>Non standard options</i>	134
8.13.3	<i>Management of the boundary conditions</i>	134
8.14	COOLING TOWER MODELLING	134
8.14.1	<i>Parameters</i>	134
8.14.2	<i>Initialisation of the variables</i>	134
8.14.3	<i>Definition of the exchange zones</i>	135
8.14.4	<i>Management of the boundary conditions</i>	135
9	<b>Key word list</b>	138
9.1	INPUT-OUTPUT	138
9.1.1	<i>"Calculation" files</i>	139
9.1.2	<i>Post-processing for EnSight or other tools</i>	141
9.1.3	<i>Chronological records of the variables on specific points</i>	143
9.1.4	<i>Time averages</i>	145
9.1.5	<i>Others</i>	146

9.2	NUMERICAL OPTIONS . . . . .	147
9.2.1	Calculation management . . . . .	147
9.2.2	Scalar unknowns . . . . .	149
9.2.3	Definition of the equations . . . . .	151
9.2.4	Definition of the time advancement . . . . .	152
9.2.5	Turbulence . . . . .	153
9.2.6	Time scheme . . . . .	158
9.2.7	Gradient reconstruction . . . . .	162
9.2.8	Solution of the linear systems . . . . .	164
9.2.9	Convective scheme . . . . .	165
9.2.10	Pressure-continuity step . . . . .	166
9.2.11	Error estimators for Navier-Stokes . . . . .	167
9.2.12	Calculation of the distance to the wall . . . . .	168
9.2.13	Others . . . . .	171
9.3	NUMERICAL, PHYSICAL AND MODELING PARAMETERS . . . . .	172
9.3.1	Numeric Parameters . . . . .	172
9.3.2	Physical parameters . . . . .	172
9.3.3	Physical variables . . . . .	173
9.3.4	Modeling parameters . . . . .	177
9.4	ALE . . . . .	181
9.5	THERMAL RADIATIVE TRANSFERS: GLOBAL SETTINGS . . . . .	181
9.6	ELECTRIC MODULE (JOULE EFFECT AND ELECTRIC ARC): SPECIFICITIES . . . . .	183
9.7	COMPRESSIBLE MODULE: SPECIFICITIES . . . . .	185
9.8	LAGRANGIAN MULTIPHASE FLOWS . . . . .	186
9.8.1	Global settings . . . . .	186
9.8.2	Specific physics models associated with the particles . . . . .	187
9.8.3	Options for two-way coupling . . . . .	189
9.8.4	Numerical modeling . . . . .	189
9.8.5	Volume statistics . . . . .	190
9.8.6	Display of trajectories and particle movements . . . . .	191
9.8.7	Display of the particle/boundary interactions and the statistics at the boundaries . . . . .	193
10	Bibliography . . . . .	196
	Index of the main variables and keywords . . . . .	198





# 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 dilatant, 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 combustion, pulverised coal combustion, electricity effects (Joule effect and electric arcs) and compressible flows.

*Code\_Saturne* is free software; you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version. *Code\_Saturne* is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details.<sup>1</sup>

*Code\_Saturne* relies on a finite volume discretisation 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).

*Code\_Saturne* is composed of three main elements and an optional GUI, as shown on figure 1:

- the Kernel module is the numerical solver
- the Preprocessor module is in charge of mesh import
- the Partitioner is in charge of optimizing domain decomposition for parallel computing (optional, but highly recommended for parallel performance)

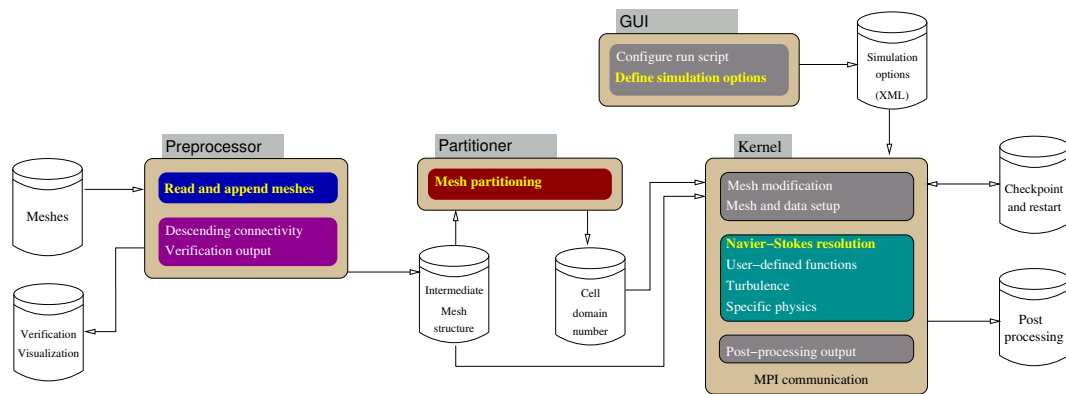


Figure 1: *Code\_Saturne* elements

*Code\_Saturne* also relies on one library (by the same team, under LGPL licence), which can also be used independently:

- PLE (Parallel Location and Exchange) for the management of code coupling

The present document is a practical user's guide for *Code\_Saturne* version 2.1.3. It is the result of the joint effort of all the members in the development team.

The aim of this document is to give practical information to the users of *Code\_Saturne*. It is therefore strictly oriented towards the usage of the code. For more details about the algorithms and their

<sup>1</sup>You should have received a copy of the GNU General Public License along with *Code\_Saturne*; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA

numerical implementation, please refer to the reports [10] and [4], and to the theoretical documentation [11], which is newer and more detailed (the latest updated version of this document is available on-line with the version of *Code\_Saturne* and accessible through the command `cs_info --guide theory`).

The present document first presents all the necessary elements to run a calculation with *Code\_Saturne* version 2.1.3. It then lists all the variables of the code which may be useful for more advanced utilisation. The user subroutines of all the modules within the code are then documented. Eventually, for each key word and user-modifiable parameter in the code, their definition, allowed values, default values and conditions for use are given. These key words and parameters are grouped under headings based on their function. An alphabetical index list is also given at the end of the document for easier consultation.

## 2 Quick start

### 2.1 Running a calculation

We assume in this section that the user has at his disposal the calculation data file (calculation set up) or already prepared it following for instance the step-by-step guidance provided in *Code\_Saturne* tutorial. The steps described below are intended to provide the user a way to run quickly on a workstation a calculation through the Graphical User Interface (GUI).

The first thing to do before running *Code\_Saturne* is to add in the user `~/.profile`, `~/.bashrc` or similar file the path leading to the chosen *Code\_Saturne* version, or define an alias to the `code_saturne` script. For example:

```
export PATH=${prefix}/bin:$PATH.
```

The second thing is to prepare the computation directories. For instance, the study directory `T_JUNCTION`, containing a single calculation directory `CASE1`, will be created by typing the command:

```
code_saturne create -s T_JUNCTION
```

The mesh files should be copied in the directory `MESH`, and the Fortran user files necessary for the calculation in the directory `CASE1/SRC`. Finally, the calculation data file `case_name.xml` read by the GUI should be copied to the directory `CASE1/DATA`. Once these steps completed, the user should go in the directory `CASE1/DATA` and type de command line `./SaturneGUI case_name.xml` to load the calculation file into the interface. A window similar to fig.2 will appear. Click on the heading “Calculation management”, select the heading “Prepare batch calculation”, see fig.3. After having chosen the number of processors, press “start calculation” to run the calculation.

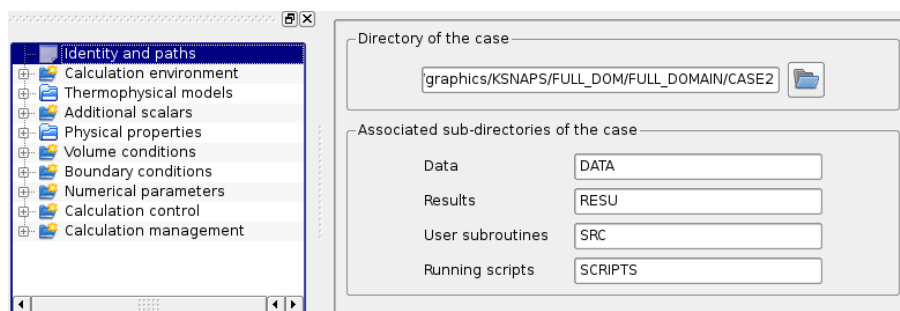


Figure 2: Identity and paths

If no problem arises, the simulation results can be found in the directory `CASE1/RESU` and be read directly by *ParaView* or *EnSight* in `CASE1/RESU/<YYYYMMDD-hhmm>/postprocessing`. Calculation history can be found in the file `<YYYYMMDD-hhmm>/listing`.

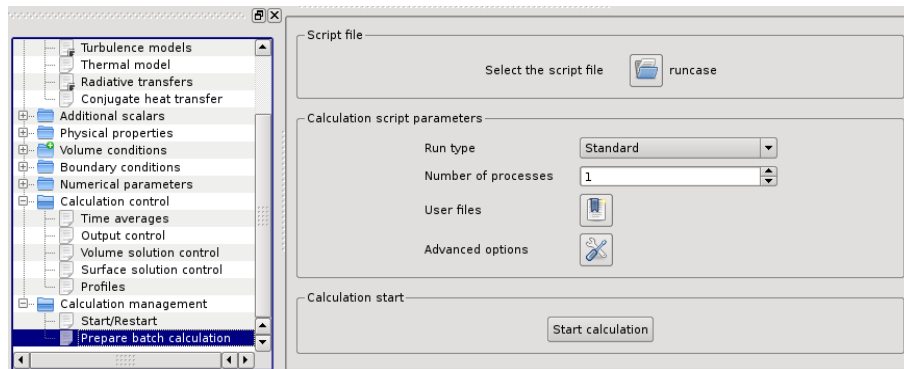


Figure 3: Prepare execution

## 2.2 Troubleshooting

If the calculation does not run properly, the user is advised to check the following points in `CASE1/RESU/<YYYYMMDD-hhmm>`:

- if the calculation stops in the pre-processor, the user should check for error messages in the file `preprocessor*.log`.
- if the problem is related to boundary conditions, the user should visualise the file `error.ensight` with *EnSight* or *Paraview*,
- if the calculation stops in the *Code\_Saturne* core, the user should look for messages at the end of the files `listing` and `error*`. In addition, the user can track the following keywords in the listing. They are specific error signals:
  - SIGFPE: a floating point exception occurred. It happens when there is a division by 0, when the calculation did not converge, or when a real number reached a value over  $10^{300}$ . Depending on the architecture *Code\_Saturne* is running on, this type of exception may be caught or ignored.
  - SIGSEGV: a memory error such as a segmentation violation occurred. An array may have exceeded its allocated memory size and a memory location in use was overwritten.

## 3 Practical information about *Code\_Saturne*

### 3.1 System Environment for *Code\_Saturne*

#### 3.1.1 Preliminary settings

In order to use *Code\_Saturne*, every user must add the following line (in their `.profile`, `.bashrc`, or equivalent, depending on the environment):

```
export PATH=${prefix}/bin:$PATH
```

or define the following alias (in their `.bashrc`, or equivalent, or `.alias` file, depending on the environment):

```
alias cs='${prefix}/bin/cs'
```

where `prefix` is the base directory where *Code\_Saturne* and its components have been installed<sup>2</sup>.

### 3.1.2 Standard directory hierarchy

The standard architecture for the simulation studies is:

An optional study directory containing:

- A directory **MESH** containing the mesh(es) necessary for the study
- A directory **POST** for the potential post-processing routines (not used directly by the code)
- One or several calculation directories

Every calculation directory contains:

- A directory **SRC** for the potential user subroutines necessary for the calculation
- A directory **DATA** for the calculation data (data file from the interface, input profiles, thermo-chemical data, ...)
- A directory **SCRIPTS** for the launch script
- A directory **RESU** for the results  
To improve the calculation traceability, the files and directories sent to **RESU** after a calculation are placed in a subdirectory named after that run's "id", which is by default based on the run date and time, using the format: `YYYYMMDD-hhmm`. It is also possible to force a specific run id, using the `--id` option to `code.saturne run`.

In the standard cases, **RESU/<run\_id>** contains a **postprocessing** directory with the post-processing (visualization) files, a **restart** directory for the calculation restart files, a **monitoring** directory for the files of chronological record of the results at specific locations (probes), **preprocessor.log** and **listing** files reporting the Preprocessor and the Kernel execution. For an tracing of the modifications in prior calculations, the user-subroutines used in a calculation are stored in a **src.saturne** subdirectory. The *Xml* Interface data file, thermo-chemical data files and launch script are also copied into the results directory. **compil.log** and **summary** are respectively reports of the compilation stage and general information on the calculation (type of machine, user, version of the code, ...).

Note that the code may be run directly in the final **RESU/<run\_id>** directory, or in a scratch directory (which may be recommended if the compute environment includes different filesystems, some better suited to data storage, others to intensive I/O). When running, the code may use additional files or directories inside its execution directory, set by the execution script, which include a **mesh\_input** file or directory, as well as a **restart** directory (which is a link or copy of a previous run's **checkpoint** directory), as well as a **run\_solver.sh** script.

For coupled calculations, whether with *Code\_Saturne* or SYRTHES, each coupled calculation domain is defined by its own directory (bearing the same name as the domain), but results are placed in a **RESU\_COUPLING** directory, with a subdirectory for each run, itself containing one subdirectory per coupled domain. Coupled cases are not run through the standard **STUDY/CASE1/SCRIPTS/runcase** script or through the `code.saturne run` command, but through a **STUDY/runcase\_coupling** script.

So in the coupled case, calculation results would not be placed in **STUDY/CASE1/RESU/20110509-1920**, but in **STUDY/RESU\_COUPLING/20110509-1920/CASE1**, with the **summary** file being directly placed in **STUDY/RESU\_COUPLING/20110509-1920** (as it references all coupled domains).

<sup>2</sup>At EDF R&D, `/home/saturne/Code_Saturne/2.1.3` is used

Below are typical contents of a case directory CASE1 in a study STUDY

STUDY/CASE1/DATA:	<b>Code_Saturne data</b>
SaturneGUI	Graphical User Interface launch script
study.xml	Graphical User Interface parameter file
REFERENCE	Example of user scripts and meteorological or thermochemical data files (used with the specific physics modules)
STUDY/CASE1/SRC:	<b>Code_Saturne user subroutines</b>
REFERENCE	Examples of a user subroutines
usclim.f90	user subroutines used for the present calculation
usini1.f90	
STUDY/CASE1/RESU/20110509-1920:	<b>results</b> for the calculation 20110509-1920
postprocessing	directory containing the <i>Code_Saturne</i> post-processing output in the <i>EnSight</i> format (both volume and boundary);
src_saturne	copy of the <i>Code_Saturne</i> user subroutines used for the calculation
monitoring	directory containing the chronological records for <i>Code_Saturne</i>
checkpoint	directory containing the <i>Code_Saturne</i> restart files
compile.log	compilation log
study.xml	Graphical User Interface parameter file used for the calculation
runcase	copy of the launch script used for the calculation
preprocessor.log	execution report for the <i>Code_Saturne</i> Preprocessor
listing	execution report for the Kernel module of <i>Code_Saturne</i>
summary	general information (machine, user, version, ...)
STUDY/CASE1/SCRIPTS:	<b>launch script</b>
runcase	launch script (which may contain batch system keywords)

Below are typical additional contents with a coupled SYRTHES case SOLID1 in a study STUDY

STUDY/runcase_coupling	coupled launch script
STUDY/SOLID1/DATA:	<b>SYRTHES data</b>
syrthes.data	SYRTHES data file
syrthes.env	SYRTHES configuration file
STUDY/RESU_COUPLING/20110509-1920/SOLID1:	<b>results (file names defined in syrthes.env)</b>
src	SYRTHES user subroutines used in the calculation
compile.log.08211921	SYRTHES compilation report
listsyr	execution log
geoms	SYRTHES solid geometry file
histos1	SYRTHES chronological records at specified probes
resus1	SYRTHES calculation restart file (1 time step)
resusc1	SYRTHES chronological solid post-processing file (may be transformed into the <i>EnSight</i> format with the <i>syrthes2ensight</i> utility)

### 3.1.3 Code\_Saturne Kernel library files

Information about the content of the *Code\_Saturne* base directories is given below. It is not of vital interest for the user, but given only as general information. Indeed, the case preparer command `code_saturne create` automatically extracts the necessary files and prepares the launch script without the user having to go directly into the *Code\_Saturne* base directories (see §3.3). The `code_saturne info` command gives direct access to the most needed information (especially the user and programmer's guides and the tutorial) without the user having to look for them in the *Code\_Saturne* directories.

The subdirectories `{prefix}/lib` and `{prefix}/bin` contain the libraries and compiled executables respectively.

The data files (for instance thermochemical data) are located in the directory **data**.

The user subroutines are available in the directory **users**, under subdirectories corresponding to each module: **base** (general routines), **cfbl** (compressible flows), **cogz** (gas combustion), **cp1v** (pulverised coal combustion), **ctwr** (cooling towers modelling), **elec** (electric module), **fuel** (heavy fuel oil combustion module), **lagr** (Lagrangian module), **pprt** (general specific physics routines) and **rayt** (radiative heat transfer). The case preparer command **code\_saturne create** copies all these files in the user directory **SRC/REFERENCE** during the case preparation.

The directory **bin** contains an example of the launch script, the compilation parameter files and various utility programs.

## 3.2 Setting up and running a calculation

### 3.2.1 Step by step calculation

This paragraph summarises the different steps which are necessary to prepare and run a standard case:

- Check the version of *Code\_Saturne* set for use in the environment variables (**code\_saturne info --version**). If it does not correspond to the desired version, update the **.profile** file to set the environment variables correctly. Log out of the session and log in again to take the modifications into account properly (cf. §3.1.1).
- Prepare the different directories using the **code\_saturne create** command (see §3.3).
- It is recommended to place the mesh(es) in the directory **MESH**, but they may be selected from other directories. Make sure they are in a format compliant with *Code\_Saturne* (see §3.4.4). There can be several meshes in case of mesh joining or coupling with SYRTHES<sup>3</sup>.
- Go to the directory **DATA** and launch the Graphical User Interface using the command **./SaturneGUI**.
- If not using the GUI, copy the **DATA/REFERENCE/cs\_user\_scripts.py** file to **DATA** and edit it, so that the correct run options and paths may be set. For advanced uses, this file may also be used in conjunction with the GUI. Just as with user Fortran subroutines below, settings defined in this file have priority over those defined in the GUI.
- Place the necessary user subroutines in the directory **SRC** (see §3.9). When not using the Interface, some subroutines are compulsory.

#### For the standard physics:

*compulsory without Graphical User Interface:*

- **usini1** to specify the calculation parameters
- **usclim** to manage the boundary conditions

*very useful:*

- **usphyv** to manage the variable physical properties (fluid density, viscosity ...)
- **usiniv** to manage the non-standard initialisations

#### For the “gas combustion” specific physics:

(not accessible through the Graphical User Interface in version 2.1.3)

*compulsory:*

- **usini1** to specify the calculation parameters
- **usppmo** to select a specific physics module and combustion model

---

<sup>3</sup>SYRTHES 3 uses meshes composed of 10-node tetrahedra (vertices and centers of edges, SYRTHES 4 uses meshes composed of 4-node tetrahedra)

EDF R&D	<b>Code_Saturne version 2.1.3 practical user's guide</b>	Code_Saturne documentation Page 15/205
---------	--	--

- `usebuc`, `usd3pc` or `uslwcc` (depending on the selected combustion model) to manage the boundary conditions of *all variables* (*i.e.* not only the ones related to the combustion model)

*very useful:*

- `usebu1`, `usd3p1` or `uslwc1` (depending on the selected combustion model) to specify the calculation options for the variables corresponding to combustion model
- `usebui`, `usd3pi` or `uslwci` (depending on the selected combustion model) to manage the initialisation of the variables corresponding to the combustion model

#### **For the “coal combustion” specific physics:**

*compulsory without Graphical User Interface:*

- `usini1` to specify the calculation parameters
- `usppmo` to select the specific physics module
- `uscpc1` or `uscplc` (depending on the specific physics module) to manage the boundary conditions of *all variables* (*i.e.* not only the ones related to the specific physics module)

*very useful:*

- `uscpi1` to specify the calculation options for the variables corresponding to the specific physics module
- `uscpi1` to manage the initialisation of the variables corresponding to the specific physics module

#### **For the “electric module” specific physics (Joule effect and electric arcs):**

(not accessible through the Graphical User Interface in version 2.1.3)

*compulsory:*

- `usini1` to specify the calculation parameters
- `usppmo` to select the specific physics module
- `uselc1` to manage the boundary conditions of *all variables* (*i.e.* not only the ones related to the electric module)
- `useliv` to initialise the enthalpy in case of Joule effect
- `uselph` to define the physical properties in case of Joule effect

*very useful:*

- `useli1` to manage the options related to the variables corresponding to the electric module
- `useliv` to manage the initialisation of the variables corresponding to the electric module

#### **For the “heavy fuel oil combustion module” specific physics:**

(not accessible through the Graphical User Interface in version 2.1.3)

*compulsory:*

- `usini1` to specify the calculation parameters
- `usppmo` to select the specific physics module
- `usfuc1` to manage the boundary conditions of *all variables* (*i.e.* not only the ones related to the specific physics module)

*very useful:*

- `usfui1` to specify the calculation options for the variables corresponding to the specific physics module
- `usfuiv` to manage the initialisation of the variables corresponding to the specific physics module

#### **For the Lagrangian module (dispersed phase):**

(the continuous phase is managed in the same way as for a case of standard physics)

(the Lagrangian module is not accessible through the Graphical User Interface in version 2.1.3)



*compulsory:*

- `uslag1` to manage the calculation conditions
- `uslag2` to manage the boundary conditions for the dispersed phase

*very useful:*

- `uslabo` to manage potential specific treatments at the boundaries (rebound conditions, specific statistics, ...)

#### **For the compressible module:**

(not accessible through the Graphical User Interface in version 2.1.3)

*compulsory:*

- `uscfx1` and `uscfx2` to manage the calculation parameters
- `uscfc1` to manage the boundary conditions
- `uscftH` to define the thermodynamics.

*very useful:*

- `uscfxi` to manage non-standard initialisations of the variables

The comprehensive list of the user subroutines and their instructions for use are given in §3.9.

- If necessary, place in the directory `DATA` the different external data (input profiles, thermochemical data files, ...)
- Prepare the launch script `runcase`, directly or through the Graphical Interface (see §3.7), or prepare the `DATA/cs.user.scripts.py` file.
- Run the calculation and analyse the results
- Purge the temporary files (in `RESU/<run_id>` or `<scratch>/<run_id>` directory).

### **3.2.2 Temporary execution directory**

During a calculation, *Code\_Saturne* may use a temporary directory for the compilation and the execution if such a “scratch” directory is defined. In that case, the result files are only copied at the end in the directory `RESU`. This is recommended if the compute environment includes different filesystems, some better suited to data storage, others to intensive I/O. If this is not the case, there is no point in running in a scratch directory rather than the results directory, as this incurs additional file copies.

*WARNING: in case of an error, the temporary directories are not deleted after a calculation, so that they may be used for debugging. They may then accumulate and may hinder the correct operation of the machine.*

**It is therefore essential to remove them regularly.**

### **3.2.3 Execution modes**

As explained before, *Code\_Saturne* is composed of two main modules, the Preprocessor and the Kernel, and an optional Partitioner. The Preprocessor reads the meshes. The Partitioner optimizes domain decomposition for parallel runs. The resulting data is transferred to the Kernel through specific files, named `mesh_input` and `domain_number_*`, where `*` is the number of processors used. In a standard calculation, the files are not copied from the temporary execution directory to the results directory, as they have no interest for data analysis, and are considered “internal” files, whose format or contents is not guaranteed not to change between *Code\_Saturne* versions.

Yet, the Preprocessor does not run in parallel and may require a large amount of memory. Similarly, the Partitioner may be run on a reduced number of processors to obtain a better partition quality, so it is sometimes useful to run the Preprocessor and Partitioner separately, on a machine or in batch queues with extended memory, and to run the proper parallel calculation on another machine or in



another batch queue. The launch scripts therefore allows specifically choosing which modules to run, either through the GUI or through `cs_user_scripts.py`:

If a `mesh_input` file or directory is defined (which may be either a `mesh_input` from a previous Preprocessor run or a `mesh_output` from a previous solver run), the script will copy or link it to the execution directory, and the Preprocessor will not be run again.

If `domain.exec_partition = False`, the Partitioner will not be run. A previous partitioning may be used by defining the `domain.partition_input` path.

If `domain.exec_kernel = False`, the Kernel will not be run. This is useful when only the mesh import and optionally partitioning stages are required.

It is encouraged to separate the mesh import and calculation runs, as this speeds up calculations by not re-importing meshes for each run. For some configurations, such as IBM Blue Gene machines with different front-end and compute nodes, mesh import may be impossible on the compute nodes (as the Preprocessor does not run in parallel, and may require too much memory), so mesh import (and serial partitioning) should be run separately on the front-end nodes, while later calculation stages should be run on compute nodes.

Note also that depending on its configuration, the Partitioner may be run either or both in serial or parallel mode. By default, serial mode is currently chosen (due to limited feedback on partitioning quality in parallel mode), but setting `domain.partition_n_procs` to a value greater than 1 in the user scripts allows running the Partitioner in parallel.

### 3.2.4 Interactive modification of the target time step

During a calculation, it is possible to change the limit time step number (`ntmabs`) specified through the Interface or in `usini1`. To do so, a file named `ficstp` must be placed in the temporary execution directory (see §3.2.2). This file must contain a blank first line and the second line indicating the value of the new limit number of time steps.

If this new limit has already been passed in the calculation, *Code\_Saturne* will stop properly at the end of the current time step (the results and restart files will be written correctly).

This procedure allows the user to stop a calculation in a clean and interactive way whenever they wish.

## 3.3 Case preparer

The case preparer command `code_saturne create` automatically creates a study directory according to the typical architecture and copies and pre-fills an example of calculation launch script.

The syntax of `code_saturne create` is as follows:

```
code_saturne create --study STUDY CASE_NAME1 CASE_NAME2...
```

creates a study directory `STUDY` with case subdirectories `CASE_NAME1` and `CASE_NAME2...`. If no case name is given, a default case directory called `CASE1` is created.

```
code_saturne create --case DEBIT3 --case DEBIT4
```

executed in the directory `STUDY` adds the case directories `DEBIT3` and `DEBIT4`.

An option `--nogui` is available for the use of *Code\_Saturne* without Graphic Interface. This option must be either the first or the last argument and appear only once.

In the directory `DATA`, the `code_saturne create` command places a subdirectory `THCH` containing examples of thermochemical data files used for pulverised coal combustion, gas combustion or electric arc. The file to be used for the calculation must be copied directly in the `DATA` directory and its name must be referenced in the launch script in the variable `THERMOCHEMISTRY_DATA`. All other files in the `DATA` or in the `THCH` will be ignored.

The `code_saturne create` command also places in the directory `DATA` the launch script for the Graphical User Interface: `SaturneGUI`.

In the directory `SRC`, the `code_saturne create` command creates a subdirectory `REFERENCE` containing all the user subroutines, classified by module type: `base`, `cfbl`, `cogz`, `cplv`, `elec`, `fuel`, `lagr`, `pprt` and `rayt`. Only the user subroutines placed directly under the directory `SRC` will be considered. The others will be ignored.

In the directory `SCRIPTS`, the `code_saturne create` command copies and pre-fills an example of the launch script: `runcase`. The study, case and user name are filled automatically in the launch script, as are the paths leading to the different directories. Other parameters must be specified in the script (see §3.7), especially the mesh file(s) to use, but everything can be specified through the Graphical Interface.

## 3.4 Supported mesh and post-processing output formats

*Code\_Saturne* supports multiple mesh formats, all of these having been requested at some time by users or projects based on their meshing or post-processing tools. All of these formats have advantages and disadvantages (in terms of simplicity, functionality, longevity, and popularity) when compared to each other. The following formats are currently supported by *Code\_Saturne*:

- SIMAIL (NOPO)
- I-deas universal
- MED
- CGNS
- EnSight 6
- EnSight Gold
- GAMBIT neutral
- Gmsh
- pro-STAR/STAR4
- STAR-CCM+

These formats are described in greater detail in the following sections. Unless a specific option is used, the Preprocessor determines the mesh format directly from the file suffix: “.case” for EnSight (6 or Gold), “.ccm” for STAR-CCM+, “.cgns” for CGNS, “.des” for SIMAIL, “.med” for MED, “.msh” for Gmsh, “.neu” for GAMBIT neutral, “.ngeom” for pro-STAR/STAR4, “.unv” for I-deas universal.

Note that the preprocessor can read gzipped mesh files directly (for Formats other than MED or CGNS, which use specific external libraries) on most machines.

### 3.4.1 Formats supported for input

#### 3.4.1.1 NOPO/SIMAIL (INRIA/SIMULOG)

This format output by SIMAIL is still heavily used at EDF. We do not currently handle cylindrical or spherical coordinates, but it seems that SIMAIL always outputs meshes in Cartesian coordinates, even if points have been defined in another system. Most “classical” element types are usable, except for pyramids.

Note that depending on the architecture on which a file was produced by SIMAIL,<sup>4</sup>, it may not be directly readable by SIMAIL on a different machine, while this is not a problem for the Preprocessor, which automatically detects the byte ordering and the 32/64 bit variant and adjusts accordingly.

<sup>4</sup> “little endian” on Intel or AMD processors, or “big endian” on most others, and starting with SIMAIL 7, 32-bit or 64-bit integer and floating-point numbers depending on architecture

Default extension:	<b>.des</b>
File type:	semi-portable “Fortran” binary (IEEE integer and floating-point numbers on 4 or 8 bytes, depending on 32 or 64 bit SIMAIL version, bytes also ordered based on the architecture)
Surface elements:	triangles, quadrangles (+ volume element face references)
Volume elements:	tetrahedra, prisms, hexahedra
Zone selection:	element face references and volume sub-domains (interpreted as numbered groups)
Compatibility:	all files of this type as long as the coordinate system used is Cartesian and not cylindrical or spherical
Documentation:	Simail user documentation and release notes or MODULEF documentation: <a href="http://www-rocq.inria.fr/modulef">http://www-rocq.inria.fr/modulef</a> Especially: <a href="http://www-rocq.inria.fr/modulef/Doc/FR/Guide2-14/node49.html">http://www-rocq.inria.fr/modulef/Doc/FR/Guide2-14/node49.html</a>

### 3.4.1.2 I-deas universal file

This format was very popular in the 1990’s and early 2000’s, and though the I-deas tool has not focused on the CFD (or even meshing) market since many years, it is handled (at least in part) by many tools, and may be considered as a major “legacy” format. It may contain many different datasets, relative to CAD, meshing, materials, calculation results, or part representation. Most of these datasets are ignored by *Code\_Saturne*, and only those relative to vertex, element, group, and coordinate system definitions are handled.

This format’s definition evolves with I-deas versions, albeit in a limited manner: some datasets are declared obsolete, and are replaced by others, but the definition of a given dataset type is never modified. Element and Vertex definitions have not changed for many years, but group definitions have gone through several dataset variants through the same period, usually adding minor additional group types not relevant to meshing. If one were to read a file generated with a more recent version of I-deas for which this definitions would have changed with no update in the Preprocessor, as the new dataset would be unknown, it would simply be ignored.

Note that this is a text format. Most element types are handled, except for pyramids.

Default extension:	<b>.unv</b>
File type:	text
Surface elements:	triangles, quadrangles
Volume elements:	tetrahedra, prisms, hexahedra
Zone selection:	colors (always) and named groups
Compatibility:	I-deas ( <i>Master Series</i> 5 to 9, <i>NX Series</i> 10 to 12) at least
Documentation:	Online I-deas NX Series documentation

### 3.4.1.3 GAMBIT neutral

This format may be produced by Ansys FLUENT’s GAMBIT meshing tool. As this tool does not export meshes to other formats directly handled by the Preprocessor (though FLUENT itself may export files to the CGNS or I-deas universal formats), it was deemed useful to enable the Preprocessor to directly read files in GAMBIT neutral format.

Note that this is a text format. “Classical” element types are usable.

Default extension:	.neu
File type:	text
Surface elements:	triangles, quadrangles
Volume elements:	tetrahedra, pyramids, prisms, hexahedra
Zone selection:	boundary conditions for faces, element groups for cells (interpreted as named groups)
Documentation:	GAMBIT on-line documentation

### 3.4.1.4 pro-STAR

This polyhedral format from CD-Adapco seems to be usable both with the **STAR-CD** and **STAR-CCM+** tools, and the **pro-STAR** tool should be able to generate it. The test meshes we have were generated by the **Comet-Design** tool, which has since been replaced by other CD-Adapco tools, especially **STAR-CD V4** and **STAR-CCM+**. The available test cases are thus not extensive in terms of functionality (especially when considering definition of descriptions), so support for this format is lightly tested.

Currently, geometric entity numbers are converted to group numbers. This tends to lead to a large number of groups.

Default extension:	.ngeom
File type:	binary file using portable XDR encoding.
Surface elements:	polygons
Volume elements:	polyhedra
Zone selection:	face and cell sets (interpreted as numbered groups)
Compatibility:	all files of this type ? (tested on purely polyhedral meshes)
Documentation:	documentation accompanying and source code provided by CD-adapco in the context of a collaboration with UMIST (now University of Manchester) and EDF R&D/MFEE

### 3.4.1.5 STAR-CCM+

This polyhedral format is the current CD-Adapco format, and is based on CD-Adapco's libccmio, which is based on ADF (the low-level file format used by CGNS prior to the shift to HDF-5). libccmio comes with a version of ADF modified for performance, but also works with a standard version from CGNS.

Currently, geometric entity numbers are converted to numbered groups, with the corresponding names printed to the Preprocessor log. Depending on whether the names were generated automatically or set by the user, it would be preferable to use the original group names rather than base their names on their numbers.

The CCMIO library is distributed freely by CD-Adapco upon demand.

Default extension:	.ccm
File type:	binary file using modified ADF library.
Surface elements:	polygons
Volume elements:	polyhedra
Zone selection:	named face and cell sets (interpreted as numbered groups, with names appearing in log)
Compatibility:	all files of this type ? (tested on purely polyhedral meshes)
Documentation:	documentation and source code provided by CD-adapco

### 3.4.1.6 EnSight 6

This format is used for output by the Harpoon meshing tool, developed by Sharc Ltd (also the distributor of EnSight for the United Kingdom). This format may represent all “classical” element types.

Designed for post processing, it does not explicitly handle the definition of surface patches or volume zones, but allows the use of many *parts* (i.e. groups of elements) which use a common vertex list. A possible convention (used at least by Harpoon) is to add surface elements to the volume mesh, using one *part* per group. The volume mesh may also be separated into several *parts* so as to identify different zones. As *part* names may contain up to 80 characters, we do not transform them into groups (whose names could be unwieldy), so we simply convert their numbers to group names.

Also note that files produced by Harpoon may contain badly oriented prisms, so the Preprocessor orientation correction option (`--reorient`) may need to be used. Meshes built by this tool also contain hanging nodes, with non-conforming elements sharing some vertices. Mesh joining must thus also be used, and is not activated automatically, as the user may prefer to specify which surfaces should be joined, and which ones should not (i.e. to conserve thin walls).

Default extension:	<code>.case</code>
File type:	text file (extension <code>.case</code> ), and text, binary, or Fortran binary file with ( <code>.geo</code> extension), describing the integers describing integers and floats in the IEEE format, using 32 bits
Surface elements:	triangles, quadrangles
Volume elements:	tetrahedra, pyramids, prisms, hexahedra
Zone selection:	part numbers interpreted as numbered groups
Compatibility:	All files of this type
Documentation:	online documentation, also available at: <a href="http://www.ensight.com/downloads/cat_view-5.html">http://www.ensight.com/downloads/cat_view-5.html</a>

### 3.4.1.7 Gmsh

This format is used by the free [Gmsh](#) tool. This tool has both meshing and post-processing functionality, but *Code\_Saturne* only imports meshes.

Note that some meshes produced by Gmsh may contain some badly oriented elements, so the Preprocessor's `-reorient` option may be necessary.

The Preprocessor handles versions 1 and 2 of this array. In version 1, two labels are associated with each element: the first defines the element's physical entity number, the second defines its elementary entity number. Using version 2, it is possible to associate an arbitrary number of labels with each element, but files produced by Gmsh use 2 labels, with the same meanings as with version 1.

We chose to convert physical entity numbers to groups. It is possible to build a mesh using Gmsh without defining any physical entities (in which case all elements will belong to the same group, but the Gmsh documentation clearly says that geometric entities are to be used so as to group elementary entities having similar “physical” meanings.

So as to obtain distinct groups with a mesh generated by Gmsh, it is thus necessary for the user to define physical entities. This requires an extra step, but allows for fine-grained control over the groups associated with the mesh, while using only elementary entities could lead to a high number of groups.

Default extension:	.msh
File type:	text or binary file
Surface elements:	triangles, quadrangles
Volume elements:	tetrahedra, pyramids, prisms, hexahedra
Zone selection:	physical entity numbers interpreted as numbered groups
Compatibility:	all files of this type
Documentation:	included documentation, also available at: <a href="http://www.geuz.org/gmsh">http://www.geuz.org/gmsh</a>

## 3.4.2 Formats supported for input or output

### 3.4.2.1 EnSight Gold

This format may represent all “classical” element types, as well as arbitrary polygons and convex polyhedra.

This format evolves slightly from one EnSight version to another, keeping backwards compatibility. For example, polygons could not be used in the same *part* as other element types prior to version 7.4, which removed this restriction and added support for polyhedra. Version 7.6 added support for material type definitions.

This format offers many possibilities not used by *Code\_Saturne*, such as defining values on part of a mesh only (using “undefined” marker values or partial values), assigning materials to elements, defining rigid motion, or defining per-processor mesh parts with ghost cells for parallel runs. Note that some libraries allowing direct EnSight Gold support do not necessarily support the whole format specification. Especially, VTK does not support material types, and has only recently added support for polyhedral elements in EnSight Gold files (interpreted as convex point sets in ParaView versions 2.4 to 2.8, and as true polyhedra starting with ParaView versions 2.10). Also, both EnSight Gold (8.2 and above) and VTK allow for automatic distribution, reducing the usefulness of pre-distributed meshes with per-processor files.

This format may be used as an input format, similar to EnSight 6. Compared to the latter, each *part* has its own coordinates and vertex connectivity, so as a convention, we consider that surface or volume zones may only be considered to be part of the same mesh if the file defines vertex IDs (which we consider to be unique vertex labels). In this case, *part* numbers are interpreted as group names. Without vertex IDs, only one part is read, and no groups are assigned.

Default extension:	directory { <i>case_name</i> }.ensight, containing a file with the .case extension
File type:	multiple binary or text files
Surface elements:	triangles, quadrangles, polygons
Volume elements:	tetrahedra, pyramids, prisms, hexahedra, convex polyhedra
Zone selection:	possibility of defining element materials (not used), or interpret part number as group name if vertex IDs are given
Compatibility:	files readable by EnSight 7.4 to 9.0, as well as tools based on the <a href="http://www.paraview.org">VTK</a> library, especially ParaView ( <a href="http://www.paraview.org">http://www.paraview.org</a> )
Documentation:	online documentation, also available at: <a href="http://www.ensight.com/downloads/cat_view-5.html">http://www.ensight.com/downloads/cat_view-5.html</a>

### 3.4.2.2 MED 2.3 or MED 3.0

Initially defined by EDF R&D, this format (*Modèle d'échanges de Données*, or *Model for Exchange of Data*) has been defined and maintained through a MED working group comprising members of EDF R&D and CEA (the *Code\_Saturne* team being represented). This is the reference format for the [SALOME](#) environment. This format is quite complete, allowing the definition of all “classical” element

types, in nodal or descending connectivity. Since MED 2.2 in 2003, this format may handle polygonal faces and polyhedral cells, as well as the definition of structured meshes.

This format, which requires a library also depending on the free HDF5 library, allows both for reading and writing meshes with their attributes (“families” of color/attribute and group combinations), as well as handling calculation data, with the possibility (unused by *Code\_Saturne*) of defining variables only on a subset (“profile”) of a mesh.

The MED library is available under a [LGPL](#) licence, and is even packaged in some Linux distributions (at least Debian and Ubuntu). Versions 2.3.5 and older require HDF5 1.6, version 2.3.6 may compile with either HDF5 1.6 or HDF5 1.8 (if the latter has HDF5 1.6 compatibility enabled).

Default extension:	<b>.med</b>
File type:	portable binary, based on the HDF5 library ( <a href="http://www.hdfgroup.org/HDF5/index.html">http://www.hdfgroup.org/HDF5/index.html</a> )
Surface elements:	triangles, quadrangles, simple polygons
Volume elements:	tetrahedra, pyramids, prisms, hexahedra, simple polyhedra
Zone selection:	element families ( <i>i.e.</i> colors and groups)
Compatibility:	MED 2.3.5, MED 2.3.6, or MED 3.0.2 and above (only unstructured nodal connectivity is supported)
Documentation:	online documentation. Download link at <a href="http://files.salome-platform.org/Salome/other/med-3.0.3.tar.gz">http://files.salome-platform.org/Salome/other/med-3.0.3.tar.gz</a>

### 3.4.2.3 CGNS 2.5 or CGNS 3.1

Promoted especially by NASA, Boeing, and ICEM CFD (as well as ONERA in France), this format (*CFD General Notation System*) is quite well established in the world of CFD. The concept is similar to that of MED, with a bigger emphasis on normalization of variable names or calculation information, and even richer possibilities. The opposite of MED, the first version of this format was limited to multibloc structured meshes, unstructured meshes having been added in CGNS 2.

Slightly older than MED, this library was free from the start, with a good English documentation, and is thus much better known. It is more focused on CFD, where MED is more generic. A certain number of tools accompany the CGNS distribution, including a mesh visualizer (which does not handle polygonal faces although the format defines them), and an interpolation tool.

We should be able to read almost any mesh written in this format, though meshes with overset interfaces may not be usable for a calculation. Other (abutting) interfaces are not handled automatically (as there are at least 3 or 4 ways of defining them, and some mesh tools do not export them<sup>5</sup>), so the user is simply informed of their existence in the Preprocessor's log file, with a suggestion to use an appropriate conformal joining option. Structured zones are converted to unstructured zones immediately after being read.

Boundary condition information is interpreted as groups with the same name. The format does not yet provide for selection of volume elements, as only boundary conditions defined in the model (and can be assigned to faces in the case of unstructured meshes, or vertices in any case). Note that boundary conditions defined at vertices are not ignored by the Preprocessor, but are assigned to the faces of which all vertices bear the same condition.<sup>6</sup>

The Preprocessor also has the capability of building additional volume or surface groups, based on the mesh sections to which cells or faces belong. This may be activated using a sub-option of the mesh selection, and allows obtaining zone selection information from meshes that do not have explicit boundary condition information but that are subdivided in appropriate zones or sections (which depends on the tool used to build the mesh).

<sup>5</sup>For example, ICEM CFD can join non-conforming meshes, but it exports joining surfaces as simple boundary faces with user-defined boundary conditions.

<sup>6</sup>If one of a face's vertices does not bear a boundary condition, that condition is not transferred to the face.



When outputting to CGNS, an unstructured connectivity is used for the calculation domain, with no face joining information or face boundary condition information.<sup>7</sup>

Though many tools support CGNS, that support is often quite dissapointing, at least for unstructured meshes. Thus, some editors seem to use different means to mark zones to associate with boundary conditions than the ones recommended in the CGNS documentation, and some behaviors are worse: for example, under EnSight Gold 8, whenever a mesh contains multiple element types, variables are assigned to the wrong cells. Regarding support of polygons (*ngons* in the CGNS standard), even the verification tools published alongside the CGNS library are unable to handle them, and report errors in valid files containing such elements. VisIt 1.11.1 reports an error when a mesh contains such faces, while EnSight Gold 8 ignores them. CGNS 3 allows for polyhedra, but as this is a recent developpement, it is not supported yet by *Code\_Saturne*.

Default extension:	.cgns
File type:	portable binary (uses the ADF library specific to CGNS, or HDF5)
Surface elements:	triangles, quadrangles, simple polygons
Volume elements:	tetrahedra, pyramids, prisms, hexahedra
Zone selection:	Surface zone selection using boundary conditions, no volume zone selection, but the Preprocessor allows creation of groups associated to zones or sections in the mesh using mesh selection sub-options
Compatibility:	CGNS 2.5 or CGNS 3.1
Documentation:	See CGNS site: <a href="http://www.cgns.org">http://www.cgns.org</a>

### 3.4.3 Meshing tools and associated formats

Most often, the choice of a mesh format is linked to the choice of a meshing tool. Still, some tools allow exporting a mesh under several formats handled by *Code\_Saturne*. This is the case of FLUENT and ICEM CFD, which can export meshes to both the I-deas universal and CGNS formats (FLUENT's GAMBIT is also able to export to I-deas universal format).

Traditionally, users exported files to the I-deas universal format, but it does not handle pyramid elements, which are often used by these tools to transition from hexahedral to tetrahedral cells in the case of hybrid meshes. The user is encouraged to export to CGNS, which does not have this limitation.

Tools related to the SALOME platform should preferably use SALOME's native MED format (export to I-deas universal is also possible, but has some limitations).

The use of files of the "Common Solver" type<sup>8</sup> is still in part possible but is deprecated. Such files are read directly from the Kernel, and this format is not handled by the launch scripts anymore. Many potentialities of *Code\_Saturne* are not usable with this file format (mesh joining with hanging nodes, periodicity, parallel computing, ...).

### 3.4.4 Meshing remarks

**WARNING:** Some turbulence models ( $k-\varepsilon$ ,  $R_{ij}-\varepsilon$  SSG, ...) used in *Code\_Saturne* are "High-Reynolds" models. Therefore the size of the cells neighboring the wall needs to be greater than the thickness of the viscous sublayer (at the wall,  $y^+ > 2.5$  is required, and  $30 < y^+ < 100$  is preferable). If the mesh does not match this constraint, the results may be false (particularly if thermal phenomena are involved). For more details on these constraints, see the keyword ITURB.

<sup>7</sup>Older versions of the documentation specified that a field must be defined on all elements of a zone, so that adding faces on which to base boundary conditions to a volume mesh would have required also defining volume fields on these faces. More recent versions of the documentation make it clear that a field must be defined on all elements of maximum dimension in a zone, not on all elements.

<sup>8</sup>File type specifically developed for the early prototype versions of *Code\_Saturne* (**tlc**) extension



### 3.5 Preprocessor command line options

The main options are:

- **--help**: gives a summary of the different command line options
- **<mesh>**: the last argumen is used to specify the name of the mesh file. The launch script automatically calls the Preprocessor for every mesh in the `MESHES []` list specified by th user.
- **--reorient**: try to re-orient badly-oriented cells if it is necessary to compensate for mesh-generation software whose output does not conform to the format specifications.

### 3.6 Kernel command line options

In the standard cases, the compilation of *Code\_Saturne* and its execution are entirely controlled by the launch script. The potential command line options are passed through user modifiable variables at the beginning of the script. This way, the user only has to fill these variables and doesn't need to search deep in the script for the Kernel command line. For more advanced usage, the main options are described below:

- **--solcom**: this option indicates that the Kernel will read the mesh directly, not using the Preprocessor output files. This is only possible with "Common Solver" type of mesh files (see §3.4.4 for restrictions).  
This option is obsolete, and is not handled by the launch script anymore.
- **--app-name**: specifies the application name. This is useful only in the case of code coupling, where the application name is used to distinguish between different code instances launched together.
- **--mpi**: specifies that the calculation is running with MPI communications. The number of processors used will be determined automatically by the Kernel. With most MPI implementations, the code will detect the presence of an MPI environment automatically, and this option is redundant. It is only kept for the rare case in which the MPI environment might not be detected.
- **--mpi-io**: specifies that if MPI-IO should be used where available, and which mode should be used (`off` to disable, `eo` for explicit offsets, and `ip` for individual file pointers). MPI-IO is only available when running with MPI, and may improve performance only on parallel file systems. In other cases, it will incur additional overhead.
- **--preprocess**: triggers the preprocessing-only mode. The code may run without any Interface parameter file nor any user subroutine. Only the initial operations such as mesh joining and modification are executed.
- **-q** or **--quality**: triggers the verification mode. The code may run without any Interface parameter file nor any user subroutine. This mode includes the preprocessing stages, and adds elementary tests:
  - the quality criteria of the mesh are calculated (non-orthogonality angles, internal faces offset, ...) and corresponding EnSight post-processing parts are created.
  - test calculation of the gradient of  $\sin(x + 2y + 3z)$ . The calculated value is compared to the exact value, and an EnSight part for the corresponding error is created. The gradient is calculated with each option `imrga` from 0 to 4.

- **--benchmark**: triggers the benchmark mode, for a timing of elementary operations on the machine. A secondary option **--mpitrace** can be added. It is to be activated when the benchmark mode is used in association with a MPI trace utility. It restricts the elementary operations to those implying MPI communications and does only one of each elementary operation, to avoid overfilling the MPI trace report.  
This command is to be placed in the **ARG\_CS\_VERIF** variable in the launch script to be added automatically to the Kernel command line.
- **--log n**: specifies the destination of the output for a single-processor calculation or for the processor of rank 0 in a parallel calculation.
  - n=0**: output directed towards the standard output
  - n=1**: output redirected towards a file **listing** (default behaviour)
 This option can be specified in the **domain.logging\_args** field of the user script.
- **--logp n**: specifies the destination of the output for the processors of rank 1 to  $N - 1$  in a calculation in parallel on  $N$  processors (*i.e.* the redirection of all but the first processor).
  - n=-1**: no output for the processors of rank 1 to  $N - 1$  (default behaviour).
  - n=0**: no redirection. Every processor will write to the standard output. This might be useful in case a debugger is used, with separate terminals for each processor.
  - n=1**: one file for the output of each processor. The output of the processors of rank 1 to  $N - 1$  are directed to the files **listing\_n0002** to **listing\_nN**. This option can be specified in the **domain.logging\_args** field of the user script.
- **-p xxx** or **--param xxx**: specifies the name of the GUI parameter file to use for the calculation. The value of **xxx** is to be defined by the **--param** option of **code\_saturne run**, either directly or in the standard **runcase** script (the file will be searched for in the **data** directory, though an absolute path name may also be defined).
- **-h** or **--help**: to display a summary of the different command line options.

### 3.7 Launch scripts

The case preparer command **code\_saturne create** places an example of launch script, **runcase**, in the **SCRIPTS** directory. This script is quite minimalist and is known to work on every architecture *Code\_Saturne* has been tested on. If a batch system is available, this script will contain options for batch submission. The script will then contain a line setting the proper **PYTHONPATH** variable for *Code\_Saturne* to run. Finally, it simply contains the **code\_saturne run** command, possible with a **--param** option when a parameters file defined by the GUI is used. Other options recognized by **code\_saturne run** may be added.

In the case of a coupled calculation, this script also exists, and may be used for preprocessing stages, but an additional **runcase\_coupling** is added in the directory above the coupled case directories, and may be used to define the list of coupled cases, as well as global options, such as MPI options of the temporary execution directory. An additional **runcase\_batch** file will contain batch submission options when a batch system is available (and is the file that should be submitted when using a batch system).

When not using the GUI, or if additional options need to be accessed, the **cs\_user\_scripts.py** file may be copied from the **DATA/REFERENCE** to the **DATA** and edited. This file contains several Python functions:

- **define\_domain\_parameter\_file** allows defining the choice of a parameters file produced by the GUI. This is generally not useful, as the parameters file may be directly defined in **runcase** or **runcase\_coupling**, or passed as an option to **code\_saturne run**, but could be useful when running more complex parametric scripts, and is provided for the sake of completeness.
- **define\_domain\_parameters** allows defining most parameters relative to case execution for the current domain, including advanced options not accessible through the GUI. This function is

the most important one in the user scripts file, and contains descriptions of the various options. Note that in most examples, setting of options is preceded by a `if domain.param == None:` line, ensuring the settings are only active if no GUI-defined parameters file is present. This is used to prevent accidental override of parameters defined by the GUI: parameters defined through the user script have priority over the GUI parameters file, so if both are used, these tests may be removed for parameters which should be defined through user scripts.

- `define_case_parameters` allows defining most parameters relative to the global calculation, such as the number of processors or the execution directory. To avoid potentially conflicting definitions, this function is ignored for coupled calculations, where the corresponding parameters may be defined in the `runcase_coupling` script.
- `define_mpi_environment` allows defining advanced MPI parameters or redefining MPI options if the automatic settings are incorrect, and its use should only rarely be necessary. To avoid potentially conflicting definitions, this function is ignored for coupled calculations, where the corresponding parameters may be defined in the `runcase_coupling` script.

## 3.8 Graphical User Interface

A Graphical User Interface is available with *Code\_Saturne*. This Interface creates or reads an XML file according to a specific *Code\_Saturne* syntax which is then interpreted by the code.

In version 2.1.3, the Graphical Interface manages calculation parameters, standard initialisation values and boundary conditions for standard physics, pulverised coal combustion and radiative transfers. The other specific physics are not yet managed by the Graphical Interface. In these particular cases, user subroutines have to be completed.

The Interface is optional. Every data that can be specified through the Interface can also still be specified in the user subroutines. In case of conflict, all calculation parameters, initialisation value or boundary condition set directly in the user subroutines will prevail over what is defined by the Interface. However, it is no longer necessary to redefine everything in the user subroutines. Only what was not set or could not be set using the Graphical Interface should be specified.

**WARNING:** There are some limitations to the changes that can be made between the Interface and the user routines. In particular, it is not possible to specify a certain number of solved variables in the Interface and change it in the user routines (for example, it is not possible to specify the use of a  $k - \varepsilon$  model in the Interface and change it to  $R_{ij} - \varepsilon$  in `usini1.f90`, or to define additional scalars in `usini1` with respect to the Interface). Also, all boundaries should be referenced in the Interface, even if the associated conditions are intended to be modified in `usclim`, and their nature (entry, outlet, wall<sup>9</sup>, symmetry) should not be changed.

For example, in order to set the boundary conditions of a calculation corresponding to a channel flow with a given inlet velocity profile, one should:

- set the boundary conditions corresponding to the wall and the output using the Graphical Interface
- set a dummy boundary condition for the inlet (uniform velocity for instance) - set the proper velocity profile at inlet in `usclim`. The wall and output areas do not need to appear in `usclim`. The dummy velocity entered in the Interface will not be taken into account.

The Graphical User Interface is launched with the `./SaturneGUI` command in the directory `DATA`. The first step is then to load an existing parameter file (in order to modify it) or to open a new one. The headings to be filled for a standard calculation are the followings:

- Identity and paths: definition of the calculation directories (`STUDY`, `CASE`, `DATA`, `SRC`, `SCRIPTS`, `MESH`).
- Calculation environment: definition of the mesh file(s), stand-alone execution of the Preprocessor module (used by the Interface to get the groups of the boundary faces).

<sup>9</sup>smooth and rough walls are considered of the same nature

- Thermophysical models: physical model, ALE mobile mesh features, turbulence model, thermal model, coupling with SYRTHES.
- Additional scalars: definition, initialisation of the scalars, and physical characteristics.
- Physical properties: reference pressure, fluid characteristics, gravity. It is also possible to write user laws for the density, the viscosity, the specific heat and the thermal conductivity in the interface through the use of a formulae interpreter.
- Volume conditions: initialization of the variables, and definition of the zones where to apply head loss.
- Boundary conditions: definition of the boundary conditions for each variable. The colors of the boundary faces may be read directly from a “listing” file created by the Preprocessor.
- Numerical parameters: number and type of time step, advanced parameters for the numerical solution of the equations.
- Calculation control: parameters concerning the time averages, time step, location of the probes where some variables will be monitored over time, definition of the frequency of the outputs in the calculation listing and in the chronological records and of the EnSight outputs. The item *Profiles* allows to save, with a given frequency, 1D profiles on an axis defined from two points provided by the user.
- Calculation management: management of the calculation restarts, updating of the launch script (temporary execution directory, parallel computing, user data or result files, ...) and interactive launch of the calculation.

The *Code\_Saturne* tutorial [14] offers a step-by-step guidance to the setting up of some simple calculations with the *Code\_Saturne* Interface.

To launch *Code\_Saturne* using an XML parameter file, the name of the file must be given to the variable **PARAM** in the launch script (see §3.7). When the launch script is edited from the Interface (Calculation management → Prepare batch analysis), the **PARAM** section is filled automatically as are the other parameters specified through the Interface.

#### NOTE: OPTION --NOGUI OF THE CODE\_SATURNE CREATE COMMAND

When a calculation is using the Interface but, for some reason, some extra parameters need to be specified in the subroutine **usini1**, the latter must be placed in the directory **SRC**. But, while doing this, all the parameters appearing in **usini1** will also be taken into account. In order to prevent the user from having to respecify in **usini1** all that he has already specified through the Interface, **code\_saturne create** automatically comments out the examples in **usini1** (**Cex** at the beginning of each line) while copying it in the directory **REFERENCE**. Therefore, the user only needs to uncomment the specific parts of **usini1** he wants to modify, and the rest of the examples will be ignored.

On the contrary, if the Interface will not be used, then all the parameters in **usini1** have to be specified. In that case, using the **--nogui** option of **code\_saturne create** will prevent it from commenting **usini1** out, thus saving the user the tedious task of uncommenting all the lines (and the risk of skipping some of them).

## 3.9 User subroutines

### 3.9.1 Preliminary comments

The user can run the calculations with or without an interface, with or without the user subroutines. Without interface, some user subroutines are needed. With interface, all the user subroutines are optional.

The parameters can be read in the interface and then in the user subroutines. In the case that a parameter is specified in the interface and in a user subroutine, it is the value in the user subroutine that is taken into account. For this reason, all the examples of user subroutines are placed in the REFERENCE directory by the case setup `code_saturne create`.

### 3.9.2 Main variables

This section presents a non-exhaustive list of the main variables which may be encountered by the user. Most of them should not be modified by the user. They are calculated automatically from the data. However it may be useful to know what they represent. Developers can also refer to [4] and [11].

These variables are listed in the alphabetical index at the end of this document.

The type of each variable is given: integer [i], real number [r], integer array [ia], real array [ra].

#### 3.9.2.1 Array sizes

- ndim:** Space dimension (ndim=3).
  
- ncel:** Number of real cells in the mesh.
- ncelet:** Number of cells in the mesh, including the ghost cells of the “halos” (see note 1).
- nfac:** Number of internal faces (see note 2).
- nfabor:** Number of boundary faces (see note 2).
- ncelbr:** Number of cells with at least one boundary face (see note 2).
  
- lndfac:** Size of the array **nodfac** of internal faces - nodes connectivity (see note 3).
- lndfbr:** Size of the array **nodfbr** of boundary faces - nodes connectivity (see note 3).
- nnod:** Number of vertices in the mesh.
  
- nfml:** Number of referenced families of entities (boundary faces, elements, ...).
- nprfml:** Number of properties per referenced entity family.
  
- nvar:** Number of solved variables (must be lower than **nvrmax**).
- nscamx:** Maximum number of scalars solutions of an advection equation, apart from the variables of the turbulence model ( $k, \varepsilon, R_{ij}, \omega, \varphi, \bar{f}$ ). That is to say the temperature and other scalars (passive or not, user-defined or not).
- nscal:** Effective number of scalars solutions of an advection equation, apart from the variables of the turbulence model ( $k, \varepsilon, R_{ij}, \omega, \varphi, \bar{f}$ ). That is to say the temperature and other scalars (passive or not, user-defined or not). These scalars can be divided into two distinct groups: **nscaus** user-defined scalars and **nscapp** scalars related to a “specific physics”. **nscal=nscaus+nscapp**, and **nscal** must be inferior or equal to **nscamx**.
- nscapp:** Effective number of scalars related to a “specific physics”. These scalars are solutions of an advection equation and distinct from the scalars of the turbulence model ( $k, \varepsilon, R_{ij}, \omega, \varphi, \bar{f}$ ). They are automatically defined by the choice of the selected specific physics model (gas combustion with Eddy Break-Up model, pulverised coal combustion, ...). For example: mass fractions, enthalpy, ....

- nscas:** Effective number of user-defined scalars. These scalars are solutions of an advection equation and distinct from the scalars of the turbulence model ( $k$ ,  $\varepsilon$ ,  $R_{ij}$ ,  $\omega$ ,  $\varphi$ ,  $\bar{f}$ ) and from the **nscapp** scalars related to the “specific physics”. For example: passive tracers, temperature (when no specific physics model is selected), ...
- nestmx:** Maximum number of error estimators for Navier-Stokes.
- npromx:** Maximum number of physical properties. They will be stored in the arrays **propce**, **propfa** or **propfb**.
- nproce:** Number of properties defined at the cells. They will be stored in the array **propce**.
- nprofa:** Number of properties defined at the internal faces. They will be stored in the array **propfa**.
- nprofb:** Number of properties defined at the boundary faces. They will be stored in the array **propfb**.
- nvisls:** Number of scalars with variable diffusivity.
- nushmx:** Maximum number of user chronological files (in the case where **ushist** is used).
- nbmomt:** Effective number of calculated time-averages. NBMOMT must be inferior or equal to **nbmomx**.
- nbmomx:** Maximum number of calculated time-averages (default value: 50).
- ndgmox:** Maximum degree of the time-averages (default value: 5).
- nclacp:** Number of coal classes for the pulverised coal combustion module. It is the total number of classes, *i.e.* the sum of the number of classes for every represented coal. **nclacp** must be inferior or equal to **nclcpm**.
- nclcpm:** Maximum number of coal classes for the pulverised coal combustion module.

#### NOTE 1: GHOST CELLS - “HALOS”

A cell (real cell) is an elementary mesh element of the spatial discretisation of the calculation domain. The mesh is made of NCEL cells.

When using periodicity and parallelism, extra “ghost” cells (called “halo” cells) are defined for temporary storage of some information (on a given processor). The total number of real and ghost cells is **ncelet**.

Indeed, when periodicity is enabled, the cells with periodic faces do not have any real neighboring cell across these particular faces. Their neighboring cell is elsewhere in the calculation domain (its position is determined by the periodicity). In order to temporarily store the information coming from this “distant” neighboring cell, a ghost cell (“halo”) is created.

The same kind of problem exists in the case of a calculation on parallel machines: due to the decomposition of the calculation domain, some cells no longer have access to all their neighboring cells, some of them being treated by another processor. The creation of ghost cells allows to temporarily store the information coming from real neighboring cells treated by other processors.

The variables are generally arrays of size **ncelet** (number of real and fictitious cells). The calculations (loops) are made on **ncel** cells (only the real cells, the fictitious cells are only used to store information).

#### NOTE 2: INTERNAL FACES

An internal face is an interface shared by two cells (real or ghost ones) of the mesh. A boundary face is a face which has only one real neighboring cell. In the case of periodic calculations, a periodic face is an internal face. In the case of parallel running calculations, the faces situated at the boundary of a partition may be internal faces or boundary faces (of the whole mesh);

#### NOTE 3: FACES-NODES CONNECTIVITY

The faces - nodes connectivity is stored by means of four integer arrays: **ipnfac** and **nodfac** for the

internal faces, `ipnfbr` and `nodfbr` for the boundary faces. `nodfac` (size `lndfac`) contains the list of all the nodes of all the internal faces; first the nodes of the first face, then the nodes of the second face, and so on. `ipnfac` (size: `nfac+1`) gives the position `ipnfac(ifac)` in `nodfac` of the first node of each internal face `ifac`. Therefore, the reference numbers of all the nodes of the internal face `ifac` are: `nodfac(ipnfac(ifac))`, `nodfac(ipnfac(ifac)+1)`, ..., `nodfac(ipnfac(ifac+1)-1)`. In order for this last formula to be valid even for `ifac=nfac`, `ipnfac` is of size `nfac+1` and `ipnfac(nfac+1)` is equal to `lndfac+1`.

The composition of the arrays `nodfbr` and `ipnfbr` is similar.

#### NOTE 4: COMMONS

**The user will not modify the existing “commons”.** This would require the recompilation of the complete version, operation which is not allowed in standard use.

### 3.9.2.2 Geometric variables

The main geometric variables are available in most of the subroutines and directly accessible through the following arrays, defined in the `mesh` module (i.e. `use mesh`).

`cdgfac(ndim,nfac)` [ra]: Coordinates of the centers of the internal faces.

`cdgfbo(ndim,nfabor)` [ra]: Coordinates of the centers of the boundary face.

`ifacel(2,nfac)` [ia]: Index-numbers of the two (only) neighboring cells for each internal face.

`ifabor(nfabor)` [ia]: Index-number of the (unique) neighboring cell for each boundary face.

`ipnfac(nfac+1)` [ia]: Position of the first node of the each internal face in the array `nodfac` (see note 3 in paragraph 3.9.2.1)..

`ipnfbr(nfabor+1)` [ia]: Position of the first node of the each boundary face in the array `nodfbr` (see note 3 in paragraph 3.9.2.1)..

`nodfac(lndfac)` [ia]: Index-numbers of the nodes of each internal face (see note 3 in paragraph 3.9.2.1)..

`nodfbr(lndfbr)` [ia]: Index-numbers of the nodes of each boundary face (see note 3 in paragraph 3.9.2.1)..

`surfaced(ndim,nfac)` [ra]: Surface vector of the internal faces. Its norm is the surface of the face and it is oriented from `ifacel(1,.)` to `ifacel(2,.)`.

`surfbbo(ndim,nfabor)` [ra]: Surface vector of the boundary faces. Its norm is the surface of the face and it is oriented outwards.

`volume(ncelet)` [ra]: Volume of each cell.

`xyzcen(ndim,ncelet)` [ra]: Coordinates of the cell centers.

`xyznod(ndim,nnod)` [ra]: Coordinates of the mesh vertices.

In addition, other geometric variables are useful for gradients reconstruction. The main variables of this type are the following:

`dijpf(ndim,nfac)` [ra]: For every internal face, the three components of the vector  $\underline{I'J'}$ , where  $I'$  and  $J'$  are respectively the orthogonal projections of the neighboring cell centers  $I$  and  $J$  on a straight line orthogonal to the face and passing through its center..

`diipb(ndim,nfabor)` [ra]: For every boundary face, the three components of the vector  $\underline{II'}$ .  $I'$  is the orthogonal projection of  $I$ , center of the neighboring cell, on the straight line perpendicular to the face and passign through its center.

`dist(nfac)` [ra]: For every internal face, dot product of the vectors  $\underline{IJ}$  and  $\underline{n}$ .  $I$  and  $J$  are respectively the centers of the first and the second neighboring cell. The vector  $\underline{n}$  is the unit vector normal



to the face and oriented from the first to the second cell.

**distbr(nfabor)** [ra]: For every boundary face, dot product between the vectors  $\underline{IF}$  and  $\underline{n}$ . I is the center of the neighboring cell. F is the face center. The vector  $\underline{n}$  is the unit vector normal to the face and oriented to the exterior of the domain.

**dofij(ndim,nfac)** [ra]: For every internal face, the three components of the vector  $\underline{OF}$ . O is the intersection point between the face and the straight line joining the centers of the two neighboring cells. F is the face center.

**icelbr(ncelbr)** [ia]: List of cells having at least one boundary face.

**ipond(nfac)** [ra]: For every internal face,  $\frac{FJ.n}{IJ.n}$ . With regard to the mesh quality, its ideal value is 0.5.

**surfan(nfac)** [ra]: Norm of the surface vector of the internal faces.

**surfbn(nfabor)** [ra]: Norm of the surface of the boundary faces.

### 3.9.2.3 Physical variables

The main physical variables are available in the majority of the subroutines and brought together according to their type in the multidimensional arrays listed below. In some particular subroutines, some variables may be given a more explicit name, in order to ease the comprehension.

**propce(ncelet,nproce)** [ra]: Properties defined at the cell centers. For instance: density, viscosity, ....

**propfa(nfac,nprofa)** [ra]: Properties defined at the internal faces. For instance: mass flow across internal faces.

**propfb(nfabor,nprofb)** [ra]: Properties defined at the boundary faces. For instance: mass flow across boundary faces, density at boundary faces, ....

**rtp(ncelet,nvar)** [ra]: Array storing the values of the solved variables at the current time step.

**rtpa(ncelet,nvar)** [ra]: Array storing the values of the solved variables at the previous time step.

#### About **rtp** and **rtpa**

The indexes allowing to mark out the different variables (from 1 to **nvar**) are integers available in a “common” file called **numvar.h**.

For example, **ipr** refers to the variable “pressure”: the pressure in the cell **iel** at the current time step is therefore **rtp(iel,ipr)**.

The list of integers referring to solved variables is given below. These variable index-numbers are not only used for the **rtp** and **rtpa** arrays, but also for some arrays of variable associated options (for instance, **blencv(ik)** is the percentage of second-order convective scheme for the turbulent energy when a corresponding turbulent model is used).

- **ipr**: pressure <sup>10</sup>.
- **iu**: velocity along the X axis.
- **iv**: velocity along the Y axis.
- **iw**: velocity along the Z axis.
- **ik**: turbulent energy, in  $k - \varepsilon$ ,  $k - \omega$  modeling or v2f ( $\varphi$ -model) modeling.

---

<sup>10</sup>**ipr** corresponds to a reduced pressure, from which the standard hydrostatic pressure has been deduced. The total pressure is stored in the **PROPCE** array



- **ir11**: Reynolds stress  $R_{11}$ , in  $R_{ij} - \varepsilon$  or SSG modeling.
- **ir22**: Reynolds stress  $R_{22}$ , in  $R_{ij} - \varepsilon$  or SSG modeling.
- **ir33**: Reynolds stress  $R_{33}$ , in  $R_{ij} - \varepsilon$  modeling.
- **ir12**: Reynolds stress  $R_{12}$ , in  $R_{ij} - \varepsilon$  modeling.
- **ir13**: Reynolds stress  $R_{13}$ , in  $R_{ij} - \varepsilon$  modeling.
- **ir23**: Reynolds stress  $R_{23}$ , in  $R_{ij} - \varepsilon$  modeling.
- **iep**: turbulent dissipation in  $k - \varepsilon$ ,  $R_{ij} - \varepsilon$  or v2f ( $\varphi$ -model) modeling.
- **iomg**: Specific dissipation rate  $\omega$ , in  $k - \omega$  SST modeling.
- **iphi**: variable  $\varphi = \overline{v^2}/k$  in v2f ( $\varphi$ -model).
- **ifb**: variable  $\overline{f}$  in v2f ( $\varphi$ -model).
- **isca(j)**: scalar  $j$  ( $1 \leq j \leq \text{nscal}$ ).

Concerning the solved scalar variables (apart from the variables pressure,  $k$ ,  $\varepsilon$ ,  $R_{ij}$ ,  $\omega$ ,  $\varphi$ ,  $\overline{f}$ ), the following are highly important:

- The designation “scalar” refers to scalar variables which are solution of an advection equation, apart from the variables of the turbulence model ( $k$ ,  $\varepsilon$ ,  $R_{ij}$ ,  $\omega$ ,  $\varphi$ ,  $\overline{f}$ ): for instance the temperature, scalars which may be passive or not, “user” or not. The mean value of the square of the fluctuations of a “scalar” is a “scalar”, too. The scalars may be divided into two groups: **nscaus** “user” scalars and **nscapp** “specific physics” scalars, with **nscal**=**nscaus**+**nscapp**. **nscal** must be inferior or equal to **nscamx**.
- The  $j^{\text{th}}$  user scalar is, in the whole list of the **nscal** scalars, the scalar number  $j$ . In the list of the **nvar** solved variables, it corresponds to the variable number **isca(j)**, its value in the cell **iel** at the current time step is given by **rtp(iel,isca(j))**.
- The  $j^{\text{th}}$  scalar related to a specific physics is, in the whole list of the **nscal** scalars, the scalar number **iscapp(j)**. In the list of the **nvar** solved variables, it corresponds to the variable number **isca(iscapp(j))**, its value in the cell **iel** at the current time step is given by **rtp(iel,isca(iscapp(j)))**.
- The temperature (or the enthalpy) is the scalar number **iscalt** in the list of the **nscal** scalars. It corresponds to the variable number **isca(iscalt)** and its value in the cell **iel** is **rtp(iel,isca(iscalt))**. if there is no thermal scalar, **iscalt** is equal to -1.
- A “user” scalar number  $j$  may represent the average of the square of the fluctuations of a scalar **k** (*i.e.* the average  $\overline{\varphi'\varphi'}$  for a fluctuating scalar  $\varphi$ ). This can be made either *via* the interface or by indicating **iscavr(j)=k** in **usini1** (if the scalar in question is not a “user” scalar, the selection is made automatically). For instance, if  $j$  and  $k$  are “user” scalars, the variable  $\varphi$  corresponding to  $k$  is the variable number **isca(k)=isca(iscavr(j))**, and its value in the cell **iel** is **rtp(iel,isca(k))=rtp(iel,isca(iscavr(j)))**.  
The variable corresponding to the mean value of the square of the fluctuations<sup>11</sup> is the variable number **isca(j)** and its value in the cell **iel** is **rtp(iel,isca(j))**.

---

<sup>11</sup>it is really  $\overline{\varphi'\varphi'}$ , and not  $\sqrt{\overline{\varphi'\varphi'}}$

About **propce**, **propfa** and **propfb** In *Code\_Saturne*, the physical properties<sup>12</sup> are stored in the **propce**, **propfa** and **propfb** arrays. Some properties, like the density, are only stored for cells and boundary faces. Some, like the mass flux, are only stored at the interior and boundary faces. To avoid having different index numbers for a physical property, depending on the array it is used in, the following structure is used in *Code\_Saturne*:

- All the properties (used or not) have a unique and distinct index-number, given automatically by the code and stored in an integer or an integer array (its size may be the maximum number of scalars or the maximum number of variables).
- The indexes referring to the different properties stored in the **propxx** arrays are given respectively by the following integer arrays:

**ipproc**(**npromx**) [ia]: Rank *i* in **propce**(.,*i*) of the properties defined at the cell centers.

**ipprof**(**npromx**) [ia]: Rank *i* in **propfa**(.,*i*) of the properties defined at the internal faces.

**ipprob**(**npromx**) [ia]: Rank *i* in **propfb**(.,*i*) of the properties defined at the boundary faces.

For instance, the index number corresponding to the density is **irom**.

In the list of the properties defined at the cell center, the density is therefore the **ipproc**(**irom**)<sup>th</sup> property: its value at the center of the cell **iel** is given by **propce**(**iel**,**ipproc**(**irom**)).

In the same way, in the list of the properties defined at the boundary faces, the density is the **ipprob**(**irom**)<sup>th</sup> property: its value at the boundary face is given by **propfb**(**iel**,**ipprob**(**irom**)).

The list of properties accessible in the **PROPxx** arrays is given below (this does not include the properties linked to the specific physics modules):

**irom** [ia]: Property number corresponding to the density (*i.e.*  $\rho$  in  $kg.m^{-3}$ ) stored at the cells and the boundary faces.

**iroma** [ia]: Property number corresponding to the density (*i.e.*  $\rho$  in  $kg.m^{-3}$ ) at the previous time step, in the case of a second-order extrapolation in time stored at the cells and the boundary faces.

**ivisc1** [ia]: Property number corresponding to the fluid molecular dynamic viscosity (*i.e.*  $\mu$  in  $kg.m^{-1}.s^{-1}$ ) stored at the cells.

**ivisla** [ia]: Property number corresponding to the fluid molecular dynamic viscosity (*i.e.*  $\mu$  in  $kg.m^{-1}.s^{-1}$ ) at the previous time step, in the case of a second-order extrapolation in time stored at the cells.

**ivisct** [ia]: Property number corresponding to the fluid turbulent dynamic viscosity (*i.e.*  $\mu_t$  in  $kg.m^{-1}.s^{-1}$ ) stored at the cells.

**ivista** [ia]: Property number corresponding to the fluid turbulent dynamic viscosity (*i.e.*  $\mu_t$  in  $kg.m^{-1}.s^{-1}$ ) at the previous time step, in the case of a second-order extrapolation in time stored at the cells.

**icp** [ia]: Property number corresponding to the specific heat, in case where it is variable (*i.e.*  $C_p$  in  $m^2.s^{-2}.K^{-1}$ ). See note below stored at the cells.

**icpa** [ia]: Property number corresponding to the specific heat, in case where it is variable (*i.e.*  $C_p$  in  $m^2.s^{-2}.K^{-1}$ ), at the previous time step, in the case of a second-order extrapolation in time.

---

<sup>12</sup>other variables are stored in the arrays **propce**, **propfa** and **propfb**. They are not “physical properties” strictly speaking, but it is convenient to have them in the same array as the proper physical properties

See note below  
stored at the cells.

**itsnsa** [ia]: In the case of a calculation run with a second-order discretisation in time with extrapolation of the source terms, property number corresponding to the source term of Navier-Stokes at the previous time step ( $kg.m^{-1}.s^{-2}$ ) stored at the cells.

**itstua** [ia]: In the case of a calculation run with a second-order discretisation in time with extrapolation of the source terms, property number corresponding to the source terms of the turbulence at the previous time step stored at the cells.

**itssca** [ia]: In the case of a calculation run with a second-order discretisation in time with extrapolation of the source terms, property number corresponding to the source terms of the equations solved for the scalars at the previous time step ( $kg.m^{-1}.s^{-2}$ ) stored at the cells.

**iestim(nestmx)** [ia]: Property number for the **nestmx** error estimators for Navier-Stokes. The estimators currently available are **iestim(iespre)**, **iestim(iesder)**, **iestim(iescor)**, **iestim(iestot)** stored at the cells.

**ifluma(nvarmx)** [ia]: Property number corresponding to the mass flow associated with each variable (*i.e.* for each face of surface  $S$ ,  $\rho \underline{u} . \underline{S}$  in  $kg.s^{-1}$ ). It must be noticed that the mass flows are associated with the variables, which allows to have a distinct convective flow for each scalar. stored at the internal faces and boundary faces.

**ifluaa(nvarmx)** [ia]: Property number corresponding to the mass flow associated with each variable at the previous time step, in the case of a second-order extrapolation in time stored at the internal faces and boundary faces.

**ivisls(nscamx)** [ia]: Property number corresponding to the diffusivity of scalars for which it is variable (*i.e.*  $\frac{\lambda}{C_p}$  for the temperature, in  $kg.m^{-1}.s^{-1}$ ). It must be noticed that the diffusivity is associated with the scalars rather than with the variables. See note below stored at the cells.

**ivissa(nscamx)** [ia]: Property number corresponding to the diffusivity of scalars for which it is variable (*i.e.*  $\frac{\lambda}{C_p}$  for the temperature, in  $kg.m^{-1}.s^{-1}$ ) at the previous time step, in the case of a second-order extrapolation in time stored at the cells.

**ismago** [i]: Property number corresponding to the variable  $C$  of the dynamic model, *i.e.* so that  $\mu_t = \rho C \Delta^2 \sqrt{2S_{ij}S_{ij}}$  (with the notations of [3]).  $C$  corresponds to  $C_s^2$  in the classical model of Smagorinsky stored at the cells.

**icour** [i]: CFL number in each cell at the present time step stored at the cells.

**ifour** [i]: Fourier number in each cell at the present time step stored at the cells.

**iprtot** [i]: Total pressure in each cell stored at the cells.

**ivisma(1 or 3)** [ia]: When the ALE method for deformable meshes is activated, **ivisma** corresponds to the “mesh viscosity”, allowing to limit the deformation in certain areas. This mesh viscosity can be isotropic or be taken as a diagonal tensor (depending on the value of the parameter **iortvm**).

stored at the cells.

**icmome(nbmomx)** [ia]: Property number corresponding to the time averages defined by the user. More precisely, it is not the time average that is stored, but a summation over time (the division by the cumulated duration is done just before the results are written)  
stored at the cells.

**icdtmo(nbmomx)** [ia]: Property number corresponding to the cumulated duration associated with each time average defined by the user, when this duration is not spatially uniform (see note below)  
stored at the cells.

#### NOTE: VARIABLE PHYSICAL PROPERTIES

Some physical properties such as specific heat or diffusivity are often constant (choice made by the user). In that case, in order to limit the necessary memory, these properties are stored as a simple real number rather than in a domain-sized array of reals.

- It is the case for the specific heat  $C_p$ .
  - If  $C_p$  is constant, it can be specified in the interface or by indicating **icp=0** in **usini1**, and the property will be stored in the real number **cp0**.
  - If  $C_p$  is variable, it can be specified in the interface or by indicating **icp=1** in **usini1**. The code will then modify this value to make **icp** refer to the effective property number corresponding to the specific heat, in a way which is transparent for the user. For each cell **iel**, the value of  $C_p$  is then given in **usphyv** and stored in the array **propce(iel,iproc(icp))**.
- It is the same for the diffusivity  $K$  of each scalar **iscal**.
  - If  $k$  is constant, it can be specified in the interface or by indicating **ivisls(iscal)=0** in **usini1**, and the property will be stored in the real number **visls0(iscal)**.
  - If  $k$  is variable, it can be specified in the interface or by indicating **ivisls(iscal)=1** in **usini1**. The code will then modify this value to make **ivisls(iscal)** refer to the effective property number corresponding to the diffusivity of the scalar **iscal**, in a way which is transparent for the user. For each cell **iel**, the value of  $k$  is then given in **usphyv** and stored in the **propce(iel,iproc(ivisls(iscal)))** array.

#### NOTE: CUMULATED DURATION ASSOCIATED WITH THE AVERAGES DEFINED BY THE USER

The cumulated duration associated with the calculation of a time averages defined by the user is often a spatially uniform value. In this case, it is stored in a simple real number: for the mean value **imom**, it is the real number **dtcmom(-idtmom(imom))** (**idtmom(imom)** is negative in this case).

When this cumulated duration is not spatially uniform (for instance in the case of a spatially variable time step), it is stored in **propce**. It must be noted that the cumulated duration associated with the calculation of the average **imom** is variable in space if **idtmom(imom)** is strictly positive. The number of the associated property in **propce** is then **icdtmo(idtmom(imom))**. For instance, for the average **imom**, the cumulated duration in the cell **iel** will be **propce(iel,icdtmo(idtmom(imom)))**.

The user may have a look to the example given in **usproj** to know how to calculate a time averages in a particular cases (printing of extreme values, writing of results, ...).

Two other variables, **hbord** and **tbord**, should be noted here, although they are relatively local (they appear only in the treatment of the boundary conditions) and are used only by developers.

**hbord(nfabor)** [ra]: Array of the exchange coefficient for temperature (or enthalpy) at the boundary faces. The table is allocated only if **isvhb** is set to 1 in **tridim**, which is done automatically, but only if the coupling with SYRTHES or the 1D thermal wall module are activated..

`tbord(nfabor)` [ra]: Temperature (or enthalpy) at the boundary faces<sup>13</sup>. The table is allocated only if `isvtb` is set to 1 in `tridim`, which is done automatically but only if the coupling with SYRTHES or the 1D thermal wall module are activated..

Tables `hbord` and `tbord` are of size `nfabor`, although they concern only the wall boundary faces.

### 3.9.2.4 Variables related to the numerical methods

The main numerical variables and “pointers”<sup>14</sup> are displayed below.

#### BOUNDARY CONDITIONS

`coefa(nfabor,*)` [ra]: Boundary conditions: see note 2.

`coefb(nfabor,*)` [ra]: Boundary conditions: see note 2.

`iclrrtp(nvarmx,2)` [ia]: For each variable `ivar` ( $1 \leq \text{ivar} \leq \text{nvar} \leq \text{nvarmx}$ ), rank in `coefa` and `coefb` of the boundary conditions. See note 2.

`icoef` [i]: Rank in `iclrrtp` of the rank in `coefa` and `coefb` of the “standard” boundary conditions. See note 2.

`icoeff` [i]: Rank in `iclrrtp` of the rank in `coefa` and `coefb` of the “flow” type boundary conditions, reserved for developers. See note 2.

`ifmfbr(nfabor)` [ia]: Family number of the boundary faces. See note 1.

`iprfml(nfml,nprfml)` [ia]: Properties of the families of referenced entities. See note 1.

`iisymph` [i]: Integer giving the rank in `ia` of the first element of the section allowing to mark out the “wall” (`itypfb=iparoi` or `iparug`) or “symmetry” (`itypfb=isymet`) boundary faces in order to prevent the mass flow (these faces are impermeable). For instance, if the face `ifac` is a wall or symmetry face, `ia(iismph+ifac-1)=0` (with `iismph=iisymph+nfabor`). Otherwise `ia(iisymph+ifac-1)=1`.

In some subroutines, an array called `isympha(nfabor)` allows to simplify the coding with `isympha(ifac)=ia(iismph+ifac-1)`.

`itrifb(nfabor)` [ia]: Indirection array allowing to sort the boundary faces according to their boundary condition type `itypfb`.

`itypfb(nfabor)` [ia]: Boundary condition type at the boundary face `ifac` (see user subroutine `usclim`).

`uetbor(nfabor)` [ra]: Friction velocity at the wall, in the case of a LES calculation with van Driest-wall damping.

#### DISTANCE TO THE WALL

`ifapat(ncelet)` [ra]: Number of the wall face (type `itypfb=iparoi` or `iparug`) which is closest to the center of a given volume when necessary ( $R_{ij} - \varepsilon$  with wall echo, LES with van Driest-wall damping, or SST  $k - \omega$  turbulence model) and when `icdpar=2`. The number of the wall face which is the closest to the center of the cell `iel` is `ifapat(iel1)`. This calculation method is not compatible with parallelism and periodicity.

<sup>13</sup>It is the physical temperature at the boundary faces, not the boundary condition for temperature. See [11] for more details on boundary conditions

<sup>14</sup>As for the geometrical variables, some variables may be accessed directly in sections of the unidimensional macro-array `ra` (for the real numbers) which is present as an argument to many subroutines. The number of the first position of these sections in `ra` is indicated by an integer stored in the `pointe` Fortran module. These integers are referred to as “pointers”

**dispar(ncelet)** [ra]: Distance between the center of a given volume and the closest wall, when it is necessary ( $R_{ij} - \varepsilon$  with wall echo, LES with van Driest-wall damping, or SST  $k - \omega$  turbulence model) and when **icdpar=1**. The distance between the center of the cell **iel** and the closest wall is **dispar(iel)**.

**yp1par** [ra]: Adimensional distance  $y^+$  between a given volume and the closest wall, when it is necessary (LES with van Driest-wall damping) and when **icdpar=1**. The adimensional distance  $y^+$  between the center of the cell **iel** and the closest wall is therefore **yp1par(iel1)**.

#### PRESSURE DROPS

**icepdc(ncepdc)** [ia]: Number of the **ncepdc** cells in which a pressure drop is imposed. See **iicepd** and the user subroutine **uskpdc**.

**ckupdc(ncepdc,6)** [ra]: Value of the coefficients of the pressure drop tensor of the **ncepdc** cells in which a pressure drop is imposed. See **ickpdc** and the user subroutine **uskpdc**.

**ncepdc** [ia]: Number of cells in which a pressure drop is imposed. See the user subroutine **uskpdc**.

#### MASS SOURCES

**icetsm(ncetsm)** [ia]: Number of the **ncetsm** cells in which a mass source term is imposed. See **icesm** and the user subroutine **ustsma**.

**itypsm(ncetsm,nvar)** [ia]: Type of mass source term for each variable (0 for an injection at ambient value, 1 for an injection at imposed value). See the user subroutine **ustsma**.

**ncetsm** [i]: Number of cells with mass sources. See the user subroutine **ustsma**.

**smacel(ncetsm,nvar)** [ra]: Value of the mass source term for pressure. For the other variables, eventual imposed injection value. See the user subroutine **ustsma**.

#### WALL 1D THERMAL MODULE

**nfpt1d** [i]: Number of boundary faces which are coupled with a wall 1D thermal module. See the user subroutine **uspt1d**.

**ifpt1d** [ia]: Array allowing to mark out the numbers of the **nfpt1d** boundary faces which are coupled with a wall 1D thermal module. The numbers of these boundary faces are given by **ifpt1d(ii)**, with  $1 \leq ii \leq \text{nfpt1d}$ . See the user subroutine **uspt1d**.

**nppt1d** [ia]: Number of discretisation cells in the 1D wall for the **nfpt1d** boundary faces which are coupled with a wall 1D thermal module. The number of cells for these boundary faces is given by **nppt1d(ii)**, with  $1 \leq ii \leq \text{nfpt1d}$ . See the user subroutine **uspt1d**.

**eppt1d** [ia]: Thickness of the 1D wall for the **nfpt1d** boundary faces which are coupled with a wall 1D thermal module. The wall thickness for these boundary faces is therefore given by **eppt1d(ii)**, with  $1 \leq ii \leq \text{nfpt1d}$ . See the user subroutine **uspt1d**.

#### OTHERS

**dt(ncelet)** [ra]: Value of the time step.

**ifmcel(ncelet)** [ia]: Family number of the elements. See note 1.

**s2kw(ncelet)** [ra]: Square of the norm of the deformation rate tensor. In the cell **iel**,  $S^2 = 2S_{ij}S_{ij}$  is given by **ra(is2kw+iel-1)**. This array is defined only with the SST  $k - \omega$  turbulence model.

**divukw** [ia]: Divergence of the velocity. In the cell **iel**,  $div(\underline{u})$  is given by **divukw(iel1)**. This array is defined only with the SST  $k-\omega$  turbulence model (because in this case it may be calculated at the same time as  $S^2$ ).

**ngrmmx** [i]: upper limit of the number of grid levels when using the multigrid solver (see **ngrmax**).

**ra(ifinra)** [ra]: Real work array.

#### NOTE: BOUNDARY CONDITIONS

The boundary conditions in *Code\_Saturne* boil down to determine a value for the current variable  $\phi$  at the boundary faces, that is to say  $\phi_f$ , value expressed as a function of  $\phi_{I'}$ , value of  $\phi$  in  $I'$ , projection of the center of the adjacent cell on the straight line perpendicular to the boundary face and crossing its center:  $\phi_f = A_{\phi,f} + B_{\phi,f}\phi_{I'}$ .

For a face **ifac**, the pair of coefficients  $A_{\phi,f}, B_{\phi,f}$  is stored in **coefa(ifac,iclvar)** and **coefb(ifac,iclvar)**, where the integer **iclvar=iclrrtp(ivar,ijcl)** determines the rank in **coefa** and **coefb** of the set of boundary conditions of the variable **ivar**.

The second index of the array **iclrrtp** allows to have several sets of boundary conditions for each variable. The “standard” boundary conditions are determined by **ijcl=icoef**, where **icoef** is a parameter which is fixed automatically by the code, and can be accessed to in the “common” file **numvar.h**. More specific or advanced boundary conditions can be accessed to with **ijcl=icoeff**.

In practice, for a variable **ivar** whose value  $\phi_{I'}$  in a boundary cell is known, the value at the corresponding boundary face **ifac** is:

$\phi_f = \text{coefa}(\text{ifac}, \text{iclvar}) + \text{coefb}(\text{ifac}, \text{iclvar}) \phi_{I'}$  with **iclvar=iclrrtp(ivar,icoef)**

### 3.9.2.5 User arrays

Modules containing user arrays accessible from all user subroutines may be defined in the **user\_modules.f90** file. This file is compiled before any other Fortran user file, to ensure modules may be accessed in other user subroutines using the **use <module>** construct.

### 3.9.2.6 Parallelism and periodicity

Parallelism is based on domain partitioning: each processor is assigned a part of the domain, and data for cells on parallel boundaries is duplicated on neighboring processors in corresponding “ghost”, or “halo” cells (both terms are used interchangeably). Values in these cells may be accessed just the same as values in regular cells. Communication is only required when cell values are modified using values from neighboring cells, as the values in the “halo” can not be computed correctly (since the halo does not have access to all its neighbors), so halo values must be updated by copying values from the corresponding cells on the neighboring processor.

Compared to other tools using a similar system, a specificity of *Code\_Saturne* is the separation of the halo in two parts: a standard part, containing cells shared through faces on parallel boundaries, and an extended part, containing cells shared through vertices, which is used mainly for least squares gradient reconstruction using an extended neighborhood. Most updates need only operate on the standard halo, requiring less data communication than those on the extended halos.

Periodicity is handled using the same halo structures as parallelism, with an additional treatment for vector and coordinate values: updating coordinates requires applying the periodic transformation to the copied values, and in the case of rotation, updating vector and tensor values also requires applying the rotation transformation. Ghost cells may be parallel, periodic, or both. The example of a pump combining parallelism and periodicity is given figure 5. In this example, all periodic boundaries match with boundaries on the same domain, so halos are either parallel or periodic.

#### **Activation**

Parallelism is activated by means GUI or of the launch scripts in the standard cases:



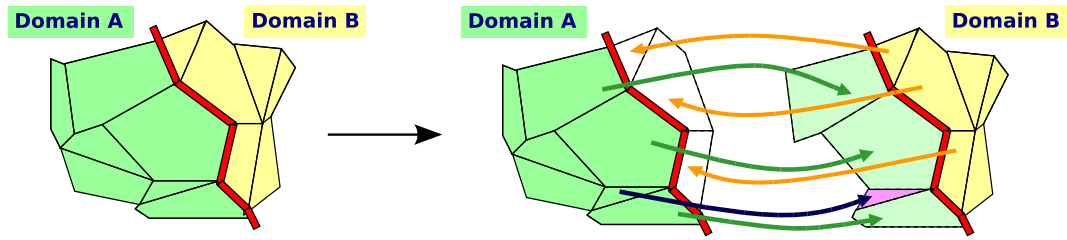


Figure 4: Parallel domain partitioning: halos

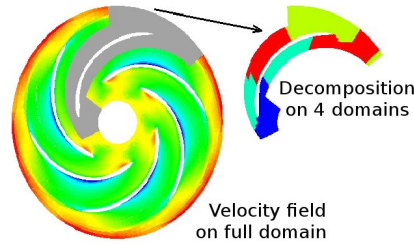


Figure 5: Combined parallelism and periodicity

- On clusters with batch systems, the launching of a parallel run requires to complete the batch cards located in the beginning of `runcase` or `runcase_batch` script, and set the number of MPI processes, or the numbers of physical nodes and processors per node (`ppn`) wanted. This can be done through the Graphical Interface or by editing the `runcase` or `runcase_batch` file directly. The number of processors defined here will override the number defined through the GUI in a non-batch environment (so that studies defined on one environment may be migrated to larger compute resources easily), but it may be overridden by the
- `define_case_parameters` function from the `cs_user_scripts.py` file, or by setting the `n_procs_weight`, `n_procs_min`, and `n_procs_max` parameters for the different domains defined in `runcase_coupling`.
- On clusters with unsupported batch systems, `runcase` file may have to be modified manually. Please do not hesitate to contact the *Code\_Saturne* support ([saturne-support@edf.fr](mailto:saturne-support@edf.fr)) so that these modifications can be added to the standard launch script to make it more general.
- A parallel calculation may be stopped in the same manner as a sequential one using a `ficstp` file (see paragraph 3.2.4).
- The standard pieces of information displayed in the listing (marked out with 'v' for the min/max values of the variables), 'c' for the data concerning the convergence and 'a' for the values before clipping) are global values for the whole domain and not related to each processor.

### User subroutines

The user can check in a subroutine

- that the presence of periodicity is tested with the variable `iperio` (=1 if periodicity is activated);
- that the presence of rotation periodicities is tested with the variable `iperot` (number of rotation periodicities);
- that running of a calculation in parallel is tested for with the variable `irangp` (`irangp` is worth -1 in the case of a non-parallel calculation and  $p - 1$  in the case of a parallel calculation,  $p$  being the number of the current processor)



EDF R&D	<b>Code_Saturne version 2.1.3 practical user's guide</b>	Code_Saturne documentation Page 41/205
---------	--	--

Attention must be paid to the coding of the user subroutines. If conventionnal subroutines like `usini1` or `usclim` usually do not cause any problem, some kind of developments are more complicated. The most usual cases are dealt with below.

Examples are given in the subroutine `usproj`.

- **Access to information related to neighboring cells in parallel and periodic cases.**

When periodicity or parallelism are brought into use, some cells of the mesh become physically distant from their neighbors. Concerning parallelism, the calculation domain is split and distributed between the processors: a cell located at the “boundary” of a given processor may have neighbors on different processors.

In the same way, in case of periodicity, the neighboring cells of cells adjacent to a periodic face are generally distant.

When data concerning neighboring cells are required for the calculation, they must first be searched on the other processors or on the other edge of periodic frontiers. In order to ease the manipulation of these data, they are stored temporarily in virtual cells called “halo” cells, as can be seen in figure 4. It is in particular the case when the following operations are made on a variable *A*:

- calculation of the gradient of *A* (use of `grdcel`);
- calculation of an internal face value from the values of *A* in the neighboring cells (use of `ifacel`).

The variable *A* needs to be exchanged before these operations can be made: to allow it, the subroutines `parcom` and `percom` need to be called **in this order**.

- **Global operations in parallel mode.**

In parallel mode, the user must pay attention during the realisation of global operations. The following list is not exhaustive:

- calculation of extreme values on the domain (for instance, minimum and maximum of some calculation values);
- test of the existence of a certain value (for instance, do faces of a certain color exist ?);
- verification of a condition on the domain (for instance, is a given flow value reached somewhere ?);
- counting out of entities (for instance, how many cells have pressure drops ?);
- global sum (for instance, calculation of a mass flow or the total mass of a pollutant).

The user may refer to the different examples present in the user subroutine `usproj`.

Care should be taken with the fact that the boundaries between subdomains consist of **internal** faces shared between two processors (these are indeed internal faces, even if they are located at a “processor boundary”). They should not be counted twice (once per processor) during global operations using internal faces (for instance, counting the internal faces per processor and summing all the obtained numbers drives into overevaluating the number of internal faces of the initial mesh).

- **Writing; operations that should be made on one processor only in parallel mode.**

In parallel mode, the user must pay attention during the writing of pieces of information. Writing to the “listing” can be done simply by using the `nfecra` logical unit (each processor will write to its own “listing” file): use `write(nfecra,...`

If the user wants an operation to be done by only one processor (for example, open or write a file), the associated instructions must be included inside a test on the value of `irangp` (generally it is the processor 0 which realises these actions, and we want the subroutine to work in non-parallel mode, too: `if (irangp.le.0) then ...`).

## Some notes about periodicity

Note that periodic faces are not part of the domain boundary: periodicity is interpreted as a “geometric” condition rather than a classical boundary condition.

Some particular points should be reminded:

- Periodicity can also work when the periodic boundaries are meshed differently (periodicity of non-conforming faces), *apart* from the case of a 180 degree rotation periodicity with faces coupled on the rotation axis.
- rotation periodicity is incompatible with
  - semi-transparent radiation,
  - reinforced velocity-pressure coupling (ipucou=1).
- although it has not been the case so far, potential problems might be met in the case of rotation periodicity with the LRR  $R_{ij} - \varepsilon$  model. They would come from the way of taking into account the orthotropic viscosity (however, this term usually has a low influence).

### 3.9.2.7 Geometry and particule arrays related to Lagrangian modeling

In this section is given a non-exhaustive list of the main variables which may be seen by the user in the Lagrangian module. Most of them should not be modified by the user. They are calculated automatically from the data. However it may be useful to know their meaning.

These variables are listed in the alphabetical index in the end of this document.

The type of each variable is given: integer [i], real number [r], integer array [ia], real array [ra].

#### SIZE OF THE LAGRANGIAN ARRAYS

- lndnod** [i]: Size of the array **icocel** concerning the cells  $\rightarrow$  faces connectivity (the faces  $\rightarrow$  nodes connectivity needs to be given to allow the construction of this connectivity. See note 3 of section 3.9.2.1).
- nbpmax** [i]: Maximum number of particles simultaneously acceptable in the calculation domain.
- nvp** [i]: Number of variables describing the particles for which a stochastic differential equation (SDE) is solved.
- nvls** [i]: Number of variables describing the supplementary user particles for which a SDE is solved.
- nvep** [i]: Number of real state variables describing the particles.
- nivep** [i]: Number of integer state variables describing the particles.
- ntersl** [i]: Number of source terms representing the backward coupling of the dispersed phase on the continuous phase.
- nvlstata** [i]: Number of volumetric statistical variables .
- nvlstas** [i]: Number of supplementary user volumetric statistical variables.
- nvisbr** [i]: Number of boundary statistical variables.
- nusbor** [i]: Number of supplementary user boundary statistical variables.
- nvgaus** [i]: Number of gaussian random variables.

#### NOTE: CONTINUOUS EULERIAN PHASE NUMBER

The current version of Lagrangian module is planned to work with only one eulerian phase. This phase carries inclusions, and source terms of backward coupling are applied to it, if necessary.

## LAGRANGIAN ARRAYS

**icocel(lndnod)** [ia]: Cells *rightarrow* internal/boundary faces connectivity. The numbers of the boundary faces are marked out in **icocel** with a negative sign.

**itycel(ncelet+1)** [ia]: Array containing the position of the first face surrounding every cell in the array **icocel** (see subroutine **lagdeb** for more details).

**ettp(nbpmax,nvp)** [ra]: Variables forming the state vector related to the particles: either at the current stage if the Lagrangian scheme is a second-order, or at the current time step if the scheme is a first-order. These variables are marked out by “pointers” whose value can vary between 1 and **nvp**:

- **jmp**: particle mass
- **jdp**: particle diameter
- **jxp, jyp, jzp**: particle coordinates
- **jup, jvp, jwp**: particle velocity components
- **juf, jvf, jwf**: locally undisturbed fluid flow velocity components
- **jtp, jtf**: particle and locally undisturbed fluid flow temperature (°C)
- **jcp**: particle specific heat
- **jhp**: coal particle temperature (°C)
- **jmch**: mass of reactive coal of the coal particle
- **jmck**: mass of coke of the coal particle
- **jvls(ii)**: *iith* supplementary user variable

**ettpa(nbpmax,nvp)** [ra]: Variables forming the state vector related to the particles: either at the previous stage if the Lagrangian scheme is a second-order, or at the previous time step if the Lagrangian scheme is a first-order.

**itepa(nbpmax,nivep)** [ia]: Integer state variables related to the particles. They are marked out by the following “pointers”:

- **jisor**: Number of the current cell containing the particle; this number is reactualised during the trajectography step
- **jinch**: Number of the coal particle

**tepa(nbpmax,nvep)** [ra]: Real state variables related to the particles. They are marked out by the following “pointers”:

- **jrtsp**: particle residence time
- **jrpoi**: particle statistic weight
- **jrdck**: coal particle shrinking core diameter
- **jrd0p**: coal particle initial diameter

→ `jrr0p`: coal particle initial density

`indep(nbpmax)` [ia]: Storage of the cell number of every particle at the beginning of a Lagrangian iteration; this data is not modified during the iteration.

`vitpar(nbpmax,3)` [ra]: At the beginning of the trajectography, `vitpar` contains the particle velocity vector components; the modifications of the particle velocity following every particle/boundary interaction are saved in this array; after the trajectography and backward coupling steps, `ettp` is updated with `vitpar`.

`vitflu(nbpmax,3)` [ra]: At the beginning of the trajectography, `vitflu` contains the locally undisturbed fluid flow velocity vector components; the modifications of the locally undisturbed fluid flow velocity following every particle/boundary interaction are saved in this array; after the trajectography and backward coupling steps, `ettp` is updated with `vitflu`.

`gradpr(ncelet,3)` [ra]: Pressure gradient of the continuous phase.

`gradvf(ncelet,9)` [ra]: Gradient of the continuous phase fluid velocity (useful if the complete model is activated: see `modcpl`).

`cpgd1(nbpmax)` [ra]: First devolatilisation term (light volatile matters) of the coal particles (useful in the case of backward coupling on the continuous phase).

`cpgd2(nbpmax)` [ra]: Second devolatilisation term (heavy volatile matters) of the coal particles (useful in the case of backward coupling on the continuous phase).

`cpght(nbpmax)` [ra]: Heterogeneous combustion term of the coal particles (useful in the case of backward coupling on the continuous phase).

`statis(ncelet,nvlsta)` [ra]: Volumetric statistics related to the dispersed phase; these statistics are the kind of results expected with the Lagrangian module. It is from these statistics that we obtain information concerning the particle cloud (the particle trajectories should only be observed on “pedagogical” account); they are marked out by the following “pointers”:

→ `ilvx,ilvy,ilvz`: mean dispersed phase velocity

→ `ilvx2,ilvy2,ilvz2`: dispersed phase velocity standard deviation

→ `ilfv`: dispersed phase volumetric concentration

→ `ilpd`: sum of the statistical weights

→ `iltp`: dispersed phase temperature (°C)

→ `ildp`: dispersed phase mean diameter

→ `ilmp`: dispersed phase mean mass

→ `ilhp`: temperature of the coal particle cloud (°C)

→ `ilmch`: mass of reactive coal of the coal particle cloud

→ `ilmck`: mass of coke of the coal particle cloud

→ `ilmck`: shrinking core diameter of the coal particle cloud

→ `ilvu(ii)`: `iith` supplementary user volumetric statistics

`parbor(nfavor,nvisbr)` [ra]: Boundary statistics related the dispersed phase; after every particle/boundary interaction it is possible to save some data and to calculate averages; the boundary statistics are marked out by the following “pointers”:

- **inbr**: number of particle/boundary interactions
- **iflm**: particle mass flow at the boundary faces
- **iang**: mean interaction angle with the boundary faces (see example in **uslabo**)
- **ivit**: mean interaction velocity with the boundary faces
- **ienc**: mass of coal deposit at the walls
- **iusb(ii)**: *iith* supplementary user boundary statistics

**tslagr(ncelet,nters1)** [ra]: Source terms corresponding to the backward coupling of the dispersed phase on the continuous phase. These source terms are marked out by the following “pointers”:

- **itsvx**, **itsvy**, **itsvz**: explicit source terms for the continuous phase velocity
- **itsli**: implicit source term for the continuous phase velocity and for the turbulent energy if the  $k - \varepsilon$  model is used
- **itske**: explicit source term for the turbulent dissipation and the turbulent energy if the  $k - \varepsilon$  turbulence model is used for the continuous phase
- **itsr11**,... **itsr33**: source terms for the Reynolds stress and the turbulent dissipation if the  $R_{ij} - \varepsilon$  turbulence model is used for the continuous phase
- **itsmas**: mass source term
- **itste**, **itsti**: explicit and implicit thermal source terms for the thermal scalar of the continuous phase
- **itsmv1(icha)**, **itsmv2(icha)**: source terms respectively for the light and heavy volatile matters
- **itsco**: source term for the carbon released during heterogeneous combustion
- **itsf**: source term for the air variance (not used at the present time)

**croule(ncelet)** [ra]: Importance function for the technique of variance reduction (cloning/fusion of particles).

**vagaus(nbpmax,nvgaus)** [ra]: Vectors of gaussian random variables.

**aux1(nbpmax,3)** [ra]: Auxiliary work array.

### 3.9.2.8 Variables saved to allow calculation restarts

The directory **checkpoint** contains:

- **main**: main restart file,
- **auxiliary**: auxiliary restart file (see **ileaux**, **iecaux**),
- **radiative\_transfer**: restart file for the radiation module,
- **lagrangian**: main restart file for the Lagrangian module,
- **lagrangian\_stats**: auxiliary restart file for the Lagrangian module (mainly for the statistics),

- **1dwall\_module**: restart file for the 1D wall thermal module,
- **vortex**: restart file for the vortex method (see **ivrtex**).

The main restart file contains the values in every cell of the mesh for pressure, velocity, turbulence variables and scalars. Its content is sufficient for a calculation restart, but the complete continuity of the solution at restart is not ensured<sup>15</sup>.

The auxiliary restart file completes the main restart file to ensure solution continuity in the case of a calculation restart. If the code cannot find one or several pieces of data required for the calculation restart in the auxiliary restart file, default values are then used. This allows in particular to run calculation restarts even if the number of faces has been modified (for instance in case of modification of the mesh merging or of periodicity conditions<sup>16</sup>). More precisely, the auxiliary restart file contains the following data:

- type and value of the time step, turbulence model,
- density value at the cells and boundary faces, if it is variable,
- values at the cells of the other variable physical properties, when they are extrapolated in time (molecular dynamic viscosity, turbulent or subgrid scale viscosity, specific heat, scalar diffusivities); for the Joule effect, the specific heat is stored automatically (in case the user should need it at restart to calculate the temperature from the enthalpy before the new specific heat has been estimated),
- time step value at the cells, if it is variable,
- mass flow value at the internal and boundary faces (at the last time step, and also at the previous time step if required by the time scheme),
- boundary conditions,
- values at the cells of the source terms when they are extrapolated in time,
- number of time-averages, and values at the cells of the associated cumulated values,
- for each cell, distance to the wall when it is required (and index-number of the nearest boundary face, depending on **icdpar**),
- values at the cells of the external forces in balance with a part of the pressure (hydrostatic, in general),
- for the D3P gas combustion model: massic enthalpies and temperatures at entry, type of boundary zones and entry indicators,
- for the EBU gas combustion model: temperature of the fresh gas, constant mixing rate (for the models without mixing rate transport), types of boundary zones, entry indicators, temperatures and mixing rates at entry,
- for the LWC gas combustion model: the boundaries of the probability density functions for enthalpy and mixing rate, types of boundary zones, entry indicators, temperatures and mixing rates at entry,
- for the pulverised coal combustion: coal density, types of boundary zones, variables **ientat**, **ientcp**, **timpat**, **x20** (in case of coupling with the Lagrangian module, **iencp** and **x20** are not saved),

---

<sup>15</sup>in other words, a restart calculation of n time steps following a calculation of m time steps will not yield strictly the same results as a direct calculation on m+n time steps, whereas it is the case when the auxiliary file is used

<sup>16</sup>imposing a periodicity changes boundary faces into internal faces

- for the electric module: the tuned potential difference `dpot` and, for the electric arc module, the tuning coefficient `coejou` (when the boundary conditions are tuned), the Joule source term for the enthalpy (with the Joule effect is activated) and the Laplace forces (with the electric arc module).

It should be noted that, if the auxiliary restart file is read, it is possible to run calculation restarts with relaxation of the density<sup>17</sup> (when it is variable), because this variable is stored in the restart file. On the other hand, it is generally not possible to do the same with the other physical properties (they are stored in the restart file only when they are extrapolated in time, or with the Joule effect for the specific heat).

Apart from `vortex` which has a different structure and is always in text format, all the restart files are binary files. Nonetheless, they may be dumped or compared using the `cs_io_dump` tool.

In the case of parallel calculations, it should be noted that all the processors will write their restart data in the same files. Hence, for instance, there will always be one and only one `main` file, whatever the number of processors used. The data in the file are written according to the initial full domain ids for the cells, faces and nodes. This allows in particular to restart using  $p$  processors a calculation begun with  $n$  processors, or to make the restart files independent of any mesh renumbering that may be carried out in each domain.

*WARNING: if the mesh is composed of several files, the order in which they appear in the launch script or in the Graphical Interface must not be modified in case of a calculation restart<sup>18</sup>.*

*NOTE: when joining of faces or periodicity is used, two nodes closer than a certain (small) tolerance will be merged. Hence, due to numerical round-up errors, two different machines may yield different results. This might change the number of faces in the global domain<sup>19</sup> and make restart files incompatible. Should that problem arise when making a calculation restart on a different architecture, the solution is to ignore the `auxiliary` file and use only the `main` file, by setting `ileaux = 0` in `usini1.f90`*

### 3.9.3 Using selection criteria in user subroutines

In order to use selection criteria (cf. §3.10) in Fortran user subroutines, a collection of utility subroutines is provided. The aim is to define a subset of the mesh, for example:

- boundary regions (cf. `usclim`, `uscpcl`, `usray2`, `uslag2`,...),
- volumic initialization (cf. `usiniv`,...),
- head-loss region (cf. `uskpdc`),
- source terms region (cf. `ustsns`, `ustssc`),
- advanced post-processing (cf. `usdpst`), `usproj`, ...),

This section explains how to define surface or volume sections, in the form of lists `lstelt` of `nlelt` elements (internal faces, boundary faces or cells). For each type of element, the user calls the appropriate Fortran subroutine: `getfbr` for boundary faces, `getfac` for internal faces and `getcel` for cells. All of these take the three following arguments:

- the character string which contains the selection criterion (see some examples below),
- the returned number of elements `nlelt`,

<sup>17</sup>such a relaxation only makes sense for a stationary calculation

<sup>18</sup>when uncertain, the user can check the saved copy of the launch script in the `RESU` directory, or the head of the `preprocessor*.log` files, which repeat the command lines passed to the Preprocessor module

<sup>19</sup>the number of cells will not be modified, it is always the sum of the number of cells of the different meshes

- the returned list of elements `lstelt`.

Several examples of possible selections are given here:

- call `getfbr('Face_1, Face_2', nlelt, lstelt)` to select boundary faces in groups Face\_1 or Face\_2,
- call `getfac('4', nlelt, lstelt)` to select internal faces of color 4,
- call `getfac('not(4)', nlelt, lstelt)` to select internal faces which have a different color from 4,
- call `getfac('4 to 8', nlelt, lstelt)` to internal faces with color between 4 and 8 internal faces,
- call `getcel('1 or 2', nlelt, lstelt)` to select cells with colors 1 or 2,
- call `getfbr('1 and y > 0', nlelt, lstelt)` to select boundary faces of color 1 which have the coordinate  $Y > 0$ ,
- call `getfac('normal[1, 0, 0, 0.0001]', nlelt, lstelt)` to select internal faces which have a normal direction to the vector (1,0,0),
- call `getcel('all[]', nlelt, lstelt)` to select all cells.

The user may then use a loop on the selected elements. For instance, in the subroutine `usclim` used to impose boundary conditions, let us consider the boundary faces of color number 2 and which have the coordinate  $X \leq 0.01$  (so that call `getfbr('2 and x <= 0.01', nlelt, lstelt)`); we can do a loop (do `ilelt = 1, nlelt`) and obtain `ifac = lstelt(ilelt)`.

#### NOTE: LEGACY METHOD USING EXPLICIT FAMILIES AND PROPERTIES

The selection method for user subroutines by prior versions of *Code\_Saturne* is still available, though it may be removed in future versions. This method was better adapted to working with colors than with groups, and is explained here:

From *Code\_Saturne*'s point of view, all the references to mesh entities (boundary faces and volume elements) correspond to a number (color number or negative of group number) associated with the entity. An entity may have several references (for instance, one entity may have one color and belong to several groups). In *Code\_Saturne*, these references may be designated as "properties".

The mesh entities are gathered in equivalence classes on the base of their properties. These equivalence classes are called "families". All the entities of one family have the same properties. In order to know the properties (in particular the color) of an entity (a boundary face for example), the user must first determine the family to which it belongs.

For instance, let's consider a mesh whose boundary faces have all been given one color (for example using SIMAIL). The family of the boundary face `ifac` is `ifml=ifmfbr(ifac)`. The first (and only) property of this family is the color `icoul`, obtained for the face `ifac` with `icoul=iprfml(ifml,1)`. In order to know the property number corresponding to a group, the utility function `numgrp(nomgrp, lngnom)` (with a name `nomgrp` of the type `character*` and its lenght `lngnom` of the type `integer`) may be used.

## 3.10 Face and cell mesh-defined properties and selection

The mesh entities may be referenced by the user during the mesh creation. These references may then be used to mark out some mesh entities according to the need (specification of boundary conditions, pressure drop zones, ...). The references are generally of one of the two following types:



- color. A color is an integer possibly associated with boundary faces and volume elements by the mesh generator. Depending on the tool, this concept may have different names, which *Code\_Saturne* interprets as colors. Most tools allow only one color per face or element.
  - I-deas uses a color number with a default of 7 (green) for elements, be they volume elements or boundary “surface coating” elements. Color 11 (red) is used for vertices, but vertex properties are ignored by *Code\_Saturne*.
  - SIMAIL uses the equivalent notions of “reference” for element faces, and “subdomain” for volume elements. By default, element faces are assigned no reference (0), and volume elements domain 1.
  - Gmsh uses “physical property” numbers.
  - EnSight has no similar notion, but if several parts are present in an EnSight 6 file, or several parts are present *and* vertex ids are given in an EnSight Gold file, the part number is interpreted as a color number by the Preprocessor.
  - The Comet Design (pro-STAR/STAR4) and NUMECA Hex file formats have a CAD section id that is interpreted as a color number. In the latter case, this notion only applies to faces, so volume elements are given color.
  - The MED format allow integer “attributes”, though many tools working with this format ignore those and only handle groups.
- groups. Named “groups” of mesh entities may also be used with many mesh generators or formats. In some cases, a given cell or face may belong to multiple groups (as some tools allow new groups to be defined by boolean operations on existing groups). In *Code\_Saturne*, every group is assigned a group number (base on alphabetical ordering of groups).
  - I-deas assigns a group number with each group, but by default, this number is just a counter. Only the group name is considered by *Code\_Saturne* (so that elements belonging to two groups with identical names and different numbers are considered as belonging to the same group).
  - CGNS allows both for named boundary conditions and mesh sections. If present, boundary condition names are interpreted as group names, and groups may also be defined based on element section or zone names using additional Preprocessor options (**-grp-cel** or **-grp-fac** followed by **section** or **zone**).
  - Using the MED format, it is preferable to use “groups” to colors, as many tools ignore the latter.

Selection criteria may be defined in a similar fashion whether using the GUI or in user subroutines. Typically, a selection criteria is simply a string containing the required color numbers or group names, possibly combined using boolean expressions. Simple geometric criteria are also possible.

A few examples are given below:

```
ENTRY
1 or 7
all[]
3.1 >= z >= -2 or not (15 or entry)
range[04, 13, attribute]
sphere[0, 0, 0, 2] and (not no_group[])
```

Strings such as group names containing whitespace or having names similar to reserved operators may be protected using “escape characters”.<sup>20</sup> More complex examples of strings with protected strings are given here:

---

<sup>20</sup>Note that for defining a string in Fortran, double quotes are easier to use, as they do not conflict with Fortran's single quotes delimiting a string. In C, the converse is true. Also, in C, to define a string such as `\plane`, the string `\\plane` must be used, as the first `\` character is used by the compiler itself. Using the GUI, either notation is easy.

"First entry" or Wall\ or\ sym  
entry or \plane or "noone's output"

The following operators and syntaxes are allowed (fully capitalized versions of keywords are also allowed, but mixed capitals/lowercase versions are not):

#### escape characters

protect next character only: \

protect string: 'string' "string"

#### basic operators

priority: ( )

not: ! !=

and: & &&

or: | || , ;

xor: ^

#### general functions

select all: all[]

entities having no group or color: no\_group[]

select a range of groups or colors: range[first, last]

range[first, last, group]

range[first, last, attribute]

For the range operator, *first* and *last* values are inclusive. For attribute (color) numbers, natural integer value ordering is used, while for group names, alphabetical ordering is used. Note also that in the bizarre (not recommended) case in which a mesh would contain for example both a color number 15 and a group named "15", using range[15, 15, group] or range[15, 15, attribute] could be used to distinguish the two.

Geometric functions are also available. The coordinates considered are those of the cell or face centers. Normals are of course usable only for face selections, not cell selections.

#### geometric functions

face normals: normal[x, y, z, epsilon]

normal[x, y, z, epsilon = epsilon]

plane,  $ax + by + cz + d = 0$  form: plane[a, b, c, d, epsilon]

plane[a, b, c, d, epsilon = epsilon]

plane[a, b, c, d, inside]

plane[a, b, c, d, outside]

plane, normal + point in plane form: plane[n<sub>x</sub>, n<sub>y</sub>, n<sub>z</sub>, x, y, z, epsilon]

plane[n<sub>x</sub>, n<sub>y</sub>, n<sub>z</sub>, x, y, z, epsilon = epsilon]

plane[n<sub>x</sub>, n<sub>y</sub>, n<sub>z</sub>, x, y, z, inside]

plane[n<sub>x</sub>, n<sub>y</sub>, n<sub>z</sub>, x, y, z, outside]

box, extents form: box[x<sub>min</sub>, y<sub>min</sub>, z<sub>min</sub>, x<sub>max</sub>, y<sub>max</sub>, z<sub>max</sub>]

box, origin + axes form: box[x<sub>0</sub>, y<sub>0</sub>, z<sub>0</sub>,

dx<sub>1</sub>, dy<sub>1</sub>, dz<sub>1</sub>, dx<sub>2</sub>, dy<sub>2</sub>, dz<sub>2</sub>, dx<sub>3</sub>, dy<sub>3</sub>, dz<sub>3</sub>]

plane[a, b, c, d, epsilon = epsilon]

plane[a, b, c, d, inside]

plane[a, b, c, d, outside]

cylinder: cylinder[x<sub>0</sub>, y<sub>0</sub>, z<sub>0</sub>, x<sub>1</sub>, y<sub>1</sub>, z<sub>1</sub>, radius]

sphere: sphere[x<sub>c</sub>, y<sub>c</sub>, z<sub>c</sub>, radius]

inequalities: >, <, >=, <= associated with x, y, z or X, Y, Z keywords and coordinate value;

x<sub>min</sub> <= x < x<sub>max</sub> type syntax is allowed.

In the current version of Code\_Saturne, all selection criteria used are maintained in a list, so that re-interpreting a criterion already encountered (such as at the previous time step) is avoided. Lists of entities corresponding to a criteria containing no geometric functions are also saved in a compact

manner, so re-using a previously used selection should be very fast. For criteria containing geometric functions, the full list of corresponding entities is not maintained, so each entity must be compared to the criterion at each time step. Heavy use of many selection criteria containing geometric functions may thus lead to reduced performance.

## 4 Importing and Preprocessing Meshes

Importing and preprocessing meshes is done both by the Preprocessor module, which is used to import meshes, and using preprocessing functions of the code Kernel.

The Preprocessor module of *Code\_Saturne* reads the mesh file(s) (under any supported format) and translates the necessary information into a Kernel input file.

When multiple meshes are used, the Preprocessor is called once per mesh, and each resulting output is added in a `mesh_input` directory (instead of a single `mesh_input` file).

The executable of the Preprocessor module is `cs_preprocess`, and the most useful options and sub-options are described briefly here. To obtain a complete and up-to-date list of options and environment variables, use the following command: `cs_preprocess -h` or `cs_preprocess --help`. Many options, such as this one, accept a short and a long version.

For the main options, the launch script `runcase` contains corresponding variables, that are used to define options for the Preprocessor. This way, the user only has to define these variables and does not detailed knowledge of the Preprocessor command line.

Nonetheless, it may be useful to call the Preprocessor manually in certain situations, especially for frequent verification when building a mesh, so its use is described here. Verification may also be done using the GUI or the mesh quality check mode of the general run script.

The Preprocessor is controlled using command-line arguments. A few environment variables allow an expert user to modify some behaviors or to obtain a trace of memory management.

### 4.1 Preprocessor options

Main choices are done using command-line options. For example:

```
cs_preprocess --num 2 fluid.med
```

means that we read the second mesh defined in the `fluid.med` file, while:

```
cs_preprocess --no-write --post-volume fluid.med fluid.msh
```

means that we read file `fluid.msh`, and do not produce a `mesh_input` file, but do output a `fluid.med` file (effectively converting a Gmsh file to a MED file).

#### 4.1.1 Mesh selection

Any use of the preprocessor requires one mesh file (except for `cs_preprocess` and `cs_preprocess -h` which respectively print the version number and list of options). This file is selected as the last argument to `cs_preprocess`, and its format is usually automatically determined based on its extension (c.f. 3.4.1 page 18) but a `--format` option allows forcing the format choice of the selected file.

For formats allowing multiple meshes in a single file, the `--num` option followed by a strictly positive integer allows selection of a specific mesh; by default, the first mesh is selected.

For meshes in CGNS format, we may in addition use the `--grp-cel` or `--grp-fac` options, followed by the `section` or `zone` keywords, to define additional groups of cell or faces based on the organization of the mesh in sections or zones. The sub-options have no effect on meshes of other formats.

### 4.1.2 Post-processing output

By default, the Preprocessor does not generate any post-processor output. By adding `--post-volume [format]`, with the optional `format` argument being one of `ensight`, `med`, or `cgns` to the command-line arguments, the output of the volume mesh to the default or indicated format is provoked.

In case of errors, output of error visualization output is always produced, and by adding `--post-error [format]`, the format of that output may be selected (from one of `ensight`, `med`, or `cgns`, assuming MED and CGNS are available),

### 4.1.3 Element orientation correction

We may activate the possible element orientation correction using the `--reorient` option.

Note that we cannot guarantee correction (or even detection) of a bad orientation in all cases. Not all local numbering possibilities of elements are tested, as we focus on “usual” numbering permutations. Moreover, the algorithms used may produce false positives or fail to find a correct renumbering in the case of highly non convex elements. In this case, nothing may be done short of modifying the mesh, as without a convexity hypothesis, it is not always possible to choose between two possible definitions starting from a point set.

With a post-processing option such as `--post-error` or, `--post-volume`, visualizable meshes of corrected elements as well as remaining badly oriented elements are generated.

## 4.2 Environment variables

Setting a few environment variables specific to the Preprocessor allows modifying its default behavior. In general, if a given behavior is modifiable through an environment variable rather than by a command-line option, it has little interest for a non-developer, or its modification is potentially hazardous. The environment variables used by the Preprocessor are described here:

#### CS\_PREPROCESS\_MEM\_LOG

Allows defining a file name in which memory allocation, reallocation, and freeing is logged.

#### CS\_PREPROCESS\_MIN\_EDGE\_LEN

Under the indicated length ( $10^{-15}$  by default), an edge is considered to be degenerate and its vertices will be merged after the transformation to descending connectivity. Degenerate edges and faces will thus be removed. Hence, the post-processed element does not change, but the Kernel may handle a prism where the preprocessor input contained a hexahedron with two identical vertex couples (and thus a face of zero surface). If the Preprocessor does not print any information relative to this type of correction, it means that it has not been necessary. To completely deactivate this automatic correction, a negative value may be assigned to this environment variable.

#### CS\_PREPROCESS\_IGNORE\_IDEAS\_COO\_SYS

If this variable is defined and is a strictly positive integer, coordinate systems in I-deas universal format files will be ignored. The behavior of the Preprocessor will thus be the same as that of versions 1.0 and 1.1. Note that in any case, non Cartesian coordinate systems are not handled yet.

### 4.2.1 System environment variables

Some system environment variables may also modify the behavior of the Preprocessor. For example, if the Preprocessor was compiled with MED support on an architecture allowing shared (dynamic) libraries, the `LD_PRELOAD` environment variable may be used to define a “priority” path to load MED or HDF5 libraries, and thus experiment with another version of these libraries without recompiling

the Preprocessor. To determine which shared libraries are used by an executable file, use the following command: `ldd {executable_path}`.

## 4.3 Optional functionality

Some functions of the Preprocessor are based on external libraries, which may not always be available. It is thus possible to configure and compile the Preprocessor so as not to use these libraries. When running the Preprocessor, the supported options are printed. The following optional libraries may be used:

- CGNS library. In its absence, [CGNS](#) format support is deactivated.
- MED-file library. In its absence, [MED](#) format is simply deactivated.
- Read compressed files using Zlib. With this option, it is possible to directly read mesh files compressed with a *gzip* type algorithm and bearing a *.gz* extension. This is limited to formats not already based on an external library (i.e. it is not usable with CGNS or MED files), and has memory and CPU time overhead, but may be practical. Without this library, files must be uncompressed before use.

## 4.4 General remarks

Note that the Preprocessor is in general capable of reading all “classical” element types present in mesh files (triangles, quadrangles, tetrahedra, pyramids, prisms, and hexahedra). Quadratic or cubic elements are converted upon reading into their linear counterparts. Vertices referenced by no element (isolated vertices or centers of higher-degree elements) are discarded. Meshes are read in the order defined by the user and are appended, vertex and element indices being incremented appropriately.<sup>21</sup>

At this stage, volume elements are sorted by type, and the fluid domain post-processing output is generated if required.

In general, groups assigned to vertices are ignored. selections are thus based on faces or cells. with tools such as SIMAIL, faces of volume elements may be referenced directly, while with I-deas or SALOME, a layer of surface elements bearing the required colors and groups must be added. Internally, the Preprocessor always considers that a layer of surface elements is added (i.e. when reading a SIMAIL mesh, additional faces are generated to bear cell face colors. When building the *faces* → *cells* connectivity, all faces with the same topology are merged: the initial presence of two layers of identical surface elements belonging to different groups would thus lead to a calculation mesh with faces belonging to two groups.

## 4.5 Files passed to the Kernel

Data passed to the Kernel by the Preprocessor is transmitted using a binary file, using “big endian” data representation, named `mesh_input` (or contained in a directory of that name).

When using the Preprocessor for mesh verification, data for the Kernel is not always needed. In this case, the `--no-write` option may be avoid creating a Preprocessor output file.

## 4.6 Mesh preprocessing

### 4.6.1 Joining of non-conforming meshes

Conforming joining of possibly non-conforming meshes may be done by the solver, and defined either using the Graphical User Interface (GUI) or the `cs_user_join` user function. In the GUI, the user needs

<sup>21</sup>Possible entity labels are not maintained, as they would probably not be unique when appending multiple meshes.

to add entries in the “Face joining” section of the “Meshes” tab in the item “Calculation environment → Meshes selection”. The user may specify faces to be joined, and can also modify basic joining parameters, see fig. 6. For a simple mesh, it is rarely useful to specify strict face selection criteria, as

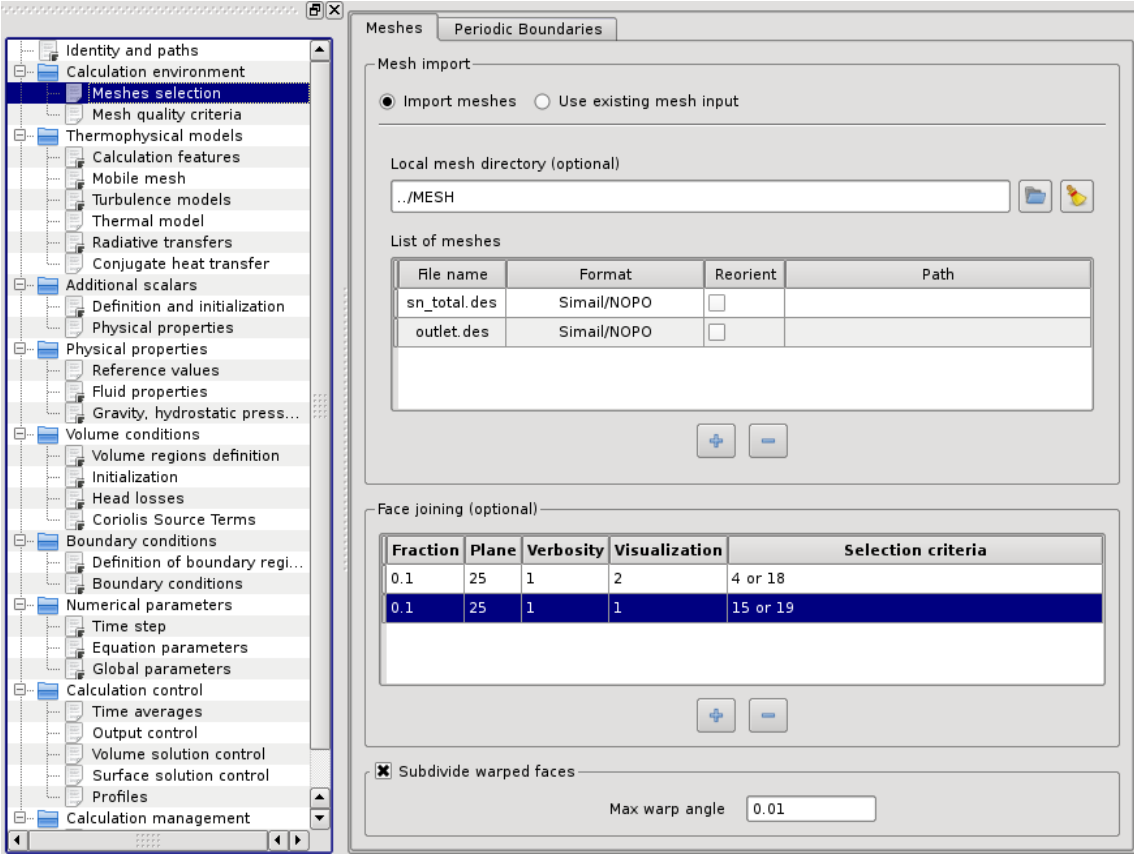


Figure 6: Conformal or non-conformal joining

joining is sufficiently automated to detect which faces may actually be joined. For a more complex mesh, or a mesh with thin walls which we want to avoid transforming into interior faces, it is recommended to filter boundary faces that may be joined by using face selection criteria. This has the additional advantage of reducing the number of faces to test for in the intersection/overlap search, and thus reduced to time required by the joining algorithm.

One may also modify tolerance criteria using 2 options:

- fraction  $r$**  assigns value  $r$  (where  $0 < r < 0.49$ ) to the maximum intersection distance multiplier (0,1 by default). The maximum intersection distance for a given vertex is based on the length of the shortest incident edge, multiplied by  $r$ . The maximum intersection at a given point along an edge is interpolated from that at its vertices, as shown on the left of figure 7;
- lane  $c$**  assigns the maximum angle between normals for two faces to be considered coplanar ( $25^\circ$  by default); this parameter is used in the second stage of the algorithm, to reconstruct conforming faces, as shown on the right of figure 7.

In practice, we are sometimes led to increase the maximum intersection distance multiplier to 0.2 or even 0.3 when joining curved surfaces, so that all intersection are detected. As this influences merging of vertices and thus simplification of reconstructed faces, but also deformation of “lateral” faces, it is recommended only to modify it if necessary. As for the **plane** parameter, its use has only been necessary on a few meshes up to now, and always in the sense of reducing the tolerance so that face

reconstruction does not try to generate faces from initial faces on different surfaces.

## 4.6.2 Periodicity

Handling of periodicity is based on an extension of conforming joining, as shown on figure 8. It is thus not necessary for the periodic faces to be conforming (though it usually leads to better mesh quality). All options relative to conforming joining of non-conforming faces also apply to periodicity. Note also that once pre-processed, 2 periodic faces have the same orientation (possibly adjusted by periodicity of rotation).

This operation can also be performed by the solver and specified either using the GUI or the `cs_user_periodicity` function.

As with joining, it is recommended to filter boundary faces to process using a selection criterion. As many periodicities may be built as desired, as long as boundary faces are present. Once a periodicity is handled, faces having periodic matches do not appear as boundary faces, but as interior faces, and are thus not available anymore for other periodicities.

## 4.6.3 Parameters for conforming or non-conforming mesh joinings

The setting of these parameters is done in the user subroutine `cs_user_join` (called once). The user can specify the parameters used for the joining of different meshes. Below is given the list of the standard parameters which can be modified:

- **fract**: the initial tolerance radius associated to each vertex is equal to the length of the shortest incident edge, multiplied by this fraction,
- **plane**: when subdividing faces, 2 faces are considered coplanar and may be joined if the angle between their unit normals (in degree) does not exceed this parameter,
- **iwarnj**: the associated verbosity level (debug level if over 3).

In the call of the function `cs_join_add`, a selection criteria for mesh faces to be joined is specified. The list of advanced modifiable parameters is given below:

- **mtf**: a merge tolerance factor, used to locally modify the tolerance associated to each vertex before the merge step. Depending on its value four scenarios are possible:
  - if  $mtf = 0$ , no vertex merge
  - if  $mtf < 1$ , the vertex merge is more strict. It may increase the number of tolerance reduction and therefore define smaller subset of vertices to merge together but it can drive to loose intersections.
  - if  $mtf = 1$ , no change occurs
  - if  $mtf > 1$ , the vertex merge is less strict. The subset of vertices able to merge is greater.
- **pmf**: a pre-merge factor. This parameter is used to define a limit under which two vertices are merged before the merge step,
- **tcm**: a tolerance computation mode. If its value is:
  - 1 (default), the tolerance is the minimal edge length related to a vertex, multiplied by a fraction.
  - 2, the tolerance is computed like for 1 with, in addition, the multiplication by a coefficient equal to the maximum between  $\sin(e1)$  and  $\sin(e2)$ ; where  $e1$  and  $e2$  are two edges sharing the same vertex  $V$  for which we want to compute the tolerance.

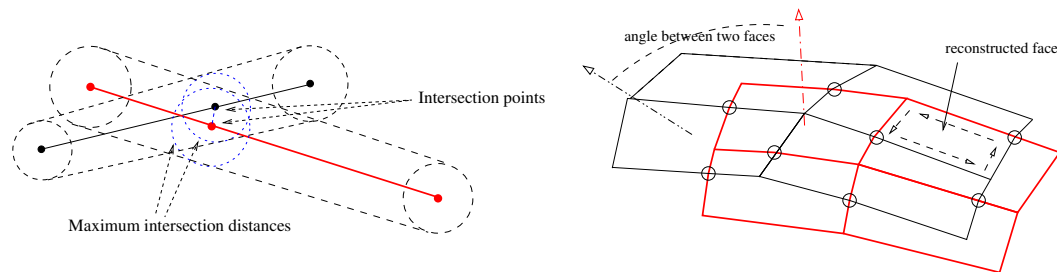


Figure 7: Maximum intersection tolerance and faces normal angle

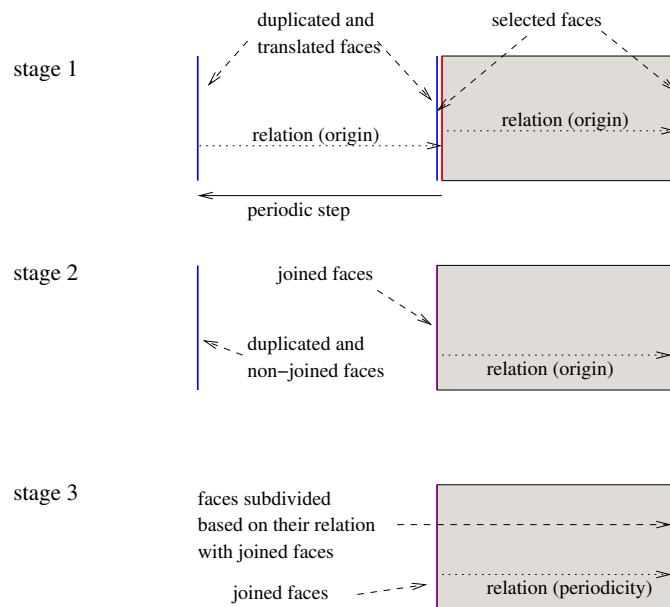


Figure 8: Matching of periodic faces



- 11, it is the same as 1 but taking into account only the selected faces.
- 12, it is the same as 2 but taking into account only the selected faces.
- **icm**: the intersection computation mode. If its value is:
  - 1 (default), the original algorithm is used. Care should taken to clip the intersection on an extremity.
  - 2, a new intersection algorithm is used. Caution should be used to avoid to clip the intersection on an extremity.
- **maxbrk**: defines the maximum number of equivalence breaks which is enabled during the merge step,
- **maxsf**: defines the maximum number of sub-faces used when splitting a selected face

The followings are advanced parameters used in the search algorithm for face intersections between selected faces (octree structure). They are useful in case of memory limitation:

- **tml**: the tree maximum level is the deepest level reachable during the tree building,
- **tmb**: the tree maximum boxes is the maximum number of bounding boxes (BB) which can be linked to a leaf of the tree (not necessary true for the deepest level),
- **tmr**: the tree maximum ratio. The building of the tree structure stops when the number of bounding boxes is superior to the product of **tmr** with the number of faces to locate. This is an efficient parameter to reduce memory consumption.

The call to the subroutine 'setajp' returns the value of these parameters.

#### 4.6.4 Parameters for the periodicity

Periodicities can be set directly in the Graphical User Interface (GUI) or using the user subroutine **cs\_user\_periodicity** (called when once during the calculation initialisation). In the GUI, the user can choose between 3 types of periodicities: translation, rotation, or mixed (see fig. 9). Then specific parameters must be set.

**cs\_user\_periodicity** can be used instead of the GUI, it allows also the user to specify the parameters used to set periodicities and gives access to more advanced parameters. Below is given the list of the main parameters which can be modified:

- **fract**: the initial tolerance radius associated to each vertex is equal to the length of the shortest incident edge, multiplied by this fraction,
- **plane**. When subdividing faces, 2 faces are considered as coplanar and may be joined if the angle between their unit normals (in degree) does not exceed this parameter,
- **iwarnj**: the associated verbosity level (debug level if over 3).

The second part of the subroutine is used to define the periodic transformations. The user provides in the subroutine 'defpro' the reference of the mesh the transformation applies to, as well as:

- the translation vector, if a periodicity of translation is used,
- the axis, the angle of rotation, and an invariant point if a periodicity of rotation is used,
- an homogeneous matrix if a general transformation is used.

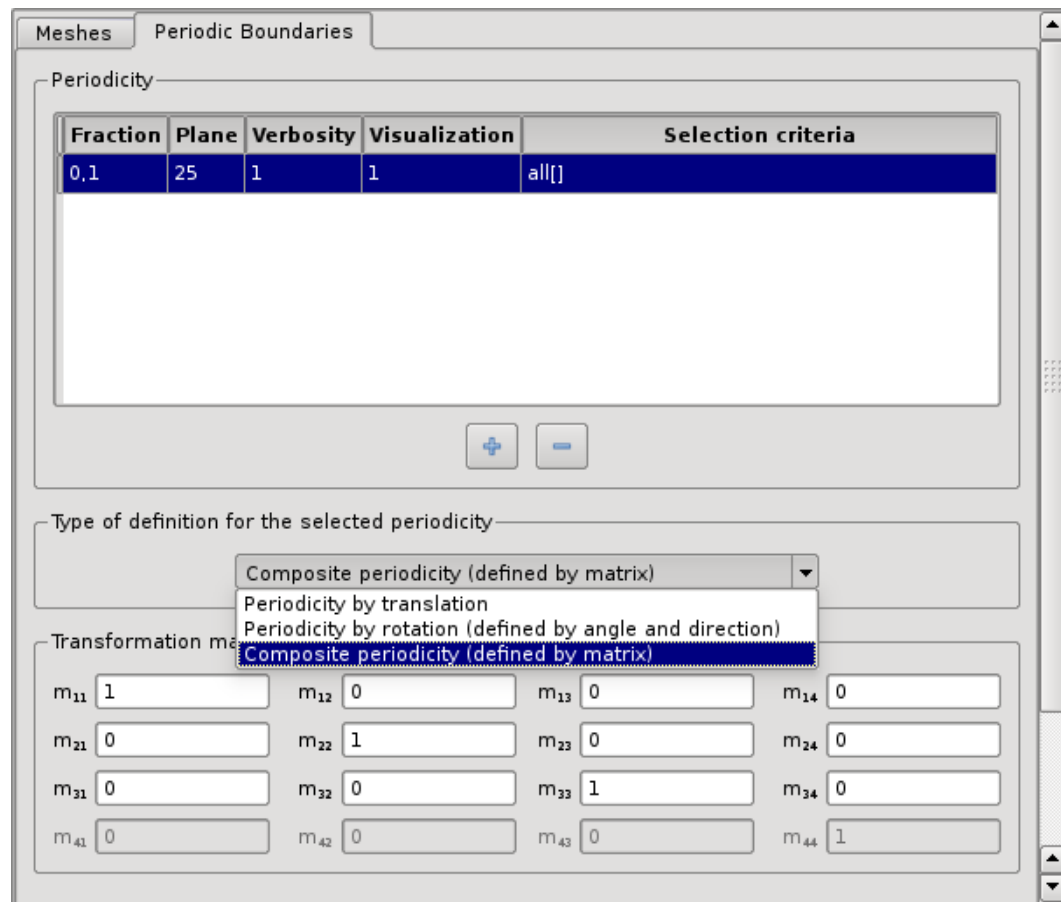


Figure 9: Periodicity

In addition, the user can modify advanced parameters in case problems occur during the joining step, or to get a better mesh quality:

- **mtf**: a merge tolerance factor, used to locally modify the tolerance associated to each vertex before the merge step. Depending on its value four scenarios are possible:
  - if  $mtf = 0$ , there is no vertex merge.
  - if  $mtf < 1$ , the vertex merge is more strict. It may increase the number of tolerance reduction and therefore define smaller subset of vertices to merge together, but it can drive to loose intersections.
  - if  $mtf = 1$ , no changes occur.
  - if  $mtf > 1$ , the vertex merge is less strict. The subset of vertices able to merge is greater.
- **pmf**: a pre-merge factor. This parameter is used to define a limit under which two vertices are merged before the merge step,
- **tcm**: a tolerance computation mode. If its value is:
  - 1 (default), the tolerance is the minimal edge length related to a vertex, multiplied by a fraction.
  - 2, the tolerance is computed like for 1 with, in addition, the multiplication by a coefficient equal to the maximum between  $\sin(e1)$  and  $\sin(e2)$ , where  $e1$  and  $e2$  are two edges sharing the same vertex  $V$  for which we want to compute the tolerance.

- 11, it is the same as 1 but taking into account only the selected faces.
- 12, it is the same as 2 but taking into account only the selected faces.
- **icm**: the intersection computation mode. If its value is:
  - 1 (default), the original algorithm is used. Care should taken to clip the intersection on an extremity.
  - 2, a new intersection algorithm is used. Caution should be used to avoid to clip the intersection on an extremity.
- **maxbrk**: defines the maximum number of equivalence breaks which are enabled during the merge step,
- **maxsf**: defines the maximum number of sub-faces used when splitting a selected face

The following are advanced parameters used in the search algorithm for face intersections between selected faces (octree structure). There are useful in case of memory limitation:

- **tml**: the tree maximum level is the deepest level reachable during the tree building
- **tmb**: the tree maximum boxes is the maximum number of bounding boxes (BB) which can be linked to a leaf of the tree (not necessary true for the deepest level)
- **tmr**: the tree maximum ratio. The building of the tree structure stops when the number of bounding boxes is superior than the product of **tmr** with the number of faces to locate. This is an efficient parameter to reduce memory consumption.

The call to the routine '**setapp**' returns the value of these parameters.

#### 4.6.5 Modification of the mesh geometry

*Subroutines called only during the calculation initialisation.*

The user subroutine **cs\_user\_mesh\_input** allows a detailed selection of imported meshes read, reading files multiple times, applying geometric transformations, and renaming groups.

The user subroutine **cs\_user\_mesh\_modify** may be used for advanced modification of the main **cs\_mesh\_t** structure.

*WARNING: Caution must be exercised when using this subroutine along with periodicity. Indeed, the periodicity parameters are not updated accordingly, meaning that the periodicity may be unadapted after one changes the mesh vertex coordinates. It is particularly true when one rescales the mesh. Rescaling should thus be done in a separate run, before defining periodicity.*

The user subroutine **cs\_user\_mesh\_thinwall** allows insertion of thin walls in the calculation mesh. Faces on each side of a thin wall will share the same vertices, so postprocessing of the main volume mesh may not show the inserted walls, though they will appear in the main boundary output mesh may be used to modify "manually" the mesh vertices coordinates, *i.e.* the array **mesh->vtx\_coord[]**.

## 5 Partitioning for parallel runs

Graph partitioning (using one of the optional METIS or SCOTCH libraries) is done using a secondary executable,

**cs\_partition**, which reads the file produced by the Preprocessor and builds one or several "cell → domain" distribution files, named **domain\_number.p** for a partitioning on *p* sub-domains.

This separation leads to extra work for the Kernel, which must redistribute data read in `mesh_input` based on the associated partitioning, but avoids requiring re-running the Preprocessor whenever running on a different number of processors.

Without partitioning (for example if neither METIS nor SCOTCH is available, or the partitioner has not been run for the required number of sub-domains), the Kernel will use a built-in partitioning using a space-filling curve (Z-curve) technique. This usually leads to partitionings of lower quality than with graph partitioning, but parallel performance remains reasonable.

## 5.1 Options

To list the partitioner's options, use the following command: `cs_partition -h`

We provide the list of required partitionings and optionally additional options. For example, to simulate a partitioning for calculations on 64 and 128 processes with no output, we may use the following command:

```
cs_partition 64 128 --no-write
```

### 5.1.1 Ignore periodicity

By default, face periodicity relations are taken into account when building the “cell → cell” connectivity graph used for partitioning. This allows better partitioning optimization, but increases the probability of having groups of cells at opposite sides of the domain in a same sub-domain. This is not an issue for standard calculations, but may degrade performance of search algorithms based on bounding boxes. It is thus possible to ignore periodicity when partitioning a mesh using the `--no-perio` option.

Note that nothing guarantees that a graph partitioner will not place disjoint cells in the same sub-domain independently of this option, but this behavior is rare.

### 5.1.2 Partitioner choice

If the Partitioner has been configured with both METIS and SCOTCH libraries, using the `--metis` or `--scotch` option allows choosing between either library. By default, *metis* is used if both choices are available.

### 5.1.3 Simulation mode

Using the `--no-write` option, we can tell the partitioner not to output a `domain_number_p` file. Partitioning is thus computed, but not saved.

### 5.1.4 Environment variables

`CS_PARTITION_MEM_LOG`

Allows defining a file name in which memory allocations, reallocations, and frees will be logged.

## 6 Basic modelling setup

### 6.1 Initialisation of the main parameters

This operation is done in the Graphical User Interface (GUI) or by using the user subroutine `usini1`. In the GUI, the initialisation is performed by filling the parameters displayed in Figs. 10 to ?? . If

the option 'Mobile mesh' is activated in Fig. 10, please see Section 8.11.4 for more details. In fig. 13, the equivalent initialisations occur in the subroutine `usiniv` when the GUI is not used. The headings filled for the initialisation are the followings:

- Thermophysical model options: ALE mobile mesh, turbulence model, thermal model, see figs. 10 to 12.
- Additional scalars: definition, initialisation of the scalars, and physical characteristics, see figs. 13 and 14. In fig. 14, the initial values are given in the subroutine `usiniv` if the GUI is not used, see Section 6.3.
- Physical properties: reference pressure, fluid characteristics, gravity, see figs. 15 to 17. If non-constant values are used for the fluid properties, and if the GUI is not used, see Section 6.5.1.
- Numerical parameters: number and type of time steps, and advanced parameters for the numerical solution of the equations, see figs. 18 to 20.
- Calculation control: parameters related to the time averages, the time step, the locations of the probes where some variables will be monitored over time (if the GUI is not used, this information is specified in Section 6.3), the definition of the frequency of the outputs in the calculation listing, the postprocessing output writer frequency and format options, and the postprocessing output meshes and variables selection, see figs. 21, 22, 23, and 24. The item "Profiles" allows to save, with a frequency defined by the user, 1D profiles on an axis defined by two points, see fig. 25.

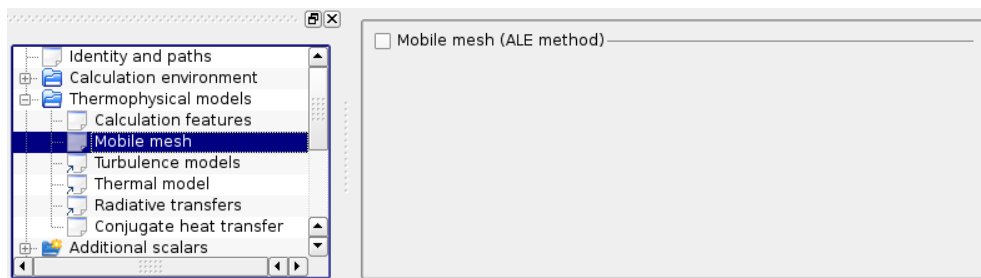


Figure 10: Mobile mesh option

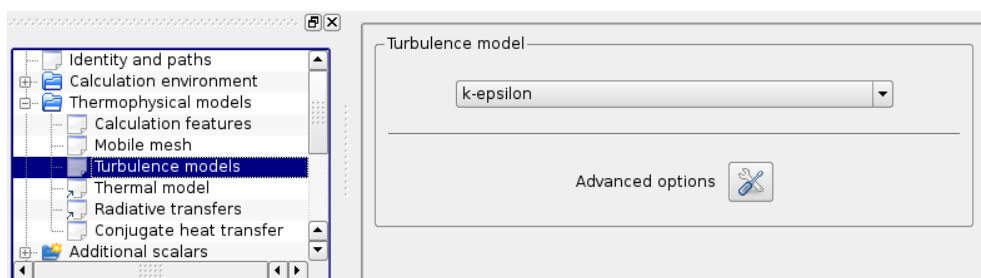


Figure 11: Turbulence model selection

In the case of a calculation launched using the interface, the subroutine `usini1` is only used to modify high-level parameters which can not be managed by the interface. In the case of a code utilisation without interface, this subroutine is compulsory and all the headings must be completed. `usini1` is used to indicate the value of different calculation basic parameters: constant and uniform physical values, parameters of numerical schemes, input-output management... It is called only during the calculation initialisation.

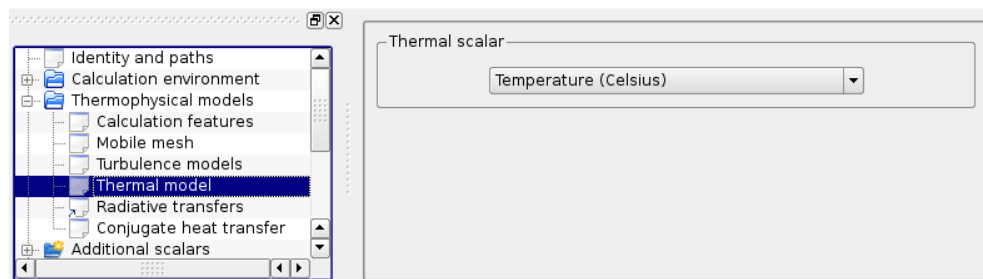


Figure 12: Thermal scalar selection

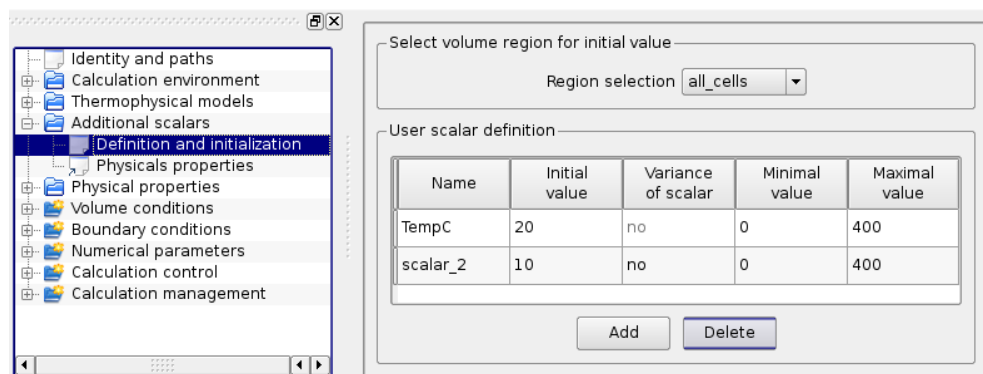


Figure 13: Definition and initialisation of the scalars

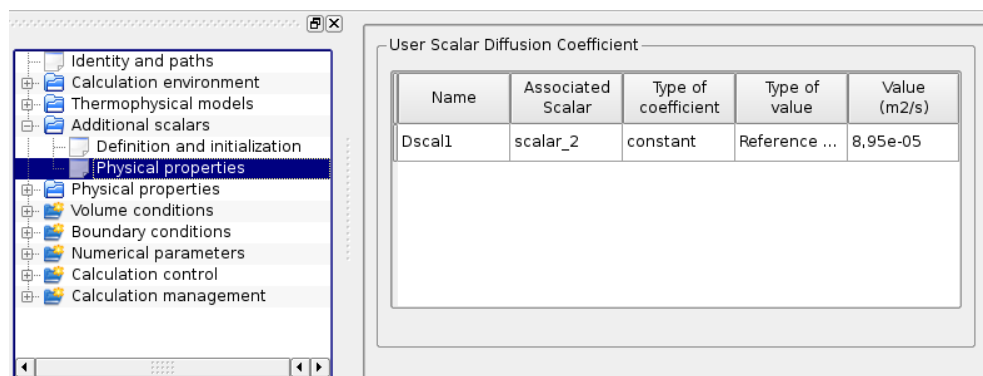


Figure 14: Associated physical properties of the scalars

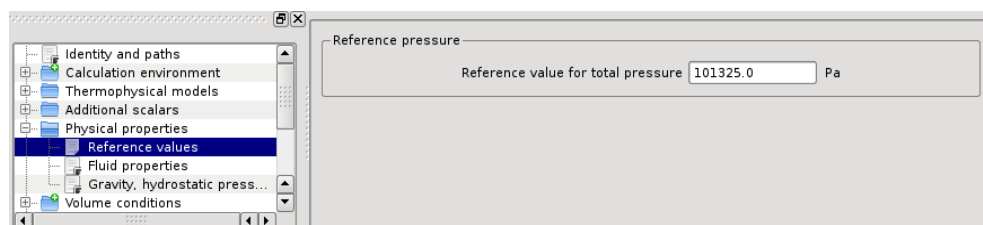


Figure 15: Setting of the reference pressure

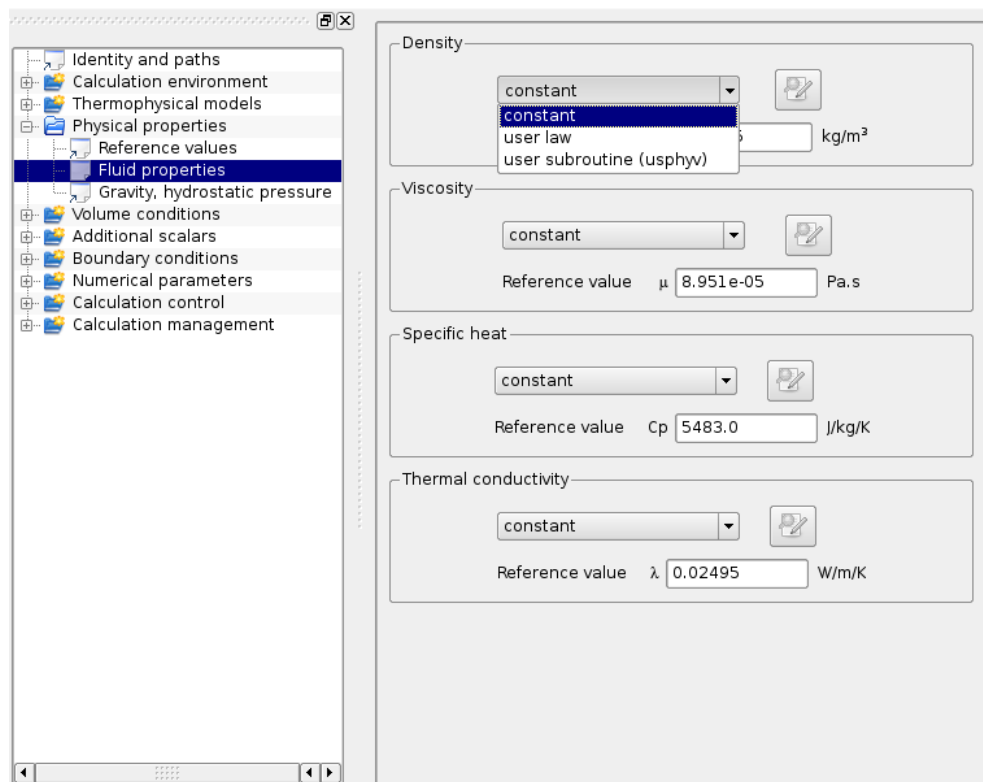


Figure 16: Fluid properties

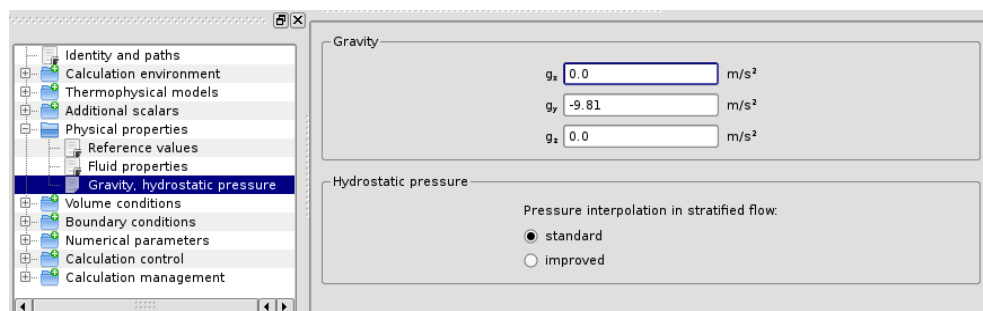


Figure 17: Settings of the gravity and of the hydrostatic pressure

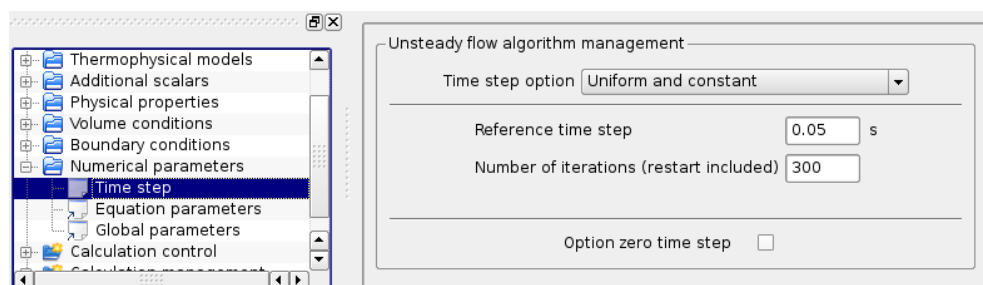


Figure 18: Time step settings



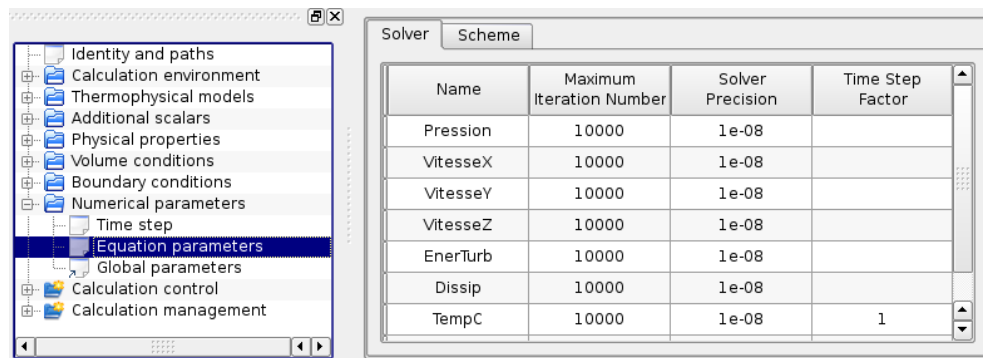


Figure 19: Numerical parameters for the main variables resolution

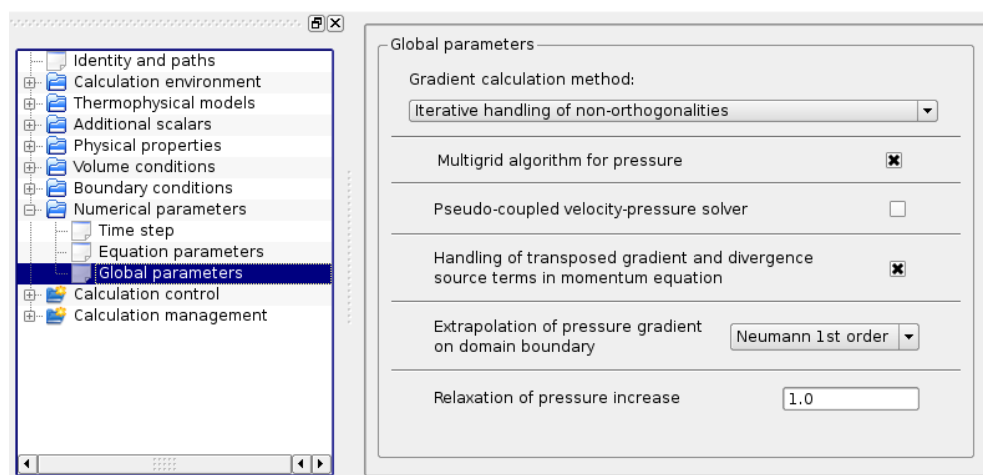


Figure 20: Global resolution parameters

For more details about the different parameters, please refer to the key word list (§9).

`usini1` is in fact constituted of 6 separate subroutines: `usipph`, `usinsc`, `usipsc`, `usipgl`, `usipsu` and `usipes`. Each one controls various specific parameters. The key words which are not featured in the supplied example can be provided by the user in `SRC/REFERENCE/base`; in this case, understanding of the comments is required to add the key words in the appropriate subroutine, it will ensure that the value has been well defined). The modifiable parameters in each of the subroutines of `usini1` are:

- `usipph`: `iturb` and `icp` (don't modify these parameters anywhere else)
- `usinsc`: `nscaus` (don't modify these parameters anywhere else)
- `usipsc`: `iscavr` and `ivisls` (don't modify these parameters anywhere else)
- `usipgl`: `idtvvar`, `ipucou`, `iphydr` and the parameters related to the error estimators (don't modify these parameters anywhere else).
- `usipsu`: physical parameters of the calculation (thermal scalar, physical properties,...), numerical parameters (time steps, number of iterations, ...), definition of the time averages.
- `usipes`: post-processing output parameters (periodicity, variable names, probe positions,...)

For more details on the different parameters, see the list of key words (§9). The names of the key words can also be seen in the help sections of the interface.

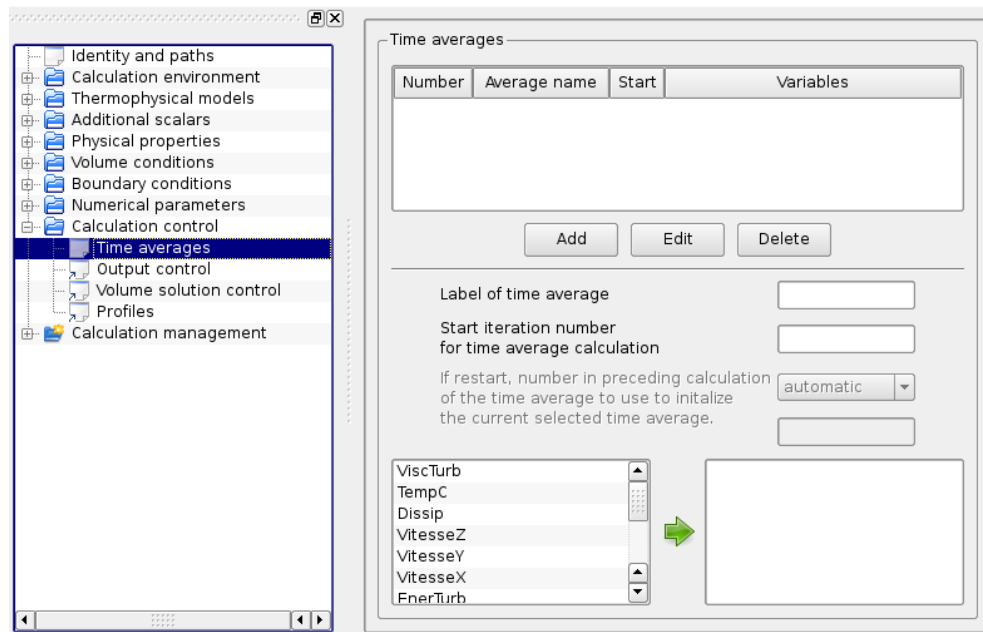


Figure 21: Management of time averaged variables

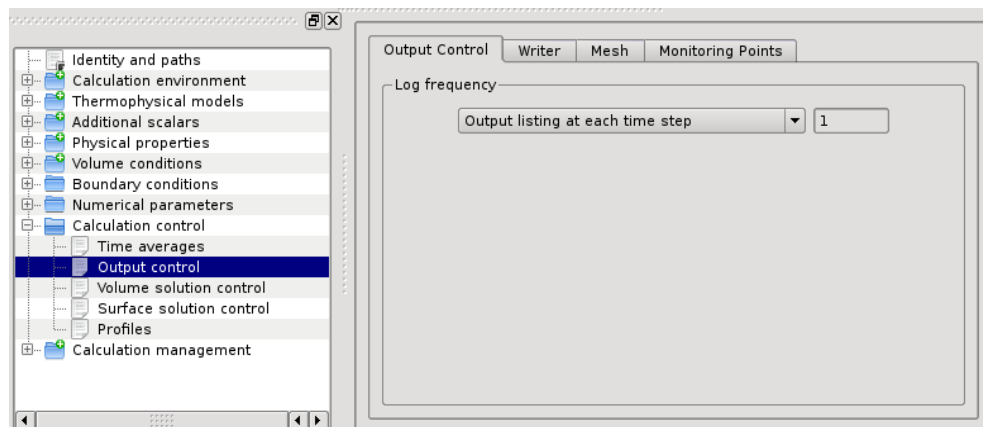


Figure 22: Parameters of chronological logging options

**NOTES**

- The table `iscavr` is filled with the user scalars which represent the mean square fluctuations of another scalar amongst the list of the `nscaus` scalars (warning, this was not the case in version 1.0). For the other scalars, `iscavr` does not need to be completed (by default, `iscavr(ii) ≤ 0`). For instance, if the scalar `jj` represents the average of the square of the fluctuations of the scalar `kk`, the user must indicate `iscavr(jj)=kk` ( $1 \leq kk \leq nscaus$ ).
- When using the interface, only the additional parameters (which can not be defined in the interface) should appear in `usini1`. To spare the user the necessity to delete the other parameters given as examples in the subroutine, the setup program `code_saturne create` comments automatically all the example lines of `usini1` with a code `!ex`. The user needs then only to remove comments at the lines which are useful for his case. This function of `code_saturne create` can be deactivated with the option `--nogui` (useful if the user knows that he will not use the interface).

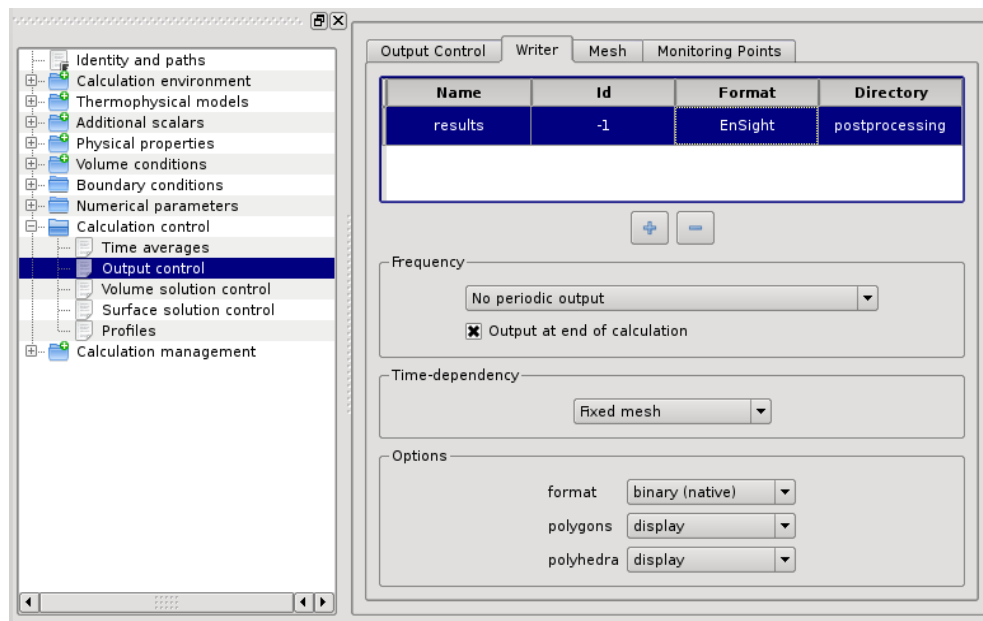


Figure 23: Management of postprocessing writers

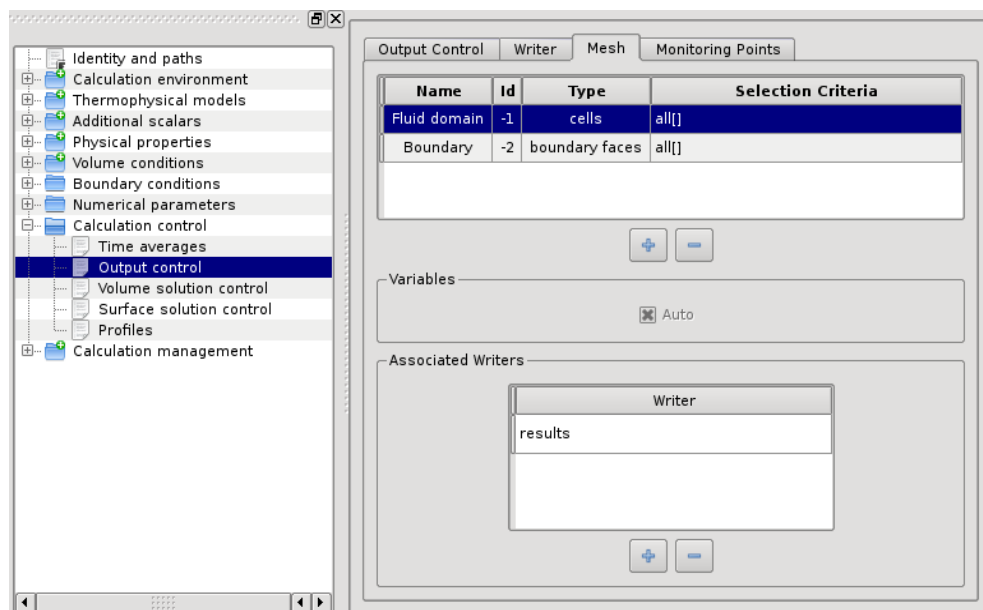


Figure 24: Management of postprocessing meshes

## 6.2 Selection of mesh inputs: `cs_user_mesh_input`

*Subroutine called only during the calculation initialisation.*

This C function may be used to select which mesh input files are read, and apply optional coordinate transformations or group renumberings to them. By default, the input read is a file or directory named `mesh_input`, but if this function is used, any file may be selected, and the same file may be read multiple times (applying a different coordinate transformation each time). All inputs read through this function are automatically concatenated, and may be later joined using the mesh joining options.

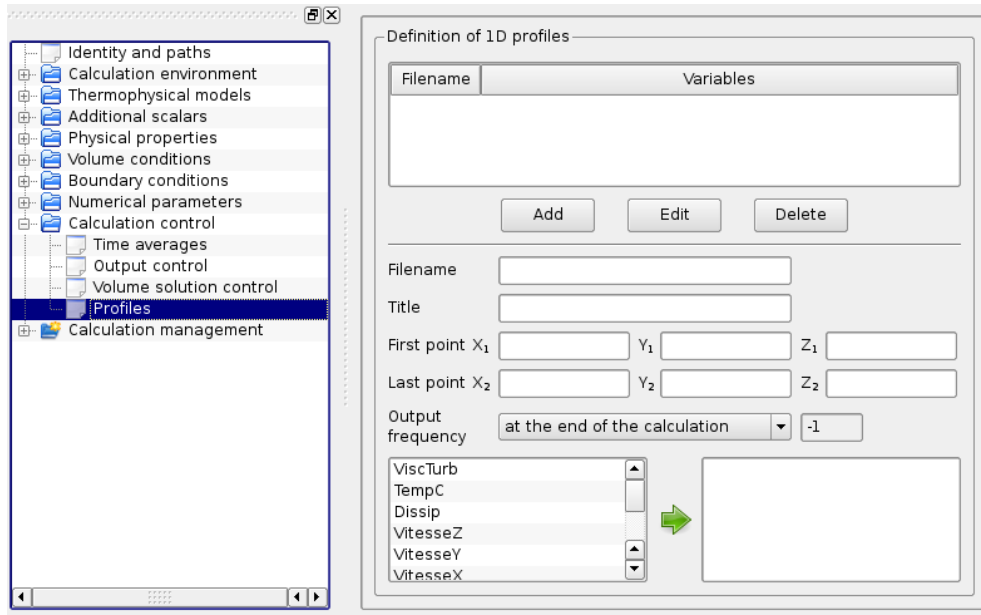


Figure 25: Management of 1D profiles of the solution

Geometric transformations are defined using a homogeneous coordinates transformation matrix. Such a matrix has 3 lines and 4 columns, with the 3 first columns describing a rotation/scaling factor, and the last column describing a translation. A 4th line is implicit, containing zeroes off-diagonal, and 1 on the diagonal. The advantages of this representation is that any rotation/translation/scaling combination may be expressed by matrix multiplication, while simple rotations or translations may still be defined easily.

### 6.3 Non-default variables initialisation

The non-default variables initialisation is performed in the subroutine `usiniv` (called only during the calculation initialisation).

At the calculation beginning, the variables are initialised automatically by the code. Velocities and scalars are set to 0 (or `scamax` or `scamin` if 0 is outside the acceptable scalar variation range), and the turbulent variables are estimated from `uref` and `almax`.

For the  $k$  in  $k - \varepsilon$ ,  $R_{ij} - \varepsilon$ , v2f or  $k - \omega$  model:

`rtp(iel,ikiph) = 1.5*(0.02*uref)**2` (in  $R_{ij} - \varepsilon$ ,  $R_{ij} = \frac{2}{3}k\delta_{ij}$ )

For the  $\varepsilon$  in  $k - \varepsilon$ ,  $R_{ij} - \varepsilon$  or v2f model:

`rtp(iel,ieiph) = rtp(iel,ikiph)**1.5*cmu/almax`

For  $\omega$  in the  $k - \omega$  model:

`rtp(iel,iomgip) = rtp(iel,ikiph)**0.5/almax`

For  $\varphi$  and  $\bar{f}$  in the v2f model:

`rtp(iel,iphiph) = 2/3`

`rtp(iel,ifbiph) = 0`

The subroutine `usiniv` allows if necessary to initialise certain variables to values closer to their estimated final values, in order to obtain a faster convergence.

This subroutine allows also to make a non-standard initialisation of physical parameters (density, viscosity, ...), to impose a local value of the time step, or to modify some parameters (time step, variable specific heat, ...) in the case of a calculation restart.

NOTE: VALUE OF THE TIME STEP

- For calculations with constant and uniform time step (`idtvar=0`), the value of the time step is `dtref`, given in the parametric file of the interface or `usini1`.
- For calculations with a non-constant time step (`idtvar=1` or `2`) which is not a calculation restart, the value of `dtref` given in the parametric file of the interface or in `usini1` is used to initialise the time step.
- For calculations with a non-constant time step (`idtvar=1` or `2`) which is a restart of a calculation whose time step type was different (for instance, restart using a variable time step of a calculation run using a constant time step), the value of `dtref`, given in the parametric file of the interface or in `usini1`, is used to initialise the time step.
- For calculations with non-constant time step (`idtvar=1` or `2`) which is a restart of a calculation whose time step type was the same (for instance, restart with `idtvar=1` of a calculation run with `idtvar=1`), the time step is read from the restart file and the value of `dtref` given in the parametric file of the interface, or in `usini1`, is not used.

It follows, that for a calculation with a non-constant time step (`idtvar=1` or `2`) which is a restart of a calculation in which `idtvar` had the same value, `dtref` does not allow to modify the time step. The user subroutine `usiniv` allows to modify the array `dt` which contains the value of the time step read from the restart file (array whose size is `ncelet`, defined at the cell centers whatever the chosen time step type is).

*WARNING: to initialise the variables in the framework of a specific physics module (`nscapp.gt.0`), one of the subroutines `usebui`, `usd3pi`, `uslwci` or `uscpiu` should be used (depending on the activated module) instead of `usiniv`.*

## 6.4 Manage boundary conditions

The boundary conditions can be specified in the Graphical User Interface (GUI) under the heading “Boundary conditions” or in the user subroutine `usclim` called every time step. With the GUI, each region and the type of boundary condition associated to it are defined in fig. 26. Then, the parameters of the boundary condition are specified in fig. 27. The colors of the boundary faces may be read directly from a “listing” file created by the Preprocessor. This file can be generated directly by the interface under the heading “Definition of boundary regions → Add from Preprocessor listing → import groups and references from Preprocessor listing”, see fig. 26.

`usclim` is the second compulsory subroutine for every calculation launched without interface (except in the case of specific physics where the corresponding boundary condition user subroutine must be used)

When the subroutine is used, `usclim` is used to define complex boundary conditions (input profiles, conditions varying in time, ...) which could not be specified by the means of the interface, and only these need to be defined. In the case of a calculation launched without the interface, all the boundary conditions must appear in `usclim`.

`usclim` is essentially constituted of loops on boundary face subsets. Several sequences of `call getfbr` (`'criterion'`, `nlelt`, `lstelt`) (cf. §3.9.3) allow to select the boundary faces with respect to their group(s), their color(s) or geometric criteria. If needed, geometric and physical variables are also available to the user. These allow him to select the boundary faces using other criteria.

For more details about the treatment of boundary conditions, the user may refer to the theoretical and computer documentation [11] of the subroutine `condli` (for wall conditions, see `clptur`) (to access this document on a workstation, use `code_saturne info --guide theory`).

From the user point of view, the boundary conditions are fully defined by three arrays<sup>22</sup>: `itypfb(nfabor)`, `icodcl(nfabor,nvar)` and `rcodcl(nfabor,nvar,3)`.

<sup>22</sup>except with Lagrangian

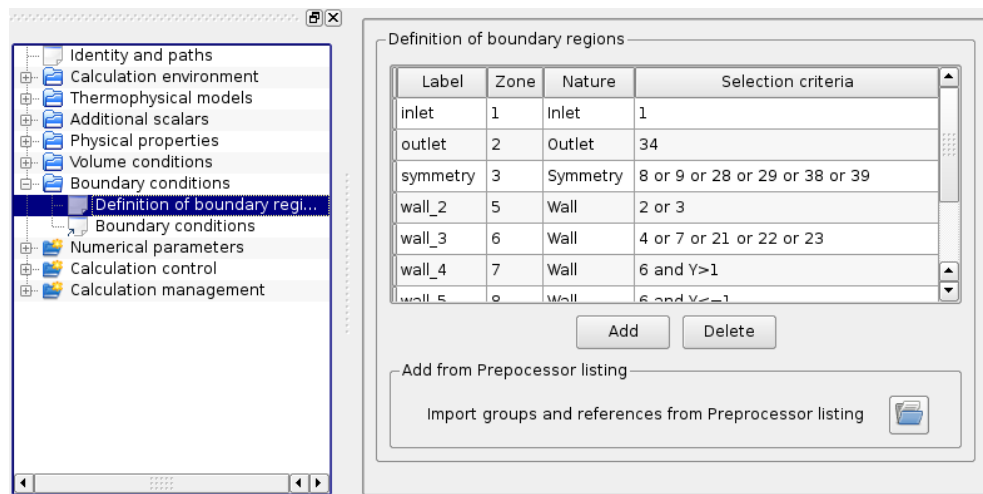


Figure 26: Definition of the boundary conditions

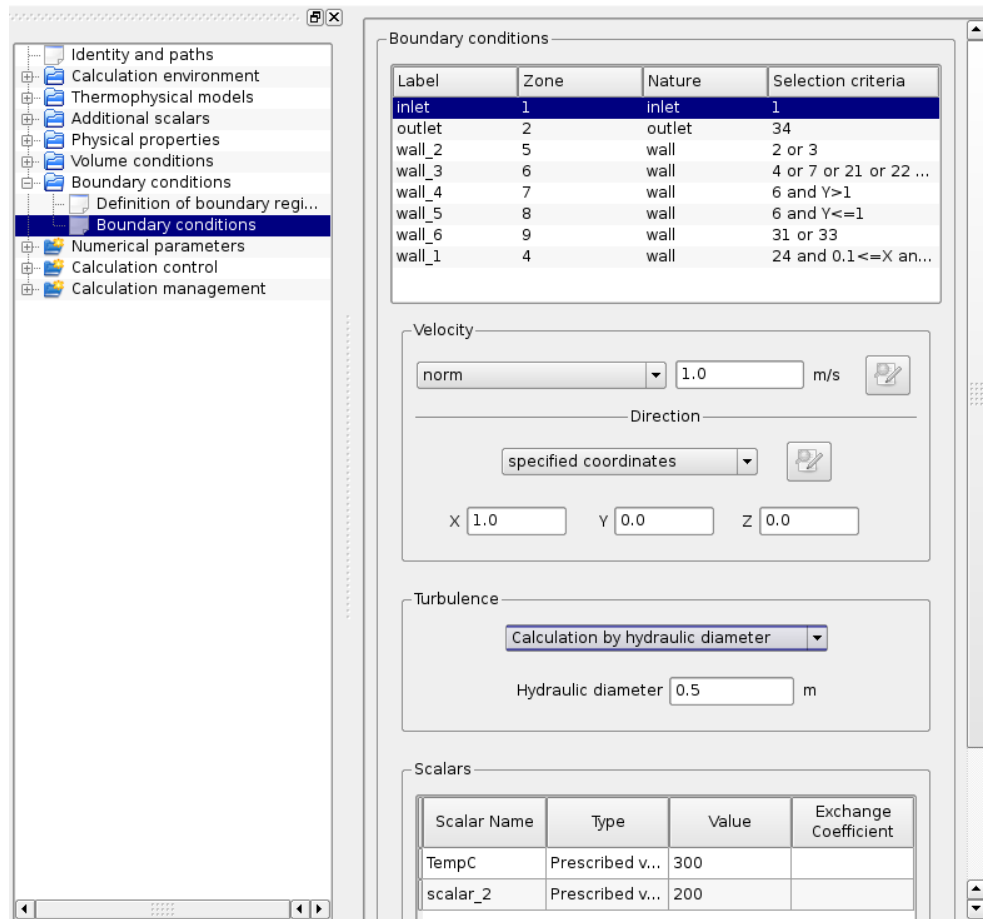


Figure 27: Parameters of the boundary conditions

- `itypfb(ifac)` defines the type of the face `ifac` (input, wall, ...).
- `icodcl(ifac,ivar)` defines the type of boundary condition for the variable `ivar` on the face

`ifac` (Dirichlet, flux ...).

- `rcodcl(ifac,ivar,...)` gives the numerical values associated with the type of boundary condition (value of the Dirichlet, of the flux ...).

In the case of standard boundary conditions (see §6.4.1), it is sufficient to complete `itypfb(ifac)` and parts of the array `rcodcl`; the array `icodcl` and most of `rcodcl` are filled automatically. For non-standard boundary conditions (see §6.4.2), the arrays `icodcl` and `rcodcl` must be fully completed.

## 6.4.1 Coding of standard boundary conditions

The standard key words used by the indicator `itypfb` are: `ientre`, `iparoi`, `iparug`, `isymet`, `isolib` and `iindef`.

- If `itypfb=ientre`: inlet face.
  - Zero-flux condition for pressure and Dirichlet condition for all other variables. The value of the Dirichlet must be given in `rcodcl(ifac,ivar,1)` for every value of `ivar`, except for `ivar=ipr`. The other values of `rcodcl` and `icodcl` are filled automatically.
- If `itypfb=iparoi`: smooth solid wall face, impermeable and with friction.
  - the potential sliding wall velocity of the face is found in `rcodcl(ifac,ivar,1)` (`ivar` being `iu`, `iv` or `iw`). The initial values of `rcodcl(ifac,ivar,1)` are zero for the three velocity components (and therefore are to be specified only if the velocity is not equal to zero).  
*WARNING: the wall sliding velocity must belong to the boundary face plane. For safety, the code only uses the projection of this velocity on the face. As a consequence, if the velocity specified by the user does not belong to the face plane, the wall sliding velocity really taken into account will be different.*
  - For scalars, two kinds of boundary conditions can be defined:
    - ↪ Imposed value at the wall. The user must write
 

```
icodcl(ifac,ivar)=5
rcodcl(ifac,ivar,1)=imposed value
```
    - ↪ Imposed flux at the wall. The user must write
 

```
icodcl(ifac,ivar)=3
rcodcl(ifac,ivar,3)=flux imposed value (depending on the variable, the user
may refer to the case icodcl=3 of Section 6.4.2 for the flux definition).
```
    - ↪ If the user does not fill these arrays, the default condition is zero flux.
- If `itypfb=iparug`: rough solid wall face, impermeable and with friction.
  - the eventual moving velocity of the wall tangent to the face is given by `rcodcl(ifac,ivar,1)` (`ivar` being `iu`, `iv` or `iw`). The initial value of `rcodcl(ifac,ivar,1)` is zero for the three velocity components (and therefore needs to be specified only in the case of the existence of a slipping velocity).  
*WARNING: the wall moving velocity must be in the boundary face plane. By security, the code uses only the projection of this velocity on the face. As a consequence, if the velocity specified by the user is not in the face plane, the wall moving velocity really taken into account will be different.*
  - The dynamic roughness must be specified in `rcodcl(ifac,iu,3)`. The values of `rcodcl(ifac,iv,3)` and `rcodcl(ifac,iw,3)` are not used.
  - For scalars, two kinds of boundary conditions can be defined:



↪ Imposed value at the wall. The user must write

```
icodcl(ifac,ivar)=6
rcodcl(ifac,ivar,1)=imposed value
rcodcl(ifac,ivar,3)=thermal roughness value
```

↪ Imposed flux at the wall. The user must write

```
icodcl(ifac,ivar)=3
rcodcl(ifac,ivar,3)=flux imposed value (for the flux definition according to
the variable, the user may refer to the case icodcl=3 of the paragraph 6.4.2).
```

↪ If the user does not complete these arrays, the default condition is zero flux.

- If `itypfb=isymet`: symmetry face (or wall without friction)

→ Nothing to be written in `icodcl` and `rcodcl`.

- If `itypfb=isolib`: free outlet face (or more precisely free inlet/outlet with forced pressure)

→ The pressure is always treated with a Dirichlet condition, calculated with the constraint  $\frac{d}{dn} \left( \frac{dP}{d\tau} \right) = 0$ . The pressure is set to  $P_0$  at the first `isolib` face met. The pressure calibration is always done on a single face, even if there are several outlets.

→ If the mass flow is coming in, the “infinite” velocity is retained and a Dirichlet condition for the scalars and the turbulent quantities is used (or zero-flux condition if no Dirichlet value has been specified).

→ If the mass flow is going out, zero-flux condition are set for the velocity, the scalars and the turbulent quantities.

→ Nothing is written in `icodcl` or `rcodcl` for the pressure or the velocity. An optional Dirichlet condition can be specified for the scalars and turbulent quantities.

- If `itypfb=iindef`: undefined type face (non-standard case)

→ Coding is done in a non-standard way by filling both arrays `rcodcl` and `icodcl` (see §6.4.2).

## NOTES

- Whatever is the value of the indicator `itypfb(ifac)`, if the array `icodcl(ifac,ivar)` is modified by the user (*i.e.* filled with a non-zero value), the code will not use the default conditions for the variable `ivar` at the face `ifac`. It will take into account only the values of `icodcl` and `rcodcl` provided by the user (these arrays must then be fully completed, like in the non-standard case).

For instance, for a normal symmetry face where scalar 1 is associated with a Dirichlet condition equal to 23.8 (with an infinite exchange coefficient):

```
itypfb(ifac)=isymet
icodcl(ifac,isca(1))=1
rcodcl(ifac,isca(1),1)=23.8
```

(`rcodcl(ifac,isca(1),2)=rinfin` is the default value, therefore it is not necessary to specify a value)  
The boundary conditions for the other variables are remain automatically defined.

- The user can define new types of boundary faces. He only needs to choose a value  $N$  and to fully specify the boundary conditions (see §6.4.2). He must specify `itypfb(ifac)=N` where  $N$  range is 1 to `ntypmx` (maximum number of boundary face types), and of course different from the values `ientre`, `iparoi`, `iparug`, `isymet`, `isolib` and `iindef` (the values of these variables are given in the file `paramx.h`). This allows to easily isolate some boundary faces, in order for instance to calculate balances.

## 6.4.2 Coding of non-standard boundary conditions

In the case a face does not correspond to a standard type, the user must fill completely the arrays `itypfb`, `icodcl` and `rcodcl`. `itypfb(ifac)` is then equal to `iundef` or another value defined by the user (see note at the end of Section 6.4.1). The arrays `icodcl` and `rcodcl` must be filled as follows:

- If `icodcl(ifac,ivar)=1`: Dirichlet condition at the face `ifac` for the variable `ivar`.
  - `rcodcl(ifac,ivar,1)` is the value of the variable `ivar` at the face `ifac`.
  - `rcodcl(ifac,ivar,2)` is the value of the exchange coefficient between the outside and the fluid for the variable `ivar`. An infinite value (`rcodcl(ifac,ivar,2)=rinf`) indicates an ideal transfer between the outside and the fluid (default case).
  - `rcodcl(ifac,ivar,3)` is not used.
  - `rcodcl(ifac,ivar,1)` has the units of the variable `ivar`, *i.e.*:
    - ↪  $m/s$  for the velocity
    - ↪  $m^2/s^2$  for the Reynolds stress
    - ↪  $m^2/s^3$  for the dissipation
    - ↪  $Pa$  for the pressure
    - ↪  $^{\circ}C$  for the temperature
    - ↪  $J.kg^{-1}$  for the enthalpy
    - ↪  $^{\circ}C^2$  for temperature fluctuations
    - ↪  $J^2.kg^{-2}$  for enthalpy fluctuations
  - `rcodcl(ifac,ivar,2)` has the following units (defined in such way that when multiplying the exchange coefficient by the variable, the given flux has the same units as the flux defined below when `icodcl=3`):
    - ↪  $kg.m^{-2}.s^{-1}$  for the velocity
    - ↪  $kg.m^{-2}.s^{-1}$  for the Reynolds stress
    - ↪  $s.m^{-1}$  for the pressure
    - ↪  $W.m^{-2}.^{\circ}C^{-1}$  for the temperature
    - ↪  $kg.m^{-2}.s^{-1}$  for the enthalpy
- If `icodcl(ifac,ivar)=3`: flux condition at the face `ifac` for the variable `ivar`.
  - `rcodcl(ifac,ivar,1)` and `rcodcl(ifac,ivar,2)` are not used.
  - `rcodcl(ifac,ivar,3)` is the flux value of `ivar` at the wall. This flux is negative if it is a source for the fluid. It corresponds to:
    - ↪  $-C_p(\frac{\lambda_T}{C_p} + \frac{\mu_t}{\sigma_T})\underline{\text{grad}} T \cdot \underline{n}$  for a temperature (in  $W/m^2$ ).
    - ↪  $-(\lambda_h + \frac{\mu_t}{\sigma_h})\underline{\text{grad}} h \cdot \underline{n}$  for an enthalpy (in  $W/m^2$ ).
    - ↪  $-(\lambda_{\varphi} + \frac{\mu_t}{\sigma_{\varphi}})\underline{\text{grad}} \varphi \cdot \underline{n}$  in the case of another scalar  $\varphi$  (in  $kg.m^{-2}.s^{-1}.[\varphi]$ , where  $[\varphi]$  are the units of  $\varphi$ ).
    - ↪  $-\Delta t \underline{\text{grad}} P \cdot \underline{n}$  for the pressure (in  $kg.m^{-2}.s^{-1}$ ).
    - ↪  $-(\mu + \mu_t)\underline{\text{grad}} U_i \cdot \underline{n}$  for a velocity component (in  $kg.m^{-1}.s^{-2}$ ).
    - ↪  $-\mu \underline{\text{grad}} R_{ij} \cdot \underline{n}$  for a  $R_{ij}$  tensor component (in  $W/m^2$ ).
- If `icodcl(ifac,ivar)=4`: symmetry condition, for the symmetry faces or wall faces without friction. This condition can only be used for velocity components ( $\underline{U} \cdot \underline{n} = 0$ ) and the  $R_{ij}$  tensor components (for other variables, a zero-flux condition type is usually used).

- If `icodcl(ifac,ivar)=5`: friction condition, for wall faces with friction. This condition can not be applied to the pressure.
  - ↪ For the velocity and (if necessary) the turbulent variables, the values at the wall are calculated from theoretical profiles. In the case of a sliding wall, the three components of the sliding velocity are given by `rcodcl(ifac,iu,1)`, `rcodcl(ifac,iv,1)`, and `rcodcl(ifac,iw,1)`.  
*WARNING: the wall sliding velocity must belong to the boundary face plane. For safety, the code uses only the projection of this velocity on the face. Therefore, if the velocity vector specified by the user does not belong to the face plane, the wall sliding velocity really taken into account will be different.*
  - ↪ For other scalars, the condition `icodcl=5` is similar to `icodcl=1`, but with a wall exchange coefficient calculated from a theoretical law. Therefore, the values of `rcodcl(ifac,ivar,1)` and `rcodcl(ifac,ivar,2)` must be specified: see [11].
- If `icodcl(ifac,ivar)=6`: friction condition, for the rough-wall faces with friction. This condition can not be used with the pressure.
  - ↪ For the velocity and (if necessary) the turbulent variables, the values at the wall are calculated from theoretical profiles. In the case of a sliding wall, the three components of the sliding velocity are given by `rcodcl(ifac,iu,1)`, `rcodcl(ifac,iv,1)`, and `rcodcl(ifac,iw,1)`.  
*WARNING: the wall sliding velocity must belong to the boundary face plane. For safety, the code uses only the projection of this velocity on the face. Therefore, if the velocity vector specified by the user does not belong to the face plane, the wall sliding velocity really taken into account will be different.*  
 The dynamic roughness height is given by `rcodcl(ifac,iu,3)` only.
  - ↪ For the other scalars, the condition `icodcl=6` is similar to `icodcl=1`, but with a wall exchange coefficient calculated from a theoretical law. The values of `rcodcl(ifac,ivar,1)` and `rcodcl(ifac,ivar,2)` must therefore be specified: see [11]. The thermal roughness height is then given by `rcodcl(ifac,ivar,3)`.
- If `icodcl(ifac,ivar)=9`: free outlet condition for the velocity. This condition is only applicable to velocity components.  
 If the mass flow at the face negative, this condition is equivalent to a zero-flux condition.  
 If the mass flow at the face is positive, the velocity at the face is set to zero (but not to the mass flow).  
`rcodcl` is not used.

#### NOTE

- A standard `isolib` outlet face amounts to a Dirichlet condition (`icodcl=1`) for the pressure, a free outlet condition (`icodcl=9`) for the velocity and a Dirichlet condition (`icodcl=1`) if the user has specified a Dirichlet value or a zero-flux condition (`icodcl=3`) for the other variables.

### 6.4.3 Checking of the boundary conditions

The code checks the main compatibilities between the boundary conditions. In particular, the following rules must be respected:

- On each face, the boundary conditions of the three velocity components must belong to the same type. The same is true for the components of the  $R_{ij}$  tensor.
- If the boundary conditions for the velocity belong to the “sliding” type (`icodcl=4`), the conditions for  $R_{ij}$  must belong to the “symmetry” type (`icodcl=4`), and vice versa.
- If the boundary conditions for the velocity belong to the “friction” type (`icodcl=5` or `6`), the boundary conditions for the turbulent variables must belong to the “friction” type, too.
- If the boundary condition of a scalar belongs to the “friction” type, the boundary condition of the velocity must belong to the “friction” type, too.

In case of mistakes, if the post-processing output is activated (which is the default setting), a special error output, similar to the mesh format, is produced in order to help correcting boundary condition definitions.

#### 6.4.4 Sorting of the boundary faces

In the code, it may be necessary to have access to all the boundary faces of a given type. To ease this kind of search, an array made of sorted faces is automatically filled (and updated at each time step): `itrifb(nfabor)`.

`ifac=itrifb(i)` is the number of the  $i^{\text{th}}$  face of type 1.

`ifac=itrifb(i+n)` is the number of the  $i^{\text{th}}$  face of type 2, if there are  $n$  faces of type 1.

... etc.

Two auxiliary arrays of size `ntypmx` are also defined.

`idebty(ityp)` is the index corresponding to the first face of type `ityp` in the array `itrifb`.

`ifinty(ityp)` is the index corresponding to the last face of type `ityp` in the array `itrifb`.

Therefore, a value `ifac0` found between `idebty(ityp)` and `ifinty(ityp)` is associated to each face `ifac` of type `ityp=itypfb(ifac)`, so that `ifac=itrifb(ifac0)`.

If there is no face of type `ityp`, the code set

`ifinty(ityp)=idebty(ityp)-1,`

which enables to bypass, for all the missing `ityp`, the loops such as

`do ii=idebty(ityp),ifinty(ityp).`

The values of all these indicators are displayed at the beginning of the code execution listing.

#### 6.4.5 Boundary conditions with LES

The subroutine `usvort` allows to generate the non-stationary inlet boundary conditions for the LES by the vortex method. The method is based on the generation of vortices in the 2D inlet plane with help from the pre-defined functions. The fluctuation normal to the inlet plane is generated by a Langevin equation. It is in the subroutine `usvort` where the parametres of this method are given.

*subroutine called for each time step*

To allow the application of the vortex method, an indicator must be informed of the method in the user subroutine `usini1(ivrtex=1)`

The subroutine `usvort` contains 3 separate parts:

- The 1st part defines the number of inlets concerned with the vortex method (`nentt`) and the number of vortex for each inlet (`nvort`), where `ient` represents the number of inlets.
- The 2nd part (`iappel=1`) defines the boundary faces at which the vortex method is applicable. The `irepvo` array is informed by `ient` which defines the number of inlets concerned with the vortex (essentially, the vortex method can be applied with many independant inlets).
- The 3rd section defines the main parameters of the method at each inlet. With the complexity of any given geometry, 4 cases are distinguished (the first 3 use the data file `ficvor` and in the final case only 1 initial velocity and energy are imposed.):
  - \* `icas=1`, For the outlet of a rectangular pipe; 1 boundary condition is defined for each side of the rectangle taking into account their interaction with the vortex.
  - \* `icas=2`, For the outlet of a circular pipe; the entry face is considered as a wall (as far as interaction with the vortex is concerned)
  - \* `icas=3`, For inlets of any geometry; no boundary conditions are defined at the inlet face (i.e no specific treatment on the interaction between the vortex and the boundary)

\* **icas**=4, similar to **icas**=3 except the data file is not used (**ficvor**); the outflow parameters are estimated by the code from the global data (initial velocity, level of turbulence and dissipation), information which is supplied by the user.

When the geometry allows, cases 1 and 2 are used. Case 4 is only used if it is not possible to use the other 3.

In the first 3 cases, the 2 base vectors in the plane of each inlet must be defined (vectors **dir1** and **dir2**). The 3rd vector is automatically calculated by the code, defined as a product of **dir1** and **dir2**. **dir1** and **dir2** must be chosen imperatively to give (**cen**, **dir1**, **dir2**) an orthogonal reference of the inlet plane and so **dir3** is oriented in the entry domain. If **icas**=2, the **cen** position must be the center of gravity of the rectangle or disc.

The reference points (**cen**, **dir1**, **dir2**, **dir3**) which define the values of the variable in the **ficvor** file.

In the case where **icas**=4, the vectors **dir1** and **dir2** are generated by the code.

If **icas**=1, the boundary conditions at the rectangle's edges must be defined. They are defined in the array **iclvor**. **iclvor(ii,ient)** represents the standard boundary conditions at the edge  $II(1 \leq II \leq 4)$  of the inlet **ient**. The code for the boundary conditions is as follows:

- \* **iclvor**=1 for a wall
- \* **iclvor**=2 for symmetry
- \* **iclvor**=3 for periodicity of translation (the face corresponding to periodicity will automatically be taken as 3)

The 4 edges are numbered relative to the directions **dir1** and **dir2** as shown in figure 28:

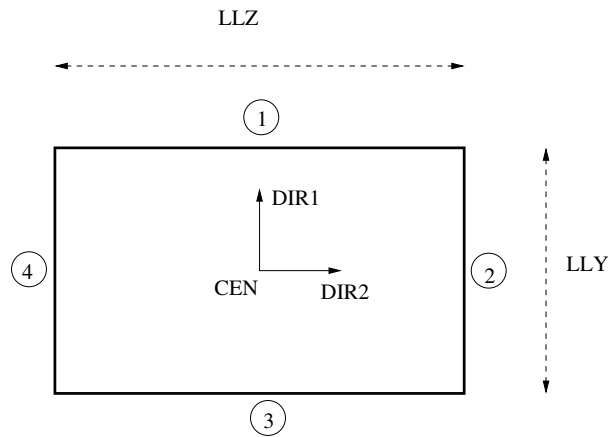


Figure 28: Numbering of the edges of a rectangular inlet(**icas**=1) treated by the vortex method

If **icas**=1, the user must define **llx** and **lly** which give the lengths of the rectangular pipe in the directions **dir1** and **dir2**.

If **icas**=2, **lld** represents the diameter of the circular pipe. If **icas**=4, **udebit**, **kdebit** and **edebit** are defined for each inlet, these give respectively, initial speed, turbulent energy level and the dissipation level. These can be used to obtain their magnitude using the correlations in the user routine **usclim** for fully developed flow in a pipe.

The case independent parameters are defined as follows:

- \* **itmpl** represents the indicator of the advancement in time of the vortex. If **itmpli**=1, the vortex will be regenerated after a fixed time of **tmplim** second (defined as **itmpli**=1). If **itmpli**=2, following the data indicated in **ficvor** file, the vortex will have a variable life span equal to  $5C_\mu \frac{k^{\frac{3}{2}}}{\varepsilon U}$ , where  $C_\mu = 0,09$  and  $k$ ,  $\varepsilon$  and  $U$  represent respectively, turbulent

energy, turbulent dissipation and the convective velocity in the direction normal to the inlet plane.

- \* **xsgmvo** represents the support functions used in the vortex method. They are representative of the eddy sizes entered in the vortex method. **isgmvo** is used to define their size: if **isgmvo**=1, **xsgmvo** will be constant across the inlet face and is defined in **usvort**, if **isgmvo**=2, **xsgmvo** will be variable and equal to the mixing length of the standard  $k-\varepsilon$  model ( $C_\mu^{\frac{3}{4}} \frac{k^{\frac{3}{2}}}{\varepsilon}$ ), if **isgmvo**=3, **xsgmvo** will be equal to the maximum of  $L_t$  et  $L_K$  where  $L_t$  and  $L_K$  are the  $\frac{\partial U}{\partial y} \frac{\partial U}{\partial y}$  Taylor and Kolmogorov co-efficients ( $L_T = (5\nu \frac{k}{\varepsilon})^{\frac{1}{2}}$ ,  $L_K = 200(\frac{\nu^3}{\varepsilon})^{\frac{1}{4}}$ ).
- \* **idepvo** gives the vortex displacement method in the 2D inlet plane (the vortex method is a langrangian method in which the eddy centers are replaced by a set velocity). If **idepvo**=1, the velocity displacement referred to by **ud** which is the vortex following a random sampling (a sample number **r**, is taken for each vortex, at each time step and for each direction and the center of the vortex is replaced by the 2 principle directions,  $\mathbf{rud}\Delta t$  where  $\Delta t$  is the time step of the calculation). If **idepvo**=2, the vortex will be convected by itself (with the speed given by the time step before the vortex method)

A data file, **ficvor**, must be defined in the cases of **icas**=1,2,3, for each inlet. The data file must contain the following data in order  $(x, y, U, \frac{\partial U}{\partial y}, k, \varepsilon)$ . The number of lines of the file is given by the integer **ndat**.  $x$  and  $y$  are the co-ordinates in the inlet plane defined by the vectors **dir1** and **dir2**.  $U$ ,  $k$  and  $\varepsilon$  are respectively, the average speed normal to the inlet, the turbulent energy and the turbulent dissipation.  $\frac{\partial U}{\partial y}$  is the derivative in the direction normal to the inlet boundary in the cases, **icas**=1, **icas**=2. Where **icas**=3 and **icas**=4 this variable is not applied (it is given the value 0) so the Langevin equations, used to generate fluctuations normal to the inlet plane, is de-activated (the fluctuations normal to the inlet is 0 on both these cases). Note that the application of many different test of the Langevin equation doesn't have a notable influence on the results and that, by contrast it simply increases the computing time per iteration and so it decreases the random sampling which slows down the pressure solver. The interpolation used in the vortex method is defined by the function **phidat**. An example is given at the end of **usvort** where the user can define the interpolation required. In the **phidat** function, **xx** and **yy** are the co-ordinates by which the value of **phidat** is calculated. **xdat** and **ydat** are the co-ordinates in the **ficvor** file. **vardat** is the value of the **phidat** function with the co-ordinates **xdat** and **ydat** (given in the **ficvor** file). Note that using an indicator **iii** accelerates the calculations (the user need not modify or delete). The user must also define the parameter **isuivo** which indicates if the vortex were started at 0 or if the file must be re-read (**ficmvo**).

### WARNING

- Be sure that the **ficvor** file and the interpolation in the user function **phidat** are compatible (in particular that all the entry region is covered by **ficvor**)
- If the user wants to use a 1D profile in the **dir2** direction, set  $x=0$  in the **ficvor** file and define the interpolation in **phidat**.

## 6.5 Manage the variable physical properties

### 6.5.1 Basic variable physical properties

When the fluid properties are not constant, the user is offered the choice to define the variation laws in the Graphical User Interface (GUI) or in the subroutine **usphyv** which is called at each time step. In the GUI, in the item "Fluid properties" under the heading "Physical properties", the variation laws

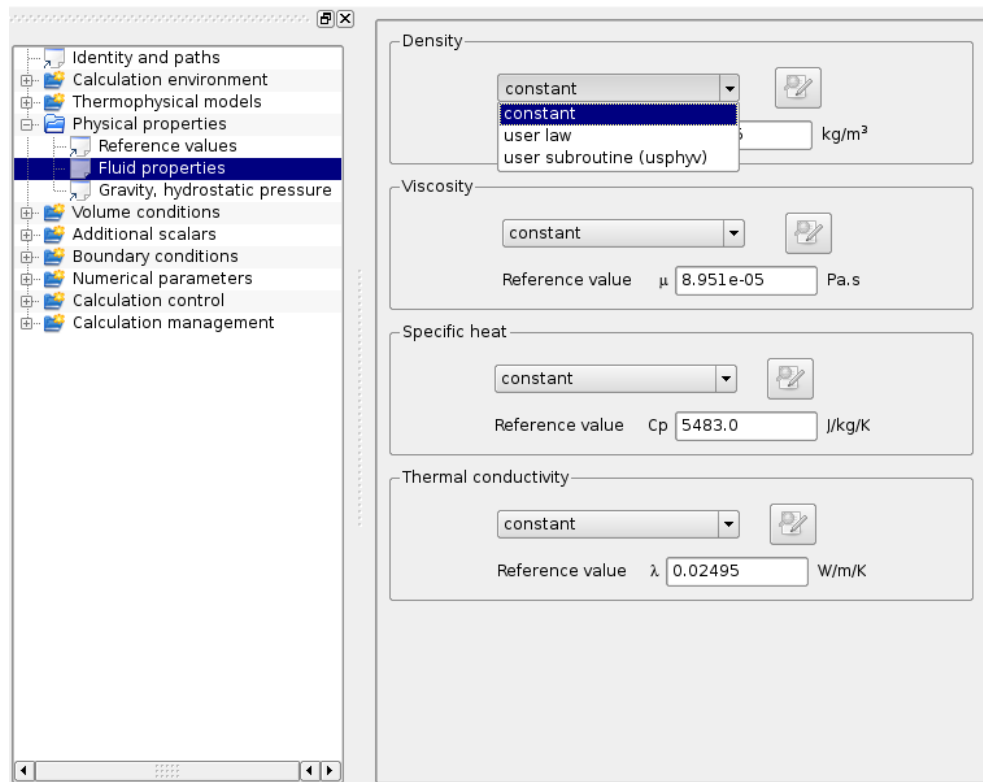


Figure 29: Physical properties - Fluid properties

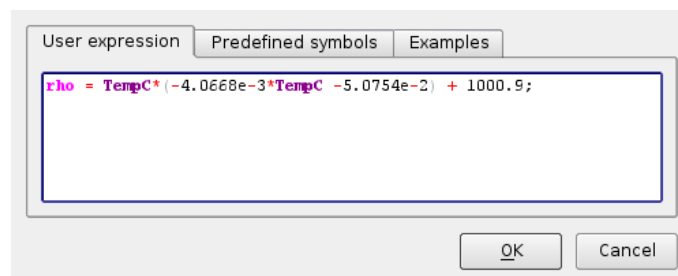


Figure 30: Definition of a user law for the density

are defined for the fluid density, viscosity, specific heat and thermal conductivity through the use of a formula editor, see figs. 29 and 30.

If necessary, all the variation laws related to the fluid physical properties are written in the subroutine `usphyv`.

The validity of the variation laws must be checked, particularly when non-linear laws are defined (for instance, a third-degree polynomial law may produce negative density values).

### **WARNING**

- If one wishes to impose a variable density or variable viscosity in `usphyv`, it must be flagged either in the interface or in `usini1(irovar=1, ivivar=1)`.



- In order to impose a physical property ( $\rho$ ,  $\mu$ ,  $\lambda$ ,  $C_p$ )<sup>23</sup> a reference value should be provided in the interface or in `usini1` (in particular for  $\rho$ , the pressure will be function of  $\rho_0 g z$ )
- By default, the  $C_p$  coefficient and the diffusivity for the scalars `iscal` ( $\lambda/C_p$  for the temperature) are considered as constant in time and uniform in space, with the values `cp0` and `visls0(iscal)` specified in the interface or in `usini1`.  
To assign a variable value to  $C_p$ , the user **must** specify it in the interface or assign the value 1 to `icp` in `usini1`, and fill for each cell `iel` the array `propce(iel,ipccp)` in `usphyv`. Completing the array `propce(iel,ipccp)` while `icp=0` induces array overwriting problems and produces wrong results.
- In the same way, to have variable diffusivities for the scalars `iscal`, the user **must** specify it in the interface or give the value 1 to `ivisls(iscal)` in `usini1`, and complete for each cell `iel` the array `propce(iel,ipcvsl)` in `usphyv`. Completing `propce(iel,ipcvsl)` while `ivisls(iscal)=0` induces memory overwriting problems and produces wrong results.

*Example:* If scalars 1 and 3 have a constant and uniform diffusivity, and if scalars 2 and 4 have a variable diffusivity, the following values must be set in `usini1`:  
`ivisls(1)=0`, `ivisls(2)=1`, `ivisls(3)=0` and `ivisls(4)=1`.  
The indicators `ivisls(2)` and `ivisls(4)` are then modified automatically by the code in order to return the rank corresponding to the diffusivity of each scalar in the list of physical properties<sup>24</sup>. The arrays `propce(iel,ipcvsl)` in `usphyv` must then be completed with `ipcvsl=ipproc(ivisls(2))` and `ipcvsl=ipproc(ivisls(4))`.

*Note:* The indicators `ivisls` must not be completed in the case of user scalars representing the average of the square of the fluctuations of another scalar, because the diffusivity of a user scalar `jj` representing the average of the square of the fluctuations of a user scalar `kk` comes directly from the diffusivity of this last scalar. In particular, the diffusivity of the scalar `jj` is variable if the diffusivity of `kk` is variable.

## 6.5.2 Modification of the turbulent viscosity

The subroutine `usvist` is used to modify the calculation of the turbulent viscosity, *i.e.*  $\mu_t$  in  $kg.m^{-1}.s^{-1}$  (this piece of information, at the mesh cell centers, is conveyed by the variable `propce(iel,ipcvst)`, with `ipcvst = ipproc(ivisct)`). The subroutine is called at the beginning of every time step, after the calculation of the physical parameters of the flow and of the “conventional” value of  $\mu_t$  corresponding to the chosen turbulence model (indicator `iturb`).

*WARNING: The calculation of the turbulent viscosity being a particularly sensible stage, a wrong use of `usvist` may seriously distort the results.*

## 6.5.3 Modification of the variable $C$ of the dynamic LES model

*Subroutine called every time step in the case of LES with the dynamic model.*

The subroutine `ussmag` is used to modify the calculation of the variable  $C$  of the LES sub-grid scale dynamic model.

Let us first remind that the LES approach introduces the notion of filtering between large eddies and small motions. The solved variables are said to be filtered in an “implicit” way. Sub-grid scale models (“dynamic” models) introduce in addition an explicit filtering.

The notations used for the definition of the variable  $C$  used in the dynamic models of *Code\_Saturne* are specified below. These notations are the ones assumed in the document [3], to which the user may refer for more details.

<sup>23</sup>except for some specific physics

<sup>24</sup>they are no longer equal to 1 but stay positive so that `ivisls>0` is synonymous with variable diffusivity



The value of  $a$  filtered by the explicit filter (of width  $\widetilde{\Delta}$ ) is called  $\widetilde{a}$  and the value of  $a$  filtered by the implicit filter (of width  $\overline{\Delta}$ ) is called  $\overline{a}$ . We define:

$$\begin{aligned}
 \overline{S}_{ij} &= \frac{1}{2} \left( \frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i} \right) & ||\overline{S}|| &= \sqrt{2\overline{S}_{ij}\overline{S}_{ij}} \\
 \alpha_{ij} &= -2\overline{\Delta} ||\overline{S}|| \overline{S}_{ij} & \beta_{ij} &= -2\overline{\Delta}^2 ||\overline{S}|| \overline{S}_{ij} \\
 L_{ij} &= \widetilde{\overline{u}_i \overline{u}_j} - \widetilde{\overline{u}_i} \widetilde{\overline{u}_j} & M_{ij} &= \alpha_{ij} - \beta_{ij}
 \end{aligned} \tag{1}$$

In the framework of LES, the total viscosity (molecular + sub-grid) in  $kg.m^{-1}.s^{-1}$  may be written in *Code\_Saturne*:

$$\begin{aligned}
 \mu_{total} &= \mu + \mu_{sub-grid} & \text{if } \mu_{sub-grid} > 0 \\
 &= \mu & \text{otherwise} \\
 \text{with } \mu_{sub-grid} &= \rho C \overline{\Delta}^2 ||\overline{S}||
 \end{aligned} \tag{2}$$

$\overline{\Delta}$  is the width of the implicit filter, defined at the cell  $\Omega_i$  by  
 $\overline{\Delta} = XLESFL * (ALES * |\Omega_i|)^{BLES}$ .

In the case of the Smagorinsky model (**iturb=40**),  $C$  is a constant which is worth  $C_s^2$ .  $C_s^2$  is the so-called Smagorinsky constant and is stored the variable *csmago*.

In the case of the dynamic model (**iturb=41**),  $C$  is variable in time and in space. It is determined by  
 $C = \frac{M_{ij}Li j}{M_{kl}M_{kl}}$ .

In practice, in order to increase the stability, the code does not use the value of  $C$  obtained in each cell, but an average with the values obtained in the neighboring cells (this average uses the extended neighborhood and corresponds to the explicit filter). By default, the value calculated by the code is

$$C = \frac{\widetilde{M_{ij}Li j}}{M_{kl}M_{kl}}$$

The subroutine **ussmag** allows to modify this value. It is for example possible to calculate the local average after having calculated the ratio

$$C = \left[ \frac{\widetilde{M_{ij}Li j}}{M_{kl}M_{kl}} \right]$$

**WARNING:** The subroutine **ussmag** can be activated only when the dynamic model is used.

## 6.6 User source terms

Let us assume that the user source terms modify the equation of a variable  $\varphi$  in the following way:

$$\rho \frac{\partial \varphi}{\partial t} + \dots = \dots + S_{impl} \times \varphi + S_{expl}$$

The example is valid a velocity component, for a turbulent variable ( $k$ ,  $\varepsilon$ ,  $R_{ij}$ ,  $\omega$ ,  $\varphi$  or  $\overline{f}$ ) and for a scalar (or for the average of the square of the fluctuations of a scalar), because the syntax of the subroutines **ustske**, **ustsri**, **ustsv2**, **ustskw** and **ustssc** is similar.

In the finite volume formulation, the solved system is then modified as follows:

$$\left( \frac{\rho_i \Omega_i}{\Delta t_i} - \Omega_i S_{impl,i} \right) \left( \varphi_i^{(n+1)} - \varphi_i^{(n)} \right) + \dots = \dots + \Omega_i S_{impl,i} \varphi_i^{(n)} + \Omega_i S_{expl,i}$$

The user needs therefore to provide the following values:

$$crvimp_i = \Omega_i S_{impl,i}$$

$$crvexp_i = \Omega_i S_{expl,i}$$

In practice, it is essential for the term  $\left(\frac{\rho_i \Omega_i}{\Delta t_i} - \Omega_i S_{impl,i}\right)$  to be positive. To ensure this property, the equation really taken into account by the code is the following:

$$\left(\frac{\rho_i \Omega_i}{\Delta t_i} - \text{Min}(\Omega_i S_{impl,i}; 0)\right) \left(\varphi_i^{(n+1)} - \varphi_i^{(n)}\right) + \dots = \dots + \Omega_i S_{impl,i} \varphi_i^{(n)} + \Omega_i S_{expl,i}$$

To make the “implication” effective, the source term decomposition between the implicit and explicit parts will be done by the user who must ensure that  $\text{crvimp}_i = \Omega_i S_{impl,i}$  is always negative (otherwise the solved equation remains right, but there will not be “implication”).

*WARNING: When the second-order in time is used along with the extrapolation of the source terms<sup>25</sup>, it is no longer possible to test the sign of  $S_{impl,i}$ , because of coherence reasons (for more details, the user may refer to the theoretical and computer documentation [11] of the subroutine `preduv`). The user must therefore make sure it is always positive (or take the risk to affect the calculation stability).*

#### PARTICULAR CASE OF A LINEARISED SOURCE TERM

In some cases, the added source term is not linear, but the user may want to linearise it using a first-order Taylor development, in order to make it partially implicit. Let us consider an equation of the type:

$$\rho \frac{\partial \varphi}{\partial t} = F(\varphi)$$

We want to make it implicit using the following method:

$$\begin{aligned} \frac{\rho_i \Omega_i}{\Delta t} \left(\varphi_i^{(n+1)} - \varphi_i^{(n)}\right) &= \Omega_i \left[ F(\varphi_i^{(n)}) + \left(\varphi_i^{(n+1)} - \varphi_i^{(n)}\right) \frac{dF}{d\varphi}(\varphi_i^{(n)}) \right] \\ &= \Omega_i \frac{dF}{d\varphi}(\varphi_i^{(n)}) \times \varphi_i^{(n+1)} + \Omega_i \left[ F(\varphi_i^{(n)}) - \frac{dF}{d\varphi}(\varphi_i^{(n)}) \times \varphi_i^{(n)} \right] \end{aligned}$$

The user must therefore specify:

$$\text{crvimp}_i = \Omega_i \frac{dF}{d\varphi}(\varphi_i^{(n)})$$

$$\text{crvexp}_i = \Omega_i \left[ F(\varphi_i^{(n)}) - \frac{dF}{d\varphi}(\varphi_i^{(n)}) \times \varphi_i^{(n)} \right]$$

*Example:*

If the equation is  $\rho \frac{\partial \varphi}{\partial t} = -K \varphi^2$ , the user must set:

$$\text{crvimp}_i = -2K \Omega_i \varphi_i^{(n)}$$

$$\text{crvexp}_i = K \Omega_i [\varphi_i^{(n)}]^2$$

### 6.6.1 In Navier-Stokes

The subroutine `ustsns` is used to add user source terms to the Navier-Stokes equations (at each time step). It is called three times every time step, once for each velocity component (`ivar` is successively `iu`, `iv` and `iw`). At each passage, the user must complete if necessary the arrays `crvimp` and `crvexp` expressing respectively the implicit and explicit part of the source term. If no other source terms apart from `ivar=iu` for example, are required, `crvimp` and `crvexp` must be read over and their 2 other components, `ivar=iv(ihpas)` and `ivar=iw` must be cancelled.

<sup>25</sup>indicator `isno2t` for the velocity, `IST02T` for the turbulence and `isso2t` for the scalars

### 6.6.2 For $k$ and $\varepsilon$

*Subroutine called every time step, in  $k - \varepsilon$  and in  $v2f$ .*

The subroutine **ustske** is used to add source terms to the transport equations related to the turbulent kinetics energy  $k$  and to the turbulent dissipation  $\varepsilon$ . This subroutine is called every time step (the treatment of the two variables  $k$  and  $\varepsilon$  is made simultaneously). The user is expected to provide the arrays **crkimp** and **crkexp** for  $k$ , and **creimp** and **creexp** for  $\varepsilon$ . These arrays are similar to the arrays **crvimp** and **crvexp** given for the velocity in the user subroutine **ustsns**. The way of making implicit the resulting source terms is the same as the one presented in **ustsns**. For  $\varphi$  and  $\bar{f}$  in the  $v2f$  model, see **ustsv2**, §6.6.4.

### 6.6.3 For $R_{ij}$ and $\varepsilon$

*Subroutine called every time step, in  $R_{ij} - \varepsilon$ .*

The subroutine **ustsri** is used to add source terms to the transport equations related to the Reynolds stress variables  $R_{ij}$  and to the turbulent dissipation  $\varepsilon$ . This subroutine is called 7 times every time step (once for each Reynolds stress component and once for the dissipation). The user must provide the arrays **crvimp** and **crvexp** for the variable **ivar** (referring successively to **ir11**, **ir22**, **ir33**, **ir12**, **ir13**, **ir23** and **iep**). These arrays are similar to the arrays **crvimp** and **crvexp** given for the velocity in the user subroutine **ustsns**. The method for impliciting the resulting source terms is the same as that presented in **ustsns**.

### 6.6.4 For $\varphi$ and $\bar{f}$

*Subroutine called every time step, in  $v2f$ .*

The subroutine **ustsv2** is used to add source terms to the transport equations related to the variables  $\varphi$  and  $\bar{f}$  of the  $v2f$   $\varphi$ -model. This subroutine is called twice every time step (once for  $\varphi$  and once for  $\bar{f}$ ). The user is expected to provide the arrays **crvimp** and **crvexp** for **ivar** referring successively to **iphi** and **ifb**. Concerning  $\varphi$ , these arrays are similar to the arrays **crvimp** and **crvexp** given for the velocity in the user subroutine **ustsns**. Concerning  $\bar{f}$ , the equation is slightly different:

$$L^2 \text{div}(\text{grad}(\bar{f})) = \bar{f} + \dots + S_{\text{impl}} \times \bar{f} + S_{\text{expl}}$$

In the finite volume formulation, the solved system is written as:

$$\int_{\partial\Omega_i} \text{grad}(\bar{f})^{(n+1)} dS = \frac{1}{L_i^2} \left( \Omega_i \bar{f}_i^{(n+1)} + \dots + \Omega_i S_{\text{impl},i} \bar{f}_i^{(n+1)} + \Omega_i S_{\text{expl},i} \right)$$

The user must then specify:

$$\text{crvimp}_i = \Omega_i S_{\text{impl},i}$$

$$\text{crvexp}_i = \Omega_i S_{\text{expl},i}$$

The way of making implicit the resulting source terms is the same as the one presented in **ustsns**.

### 6.6.5 For $k$ and $\omega$

*Subroutine called every time step, in  $k - \omega$ .*

The subroutine **ustskw** is used to add source terms to the transport equations related to the turbulent kinetics energy  $k$  and to the specific dissipation rate  $\omega$ . This subroutine is called every time step (the treatment of the two variables  $k$  and  $\omega$  is made simultaneously). The user is expected to provide the arrays **crkimp** and **crkexp** for the variable  $k$ , and the arrays **crwimp** and **crwexp** for the variable  $\omega$ . These arrays are similar to the arrays **crvimp** and **crvexp** given for the velocity in the user subroutine **ustsns**. The way of making implicit the resulting source terms is the same as the one presented in **ustsns**.

## 6.6.6 For user scalars

*Subroutine called every time step.*

The subroutine `ustssc` is used to add source terms to the transport equations related to the user scalars (passive or not, average of the square of the fluctuations of a scalar, ...). In the same way as `ustsns`, this subroutine is called every time step, once for each user scalar. The user needs to provide the arrays `crvimp` and `crvexp` related to each scalar. `cvimp` and `crvexp` must be set to 0 for the scalars on which it is not wished for the user source term to be applied (the arrays are initially set to 0 at each inlet in the subroutine.)

## 6.7 Pressure drops (head losses)

Pressure drops can be defined in the Gaphical User Interface (GUI) or in the subroutine `uskpdc` (called three times every time step). In the GUI, under the heading “Volume conditions”, the item “Volume regions definition” allows to define areas where pressure drops occur, see an example in fig 31. The item “Head losses” allows to specify the head loss coefficients, see fig 32. The tensor representing the pressure drops is supposed to be symmetric and positive.

If necessary, the pressure drops are written in the subroutine `uskpdc`.

- During the first call, all the cells are checked to know the number of cells in which a pressure drop is present. This number is called `nceptdp` in `uskpdc` (and corresponds to `nceptdc`). It is used to lay out the arrays related to the pressure drops. If there is no pressure drop, `nceptdp` must be equal to zero (it is the default value, and the rest of the subroutine is then useless).
- During the second call, all the cells are checked again to complete the array `icepdp` whose size is `nceptdp`. `icepdc(ielpdc)` is the number of the `ielpdc`<sup>th</sup> cell containing pressure drops.
- During the third call, all the cells containing pressure drops are checked in order to complete the array containing the components of the tensor of pressure drops `ckupdc(nceptdp, 6)`. This array is so that the equation related to the velocity may be written:

$$\rho \frac{\partial}{\partial t} \underline{u} = \dots - \rho \underline{\underline{K}}_{pdc} \cdot \underline{u}$$

The tensor components are given in the following order (in the general reference frame): `k11`, `k22`, `k33`, `k12`, `k13`, `k23` with `k12`, `k13` and `k23` being zero if the tensor is diagonal.

The three calls are made every time step, so that variable pressure drop zones or values may be treated.

## 6.8 Management of the mass sources

The subroutine `ustsma` is used to add a density source term in some cells of the domain (called at each time step). The mass conservation equation is then modified as follows:

$$\frac{\partial \rho}{\partial t} + \text{div}(\rho \underline{u}) = \Gamma$$

$\Gamma$  is the mass source term expressed in  $kg.m^{-3}.s^{-1}$ .

The presence of a mass source term modifies the evolution equation of the other variables, too. Let  $\varphi$  be a any solved variable apart from the pressure (velocity component, turbulent energy, dissipation, scalar, ...). Its evolution equation becomes:

$$\rho \frac{\partial \varphi}{\partial t} + \dots = \dots + \Gamma(\varphi_i - \varphi)$$

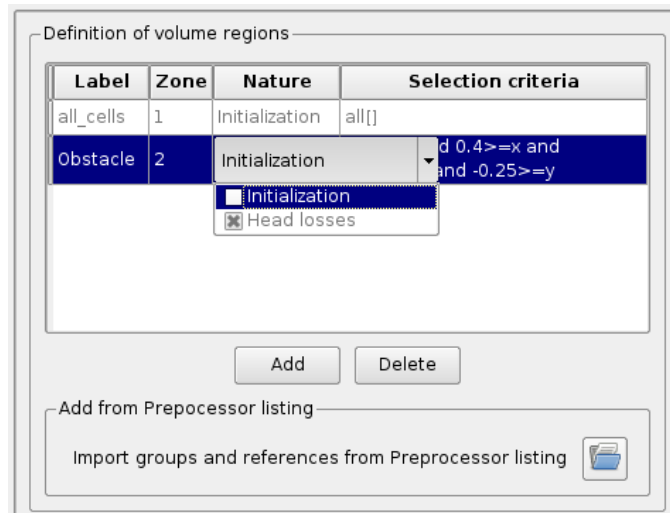
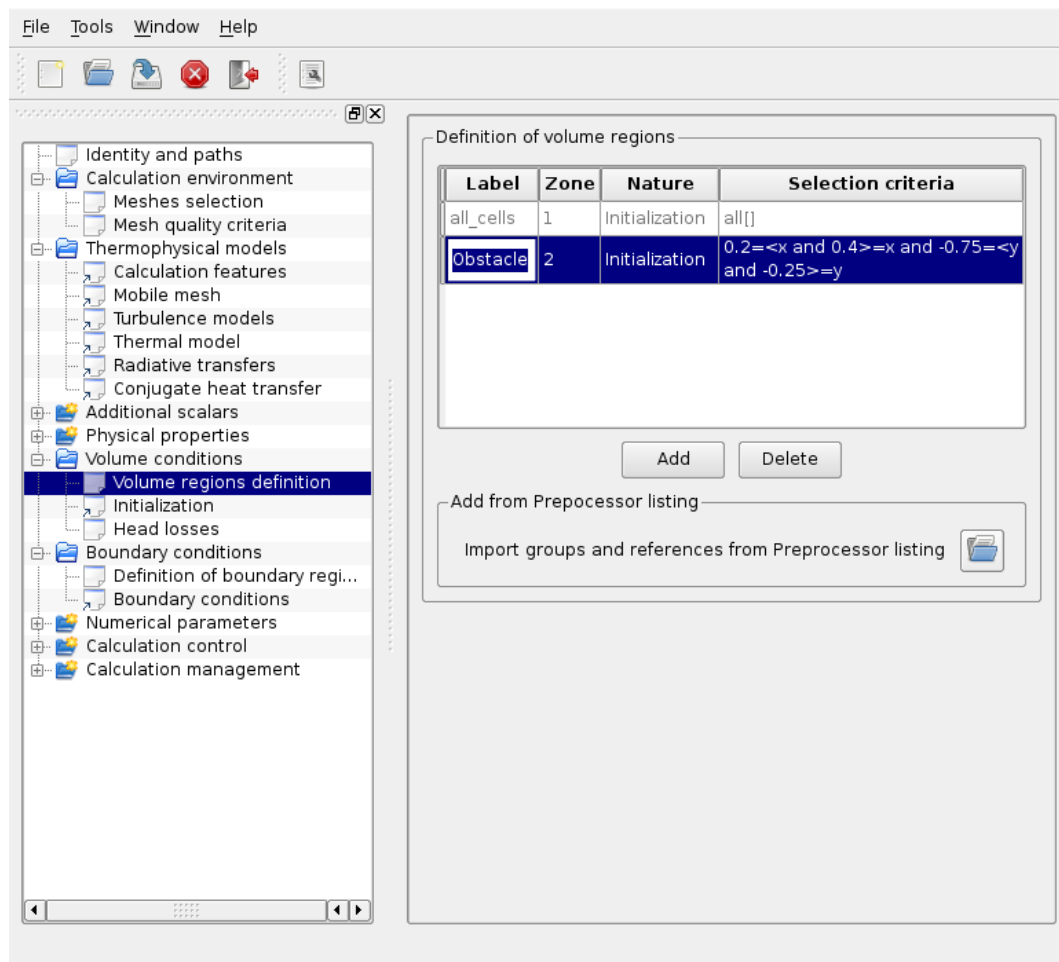


Figure 31: Creation of head losses region

$\varphi_i$  is the value of  $\varphi$  associated with the mass entering or leaving the domain. After discretisation, the

Select volume zone for head losses

Label	Zone	Selection criteria
Obstacle	2	0.2=<x and 0.4>=x and -0.75=...

Tensor coefficients

Head losses coefficients:  $K_{ii} = 0.5 \alpha_{ii} |U|$

$\alpha_{xx}$  
 $\alpha_{yy}$  
 $\alpha_{zz}$

☐ Reference frame transformation matrix

Figure 32: Head losses coefficients

equation may be written:

$$\rho \frac{\varphi^{(n+1)} - \varphi^{(n)}}{\Delta t} + \dots = \dots + \Gamma(\varphi_i - \varphi^{(n+1)})$$

For each variable  $\varphi$ , there are two possibilities:

- We can consider that the mass is added (or removed) with the ambient value of  $\varphi$ . In this case  $\varphi_i = \varphi^{(n+1)}$  and the equation of  $\varphi$  is not modified.
- Or we can consider that the mass is added with an imposed value  $\varphi_i$  (this solution is physically correct only when the mass is effectively added,  $\Gamma > 0$ ).

This subroutine is called three times every time step.

- During the first call, all the cells are checked to know the number of cells containing a mass source term. This number is called `ncesmp` in `ustsma` (and corresponds to `ncetsm`). It is used to lay out the arrays related to the mass sources. If there is no mass source, `ncesmp` must be equal to zero (it is the default value, and the rest of the subroutine is then useless).
- During the second call, all the cells are checked again to complete the array `icetsm` whose dimension is `ncesmp`. `icetsm(ieltsm)` is the number of the `ieltsm`<sup>th</sup> cell containing a mass source.
- During the third call, all the cells containing mass sources are checked in order to complete the arrays `itypsm(ncesmp,nvar)` and `smacel(ncesmp,nvar)`:
  - `itypsm(ieltsm,ivar)` is the flow type associated with the variable `ivar` in the `ieltsm`<sup>th</sup> cell containing a mass source.
    - `itypsm=0`:  $\varphi_i = \varphi^{(n+1)}$  condition
    - `itypsm=1`: imposed  $\varphi_i$  condition
    - `itypsm` is not used for `ivar=ipr`
  - `(ieltsm,ipr)` is the value of the mass source term  $\Gamma$ , in  $kg.m^{-3}.s^{-1}$ .
  - `smacel(ieltsm,ivar)`, for `ivar` different from `ipr`, is the value of  $\varphi_i$  for the variable `ivar` in the `ieltsm`<sup>th</sup> cell containing a mass source.

#### NOTES

- If `itypsm(ieltsm,ivar)=0`, `smacel(ieltsm,ivar)` is not used.

- If  $\Gamma = \text{smacel}(\text{ieltsm}, \text{ipr}) < 0$ , mass is removed from the system, and *Code\_Saturne* considers automatically a  $\varphi_i = \varphi^{(n+1)}$  condition, whatever the values given to  $\text{itypsm}(\text{ieltsm}, \text{ivar})$  and  $\text{smacel}(\text{ieltsm}, \text{ivar})$  (the extraction of a variable is done at ambient value).

The three calls are made every time step, so that variable mass source zones or values may be treated.

For the variance, do not take into account the scalar  $\varphi_i$  in the environment where  $\varphi \neq \varphi_i$  generates a variance source.

## 7 Results analysis

### 7.1 Management of the post-processing intermediate outputs

The subroutine `usnpst` is used to specify when post-processing outputs will be generated (it is called at each time step even if the user hasn't moved it to the directory SRC). By default, it tests if the current time step number (`ntcabs`) is a multiple of the chosen output frequency (`ntchr`). If it is the case, the indicator `iipost` turns to 1, which triggers the writing of an intermediate output. If the frequency is given a negative value, the test is not performed.

For instance, a user who wants to generate post-processing outputs (also called “chronological outputs”) at the time step number 36 and around the physical time  $t=12$  seconds may use the following test:

<code>iipost = 0</code>	No output by default.
<code>if (ntcabs.eq.36) then</code>	If the current time step is the 36 <sup>th</sup> ,
<code>iipost=1</code>	generate an output.
<code>endif</code>	End of the test on the time step number.
<code>if (abs(ttcabs-12.d0).le.0.01d0) then</code>	If the physical time is 12s +/- 0.01s,
<code>iipost=1</code>	generate an output.
<code>endif</code>	End of the test on the physical time.

In any case, a post-processing output is generated after the last time step, `usnpst` being used or not.

## 7.2 Definition of post-processing and mesh zones

The functions defined in `cs_user_postprocess.c`, namely `cs_user_postprocess_writers`, `cs_user_postprocess_mesh` and `cs_user_postprocess_activate` allow for the definition of postprocessing output formats and frequency, and for the definition of surface or volume sections, in the form of lists of `nlfac` internal faces (`1stfac`) and `nlfab` boundary faces (`1stfab`), or of `nlcel` cells (`1stcel`), in order to generate chronological outputs in *EnSight*, *MED* or *CGNS* format.

One or several writers can be associated with each post-processing mesh, or “part” created. The arguments of the function `cs_post_define_writer` are as follows:

- **writer\_id**: id the the associated writer.  
negative ids are reserved (-1 for the main output), but the matching writer's options may be redefined by calls to this function.
- **case\_name**: basic name of the associated case.  
*WARNING*: depending on the chosen format, this name may be shortened (maximum number of characters: 32 for *MED*, 19 for *EnSight*) or modified automatically (whitespaces or forbidden characters will be replaced by '\_')
- **dir\_name**: name of the output directory
- **fmt\_name**: choice of the output format:
  - *EnSight Gold* (*EnSight* also accepted)
  - *MED-fichier* (*MED* also accepted)
  - *CGNS*

The options are not case-sensitive, so *ensight* or *cgns* are valid, too.

- **fmt\_opts**: character string containing a list of options related to the format, separated by commas; for the *EnSight Gold* format, these options are:
  - *binary* for a binary format version (by default)
  - *text* for a text format version
  - *discard\_polygons* to prevent from exporting faces with more than four edges (which may not be recognized by some post-processing tools); such faces will therefore not appear in the post-processing mesh.
  - *discard\_polyhedra* to prevent from exporting elements which are neither tetrahedra, prisms, pyramids nor hexahedra (which may not be recognized by some post-processing tools); such elements will therefore not appear in the post-processing mesh.
  - *divide\_polygons* to divide faces with more than four edges into triangles, so that any post-processing tool can recognize them
  - *divide\_polyhedra* to divide elements which are neither tetrahedra, prisms, pyramids nor hexahedra into simpler elements (tetrahedra and pyramids), so that any post-processing tool can recognize them
  - *split\_tensor* to export the components of a tensor variable as a series of independent variables (a variable is recognised as a tensor if its dimension is 6 or 9); not implemented yet.
- **time\_dep**: indicates if the post-processing (i.e. visualization) meshes (or “parts”) are:
  - `FVM_WRITER_FIXED_MESH` fixed (usual case)
  - `FVM_WRITER_TRANSIENT_COORDS` deformable (the vertex positions may vary over time)
  - `FVM_WRITER_TRANSIENT_CONNECT` modifiable: (the lists of cells or faces defining these meshes can be changed over time)



- **output\_at\_end**: force output at calculation end if not 0
- **frequency\_n**: default output frequency in time steps associated with this writer, or  $< 0$  (the output may be forced or prevented at any time step using the function `cs_user_postprocess_activate`)
- **frequency\_t**: default output frequency in seconds associated with this writer, or  $< 0$  (has priority over **frequency\_n**, and the output may be forced or prevented at any time step using the function `cs_user_postprocess_activate`)

In order to allow the user to add an output format to the main output format, or to add a mesh to the default output, the lists of standard and user meshes and writers are not separated. Negative numbers are reserved for the non-user items. For instance, the mesh numbers -1 and -2 correspond respectively to the global mesh and to boundary faces, generated by default, and the writer -1 corresponds to the usual post-processing case defined *via* `usini1` or *via* the interface.

The user chooses the numbers corresponding to the post-processing meshes and writers he wants to create. These numbers must be positive integers. It is possible to associate a user mesh with the standard post-processing case (-1), or to ask for outputs regarding the boundary faces (-2) associated with a user writer.

For safety, the output frequency and the possibility to modify the post-processing meshes are associated with the writers rather than with the meshes. This logic avoids unwanted generation of inconstituent post-processing outputs. For instance EnSight would not be able to read a case in which one field is output to a given part every 10 time steps while another field is output to the same part every 200 time steps.

The possibility to modify a mesh over time is limited by the most restrictive writer which is associated with. For instance, if writer 1 allows the modification of the mesh topology (argument `time_dep = FVM_WRITER_TRANSIENT_CONNECT` in the call to `cs_post_define_writer`) and writer 2 allows no modification (`time_dep = FVM_WRITER_FIXED_MESH`), a user post-processing mesh associated with the writers 1 and 2 will not be modifiable, but a mesh associated only with the writer 1 will be modifiable. The modification is done by means of the user subroutine `usmpst`, which is called once per time step and per modifiable mesh.

It is possible to output variables which are normally automatically output on the main volume or boundary meshes to a user mesh which is a subset of one of these by setting the `auto_variables` argument of one of the `cs_post_define..._mesh` to `true`.

It is also possible to define an alias of a post-processing mesh. An alias shares all the attributes of its parent mesh (without duplication), except its number. This may be used to output different variables on a same mesh with 2 different writers: the choice of output variables is based on the mesh, so if  $P_a$  is associated with writer  $W_a$ , all that is needed is to define an alias  $P_b$  to  $P_a$  and associate it with writer  $W_b$  to allow a different output variable selection with each writer. An alias may be created using the `pstalm` subroutine.

Modification of a postprocessing mesh or its alias over time is always limited by the most restrictive "writer" to which its meshes have been associated (parts of the structures being shared in memory). It is possible to define as many aliases as are required for a true mesh, but an alias cannot be defined for another alias.

It is not possible to mix cells and faces in the same mesh (most of the post-processing tools being perturbed by such a case)<sup>26</sup>.

For a better understanding, the user may refer to the examples given in `cs_user_postprocess_meshes`. We can note that the whitespaces in the beginning or in the end of the character strings given as arguments of the functions called are suppressed automatically.

<sup>26</sup>actually, faces adjacent to selected cells and belonging to face or cell groups may be selected when the `add_groups` of `cs_post_define..._mesh` is set to `true`, so as to maintain group information, but those faces will only be written for formats supporting this (such as MED), and will only bear groups, not variable fields

The additional variables to post-process on the defined meshes will be specified in the subroutine `usvpst`. “

*WARNING In the parallel case, some meshes may not contain any local elements on a given processor. This is not a problem at all, as long as the mesh is defined for all processors (empty or not). It would in fact not be a good idea at all to define a postprocessing mesh only if it contains local elements, global operations on that mesh would become impossible, leading to probable deadlocks or crashes.*

## 7.3 Modification of the mesh zones to post-process

*Subroutine called only for each modifiable “part”, at every active time step of an associated “writer”.*

The subroutine `usmpst` is used to modify the lists of cells, internal and boundary faces which define a “user part” (or post-processing mesh) defined through the user subroutine `usdpst` and associated only with “writers”; allowing “part” modifications over time (*i.e.* created with the parameter `indmod` = 2).

At first, the corresponding lists contain the previously defined values. If these lists are modified for a given post-processing mesh, the argument `imodif` must be given the value 1. If this argument maintains its initial value of 0, the code will not consider this “part” to have been modified away from that call and it will offer to bring it up to date. It is in fact at the end of an optimisation so there is no need to modify these “parts” within the definite and modifiable assembly (if in doubt, let `imodif`=1).

Note that the `itypps` flag can be used to determine whether the current post-processing mesh contains cells (`itypps`(1) = 1), internal faces (`itypps`(2) = 1), or boundary faces (`itypps`(2) = 1) globally (as the number of local cells or faces of a processor could be 0, it doesn’t provide sufficient information). If at any time, a given part contains no element of any type, all the values of `itypes` will be 0 and that number cannot be put in the part (`nummai`) to determine if it will affect the cells or faces<sup>27</sup>.

The user may refer to the example, in which cells are selected according to a given criterion:

- For a volume “part”, cells for which the velocity exceeds a certain value.
- For a surface “part”, interior faces which are between a cell in which the velocity exceeds a certain value and a cell in which the velocity is lower than this value (and boundary faces neighboring a cell in which the velocity exceeds this value). This surface post-processing mesh corresponds therefore to an approximation of a velocity isosurface.

## 7.4 Definition of the variables to post-process

For the parts defined in `usdpst`, the subroutine `usvpst` is used to specify the variables to post-process (called for each “part”, at every active time step of an associated “writer”, see `usdpst`).

The output of a given variable is generated by means of a call to `psteval`, whose arguments are:

- `nummai`: current “part” number (input argument in `usvpst`).
- `namevr`: name to give to the variable.
- `idimt`: dimension of the variable (3 for a vector, 1 for a scalar).
- `ientla`: indicates if the stored arrays are “interlaced” or not:
  - 0: not interlaced, in the form  $\{x_1, x_2, \dots, x_n, y_1, y_2, \dots, y_n, z_1, z_2, \dots, z_n\}$  (case of all variables defined in `rtp`).
  - 1: interlaced, in the form  $\{x_1, y_1, z_1, x_2, y_2, z_2, \dots, x_n, y_n, z_n\}$  (case of the geometric parameters, like `xyzcen`, `surfbo`, ...).

For a scalar variable, this argument does not matter.

<sup>27</sup>It is not expressly forbidden to associate cells with the “part” at a certain timestep and faces at another, but this has not been tested

- **ivarpr**: indicates if the variable is defined on the “parent” mesh or locally:
  - 0: variable generated by the user in the given work arrays **tracel**, **trafac**, and **trafbr** (whose size is respectively the number of cells, internal faces and boundary faces of the “part”,  $\times 3$ ). The arrays **lstcel**, **lstfac**, and **lstfbr** can be used to get the numbers corresponding to the cells, internal faces and boundary faces associated with the “part” and to generate the appropriate post-processing variable.
  - 1: variable already defined in the main mesh (“parent” mesh of the “parts”), for example the variables in the **rtp** array. Instructions in the report which list **lstcel**, **lstfac**, and **lstfbr** will be treated directly by the sub routine, avoiding unused copies and simplifying the code
- **ntcabs**: absolute current time step number. If a negative value is given (usually -1), the variable will be regarded as time-independent (and we will have to make sure this call is only made once).
- **ttcabs**: current physical time value. It is not taken into account if **ntcabs** < 0.
- **tracel**: array containing the values of the variable at the cells. If **ivarpr** = 1, this argument will be replaced by the position of the beginning of the array on which the variable is defined, for instance **rtp(1, iu(1))** for the velocity.
- **trafac**: equivalent of **tracel** for the internal faces.
- **trafbr**: equivalent of **tracel** for the boundary faces.

The user may refer to the example, which presents the different ways of generating an output of a variable.

*WARNING: Apart from the time-independent variables, it is not recommended not to generate the same variables at every call (corresponding to an active time step) for a given mesh, because the post-processing tool may have difficulties to deal with such a case. To generate outputs of different variables on the same mesh with different frequencies, it is recommended to create an alias of this mesh and to associate it with a different “writer” in the subroutine **usdpst**.*

## 7.5 Modification of the variables at the end of a time step

The subroutine **usproj** is called at the end of every time step. It is used to print or modify any variable at the end of every time step.

Several examples are given:

- Calculation of a thermal balance at the boundaries and in the domain (including the mass source terms)
- Modification of the temperature in a given area starting from a given time
- Extraction of a 1D profile, see fig. 25
- Printing of a moment
- Utilisation of utility subroutines useful in the case of a parallel calculation (calculation of a sum on the processors, of a maximum, ...)

*WARNING: As all the variables (solved variables, physical properties, geometric parameters) can be modified in this subroutine, a wrong use may distort totally the calculation.*

The thermal balance example is particularly interesting.

- It can be easily adapted to another scalar (only three simple modifications to do, as indicated in the subroutine).
- It shows how to make a sum on all the subdomains in the framework of a parallel calculation (see the calls to the subroutines `par*`).
- It shows the precautions to take before doing some operations in the framework of periodic or parallel calculations (in particular when we want to calculate the gradient of a variable or to have access to values at the cells neighboring a face).
- Finally it must not be forgotten that the resolution with temperature as a solved variable is questionable when the specific heat is not constant.

## 7.6 Non-standard management of the chronological record files

The interface and the subroutine `usini1` allow to manage the “automatic” chronological record files in an autonomous way: position of the probes, printing frequency and related variables. The results are written in a different file for each variable. These files are written in *xmgrace* or *gnuplot* format and contain the profiles corresponding to every probe. This type of output format may not be well adapted if, for instance, the number of probes is too high. The subroutine `ushist`, called at each time step, allows then to personalise the output format of the chronological record files. The version given as example in the directory works as follows:

- Positionning of the probes (only at the first passage): the index `ii` varies between 1 and the number of probes. The coordinates `xx`, `yy` and `zz` of each probe are given. The subroutine `findpt` gives then the number `icapt(ii)` of the cell center which is the closest to the defined probe.
- Opening of the output files (only at the first pass): in the version given as example, the program opens a different file for all the `nvar` variables. `ficush(j)` contains the name of the  $J^{\text{th}}$  file and `impush(j)` its unit number (`impush` is initialised by default so that the user has at his disposal specific unit numbers and does not run the risk to overwrite an already open file).
- Writing to the files: in the version given as example, the program writes the time step number, the physical time step (based on the standard time step in the case of a variable time step) and the value of the selected variable at the different probes.
- Closing of the files (only at the last time step).

*WARNING: The use of `ushist` neither erases nor replaces the parameters given in the interface or in `usini1`. Therefore, in the case of the use of `ushist`, and to avoid the creation of useless files, the user should set `ncapt=0` in the interface or in `usini1` to deactivate the automatic production of chronological records.*

In addition, `ushist` generates supplementary result files.

## 8 Advanced modelling setup

### 8.1 Use of a specific physics

Specific physics such as dispersed phase, atmospheric flows and coal combustion models can be added by the user from the interface, or by using the subroutine `usppmo` (called only during the calculation initialisation). With the interface, when a specific physics is activated in fig. 33, additional items or headings may appear (see for instance Sections 8.6.1 and 8.2.0.1).

When the interface is not used, `usppmo` is one of the three subroutines which must be obligatory completed by the user in order to use a specific physics module. Also, some specific physics modules

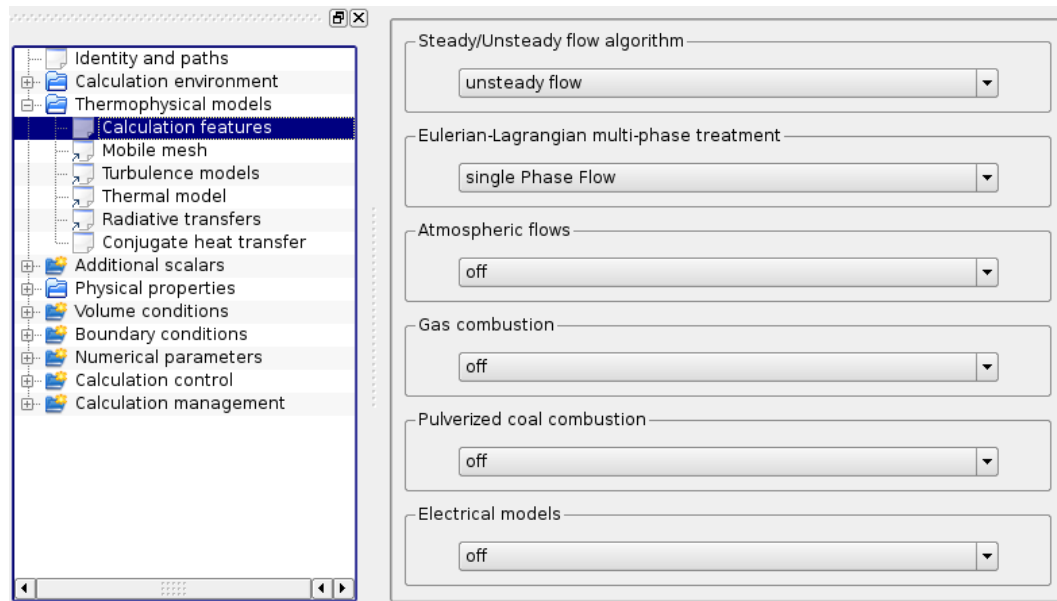


Figure 33: Thermophysical models selection

can not yet be activated through the interface such as the modules listed below which were not quoted at the beginning of this section. At the moment, *Code\_Saturne* allows to use two “pulverised coal” modules (with Lagrangian coupling or not), two “gas combustion” modules, two “electrical” modules, a “compressible” module, an “cooling towers” module and an “atmospheric” module. To activate one of these modules, the user needs to complete one (and only one) of the indicators `ippmod(i.....)` in the subroutine `usppmo`. By default, all the indicators `ippmod(i.....)` are initialised at -1, which means that no specific physics is activated.

- Diffusion flame in the framework of “3 points” rapid complete chemistry: indicator `ippmod(icod3p)`
  - `ippmod(icod3p) = 0` adiabatic conditions
  - `ippmod(icod3p) = 1` permeatic conditions (enthalpy transport)
  - `ippmod(icod3p) = -1` module not activated
- Eddy Break Up pre-mixed flame: indicator `ippmod(icoebu)`
  - `ippmod(icoebu) = 0` adiabatic conditions at constant richness
  - `ippmod(icoebu) = 1` permeatic conditions at constant richness
  - `ippmod(icoebu) = 2` adiabatic conditions at variable richness
  - `ippmod(icoebu) = 3` permeatic conditions at variable richness
  - `ippmod(icoebu) = -1` module not activated
- Libby-Williams pre-mixed flame: indicator `ippmod(icolwc)`
  - `ippmod(icolwc)=0` two peak model with adiabatic conditions.
  - `ippmod(icolwc)=1` two peak model with permeatic conditions.
  - `ippmod(icolwc)=2` three peak model with adiabatic conditions.
  - `ippmod(icolwc)=3` three peak model with permeatic conditions.
  - `ippmod(icolwc)=4` four peak model with adiabatic conditions.
  - `ippmod(icolwc)=5` four peak model with permeatic conditions.

- `ippmod(icolwc)=-1` module not activated.
- Multi-coals and multi-classes pulverised coal combustion: indicator `ippmod(icp3pl)` The number of different coals must be inferior or equal to `ncharm = 3`. The number of particle size classes `nclpch(icha)` for the coal `icha`, must be inferior or equal to `ncpcmx = 10`.
  - `ippmod(icp3pl) = 0` imbalance between the temperature of the continuous and the solid phases
  - `ippmod(icp3pl) = 1` otherwise
  - `ippmod(icp3pl) = -1` module not activated
- Lagrangian modeling of multi-coals and multi-classes pulverised coal combustion: indicator `ippmod(icpl3c)` The number of different coals must be inferior or equal to `ncharm = 3`. The number of particle size classes `nclpch(icha)` for the coal `icha`, must be inferior or equal to `ncpcmx = 10`.
  - `ippmod(icpl3c) = 1` coupling with the Lagrangian module, with transport of  $H_2$
  - `ippmod(icpl3c) = -1` module not activated
- Electric arc module (Joule effect and Laplace forces): indicator `ippmod(ielarc)`
  - `ippmod(ielarc) = 1` determination of the magnetic field by means of the Ampere's theorem (not available)
  - `ippmod(ielarc) = 2` determination of the magnetic field by means of the vector potential
  - `ippmod(ielarc) = -1` module not activated
- Joule effect module (Laplace forces not taken into account): indicator `ippmod(ieljou)`
  - `ippmod(ieljou) = 1` use of a real potential
  - `ippmod(ieljou) = 2` use of a complex potential
  - `ippmod(ieljou) = 3` use of real potential and specific boundary conditions for transformers.
  - `ippmod(ieljou) = 4` use of complex potential and specific boundary conditions for transformers.
  - `ippmod(ieljou) = -1` module not activated
- Compressible module: indicator `ippmod(icompf)`
  - `ippmod(icompf) = 0` module activated
  - `ippmod(icompf) = -1` module not activated
- atmospheric flow module: indicator `ippmod(iatmos)`
  - `ippmod(iatmos) = -1` module not activated
  - `ippmod(iatmos) = 0` standard modelling
  - `ippmod(iatmos) = 1` dry atmosphere
  - `ippmod(iatmos) = 2` humid atmosphere (NOT functional)
- cooling towers module: indicator `ippmod(iaeros)`
  - `ippmod(iaeros) = -1` module not activated
  - `ippmod(iaeros) = 0` no model (NOT functional)
  - `ippmod(iaeros) = 1` Poppe's model
  - `ippmod(iaeros) = 2` Merkel's model

**WARNING:** Only one specific physics module can be activated at the same time.

In the framework of the gas combustion modeling, the user may impose his own enthalpy-temperature tabulation (conversion law). He needs then to give the value zero to the indicator `indjon` (the default value being 1). For more details, the user may refer to the following note (thermo-chemical files).

NOTE: THE THERMO-CHEMICAL FILES

The user must not forget to place in the directory DATA the thermo-chemical file `dp_FCP`, `dp_C3P`, `dp_C3PSJ` or `dp_ELE` (depending on the specific physics module he activated) and to specify the name of this file in the variable `THERMOCHEMISTRY_DATA` in the launch script (for instance: `THERMOCHEMISTRY_DATA"dp_C3P"`). Some example files are placed in the directory `DATA/THCH` at the creation of the study case. Their content is described below.

- Example of file for the gas combustion:

→ if the enthalpy-temperature conversion data base JANAF is used: `dp_C3P` (see array 1).

Lines	Examples of values	Variables	Observations
1	5	<code>ngaze</code>	Number of current species
2	10	<code>npo</code>	Number of points for the enthalpy-temperature tabulation
3	300.	<code>tmin</code>	Temperature inferior limit for the tabulation
4	3000.	<code>tmax</code>	Temperature superior limit for the tabulation
5			Empty line
6	CH4 O2 CO2 H2O N2	<code>nomcoe(ngaze)</code>	List of the current species
7	.35 .35 .35 .35 .35	<code>kabse(ngaze)</code>	Absorption coefficient of the current species
8	4	<code>nato</code>	Number of elemental species
9	.012 1 0 1 0 0	<code>wmolat(nato),</code>	Molar mass of the elemental species (first column)
10	.001 4 0 0 2 0	<code>atgaze(ngaze,nato)</code>	Composition of the current species as a function of the elemental species ( <code>ngaze</code> following columns)
11	.016 0 2 2 1 0		
12	.014 0 0 0 0 2		
13	3	<code>ngazg</code>	Number of global species Here, <code>ngazg</code> = 3 (Fuel, Oxidiser and Products)
14	1. 0. 0. 0. 0.	<code>compog(ngaze,ngazg)</code>	Composition of the global species as a fonction of the current species of the line 6 In the order: Fuel (line 15), Oxidiser (line 16) and Product (line 17)
15	0. 1. 0. 0. 3.76		
16	0. 0. 1. 2. 7.52		
17	1	<code>nrgaz</code>	Number of global reactions Here <code>nrgaz</code> = 1 (always equal to 1 in this version)
18	1 2 -1 -9.52 10.52	<code>igfuel(nrgaz),</code> <code>igoxy(nrgaz),</code>  <code>stoeg(ngazg,nrgaz)</code>	Numbers of the global species concerned by the stoichiometric ratio (first 2 integers) Stoichiometry in reaction global species. Negative for the reactants (here "Fuel" and "Oxidiser") and positive for the products (here "Products")

Table 1: Example of file for the gas combustion when JANAF is used: `dp_C3P`

→ if the user provides his own enthalpy-temperature tabulation (there must be three chemical species and only one reaction): `dp_C3PSJ` (see array 2). This file replaces `dp_C3P`.

- Example of file for the pulverised coal combustion: `dp_FCP` (see array 3).



Lines	Examples of values	Variables	Observations
1	6	<b>npo</b>	Number of tabulation points
2	50. -0.32E+07 -0.22E+06 -0.13E+08	<b>th(npo), ehgazg(1,npo), ehgazg(2,npo), ehgazg(3,npo)</b>	Temperature(first column), mass enthalpy of fuel, oxidiser and products (columns 2,3 and 4) from line 2 to line <b>npo</b> +1
3	250. -0.68E+06 -0.44E+05 -0.13E+08		
4	450. 0.21E+07 0.14E+06 -0.13E+08		
5	650. 0.50E+07 0.33E+06 -0.12E+08		
6	850. 0.80E+07 0.54E+06 -0.12E+08		
7	1050. 0.11E+08 0.76E+06 -0.11E+08		
8	.00219 .1387 .159	<b>wmolg(1), wmolg(2), wmolg(3)</b>	Molar mass of fuel, oxidiser and products
9	.11111	<b>fs(1)</b>	Mixing rate at the stoichiometry (relating to Fuel and Oxidiser)
10	0.4 0.5 0.87	<b>ckabsg(1), ckabsg(2), ckabsg(3)</b>	Absorption coefficient of fuel, oxidiser and products
11	1. 2.	<b>xco2, xh2o</b>	Molar coefficients of $CO_2$ and $H_2O$ in the products (radiation using Modak)

Table 2: Example of file for the gas combustion when the user provides his own enthalpy-temperature tabulation (there must be three species and only one reaction): **dp\_C3PSJ** (this file replaces **dp\_C3P**)

- Example of file for the electric arc: **dp\_ELE** (see array 4).



Lines	Examples of values	Variables	Observations
1	THERMOCHEMISTRY		Comment line
2	8	ncoel	Number of current species
3	8	npo	Number of points for the enthalpy-temperature tabulation
4	CURRENT SPECIES		Comment line
5	CH4 C2H4 CO O2 CO2 H2O N2 C(S)	nomcoel(ncoel)	List of the current species
6	300.	tmin	Temperature inferior limit (Kelvin) for the enthalpy-temperature tabulation
7	2400.	tmax	Temperature superior limit (Kelvin) for the enthalpy-temperature tabulation
8	4	nato	Number of elementary species
9	.012 1 2 1 0 1 0 0 1		Molar mass of the elemental species (first column)
10	.001 4 4 0 0 0 2 0 0	vmolat(nato),	and composition of the current species as a function of the elemental species
11	.016 0 0 1 2 2 1 0 0	atcoel(ncoel,nato)	
12	.014 0 0 0 0 0 0 2 0		
13	RADIATION		Comment line
14	0.1	ckabs1	Constant absorption coefficient for the gas mixture
15	COAL CHARACTERISTICS		Comment line
16	2	ncharb	Number of coal types
17	1 1	nclpch(ncharb)	Number of classes for each coal (each column corresponding to one coal type )
18	50.E-6 50.E-6	diam20(nclacp)	Initial diameter of each class (m) nclacp is the total number of classes. All the diameters are written on the same line (sucessively for each coal, we give the diameter corresponding to each class)
19	74.8 60.5	cch(ncharb)	Composition in C (mass.-%, dry) of each coal
20	5.1 4.14	hch(ncharb)	Composition in H (mass.-%, dry) of each coal
21	12.01 5.55	och(ncharb)	Composition in O (mass.-%, dry) of each coal
22	0 31524000. 0 31524000.	ipci(ncharb) pcich(ncharb)	Value of the PCI ( $Jkg^{-1}$ ) for each coal, the first integer indicating if this value refers to pure (0) or dry coal (1)
23	1800. 1800.	cp2ch(ncharb)	Heat-storage capacity at constant pressure ( $Jkg^{-1}K^{-1}$ ) for each coal
24	1200. 1200.	rho0ch(ncharb)	Initial density ( $kgm^{-3}$ ) of each
25	Coke		Comment line
26	0. 0.	cck(ncharb)	Composition in C (mass.-%, dry) of the coke for each coal
27	0. 0.	hck(ncharb)	Composition in H (mass.-%, dry) of the coke for each coal
28	0. 0.	ock(ncharb)	Composition in O (mass.-%, dry) of the coke for each coal
29	0. 0.	pcick(ncharb)	PCI of the dry coke ( $Jkg^{-1}$ ) for each coal
30	Ashes		Comment line
31	6.3 6.3	xashch(ncharb)	Ash mass fraction (mass.-%, dry) in each coal
32	0. 0.	h0ashc(ncharb)	Ash formation enthalpy ( $Jkg^{-1}$ ) for each coal
33	0. 0.	cpashc(ncharb)	CP of the ashes ( $Jkg^{-1}K^{-1}$ ) for each coal
34	0. 0.	xwatch(ncharb)	humidity rate of the ashes (mass.-%) for each coal
35	Devolatilisation (Kobayashi)		Comment line
36	1 0.37 0 0.37	iy1ch(ncharb), y1ch(ncharb)	For each coal, pairs (iy1ch, y1ch). The real y1ch is the adimensional stoich. coefficient If the integer iy1ch is worth 1, the provided value of y1ch is adopted and the composition of the light volatile matters is calculated automatically. If the integer iy1ch is worth 0, the provided value of y1ch is ignored: y1ch is calculated automatically (the light volatiles are then composed of $CH_4$ , $CO$ ).
37	1 0.74 1 0.74	iy2ch(ncharb), y2ch(ncharb)	For each coal, pairs (iy2ch, y2ch). The real y2ch is the adimensional stoich. coefficient If the integer iy2ch is worth 1, the provided value of y2ch is adopted and the composition of the heavy volatile matters is calculated automatically. If the integer iy2ch is worth 0, the provided value of y2ch is ignored: y2ch is calculated automatically (the heavy volatiles are then composed of $C_2H_4$ , $CO$ ).
38	370000. 410000.	a1ch(ncharb)	Devolatilisation pre-exponential factor A1 ( $s^{-1}$ ) for each coal (light volatile matters)
39	1.3E13 1.52E13	a2ch(ncharb)	Devolatilisation pre-exponential factor A2 ( $s^{-1}$ ) for each coal (heavy volatile matters)
40	74000. 80000.	e1ch(ncharb)	Devolatilisation activation energy E1 ( $Jmol^{-1}$ ) for each coal (light volatile matters)
41	250000. 310000.	e2ch(ncharb)	Activation energy E2 ( $Jmol^{-1}$ ) of devolatilisation for each coal (heavy volatile matters)
42	heterogeneous combustion $O_2$		Comment lign
43	17.88 17.88	ahetch(ncharb)	Char burnout pre-exponential constant ( $kgm^{-2}s^{-1}atm^{-1}$ ) for each coal
44	16.55 16.55	ehetch(ncharb)	Char burnout activation energy ( $kcalmol^{-1}$ ) for each coal
45	1 1	iochet(ncharb)	Char burnout reaction order for each coal 0.5 if iochet = 0 and 1 if iochet = 1
46	heterogeneous combustion $CO_2$		Comment lign
47	1.788 1.788	ahetch(ncharb)	Char burnout pre-exponential constant ( $kgm^{-2}s^{-1}atm^{-1}$ ) for each coal
48	1.655 1.655	ehetch(ncharb)	Char burnout activation energy ( $kcalmol^{-1}$ ) for each coal
49	1 1	iochet(ncharb)	Char burnout reaction order for each coal 0.5 if iochet = 0 and 1 if iochet = 1
50	OXYDIZERS CHARACTERISTICS		Comment lign
51	3	noxyd	Number of oxydizers (mixtures of $O_2$ , $N_2$ , $H_2O$ , $CO_2$ )
52	1. 0. 1.	oxyo2(noxyd)	Composition in $O_2$ of each oxydizer (moles)
53	0. 0. 1.	oxyn2(noxyd)	Composition in $N_2$ of each oxydizer (moles)
54	0. 0. 1.	oxyh2o(noxyd)	Composition in $H_2O$ of each oxydizer (moles)
55	2.39 1. 1.	oxycO2(noxyd)	Composition in $CO_2$ of each oxydizer (moles)

Table 3: Example of file for the pulverised coal combustion: dp\_FCP

Lines	Examples of values	Variables	Observations
1	# Fichier ASCII format libre ...		Free comment
2	# Les lignes de commentaires ...		Free comment
3	# ...		Free comment
4	# Proprietes de l'Argon ...		Free comment
5	# ...		Free comment
6	# Nb d'especes NGAZG et Nb ...		Free comment
7	# NGAZG NPO ...		Free comment
8	1 238	ngazg npo	Number of species Number of given temperature points for the tabulated physical properties (npo ≤ npot set in ppthch.h) So there will be ngazg blocks of npo lines each
9	# ...		Free comment
14	0	ixkabe	Radiation options for xkabe
15	# ...		Free comment
16	# Proprietes ...		Free comment
17	# T H ...		Free comment
18	# Temperature Enthalpie ...		Free comment
19	# ...		Free comment
20	# K J/kg ...		Free comment
21	# ...		Free comment
22	300. 14000. ...	h roel cpel sigel visel xlabel xkabel	Tabulation in line of the physical properties as a function of the temperature in Kelvin for each of the ngazg species Enthalpy in J/kg Density in kg/m3 Specific heat in J/(kg K) Electric conductivity in Ohm/m Dynamic viscosity in kg/(m s) Thermal conductivity in W/(m K) Absorption coefficient (radiation)

Table 4: Example of file for the electric arc module: **dp\_ELE**

## 8.2 Pulverised coal and gas combustion module

### 8.2.0.1 Initialisation of the variables

For coal combustion, it is possible to initialise the specific variables in the Graphical User Interface (GUI) or in the subroutines `usebui`, `usd3pi`, `uslwc` and `uscpi`. In the GUI, when a coal combustion physics is selected in the item “Calculation features” under the heading “Thermophysical models”, an additional item appears: “Pulverized coal combustion”. In this item the user can define coal types, its composition, the oxydant and reactions parameters, see figs. 34 to 38.

The screenshot shows the 'Coal' tab selected in the 'Oxydant' section. The 'Coal combustion' section contains a 'Number of coal types' input field set to 2, with a list showing 'Coal 1' and 'Coal 2'. There are 'Add' and 'Delete' buttons. Below this is a 'Classes' section with tabs for 'Coal', 'Coke', 'Ashes', 'Devolatilisation', and 'Heterogeneous combustion'. The 'Coal' tab is active, showing a table with 'Number of classes' and 'Initial diameter' columns. The table lists 'Class 1' with an initial diameter of 0,002 and 'Class 2' with an initial diameter of 0,0001. There are 'Add' and 'Delete' buttons for the classes.

Figure 34: Thermophysical models - Pulverized coal combustion, coal classes

The screenshot shows the 'Coke' tab selected in the 'Classes' section. The 'Coke composition' section contains three input fields for 'Composition over C on dry', 'Composition over H on dry', and 'Composition over O on dry', all set to 0.0%. Below this is the 'Coke properties' section with an input field for 'PCI' set to 0.0 J/kg.

Figure 35: Pulverized coal combustion, coke

If the user deals with gas combustion or if he (or she) does not want to use the GUI for coal combustion, the subroutines `usebui`, `usd3pi`, `uslwc` and `uscpi` are used (only during the calculation initialisation).

In this section, “specific physics” will refer to gas combustion or to pulverised coal combustion.

The screenshot shows the 'Coal' tab selected in the 'Classes' menu. The 'Coal composition' section contains three input fields for 'Composition over C on dry', 'Composition over H on dry', and 'Composition over O on dry', all set to 0.0%. The 'Coal properties' section contains three input fields: 'PCI' (0.0 J/kg), 'Cp' (0.0 J/kg/K), and 'Density' (0.0 kg/m³). A dropdown menu for 'on pure' is also visible next to the PCI field.

Figure 36: Pulverized coal combustion, coal composition

The screenshot shows the 'Heterogeneous combustion' tab selected in the 'Classes' menu. The 'Parameters for O2' section contains three input fields: 'Pre-exponential constant' (0.0 kg/m²/s/atm), 'Activation energy' (0.0 kcal/mol), and 'Reaction order' (0.5). The 'Parameters for CO2' section contains three input fields: 'Pre-exponential constant' (0.0 kg/m²/s/atm), 'Activation energy' (0.0 kcal/mol), and 'Reaction order' (0.5).

Figure 37: Pulverized coal combustion, reaction parameters

These subroutines allow the user to initialise some variables specific to the specific physics activated *via usppmo*. As usual, the user may have access to several geometric variables to discriminate between different initialisation zones if needed.

*WARNING: in the case of a specific physics modeling, all the variables will be initialised here, even the potential user scalars: usiniv is no longer used.*

- in the case of the EBU pre-mixed flame module, the user can initialise in every cell `iel`: the mixing rate `rtp(iel,isca(ifm))` in variable richness, the fresh gas mass fraction `rtp(iel,isca(iygm))` and the mixture enthalpy `rtp(iel,isca(ihm))` in permeatic conditions
- in the case of the rapid complete chemistry diffusion flame module, the user can initialise in

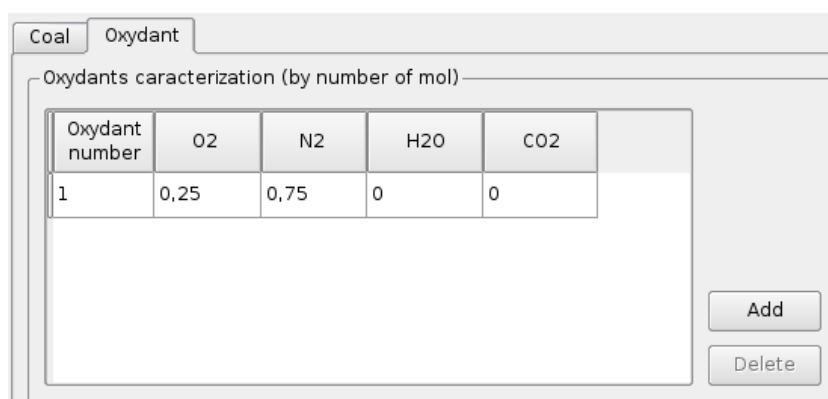


Figure 38: Pulverized coal combustion, oxydant

every cell *iel*: the mixing rate `rtp(iel,isca(ifm))`, its variance `rtp(iel,isca(ifp2m))` and the mixture mass enthalpy `rtp(iel,isca(ihm))` in permeatic conditions

- in the case of the pulverised coal combustion module, the user can initialise in every cell *iel*:

→ the transport variables related to the solid phase

`rtp(iel,isca(ixch(icla)))` the reactive coal mass fraction related to the class *icla* (*icla* from 1 to *nclacp* which is the total number of classes, *i.e.* for all the coal type)

`rtp(iel,isca(ixck(icla)))` the coke mass fraction related to the class *icla*

`rtp(iel,isca(inp(icla)))` the number of particles related to class *icla* per kg of air-coal mixture

`rtp(iel,isca(ih2(icla)))` the mass enthalpy related to the class *icla* in permeatic conditions

→ `rtp(iel,isca(ihm))` the mixture enthalpy

→ the transport variables related to the gas phase

`rtp(iel,isca(if1m(icha)))` the mean value of the tracer 1 representing the light volatile matters released by the coal *icha*

`rtp(iel,isca(if2m(icha)))` the mean value of the tracer 2 representing the heavy volatile matters released by the coal *icha*

`rtp(iel,isca(if3m))` the mean value of the tracer 3 representing the carbon released as CO during coke burnout

`rtp(iel,isca(if4p2m))` the variance associated with the tracer 4 representing the air (the mean value of this tracer is not transported, it can be deduced directly from the three others)

`rtp(iel,isca(ifp3m))` the variance associated with the tracer 3

## 8.2.1 Boundary conditions

In this section, “specific physics” refers to gas combustion or to pulverised coal combustion.

For coal combustion, it is possible to manage the boundary conditions in the Graphical User Interface (GUI). When the coal combustion physics is selected in the heading “Thermophysical models”, specific boundary conditions are activated for inlets, see fig. 39. The user fills for each type of coal previously defined (see Section 8.2.0.1) the initial temperature and initial composition of the inlet flow, as well as the mass flow rate.

For gas combustion or if the GUI is not used for coal combustion, the use of `usebuc` (called at every time step), `usd3pc`, `uslwcc`, `uscpc1` or `uscplc` is as mandatory as `usini1` and `usppmo` to run a calculation

Boundary conditions

Label	Zone	Nature	Selection criteria
wall	3	wall	5
cold_inlet	1	inlet	2
hot_inlet	2	inlet	6
outlet	5	outlet	7

Flows and temperatures

Oxydant and coal

Mass flow rate and temperature for oxydant

Norm
0.03183 m/s

Oxydant number 1
Temperature 1273.15 K

Direction
Normal to the inlet

Mass flow rate and temperature of coals

Coal number	Flow (kg/s)	Temperature (K)
Coal 1	1	1273,15
Coal 2	1	1273,15

Ratio of mass distribution for each class of coal

	Coal 1	Coal 2
Class 1	30	100
Class 2	70	

Figure 39: Boundary conditions for the combustion of coal

involving specific physics. The way of using them is the same as using `usclim` in the framework of standard calculations, that is, run several loops on the boundary faces lists (cf. §3.9.3) marked out by their colors, groups, or geometrical criterion, where the type of face, the type of boundary condition for each variable and eventually the value of each variable are defined.

*WARNING: In the case of a specific physics modeling, all the boundary conditions for every variable must be defined here, even for the eventual user scalars: `usclim` is not used at all.*

In the case of a specific physics modeling, a zone number **izone**<sup>28</sup> (for instance the color **icoul**) is associated with every boundary face, in order to gather together all the boundary faces of the same type. In comparison to **usclim**, the main change from the user point of view concerns the faces whose boundary conditions belong to the type **itypfb=ientre**:

- for the EBU pre-mixed flame module:

- the user can choose between the “burned gas inlet” type (marked out by the burned gas indicator **ientgb(izone)=1**) and the “fresh gas inlet” type (marked out by the fresh gas indicator **ientgf(izone)=1**)
- for each inlet type (fresh or burned gas), a mass flow or a velocity must be imposed:
  - to impose the mass flow,
    - the user gives to the indicator **qimp(izone)** the value 1,
    - the mass flow value is set in **qimp(izone)** (positive value, in  $kg\cdot s^{-1}$ )
    - finally he imposes the velocity vector direction by giving the components of a direction vector in **rcodcl(ifac,iu)**, **rcodcl(ifac,iv)** and **rcodcl(ifac,iw)**

**WARNING:**

- the variable **qimp(izone)** refers to the mass flow across the whole zone **izone** and not across a boundary face (specifically for the axisymmetric calculations, the inlet surface of the mesh must be broken up)
- the variable **qimp(izone)** deals with the inflow across the area **izoz** and only across this zone; it is recommended to pay attention to the boundary conditions.
- the velocity direction vector is neither necessarily normed, nor necessarily incoming.
- to impose a velocity, the user must give to the indicator **qimp(izone)** the value 0 and set the three velocity components (in  $m\cdot s^{-1}$ ) in **rcodcl(ifac,iu)**, **rcodcl(ifac,iv)** and **rcodcl(ifac,iw)**
- finally he specifies for each gas inlet type the mixing rate **fment(izone)** and the temperature **tkent(izone)** in Kelvin

- for the “3 points” diffusion flame module:

- the user can choose between the “oxydiser inlet” type marked out by **ientox(izone)=1** and the “fuel inlet” type marked out by **ientfu(izone)=1**
- concerning the input mass flow or the input velocity, the method is the same as for the EBU pre-mixed flame module
- finally, the user sets the temperatures **tinoxy** for each oxydiser inlet and **tinfuel**, for each fuel inlet

*Note: In the standard version, only the cases with only one oxydising inlet type and one fuel inlet type can be treated. In particular, there must be only one input temperature for the oxidiser (**tinoxy**) and one input temperature for the fuel (**tinfuel**).*

- for the pulverised coal module:

- the inlet faces can belong to the “primary air and pulverised coal inlet” type, marked out by **ientcp(izone)=1**, or to the “secondary or tertiary air inlet” type, marked out by **ientat(izone)=1**
- in a way which is similar to the process described in the framework of the EBU module, the user chooses for every inlet face to impose the mass flow or not (**qimp(izone)=1** or 0). If the mass flow is imposed, the user must set the air mass flow value **qimpat(izone)**,

<sup>28</sup>**izone** must be less than the maximum number of boundary zone allowable by the code, **nozppm**. This is fixed at 2000 in **pppvar.h**; not to be modified

its direction in `rcodcl(ifac,iu)`, `rcodcl(ifac,iv)` and `rcodcl(ifac,iw)` and the incoming air temperature `timpat(izone)` in Kelvin. If the velocity is imposed, he has to set `rcodcl(ifac,iu)`, `rcodcl(ifac,iv)` and `rcodcl(ifac,iw)`.

→ if the inlet belongs to the “primary air and pulverised coal” type (`ientcp(izone) = 1`) the user must also define for each coal type `icha`: the mass flow `qimpcp(izone,icha)`, the granulometric distribution `distch(izone,icha,iclapc)` related to each class `iclapc`, and the injection temperature `timpcp(izone,icha)`

## 8.2.2 Initialisation of the options of the variables

In the case of coal combustion, time averages, chronological records and listings follow-ups can be set in the Graphical User Interface (GUI) or in the subroutines `usebu1`, `usd3p1`, `uslwc1`, `uscpi1` and `uscpl1`. In the GUI, under the heading “Calculation control”, additional variables appear in the list in the items “Time averages” and “Profiles”, as well as in the item Volume solution control”, see figs. 40 and 41.

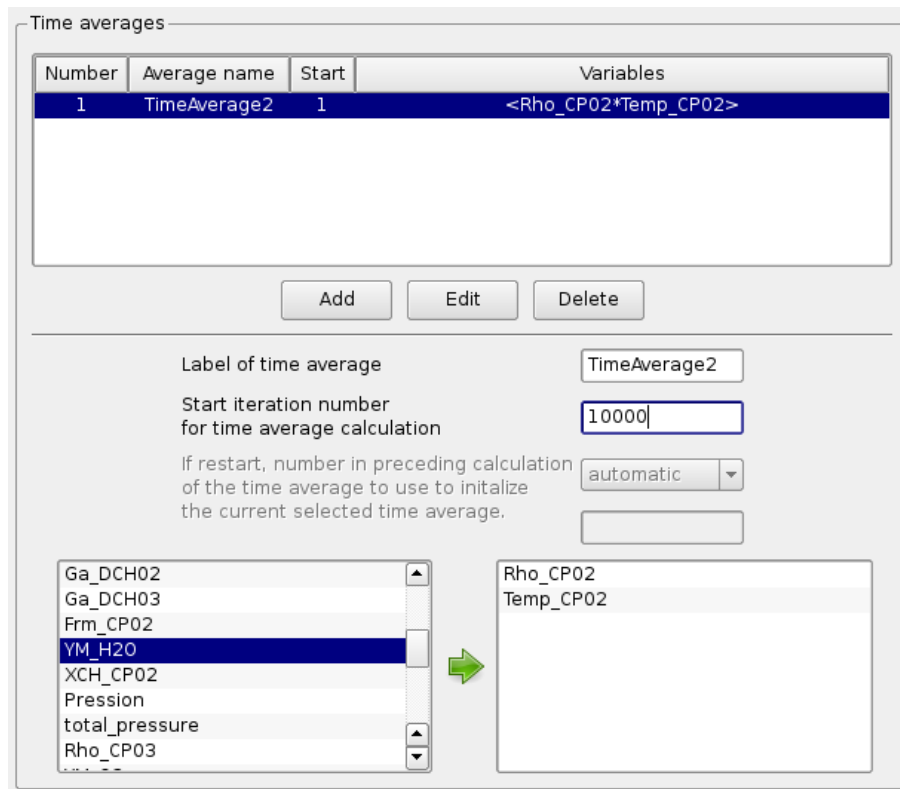


Figure 40: Calculation control - Time averages

In this section, “specific physics” refers to gas combustion or pulverised coal combustion.

For gas combustion or if the GUI is not used for coal combustion, the 3 subroutines `usebu1`, `usd3p1`, `uslwc1`, `uscpi1` and `uscpl1` can be used to complete `usini1` for the considered specific physics. These subroutines are called at the calculation start. They allow to:

- generate, for the variables which are specific to the activated specific physics module, chronological outputs (indicators `ichrvr(ipp)`), follow-ups in the listings (indicator `ilisvr(ipp)`) and to activate chronological records at the probes defined in `usini1` (indicators `ihisvr(ipp)`).



Solution control			
Name	Print in listing	Post-processing	Probes
Fr_HET_02	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
Enthalpy	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
NP_CP01	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
NP_CP02	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
NP_CP03	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
XCH_CP01	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
XCH_CP02	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
XCH_CP03	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
XCK_CP01	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
XCK_CP02	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
XCK_CP03	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
ENT_CP01	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
ENT_CP02	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
ENT_CP03	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
Fr_MV101	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
Fr_MV102	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
Fr_MV201	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
Fr_MV202	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
Var_AIR	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
Temp_GAZ	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
ROM_GAZ	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
YM_CHx1m	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4
YM_CHx2m	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1 2 3 4

Figure 41: Calculation control - Volume solution control

The way of doing it is the same as in `usini1` and the writing frequencies of these outputs are set by `usini1`. The values of the indicators `ipp` are `ipp=ipppro(ipproc(ivar))`, with `ivar` the number of the specific physics variable. Concerning the main variables (velocity, pressure, etc ...) the user must still complete `usini1` if he wants to get chronological records, printings in the listing or chronological outputs. The variables which can be activated by the user for each specific physics are listed below. The calculation variables `ivar` (defined at the cell `iel` by `rtp(iel,ivar)`) and the properties `iprop` (defined at the cell `iel` by `propce(iel,ipproc(iprop))`) are listed now:

→ EBU pre-mixed flame modeling:

- Calculation variables `rtp(iel,ivar)`
  - `ivar = isca(iygfm)` fresh gas mass fraction
  - `ivar = isca(ifm)` mixing rate
  - `ivar = isca(ihm)` enthalpy, if transported
- Properties `propce(iel,ipproc(iprop))`
  - `iprop = itemp` temperature
  - `iprop = iym(1)` fuel mass fraction
  - `iprop = iym(2)` oxidiser mass fraction

EDF R&D	<b>Code_Saturne version 2.1.3 practical user's guide</b>	Code_Saturne documentation Page 104/205
---------	--	---

`iprop = iym(3)` product mass fraction

`iprop = ickabs` absorption coefficient, when the radiation modeling is activated

`iprop = it3m` and `it4m` “ $T^3$ ” and “ $T^4$ ” terms, when the radiation modeling is activated

→ rapid complete chemistry diffusion flame modeling:

everything is identical to the “EBU” case, except the fresh gas mass fraction which is replaced by the variance of the mixing rate `ivar=isca(ifp2m)`

→ pulverised coal modeling with 3 combustibles:

*variables shared by the two phases:*

- Calculation variables `rtp(iel,ivar)`

`ivar = isca(ihm)`: gas-coal mixture enthalpy

`ivar = isca(imm1)`: molar mass of the gas mixture

*variables specific to the dispersed phase:*

- Calculation variables `rtp(iel,ivar)`

`ivar = isca(ixck(icla))`: coke mass fraction related to the class `icla`

`ivar = isca(ixch(icla))`: reactive coal mass fraction related to the class `icla`

`ivar = isca(inp(icla))`: number of particles of the class `icla` per kg of air-coal mixture

`ivar = isca(ih2(icla))`: mass enthalpy of the coal of class `icla`, if we are in permeatic conditions

- Properties `propce(iel,iproc(iprop))`

`iprop = imm1`: molar mass of the gas mixture

`iprop = itemp2(icla)`: temperature of the particles of the class `icla`

`iprop = irom2(icla)`: density of the particles of the class `icla`

`iprop = idiam2(icla)`: diameter of the particles of the class `icla`

`iprop = igmdch(icla)`: disappearance rate of the reactive coal of the class `icla`

`iprop = igmdv1(icla)`: mass transfer caused by the release of light volatiles from the class `icla`

`iprop = igmdv2(icla)`: mass transfer caused by the release of heavy volatiles from the class `icla`

`iprop = igmhet(icla)`: coke disappearance rate during the coke burnout of the class `icla`

`iprop = ix2(icla)`: solid mass fraction of the class `icla`

*variables specific to the continuous phase:*

- Calculation variables `rtp(iel,ivar)`

`ivar = isca(if1m(icha))`: mean value of the tracer 1 representing the light volatiles released by the coal `icha`

`ivar = isca(if2m(icha))`: mean value of the tracer 2 representing the heavy volatiles released by the coal `icha`

`ivar = isca(if3m)`: mean value of the tracer 3 representing the carbon released as CO during coke burnout

`ivar = isca(if4pm)`: variance of the tracer 4 representing the air

`ivar = isca(if3p2m)`: variance of the tracer 3

- Properties `propce(iel,iproc(iprop))`

`iprop = itemp1`: temperature of the gas mixture

`iprop = iym1(1)`: mass fraction of  $CH_{X1m}$  (light volatiles) in the gas mixture

`iprop = iym1(2)`: mass fraction of  $CH_{X2m}$  (heavy volatiles) in the gas mixture

`iprop = iym1(3)`: mass fraction of CO in the gas mixture

`iprop = iym1(4)`: mass fraction of  $O_2$  in the gas mixture

```

iprop = iym1(5): mass fraction of CO2 in the gas mixture
iprop = iym1(6): mass fraction of H2O in the gas mixture
iprop = iym1(7): mass fraction of N2 in the gas mixture

```

- set the relaxation coefficient of the density **srrom**, with  
 $\rho^{n+1} = \text{srrom} * \rho^n + (1 - \text{srrom})\rho^{n+1}$   
(by default, the adopted value is **srrom** = 0.8. At the beginning of a calculation, a sub-relaxation of 0.95 may reduce the numerical “shocks”).
- set the dynamic viscosity **dift10**. By default **dift10** =  $4.25 \text{ kg m}^{-1} \text{ s}^{-1}$  (the dynamic diffusivity being the ratio between the thermal conductivity  $\lambda$  and the mixture specific heat  $C_p$  in the equation of enthalpy).
- set the value of the constant **cebu** of the Eddy Break Up model (only in **usebu1**. By default **cebu**=2.5)

## 8.3 Heavy fuel oil combustion module

### 8.3.1 Initialisation of transported variables

To initialise or modify (in case of a continuation) values of transported variables and of the time step, the subroutine **usfuiv** is used. It is similar to **usiniv**. It is called at the beginning of every computation (new or continuation) before the time loop.

Physical properties are stored in **propce** (cell center), **propfa** (inner face) and **propfb**. For instance, **propce(iel, ipproc(irom ))** is **rom(iel)**, the mean density (in  $\text{kg.m}^{-3}$ ), and **propfa(ifac, ipprof(ifuluma(ivar))** is **flumas(IFAC,IVAR)**, the convective flux of the variable **ivar**.

Physical properties (**rom**, **viscl**, **cp**, ...) are computed in **ppphyv** and are not to be modified here.

All cells can be identified by using the subroutine '**getcel**'. All boundary faces may be identified using the '**getfbr**' subroutine. All internal faces may be identified using the '**getfac**' subroutine. Details of the syntax of these three subroutines are given in **usfuiv**.

In **usfuiv** the user initialise quantities related to the turbulent model chosen, and to gaseous species and droplets compositions. Exemples are provided in the subroutine.

### 8.3.2 Boundary conditions

Boundary conditions are defined on a per-face basis in **usfucl**. Boundary faces may be identified using the '**getfbr**' subroutine. **usfucl** is very similar to **uscpcl**, see Section 8.2.1. Boundary conditions may be assigned in two ways:

- . for “standard” boundary conditions (inlet, free outlet, wall, symmetry): a code is defined in the array **itypfb** (of dimensions equal to the number of boundary faces). This code will then be used by a non-user subroutine to assign the conditions.
- . for “non-standard” conditions: see details given in **usfucl**.

### 8.3.3 Initialisation of the options of the variables

The presence of a fuel combustion module variable in the listing, *histo* files, and the output frequency are set in the subroutine **usfui1**. If the vectors below are not allocated, default values will be used:

- **ichrvr**: chronological output (1:yes / 0:no)
- **ilisvr**: listing output (1:yes / 0:no)

- `ihisvr`: *histo* output (number of probes and probe numbers), if = -1, every probes defined in `usini1` will be found in the *histo* files

Calculation options such as a the relaxation parameter the for density (recommended when starting a combustion computation but forbidden for unstationnary computations) can also be set, as well as physical constants like the the laminar viscosity for the enthalpy.

## 8.4 Radiative thermal transfers in semi-transparent gray media

### 8.4.1 Initialisation of the radiation main parameters

The main radiation parameters can be initialise in the Graphical User Interface (GUI) or in the user subroutine `usray1`. In the GUI, under the heading “Thermophysical models”, when one of the two thermal radiative transfers models is selected, see fig. 42, additional items appear. The user is asked to choose the number of directions for angular discretisation, to define the absorption coefficient and select if the radiative calculation are restarted or not, see figs. 43 and 45. When “Advanced options” is selected for both models figs. 44 or 46 appear, the user must fill the resolution frequency and verbosity levels. In addition, the activation of the radiative transfer leads to the creation of an item “Surface solution control” under the heading “Calculation control”, see fig. 47, where radiative transfer variables can be selected to appear in the output listing.

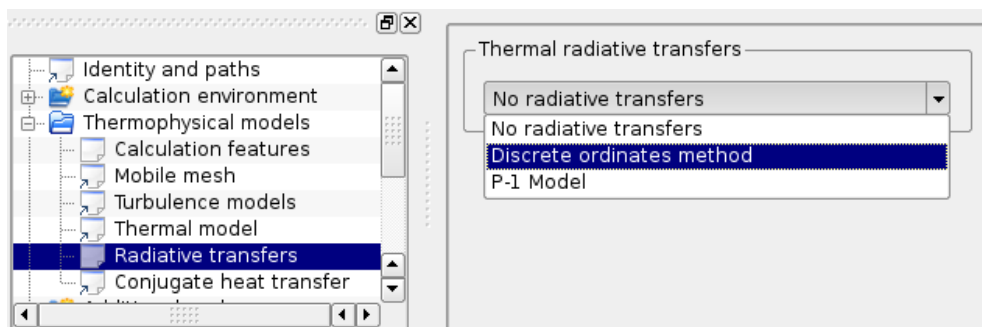


Figure 42: Radiative transfers models

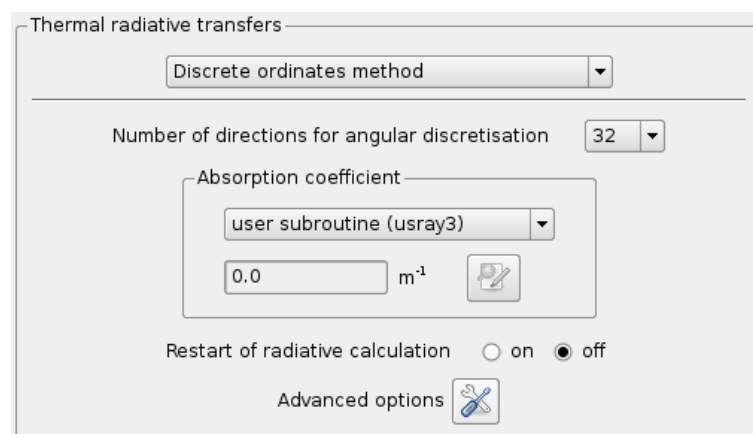


Figure 43: Radiative transfers - parameters of the DO method

If the GUI is not used, `usray1` is one of the two subroutine which must be completed by the user for all calculations including radiative thermal transfers. It is called only during the calculation initialisation.

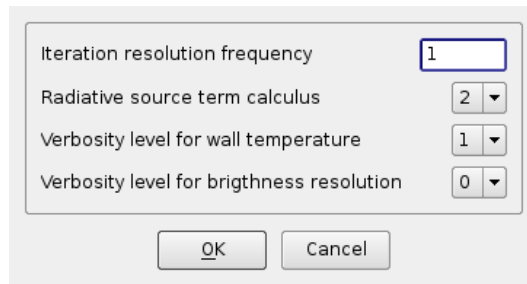


Figure 44: Radiative transfers - advanced parameters of the DO method

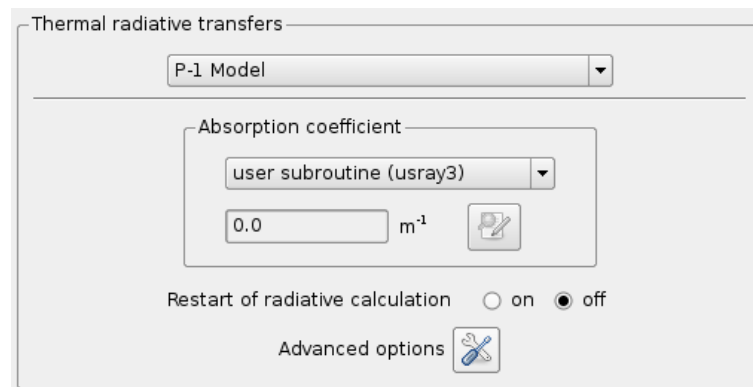


Figure 45: Radiative transfers - parameters of the P-1 model

It is composed of three headings. The first one is dedicated to the activation of the radiation module, only in the case of classic physics.

*WARNING: when a calculation is ran using a specific physics module, this first heading must not be completed. The radiation module is then activated or not, according to the parameter file related to the considered specific physics.*

In the second heading the basic parameters of the radiation module are indicated.

Finally, the third heading deals with the selection of the post-processing graphic outputs. The variables to treat are splitted into two categories: the volumetric variables and those related to the boundary faces.

For more details about the different parameters, the user may refer to the key word list (§9).

## 8.4.2 Radiative transfers boundary conditions

These informations can be filled by the user through the Graphical User Interface (GUI) or by using the subroutine `usray2` (called every time step). If the interface is used, when one of the “Radiative transfers” options is selected in fig. 42, it activates specific boundary conditions each time a “Wall” is defined, see fig. 48. The user can then choose between 3 cases. The parameters the user must specify are displayed for one of them in fig. 49.

When the GUI is not used, `usray2` is the second subroutine necessary for every calculation which includes radiative thermal transfers. It is used to give all the necessary parameters concerning, in the one case, the wall temperature calculation, and in the other, the coupling between the thermal scalar (temperature or enthalpy), and the radiation module at the calculation domain boundaries. It must

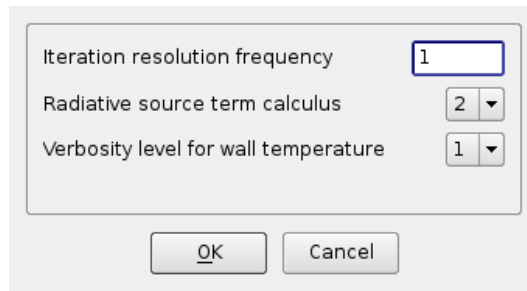


Figure 46: Radiative transfers - advanced parameters of the P-1 model

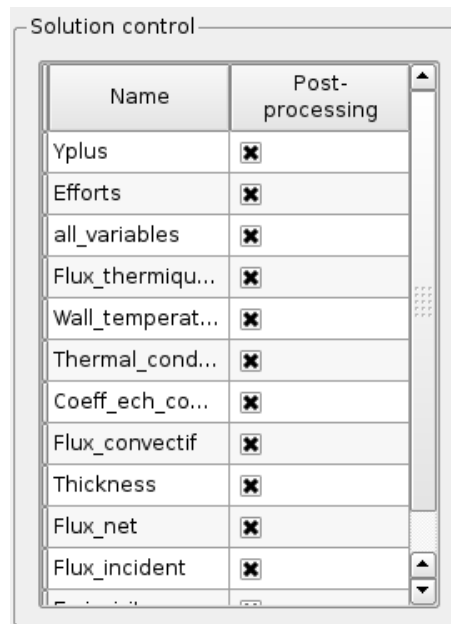


Figure 47: Calculation control - Radiative transfers postprocessing output

be noted that the boundary conditions concerning the thermal scalar which may have been defined in the subroutine `usclim` will be modified by the radiation module according to the data given in `usray2` (cf. §3.9.3).

A zone number must be given to each boundary face <sup>29</sup>and, specifically for the walls, a boundary condition type and an initialisation temperature (in Kelvin). The initialisation temperature is only used to make the solving implicit at the first time step. The zone number allows to assign an arbitrary integer to a set of boundary faces having the same radiation boundary condition type. This gathering is used by the calculation, and in the listing to print some physical values (mean temperature, net radiative flux ...). An independent graphic output in *EnSight* format is associated with each zone and allows the display on the boundary faces of the variables selected in the third heading of the subroutine `usray1`.

A boundary condition type stored in the array `ISOTHP` is associated with each boundary face. There are five different types:

- `itpimp`: wall face with imposed temperature,
- `ipgrno`: for a gray or black wall face, calculation of the temperature by means of a flux balance,

<sup>29</sup>this must be less than the maximum allowable by the code, `nozrdm`. This is fixed at 2000 in `radiat.h` and cannot be modified.

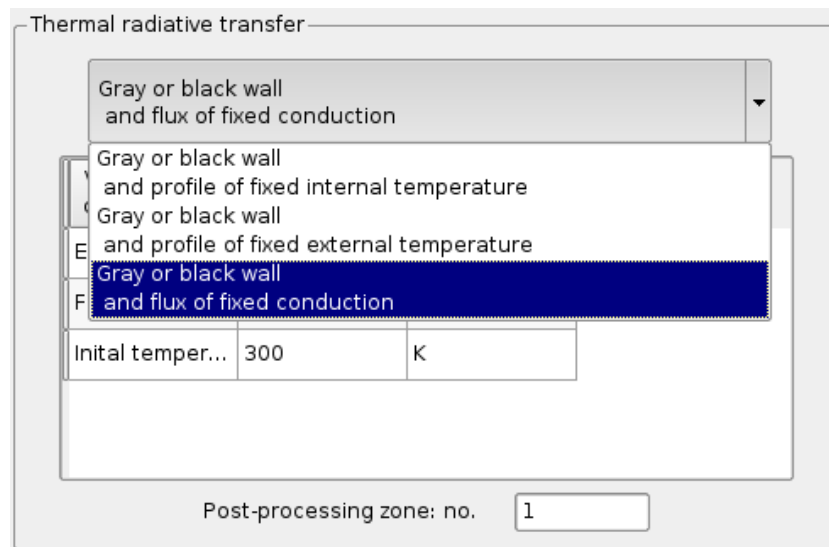


Figure 48: Boundary conditions - choice of wall thermal radiative transfers

- **iprefl**: for a reflecting wall face, calculation of the temperature by means of a flux balance. This is fixed at 2000 in **radiat.h** and cannot be modified.
- **ifgrno**: gray or black wall face to which a conduction flux is imposed,
- **ifrefl**: reflecting wall face to which a conduction flux is imposed, which is equivalent to impose this flux directly to the fluid.

Depending on the selected boundary condition type at every wall face, the code needs to be given some supplementary pieces of information:

- **itpimp**: the array **tintp** must be completed with the imposed temperature value and the array **epsp** must be completed with the emissivity value (strictly positive).
- **ipgrno**: must be given: an initialisation temperature in the array **tintp**, the wall emissivity (strictly positive, in **epsp**), thickness (in **epap**), thermal conductivity (in **xlamp**) and an external temperature (in **textp**) in order to calculate a conduction flux across the wall.
- **iprefl**: must be given: an initialisation temperature (in **tintp**), the wall thickness (in **epap**) and thermal conductivity (in **xlamp**) and an external temperature (in **textp**).
- **ifgrno**: must be given: an initialisation temperature (in **tintp**), the wall emissivity (in **epsp**) and the conduction flux (in  $W/m^2$  whatever the thermal scalar, enthalpy or temperature) in the array **rcodcl**. The value of **rcodcl** is positive when the conduction flux is directed from the inside of the fluid domain to the outside (for instance, when the fluid heats the walls). If the conduction flux is null, the wall is adiabatic.
- **ifrefl**: must be given: an initialisation temperature (in **tintp**) and the conduction flux (in  $W/m^2$  whatever the thermal scalar) in the array **rcodcl**. The value of **rcodcl** is positive when the conduction flux is directed from the inside of the fluid domain to the outside (for instance, when the fluid heats the walls). If the conduction flux is null, the wall is adiabatic. The flux received by **rcodcl** is directly imposed as boundary condition for the fluid.

*WARNING: it is mandatory to set a zone number to every boundary face, even those which are not wall faces. These zones will be used during the printing in the listing. It is recommended to gather together the boundary faces of the same type, in order to ease the reading of the listing.*

Wall radiative characteristics	Value	Unit
Emissivity	0,8	
Conductivity	3	W/m/K
Thickness	0,1	m
Profile of ext...	300	K
Profile of int...	300	K

Post-processing zone: no.

Figure 49: Boundary conditions - example of wall thermal radiative transfer

### 8.4.3 Absorption coefficient of the medium, boundary conditions for the luminance and calculation of the net radiative flux

When the absorption coefficient is not constant, the subroutine `usray3` is called instead at each time step. It is composed of three parts. In the first one, the user must provide the absorption coefficient of the medium in the array `CK`, for each cell of the fluid mesh. By default, the absorption coefficient of the medium is 0, which corresponds to a transparent medium.

*WARNING: when a specific physics is activated, it is forbidden to give a value to the absorption coefficient in this subroutine. In this case, it is calculated automatically, or given by the user via a thermo-chemical parameter file (`dp-C3P` or `dp-C3PSJ` for gas combustion, and `dp-FCP` for pulverised coal combustion).*

The two following parts of this subroutine concern a more advanced use of the radiation module. It is about imposing boundary conditions to the equation of radiative transfer and net radiative flux calculation, in coherence with the luminance at the boundary faces, when the user wants to give it a particular value. In most cases, the given examples do not need to be modified.

### 8.4.4 Encapsulation of the temperature-enthalpy conversion

*Subroutine called every time step.*

The user subroutine `usray4` is used to call the user subroutine `usthht`. `usthht` is used to encapsulate a simple enthalpy-temperature conversion law and its inverse. The user can implement his own conversion formulas into it.

This subroutine is useless when the thermal scalar is the temperature.

*WARNING: when a specific physics is activated, it is forbidden to use this subroutine. In this case, `usray4` is replaced by `ppray4`, which is not a user subroutine.*

The value of the argument `mode` allows to know in which direction the conversion will be made:



- `mode = 1`: the fluid enthalpy in the cell must be converted into temperature (in Kelvin),
- `mode = -1`: the wall temperature (`text` or `tparoi`, in Kelvin) must be converted into enthalpy.

*WARNING: the value of `mode` is passed as argument and must not be modified by the user.*

## 8.4.5 Input of radiative transfer parameters

*The routine `usray5` is called twice. The first time is for boundary conditions. The second time is for the net radiation flux computation*

In this subroutine, during the first call (`iappel=1`), the boundary conditions are filled:

- the radiative intensity must be set in the array `cofrua` when the discrete ordinates model is used; an example is given in `usray5` for an isotropic radiation field on a gray wall. Proposed boundary conditions for the intensity in `usray5` are: symmetry, inlet/outlet, and wall boundary,
- the entering intensity for free boundaries is set to zero in `cofrua` (if the user has more information, he can improve it),
- arrays `cofrua` and `cofrub` must be filled when the P-1 model is used. The boundary conditions proposed are the same as with the discret ordinates model.

During the second call (`iappel=2`), the density of the net radiation flux must be calculated consistently with the boundary conditions of the intensity considering that the density of net flux is the balance between the radiative emitting part of a boundary face (and not the reflecting one) and the radiative absorbing part. The provided example is consistent with the example of the intensity boundary conditions given when the discret ordinates model is used.

## 8.5 Conjugate heat transfers

### 8.5.1 Thermal module in a 1D wall

*subroutine called at every time step*

This subroutine takes into account the affected thermal inertia by a wall. Some boundary faces are treated as a solid wall with a given thickness, on which the code resolves an undimensional equation for the heat conduction. The coupling between the 1D module and the fluid works in a similar way to the coupling with the SYRTHES. In construction, the user is not able to account for the heat transfer between different parts of the wall. A physical analysis of each problem, case by case is required to evaluate the relevance of its usage by way of a report of the simple conditions (temperature, zero-flux ) or a coupling with SYRTHES.

The use of this code requires that the thermal scalar is defined as (`iscalt > 0`).

*WARNING: The 1D thermal module is developped assuming the thermal scalar as a temperature. If the thermal scalar is an enthalpy, the code calls the subroutine `usthht` for each transfer of information between the fluid and the wall in order to convert the enthalpy to temperature and vice-versa. This function has not been tested and is firmly discouraged. If the thermal variable is the total (compressible) energy, the thermal module will not work.*

This procedure is called twice, on initialisation and again at each time step.

- The 1st call (initialisation) all the boundary faces that will be treated as a coupled wall are marked out. This figure is written noted as `nfkpt1d`. It applies dimension to the arrays in the

thermal module. `nfkpt1d` will be at 0 if there are no coupled faces (it is in fact the default value, the remainder of the subroutine is not used in this case). The parameter `isuit1` also need to be defined, this indicates if the temperature of the wall must be initialised or written in the file (stored in the variable `filmt1`).

- The 2nd call (initialisation) again concern the wall faces, it completes the `ifpt1d` array of dimension `nfpt1d`. `ifpt1d(ifbt1d)` is the number `ifbt1d`<sup>th</sup> boundary faces coupled with the thermal module of a 1D wall. The directional parameters are then completed for a pseudo wall associated to each face

- `nppt1d(nfpt1d)`: number of cells in the 1D mesh associated to the pseudo wall.
- `eppt1d(nfpt1d)`: thickness of the pseudo wall.
- `rgpt1d(nfpt1d)`: geometry of the pseudo wall mesh (refined as a fluid if `rgt1d` is smaller than 1)
- `tppt1d(nfpt1d)`: initialisation temperature of the wall (uniform in thickness). In the course of the calculation, the array stores the temperature of the solid at the fluid/solid interface.

Other than for re-reading a file (`ficmt1`), `tppt1d` is not used. `nppt1d`, `ifpt1d`, `rgpt1d` and `eppt1d` are compared to data from the follow-up file and they must be identical.

*WARNING: The test in `ifpt1d` implicitly assumes that the array is completed in ascending order (i.e `ifpt1d(ii) > ifpt1d(jj)` if `ii > jj`. This will be the case if the coupled faces are defined starting from the unique loop on the boundary faces (as in the example). If this is not the case, contact the development team to short circuit the test.*

- The 3rd call (at each time step) is for the confirmation that all the arrays involving physical parameter and external boundary conditions have been completed.
  - `iclt1d(nfpt1d)`: Typical boundary condition at the external (pseudo) wall: Dirichlet condition (`iclt1d=1`) or flux condition (`iclt1d=3`)
  - `tept1d(nfpt1d)`: External temperature of the pseudo wall in the Dirichlet case.
  - `hept1d(nfpt1d)`: External coefficient of transfer in the pseudo wall under Dirichlet conditions (in  $W.m^{-2}.K$ ).
  - `fept1d(nfpt1d)`: External heat flux in the pseudo wall under the flux conditions (in  $W.m^{-2}$ , negative value for energy entering the wall).
  - `xlmt1d(nfpt1d)`: Conductivity  $\lambda$  of the wall uniform in thickness (in  $W.m^{-1}.K^{-1}$ ).
  - `rcpt1d(nfpt1d)`: Volumetric heat capacity  $\rho C_p$  of the wall uniform in thickness (in  $J.m^{-3}.K^{-1}$ ).
  - `dtpt1d(nfpt1d)`: Physical time step associated with the solved 1D equation of the pseudo wall (which can be different from the time step in the calculation).

The 3<sup>rd</sup> call, done at each time step, allows to impose boundary conditions and physical values in time.

## 8.5.2 Fluid-Thermal coupling with SYRTHES

When the user wishes to couple *Code\_Saturne* with SYRTHES to include heat transfers, it can be done in the Graphical User Interface (GUI) or in the user subroutine `cs_syrthes_coupling`. In the GUI, to set such a coupling, a thermal scalar must be selected first in the item “Thermal scalar” under the heading “Thermophysical models”. Then the item “Conjugate heat transfer” will appear, see fig. 50. The zones where the coupling occurs must be defined and a projection axis can be specified in case of 2D coupling.

If the subroutine `ussyrc` is used, the user must specify the arguments passed to the subroutine `'defsyr'`. These arguments are:

- `numsyrc` is the matching SYRTHES application id number, or `-1`,

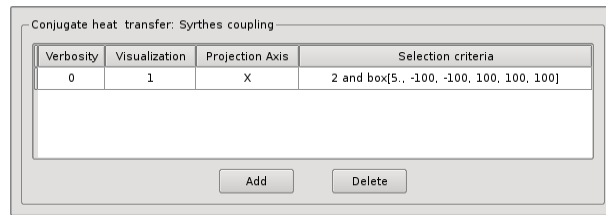


Figure 50: Thermophysical models - coupling with SYRTHES

- `namsyr` is the matching SYRTHES application name,
- `cprjsy`: ' ' if the user wishes to use a 3D standard coupling, or specify 'x', 'y', or 'z' as the projection axis if a 2D coupling with SYRTHES is used,
- `critsu` is the surface selection criteria,
- `critvl` is the volume selection criteria (only with SYRTHES 4),
- `iwarns` is the verbosity level.

Examples are provided in 'ussyrc'.

## 8.6 Lagrangian modeling of multiphase flows with dispersed inclusions

### 8.6.1 Initialisation of the Lagrangian modeling parameters

The initialisation of the Lagrangian module parameters can be performed in the Graphical User Interface (GUI) or in the user subroutine `uslag1` (called only during the calculation initialisation). In the GUI, the selection of the Lagrangian module in the item "Calculation features" under the heading "Thermophysical models" activates the heading "Particle and droplets tracking". The initialisation is performed in the three items included in this heading. In "Global settings", the user defines the Eulerian/Lagrangian multi-phase treatment, the main parameters, the specific physics associated with the particles and numerical advanced options, see figs. 51 to 53. In the item "Statistics", names are associated to volume and boundary statistical variables for listing and post-processing, see fig. 54. In the item "Output", the user defines the output frequency, post-processing options for particles and selects the variables that will appear in the listing, see fig. 55.

When the GUI is not used, `uslag1` is one of the two subroutines which must be completed in the case of a calculation using a Lagrangian multiphase flow model. This subroutine gathers in different headings all the key word which are necessary to configure the Lagrangian module. The different headings refer to:

- the global configuration parameters
- the specific physical models describing the particle behaviour
- the backward coupling (influence of the dispersed phase on the continuous phase)
- the numerical parameters
- the volumetric statistics
- the boundary statistics
- the postprocessing in trajectory mode

Figure 51: Lagrangian module - global settings

For more details about the different parameters, the user may refer to the key word list (§9.8).

The results of the lagrangian module consist in some information about the particle cloud. These pieces of information are displayed in the form of statistics. It is therefore necessary to activate the calculation of the statistics at a given instant during the simulation. To do so, there are different strategies which are strongly related to the flow nature, stationary or not.

Except from the cases where the injection conditions depend on the time, it is generally recommended to realise a first Lagrangian calculation whose aim is to get a nearly constant particle number in the calculation domain. In a second step, a calculation restart is done to calculate the statistics.

When the single-phase flow is steady and the inclusion presence rate is low enough to neglect their influence on the continuous phase behaviour, it is better to realise a Lagrangian calculation on a fixed field. It is then possible to calculate stationary volumetric statistics and to give a statistical weight higher than 1 to the particles, in order to reduce the number to treat while keeping the right concentrations.

Otherwise, when the continuous phase flow is stationary, but the backward coupling must be taken into consideration, it is still possible to activate stationary statistics.

When the continuous phase flow is non-stationary, it is no longer possible to use stationary statistics. To have correct statistics at every moment in the whole calculation domain, it is imperative to have an established particle seeding and it is recommended (when it is possible) not to impose statistical weights different from the unity.

Finally, when the complete model is used for the turbulent dispersion modeling, the user must make sure that the volumetric statistics are directly used for the calculation of the locally undisturbed fluid flow field.

When the thermal evolution of the particles is activated, the associated particulate scalars are always the inclusion temperature and the locally undisturbed fluid flow temperature expressed in degrees Celsius, whatever the thermal scalar associated with the continuous phase is (temperature or enthalpy).

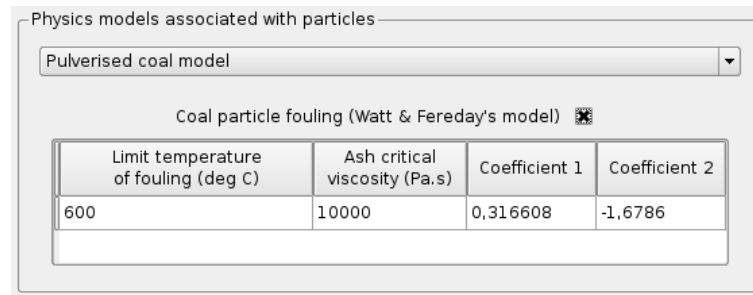


Figure 52: Lagrangian module - global settings, specific physics

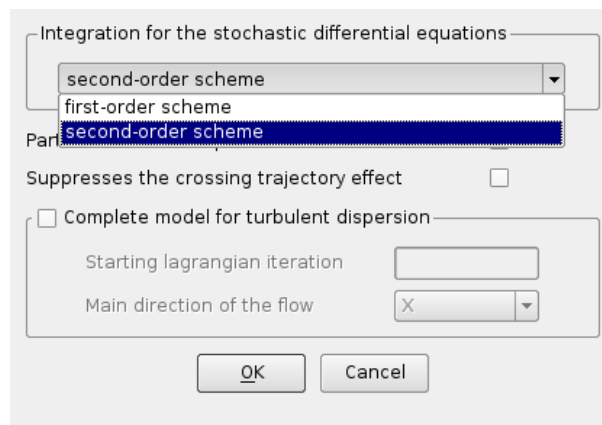


Figure 53: Lagrangian module - global settings, advanced numerical options

If the thermal scalar associated with the continuous phase is the temperature in Kelvin, the unit change is done automatically. If the thermal scalar associated with the continuous phase is the enthalpy, the enthalpy-temperature conversion subroutine `usthht` must be completed for `mode=1`, and must express temperatures in degrees Celsius.

In all cases, the thermal backward coupling of the dispersed phase on the continuous phase is adapted to the thermal scalar transported by the fluid.

*WARNING: Up to now, parallelism and periodicity are not compatible with the Lagrangian module. This compatibility will be soon implemented. It is however possible, in the framework of a Lagrangian calculation on a fixed field, to realise in a first step the calculation of the continuous phase using parallelism, and to conduct in a second step the Lagrangian calculation by doing a restart on only one processor.*

## 8.6.2 Management of the boundary conditions related to the particles

The boundary conditions related to particles can be defined in the Graphical User Interface (GUI) or in the subroutines `uslag2` and `uslain`. In the GUI, the selection of the Lagrangian module in the item "Calculation features" under the heading "Thermophysical models" activates the item "Particle boundary conditions" under the heading "Boundary conditions". Different options are available depending on the type of standard boundary conditions selected (wall, inlet/outlet, etc...), see fig. 56.

In the framework of the multiphase lagrangian modeling, the management of the boundary conditions concerns the particle behaviour when there is an interaction between its trajectory and a boundary face. These boundary conditions may be imposed independently of those concerning the eulerian fluid phase (they are of course generally coherent). The boundary condition zones are actually redefined by

Particles statistics read from restart file ☐

Number of particles cluster

Volumes Boundaries

☒ Volume statistics

Iteration starting

Threshold for statistical weight

Variable names

Name	Mean value name	Variance name	Recording
statistical_weight	statistical_weight		<input checked="" type="checkbox"/>
velocity_U	mean_velocity_U	variance_velocity...	<input checked="" type="checkbox"/>
velocity_V	mean_velocity_V	variance_velocity...	<input checked="" type="checkbox"/>
velocity_W	mean_velocity_W	variance_velocity...	<input checked="" type="checkbox"/>
mass_fraction	mean_mass_frac...	variance_mass_f...	<input checked="" type="checkbox"/>
resident_time	mean_resident t	variance_reside	<input checked="" type="checkbox"/>

Figure 54: Lagrangian module - statistics

the Lagrangian module (cf. §3.9.3), and a type of particle behaviour is associated with each one. The management of the Lagrangian boundary conditions is done by means of several user subroutines: `uslag2` for the classic conditions and `uslain` to specify profiles if necessary. Otherwise, the subroutine `uslabo` allows to define the type of particle/wall interaction. It will be described in a specific paragraph.

#### SUBROUTINE USLAG2

*Subroutine called every time step.*

It is the second indispensable subroutine for every calculation using the Lagrangian module. The main numerical variables are described below.

**ifrlag(nfabor) [ia]:** In the Lagrangian module, the user defines **nfrlag** boundary zones from the color of the boundary faces, or more generally from their properties (colors, groups ...), from the boundary conditions defined in `usclim`, or even from their coordinates. To do so, the array **ifrlag(nfabor)** giving for each face **ifac** the number **ifrlag(ifac)** corresponding to the zone to which it belongs, is completed. The zone numbers (*i.e.* the values of **ifrlag(ifac)**) are chosen freely by the user, but must be strictly positive integers inferior or equal to **nflagm** (parameter stored in `lagpar.h`, whose default value is 100). A zone type is associated with every zone; it will be used to impose global boundary conditions.  
*WARNING: it is essential that every boundary face belongs to a zone..*

**iusncl(nflagm) [ia]:** For all the **nfrlag** boundary zones previously identified, the number of classes **nbclas**<sup>30</sup> of entering particles is given: **iusncl(izone) = nbclas**. By default, the number of particle classes is zero. The maximum number of classes is **nclagm** (parameter stored in `lagpar.h`, whose default value is 20)..

<sup>30</sup> a class is a set of particles sharing the same physical properties and the same characteristics concerning the injection in the calculation domain

Output listing

Output listing at each time step 1

Post-processing for particles

Trajectory mode ☐

Displacement mode ☐

Output frequency 1

Number of particles for post-processing 500

Format EnSight

Options ascii

Variables selection

Particle velocity ☐

Fluid velocity seen by particles ☐

Residence time ☐

Particle diameter ☐

Particle temperature ☐

Particle mass ☐

Temperature of the coal particles ☐

Shrinking core diameter of the coal particles ☐

Mass of reactive coal of the coal particles ☐

Mass of char of the coal particles ☐

Figure 55: Lagrangian module - output

`iusclb(nflagm) [ia]`: For all the `nflag` boundary zones previously identified, a particle boundary condition type is given. There are two categories of particle boundary condition types: those predefined in the subroutine `uslabo` (marked out by the key words `ientrl`, `isortl`, `irebol`, `idepo1`, `idepo2`, `idepo3`, `iencrl`) and the user boundary condition types (marked out by the key words `jbord1` to `jbord5`), whose corresponding particle behaviour must be defined in the subroutine `uslabo`.

- if `iusclb(izone) = ientrl`, `izone` is a particle injection zone. For each particle class associated with this zone, some pieces of information must be given (see below). If a particle trajectory crosses an injection zone, then we consider that this particle leaves the calculation domain.
- if `iusclb(izone) = isortl`, the particles interacting with the zone `izone` leave the calculation domain.
- if `iusclb(izone) = irebol`, the particles undergo an elastic rebound on the boundary zone `izone`.
- if `iusclb(izone) = idepo1`, the particles settle definitely on the boundary zone `izone`. These particles can not be put in suspension again, and we consider that they leave the calculation domain.
- if `iusclb(izone) = idepo2`, the particles settle definitely on the boundary zone `izone`, but they are kept in the calculation domain. This distinction with the type `idepo1` is useful only when post-processings in movement mode (`ifensi2 = 1`) are realised: the particles do not disappear after touching the boundary zone. However, using `idepo2` type zones necessitates more memory than using `idepo1` type zones.

Lagrangian boundary condition			
Label	Nature	Particle-boundary interaction	Number of classes
wall	wall	Particles deposit	0
cold_inlet	inlet	Particles injection zone	0
hot_inlet	inlet	Particles deposit + memory	0
		Particles deposit + suspension	0
		Particles rebound zone	0
outlet	outlet	Particles deposit	0
		Particles depo...achment force	

Figure 56: Lagrangian module - boundary conditions

- if `iusclb(izone) = idepo3`, the particles settle on the boudary zone `izone`, but can be put in suspension again depending on the local description of the continuous phase flow.
- if `iusclb(izone) = iencrl`, the particles which are coal particles (if `iphyla = 2`) can become fouled up on the zone `izone`. The slagging is a `idepo1` type deposit of the coal particle if a certain criterion is respected. Otherwise, the coal particle rebounds (`irebol` type behaviour). This boundary condition type is available if `iencra = 1`. A limit temperature `tprenc`, a critical viscosity `visref` and the coal composition in mineral matters must be given in the subroutine `uslag1`. The slagging criterion given by default may be modified in the subroutine `uslabo`.
- if `iusclb(izone) = jbord1` to `jbord5`, then the particle interaction with the boundary zone `izone` is given by the user. The particle behaviour associated with each type `jbord*` must be defined in the subroutine `uslabo`.

`iuslag(nclagm, nflagm, ndlaim) [ia]`: Some pieces of information must be given for each particle class associated with an injection zone. The first part consists in integers contained in the array `iuslag`. There are at the most `ndlaim` integers. These pieces of information must be provided for each class `iclas` and each particle injection zone `izone`. They are marked out by means of "pointers":

- `iuslag(iclas,izone,ijnbp)`: number of particles to inject in the calculation domain per class and per zone.
- `iuslag(iclas,izone,ijfre)`: injection period (expressed in number of time steps). If the period is null, then there is injection only at the first absolute Lagrangian time step (including the restart calculations).
- `iuslag(iclas,izone,ijuvw)`: type of velocity condition:
  - if `iuslag(iclas,izone,ijuvw) = 1`, the particle velocity vector is imposed, and its components must be given in the array `ruslag` (see below).
  - if `iuslag(iclas,izone,ijuvw) = 0`, the particle velocity is imposed perpendicular to the injection boundary face and with the norm `ruslag(iclas,izone,iuno)`.
  - if `iuslag(iclas,izone,ijuvw) = -1`, the particle injection velocity is equal to the fluid velocity at the center of the cell neighboring the injection boundary face.
- `iuslag(iclas,izone,inuchl)`: when the particles are coal particles (`iphyla = 2`), this part of the array contains the coal index-number, between 1 and `ncharb` (defined by the user in the thermo-chemical file `dp_FCP`, with `ncharb ≤ ncharm = 3`).



`ruslag(nclagm, nflagm, ndlagm)` [ra]: Some pieces of information must be given for each particle class associated with an injection zone. The second and last part consists in real numbers contained in the array `ruslag`. There are at the most `ndlagm` such real numbers. These pieces of information must be provided for each class `iclas` and each particle injection zone `izone`. They are marked out by means of “pointers”:

- `ruslag(iclas,izone,iuno)`: norm of the injection velocity, useful if `iuslag(iclas,izone,ijuvw) = 0`.
- `ruslag(iclas,izone,iupt)`, `ruslag(iclas,izone,ivpt)`, `ruslag(iclas,izone,iwpt)`: components of the particle injection vector, useful if `iuslag(iclas,izone,ijuvw) = 1`.
- `ruslag(iclas,izone,idebt)`: allows to impose a particle mass flow. According to the number of injected particles, the particle statistical weight `tepa(npt,jrpoi)` is recalculated in order to respect the required mass flow (the number of injected particles does not change). When the mass flow is null, it is not taken into account.
- `ruslag(iclas,izone,ipoit)`: particle statistical weight per class and per zone.
- `ruslag(iclas,izone,idpt)`: particle diameter. When the particles are coal particles (`iphyla = 2`), this diameter is provided by the thermo-chemical file `dp_FCP` via the array `diam20(iclg)`, where `iclg` is the “pointer” on the total class number (*i.e.* for all the coal types). When the standard deviation of the particle diameter is different from zero, this diameter becomes a mean diameter.
- `ruslag(iclas,izone,ivdpt)`: standard deviation of the injection diameter. To impose this standard deviation allows to respect granulometric distribution: the diameter of each particle is calculated from the mean diameter, the standard deviation and a gaussian random number. In this case, it is strongly recommended to intervene in the subroutine `uslain` to restrict the diameter variation range, in order to avoid aberrant values. If this standard deviation is null, then the particle diameter is constant per class and per zone.
- `ruslag(iclas,izone,iropt)`: particle density. When the particles are coal particles (`iphyla = 2`), this density is set in the thermo-chemical file `dp_FCP` via the array `rho0ch(icha)`, where `icha` is the coal number.
- `ruslag(iclas,izone,itpt)`: particle injection temperature in °C. Useful if `iphyla = 1` and if `itpvar = 1`.
- `ruslag(iclas,izone,icpt)`: particle injection specific heat. Useful if `iphyla = 1` and if `itpvar = 1`. When the particles are coal particles (`iphyla = 2`), the specific heat is set in the thermo-chemical file `dp_FCP` via the array `cp2ch(icha)`.
- `ruslag(iclas,izone,iepsi)`: particle emissivity. Useful if `iphyla = 1` and if `itpvar = 1`, and if the radiation module is activated for the continuous phase (note: when `iphyla = 2`, the coal particle emissivity is given the value 1).
- `ruslag(iclas,izone,ihpt)`: particle injection temperature in °C when these particles are coal particles. The array `ruslag(iclas,izone,itpt)` is then no longer active. Useful if `iphyla = 2`.
- `ruslag(iclas,izone,imcht)`: mass of reactive coal. Useful if `iphyla = 2`.
- `ruslag(iclas,izone,imckt)`: mass of coke. This mass is null if the coal did not begin to burn before its injection. Useful if `iphyla = 2`.

`iusvis(nflagm)` [ia]: In order to display the variables at the boundaries defined in the subroutine `uslag1`, this array allows to select the boundary zones on which a display is wanted. To do so, a number is associated with each zone `izone`. If this number is strictly positive, the corresponding zone is selected; if it is null, the corresponding zone is eliminated. If several zones are associated with the same number, they will be displayed together in the same selection with *EnSight*. Each selection will be split in *EnSight* parts according to the geometric types of the present boundary faces ((*i.e.* 'tria3', 'quad4' and 'nsided')..

#### SUBROUTINE USLAIN

*Subroutine called every time step.*

It is not mandatory to intervene in this subroutine.

`uslain` is used to complete `uslag2` when the particles must be injected in the domain according to fine constraints (profile, position, ...): the arrays `ettp`, `tepa` and `itepa` can be modified here for the new particles (these arrays were previously completed automatically by the code from the data provided by the user in `uslag2`).

In the case of a more advanced utilisation, it is possible to modify here all the arrays `ettp`, `tepa` and `itepa`. The particles already present in the calculation domain are marked out by an index varying between 1 and `nbpart`. The particles entering the calculation domain at the current iteration are marked out by an index varying between `nbpart+1` and `nbpnew`.

### 8.6.3 Treatment of the particle/boundary interaction

The subroutine `uslabo` is not mandatory but is required in four different cases. It is called for each particle/boundary interaction.

Firstly, an intervention is required when `jbord*` type boundary conditions are used: it is then necessary to code in this subroutine the corresponding particle/boundary interactions.

Secondly, it is possible to select the particle/boundary interaction types (`irebol`, `idepo1`, ...) for which the user wants to save the wall statistics activated in the subroutine `uslag1`.

Thirdly, if user boundary statistics are activated *via* the key word `nusbor` in the subroutine `uslag1`, it is then necessary to program them in the subroutine `uslabo`. When the boundary statistics are stationary, these new boundary statistics are added using the array `parbor`. When they are non-stationary (number of Lagrangian iterations lower than `nstbor`, or `isttio` = 0), the array `parbor` is reset at every iteration.

Fourthly, when the user wants to modify the formulation of the wall slugging by the coal particles, it is then necessary to program the new laws in the subroutine `uslabo`.

#### CONSTRUCTION RULES OF A NEW PARTICLE/BOUNDARY INTERACTION

1. The real numbers `kx`, `ky`, `kz` provide the coordinates of the intersection point between the current particle trajectory and the interacting boundary face.
2. If the user wants to modify the particle position, it can be done directly *via* the arrays `ettp` and `ettpa`:
  - new departure point of the current trajectory segment:  
`ettpa(npt,jxp)`, `ettpa(npt,jyp)`, `ettpa(npt,jzp)`
  - new arrival point of the current trajectory segment:  
`ettp(npt,jxp)`, `ettp(npt,jyp)`, `ettp(npt,jzp)`
3. The particle and the fluid velocities may be modified according to the desired interaction *via* the arrays `vitpar` and `vitflu`, they **must not** be modified *via* `ettp` and `ettpa` in this subroutine.
4. For a given interaction, it is necessary to specify the key word `isuivi`:
  - `isuivi` = 0 if the particle does not need to be followed in the mesh after the interaction between its trajectory and the boundary face (by default, it is the case for `ientrl`, `isortl`, `idepo1`, `idepo2`);
  - `isuivi` = 1 to continue to follow the particle in the mesh after its interaction (by default, it is the case for `irebol` and `idepo3`). The value of `isuivi` may be a function of the particle and boundary state (for instance, `isuivi` = 0 or 1 depending on the physical properties for the interaction type `iencrl`).
5. The array zone `itepa(npt,jisor)`, containing the index-number of the cell where the particle is, must be updated. Generally:
  - `itepa(npt,jisor)` = `ifabor(kface)` when the particle stays in the calculation domain (`kface` is the number of the interacting boundary face).
  - `itepa(npt,jisor)` = 0 to eliminate definitively the particle from the calculation domain.

#### NOTE: ORDER OF THE NUMERICAL SCHEME AFTER A PARTICLE/BOUNDARY INTERACTION

When a particle interacts with a boundary face, the integration order of the associated stochastic equations is always a first-order, even if a second-order scheme is used elsewhere.

## 8.6.4 Option for particle cloning/merging

*Subroutine called every Lagrangian iteration.*

An intervention in the subroutine **uslaru** is required when the particle cloning/merging option is activated *via* the key word **iroule**. The important function '**croule**' must then be completed.

The aim of this technique is to reduce the number of particles to treat in the whole flow and to refine the description of the particle cloud only where the user wants to get more accurate volumetric statistics than in the rest of the calculation domain.

The values given to the importance function are strictly positive real numbers allowing to classify the zones according to their importance. The higher the value given to the importance function, the more important the zone.

For instance, when a particle moves from a zone of importance 1 to a zone of importance 2, it undergoes a cloning: the particle is replaced by two identical particles, whose statistical weight is the half of the initial particle. When a particle moves from a zone of importance 2 to a zone of importance 1, it undergoes a fusion: the particle survives to its passing through with a probability of 1/2. A random dawning is used to determine if the particle will survive or disappear.

In the same way, when a particle moves from a zone of importance 3 to a zone of importance 7, it undergoes a cloning. The particle is cloned in  $\text{Int}(7/3)=2$  or  $\text{Int}(7/3)+1=3$  particles with a probability of respectively  $1-(7/3-\text{Int}(7/3))=2/3$  and  $7/3-\text{Int}(7/3)=1/3$ . If the particle moves from a zone of importance 7 to a zone of importance 3, it undergoes a fusion: it survives with a probability of 3/7.

*WARNING: The importance function must be a strictly positive real number in every cell*

## 8.6.5 Manipulation of particulate variables at the end of an iteration and user volumetric statistics

**uslast**: *subroutine called at the end of every Lagrangian iteration*

**uslaen**: *subroutine called at every chronological output and every listing printing*

The subroutine **uslast** is called at the end of every Lagrangian iteration, it allows therefore the modification of variables related to the particles, or the extraction and preparation of data to display in the listing or the post-processing.

An intervention in both subroutines **uslast** and **uslaen** is required if supplementary user volumetric statistics are wanted.

### USER VOLUMETRIC STATISTICS:

The volumetric statistics are calculated by means of the array **statis**. Two situations may happen:

- the calculation of the statistics is not stationary: **statis** is reset at every Lagrangian iteration;
- the calculation of the statistics is stationary: the array **statis** is used to store cumulated values of variables, which will be averaged at the end of the calculation in the subroutine **uslaen**.

According to the user parameter settings, it may happen that during the same calculation, the statistics will be non-stationary in a first part and stationary in second part.

### • USER VOLUMETRIC STATISTICS: SUBROUTINE USLAST

In this subroutine, the variable whose volumetric statistic is wanted is stored in the array **statis**. In the framework of stationary statistics, the average itself is calculated in the subroutine **uslaen**.

This average is obtained through the division of the cumulated value by:

- either the duration of the stationary statistics calculation stored in the variable `tstat`,
- or the number of particles in statistical weight.

This method of averaging is applied to every piece in the listing and to the post-processing outputs.

- USER VOLUMETRIC STATISTICS: SUBROUTINE `USLAEN`

In this subroutine is calculated the average corresponding to the cumulated value obtained in the subroutine `uslast`. This subroutine is also used for the standard volumetric statistics. Several examples are therefore described.

### 8.6.6 User stochastic differential equations

An intervention in the subroutine `uslaed` is required if supplementary user variables are added to the particle state vector (arrays `ettp` and `ettpa`). This subroutine is called at each Lagrangian sub-step.

The integration of the stochastic differential equations associated with supplementary particulate variables is done in this subroutine.

When the integration scheme of the stochastic differential equations is a first-order (`nordre = 1`), this subroutine is called once every Lagrangian iteration, if it is a second-order (`nordre = 2`), it is called twice.

The solved stochastic differential equations must be written in the form:

$$\frac{d\Phi_p}{dt} = -\frac{\Phi_p - \Pi}{\tau_\phi}$$

where  $\Phi_p$  is the  $I$ th supplementary user variable (`nvls` in total) available in `ettp(nbpmax, jvls(i))` and in `ettpa(nbpmax, jvls(i))`,  $\tau_\phi$  is a quantity homogen to a characteristic time, and  $\Pi$  is a coefficient which may be expressed as a function of the other particulate variables contained in `ettp` and `ettpa`. In order to do the integration of this equation, the following parameters must be provided:

- $\tau_\phi$ , equation characteristic time, in the array `aux11` for every particle,
- $\Pi$ , equation coefficient, in the array `aux12`. If the integration scheme is a first-order, then  $\Pi$  is expressed as a function of the particulate variables at the previous iteration, stored in the array `ettpa`. If the chosen scheme is a second-order, then  $\Pi$  is expressed at the first call of the subroutine (prediction step `nor = 1`) as a function of the variables at the previous iteration (stored in `ettpa`), then at the second call (correction step `nor = 2`) as a function of the predicted variables stored in the array `ettp`.

If necessary, the thermal characteristic time  $\tau_c$ , whose calculation can be modified by the user in the subroutine `uslatc`, is stored for each particle in the part `tempct(nbpmax, 1)` of the array `tempct`.

### 8.6.7 Particle relaxation time

An intervention in this subroutine is not mandatory.

The particle relaxation time may be modified in the subroutine `uslatp` according to the chosen formulation of the drag coefficient.

The particle relaxation time, modified or not by the user, is available in the array `taup`.

## 8.6.8 Particle thermal characteristic time

An intervention in this subroutine is not mandatory.

The particle thermal characteristic time may be modified in the subroutine `uslatc` according to the chosen correlation for the calculation of the Nusselt number. This subroutine is called at each Lagrangian sub-step.

The thermal characteristic time, modified or not by the user, is available in the zone `tempct(nbpmax, 1)` of the array `tempct`.

## 8.7 Compressible module

When the compressible module<sup>31</sup> is activated, it is recommended to:

- use the option “time step variable in time and uniform in space” (`idtvar=1`) with a maximum Courant number of 0.4 (`coumax=0.4`): these choices must be written in `usini1`
- keep the convective numerical schemes proposed by default.

### 8.7.1 Initialisation of the options of the variables

*Subroutines called at each time step.*

The subroutines `uscfx1` and `uscfx2` complete `usini1`.

`uscfx1` allows to set non standard calculation options related to the compressible module, and in particular to fill in the key word `icfgrp` allowing to take into account the hydrostatic equilibrium in the boundary conditions.

`uscfx2` allows to specify for the molecular thermal conductivity and the volumetric viscosity the following pieces of information:

- variable or not (`iviscv`)
- reference value (`viscv0`)

### 8.7.2 Management of the boundary conditions

*Subroutine called every time step.*

The use of `uscfcl` is compulsory when running a calculation that uses the compressible module, just as it is in both `usini1` and `usppmo`. The way of using it is the same as the way of using `usclim` in the framework of standard calculations, that is to say several loops on the boundary faces lists (cf. §3.9.3) marked out by their colors, groups, or geometrical criterion, where the type of face, the type of boundary condition for each variable and eventually the value of each variable are defined.

*WARNING: in the case of a calculation using the compressible module, the boundary conditions of all the variables are defined here, even those of the eventual user scalars: `usclim` is not used at all.*

In the compressible module, the different available boundary conditions are the followings:

- inlet/outlet for which everything is known
- supersonic outlet

---

<sup>31</sup>For more details concerning the compressible version, the user may refer to the document “Implantation d’un algorithme compressible dans *Code\_Saturne*”, Rapport EDF 2003, HI-83/03/016/A, P. Mathon, F. Archambeau et J.-M. Hérard.

- subsonic inlet
- subsonic wall
- wall
- symmetry

### 8.7.3 Initialisation of the variables

The subroutine `uscfxi`, called during the calculation initialisation, is used to initialise some variables specific to the specific physics activated *via* `usppmo`. As usual, the user may have access to several geometric variables to discriminate between different initialisation zones if needed.

*WARNING: in the case of a specific physics modeling, all the variables are initialised here: `usiniv` is not used at all.*

This subroutine works like `usiniv` for velocity, turbulence and passive scalars. Concerning pressure, density, temperature and specific total energy, only 2 variables out of the 4 are independant. The user may also initialise the variable pair he wants (apart from temperature-energy) and the two other variables will be calculated automatically by giving the right value to the variable `iccfth` used for the call to `uscfth`.

### 8.7.4 Thermodynamics

*The subroutine `uscfth` is called several times at each time step (boundary conditions, physical properties, solving of the energy equation, ...).*

This subroutine is used to set the thermodynamics parameters. By default, the perfect gas laws are implemented. If the user needs to use other laws (perfect gas with variable Gamma, Van der Waals), he (or she) must modify this subroutine.

### 8.7.5 Management of variable physical properties

If necessary, all the variation laws of the fluid physical properties (viscosity, specific heat, ...) can be described in the subroutine `uscfpv` which is then called at each time step. This subroutine replaces and is similar to `usphyv`.

The user should make sure that the defined variation laws are valid for the whole variation range of the variables.

## 8.8 Management of the electric arc module

### 8.8.1 Initialisation of the variables

*subroutine called only at the initialisation of the calculation*

The subroutine `useliv` allows the user to initialise some of the specific physics variables prompted via `usppmo`. It is called only during the initialisation of the calculation. The user has access, as usual, to many geometric variables so that the zones can be treated separately if needed.

*WARNING: For the specific physics, it is here that all variables are initialised: `usiniv` is not used*

This subroutine works like `usiniv`. The values of potential and its constituents are initialised if required.

It should be noted that the enthalpy is relevant.

- For the electric arc module, the enthalpy value is taken from the temperature of reference `t0` (given in `usini1`) from the temperature-enthalpy tables supplied in the data file `dp_ELE`. The user must not intervene here.
- For the Joule effect module, the value of enthalpy must be specified by the user. An example is given of how to obtain the enthalpy from the temperature of reference `t0`(given in `usini1`), the temperature-enthalpy law must be supplied. A code is suggested in the sub routine `usthht`(which is there for the determination of physical properties).

## 8.8.2 Variable physical properties

All the laws of the variation of physical data of the fluid are written (when necessary) in the subroutine `uselph...`. The subroutine replaces `usphyvv` and works in a similar manner. It is called at each time step.

*WARNING: For the electric module, it is here that all the physical variables are defined (including the relative cells and the eventual user scalars): `usepelph` is not used.*

The user should ensure that the defined variation laws are valid for the whole range of variables. Particular care should be taken with non-linear laws (for example, a 3<sup>rd</sup> degree polynomial law giving negative values of density)

*WARNING: in the electric module, all the physical properties are considered as variables and are therefore stored in the `propce` array. `cp0`, `viscls0`, `viscl0` are not used*

For the Joule effect, the user is required to supply the physical properties in the sub- routine. Examples are given which are to be adapted by the user. If the temperature is to be determined to calculate the physical properties, the solved variable, enthalpy must be deduced. The preferred temperature-enthalpy law can be selected in the subroutine `usthht` (an example of the interpolation is given from the law table. This subroutine can be re-used for the initialisation of the variables(`useliv`)) For the electric arc module, the physical properties are interpolated from the data file `dp_ELE` supplied by the user. Modifications are generally not necessary.

## 8.8.3 Boundary Conditions

### SUBROUTINE USELCL

*subroutine called at each time step.*

As much as `usini1` and `usppmo`, the use of `uselcl` is required to run an electric calculation. The main use is the same as occurs in `usclim` for the standard *Code\_Saturne* calculations, for which different loops on the boundary faces is defined. Each faces list is built with the use of selection criteria (cf. §3.9.3), and is referenced by their group(s), their color(s) or geometrical criterions. The face type, the boundary conditions for each variable, and finally the value of each variable or imposed flow are fixed.

*WARNING: for the electric module, the boundary conditions of all the variables are defined here, even for those of the eventual user scalars: `usclim` is not used at all.*

For the electric module, each boundary face is associated with a number `izone`<sup>32</sup>(the color `icoul` for example) in order to group together all the boundary faces of the same type. In the report `usclim`, the main change from the users point of view concerns the specification of the boundary conditions of the potential, which isn't implied by default. The Dirichlet and Neumann conditions must be imposed explicitly using `icodc1` and `rcodc1` (as would be done for the classical scalar).

Whats more, if one wishes to slow down the power dissipation(Joule effect module) or the current (electric arc module) from the imposed values (`puismp` and `couimp` respectively), they can be changed by the potential scalar as shown below:

<sup>32</sup>`izone` must be less than the maximum value allowed by the code, `nozzppm`. This is fixed at 2000 in `ppvar.h` and cannot be modified.



EDF R&D	<b>Code_Saturne version 2.1.3 practical user's guide</b>	Code_Saturne documentation Page 127/205
---------	--	---

- For the electric arc, the imposed potential difference can be a fixed variable: for example, the cathode can be fixed at 0 and the potential at the anode contains the variable **dpot**. This variable is initialised in **usel11** by an estimated potential difference. If **ielcor=1** (see **usel11**), **dpot** is updated automatically during the calculation to obtain the required current.
- For the Joule module effect, **dpot** is again used with the same signification as in the electric arc module. If **dpot** is not wanted in the setting of the boundary conditions, the variable **coejou** can be used. **coejou** is the coefficient by which the potential difference is multiplied to obtain the desired power dissipation. By default this begins at 1 and is updated automatically. If **ielcor=1** (see **usel11**), multiply the imposed potentials in **uselc1** by **coejou** at each time step to achieve the desired power dissipation.

*WARNING: In alternative current, attention should be paid to the values of potential imposed at the limits: the variable named "real potential" represents an effective value if the current is in single phase, and a "real part" if not.*

- For the Joule studies, a complex potential is sometimes needed (**ippmod(ieljou)=2**): this is the case in particular where the current has 3 phases. To have access to the phase of the potential, and not just to its amplitude, the 2 variables must be deleted: in *Code\_Saturne*, there are 2 arrays specified for this role, the real part and the imaginary part of the potential. For use in the code, these variables are named "real potential" and "imaginary potential". For an alternative sinusoidal potential  $Pp$ , the maximum value is noted as  $Pp_{\max}$ , the phase is noted as  $\phi$ , the real potential and the imaginary potential are respectively  $Pp_{\max} \cos\phi$  and  $Pp_{\max} \sin\phi$ .
- For the Joule studies in which one does not have access to the phases, the real potential (imaginary part =0) will suffice (**ippmod(ieljou)=1**): this is obviously the case with continuous current, but also with single phase alternative current. In *Code\_Saturne* there is only 1 variable for the potential, called "real potential". Pay attention to the fact that in alternate current, the "real potential" represents a effective value of potential,  $\frac{1}{\sqrt{2}} Pp_{\max}$  (in continuous current there is no such ambiguity).

#### SUBROUTINE USETCL

*Subroutine called every time step.*

This subroutine is compulsory when the electrical module is used. It manages the boundary conditions for variables unknown by **usclim**. It calculates:

- the intensity at each electrode
- the voltage on each termin of transformers. To achieve it, the intensity, the rvoltage at each termin, the Rvoltage, and the total intensity of the transformer are calculated.

Finally, a test is performed to check if the offset is zero or if a boundary face is in contact with the ground.

### 8.8.4 Initialisation of the variable options

The subroutine **usel11** is completed in **usini1** for the specific physics. It is called at each time step. It allows:

- to activate the variables in the specific physics module, the chronological outputs (**ichrvr(ipp)** indicators), the listings (**ilisvr(ipp)** indicators) and the historical exits at the probes defined in **usini1** (**ihisvr(ipp)** indicators). The functions are the same as in **usini1** and the script frequency of the exits are fixed using **usini1**. The indicators **ipp** are for the value **ipp=ipppro**

(`ipproc(ivar)`), with `ivar`, the number of specific physics variables. With the main variables which concern the user (velocity, pressure, etc), the user must always use `usini1` if the history, the listings, or the chronological files are required. The variables which the user can activate are marked out. The number of variables in the calculation is given in `ivar` (defined by `propce(iel,ipproc(iprop))` for cell `iel`):

→ Electric Arc Module:

- Calculation variables `rtp(iel,ivar)`
  - `ivar = isca(ihm)` enthalpy
  - `ivar = isca(ipotr)` real potentiel
  - `ivar = isca(ipotva(i))` solved components of the potential vector.
  - `ivar = isca(iycoel(iesp))` the mass fraction of `ngazg` composites if there are more than 1
- Properties `propce(iel,ipproc(iprop))`
  - `iprop = itemp` temperature
  - `iprop = iefjou` power dissipation by the Joule effect.
  - `iprop = ilapla(i)` components of the laplace forces.

→ Joule Module effect :

- Calculation variables `rtp(iel,ivar)`
  - `ivar = isca(ihm)` enthalpy
  - `ivar = isca(ipotr)` real potential
  - `ivar = isca(ipoti)` imaginary potential if its to be taken into account
  - `ivar = isca(iycoel(iesp))` the mass fraction of `ngazg` composites if there are more than 1
- Properties `propce(iel,ipproc(iprop))`
  - `iprop = itemp` temperature
  - `iprop = iefjou` volumic power dissipation by Joule effect.

- to give the coefficient of relaxation of the density `srrom`:  

$$\rho^{n+1} = \text{srrom} * \rho^n + (1 - \text{srrom})\rho^n$$
(for the electric arc, the sub-relaxation is taken into account during the 2nd time step; for the Joule effect the sub relaxation is not accounted for unless the user specifies in `uselph`)
- indicates if the data will be fixed in the power dissipation or in the current, done in `ielcor`.
- target current fixed as `couimp` (electric arc module) or the power dissipation `puism` (Joule module effect).
- Fix the initial value of potential difference `dpot`, the for the calculations with a single fixed parameter as `couimp` or `puism`.

## 8.8.5 Post-processing output

The subroutine `uselen` allows the addition on  $n$  variables in the preprocessing output and works like the subroutine `usvpst` (with the electric module, it is however also possible to use `usvpst`. It is called at each chronological output

The algebraic variables related to the electric module are provided by default provided that they are not explicitly contained in the `propce` array:

- gradient of real potential in  $Vm^{-1}$  (`grad PotR = -E`)
- density of real current in  $Am^{-2}$  (`j = σE`)

specifically for the Joule module effect with `ippmod(ieljou)=2` :

- gradient of imaginary potential in  $Vm^{-1}$
- density of real current in  $Am^{-2}$

specifically for the electric arc module with `ippmod(ielarc)=2` :

- magnetic field in  $T$  ( $\underline{B} = \text{rot } \underline{A}$ )

If it is convenient for the user, there is no need to add this subroutine into the SRC directory: the post-processing will be done automatically (at the same frequency (`NTCHR`) as the other calculation variables)

## 8.9 Code\_Saturne-Code\_Saturne coupling

*Subroutine called once during the calculation initialisation.*

This user subroutine `ussatc` is used to couple `Code_Saturne` with itself. It is used for turbomachine applications for instance, the first `Code_Saturne` managing the fluid around the rotor and the other the fluid around the stator. In the case of a coupling between two `Code_Saturne` instances, the `numsat` and `namsat` arguments of the subroutine '`defsat`' are ignored. In case of multiple couplings, a coupling will be matched with available `Code_Saturne` instances prioritarily based on the `namsat` (`Code_Saturne` instance name) argument, then on the `numsat` (`Code_Saturne` instance application number) argument. If `namsat` is empty, matching will be based on `numsat` only.

The arguments of '`defsat`' are:

- `numsat`: the matching `Code_Saturne` application id, or `-1`,
- `namsat`: the matching `Code_Saturne` application name,
- `crtcsu`: the cell selection criteria for support,
- `crtfsu`: the boundary face selection criteria for support (not functional),
- `crtccp`: the cell selection criteria for coupled cells,
- `crtfcp`: the boundary face selection criteria for coupled faces,
- `iwarns`: the verbosity level.

## 8.10 Fluid-Structure external coupling

*Subroutine called only once or at each iteration.*

The subroutine `usaste` belongs to the module dedicated to external Fluid-Structure coupling with `Code_Aster`. Here one defines the boundary faces coupled with `Code_Aster` and the fluid forces components which are given to structural calculation. When using external coupling with `Code_Aster`, structure number necessarily needs to be negative; the references of coupled faces being i.e. `-1`, `-2`, etc... The subroutine performs the following operations:

- '`getfbr`' is called to get a list of elements matching a geometrical criterion or reference number then a colour (negative value) is associated to these elements.
- the value passed to `asddlf`, for user-chosen component, for every negative colour, defines the movement imposed to the external structure.
- the user specify with the value of `isyncp` if `Code_Saturne` and `Code_Aster` use synchronised chronological output or not.

## 8.11 ALE module

### 8.11.1 Initialisation of the options

This initialisation can be performed in the Graphical User Interface (GUI) or in the subroutines `usalin` and `usstr1`. First of all, in the GUI when the “Mobile mesh” is selected in the “Thermophysical models” heading, additional options are displayed. The user must choose a type of mesh viscosity and how to describe its spatial distribution, see fig. 57.

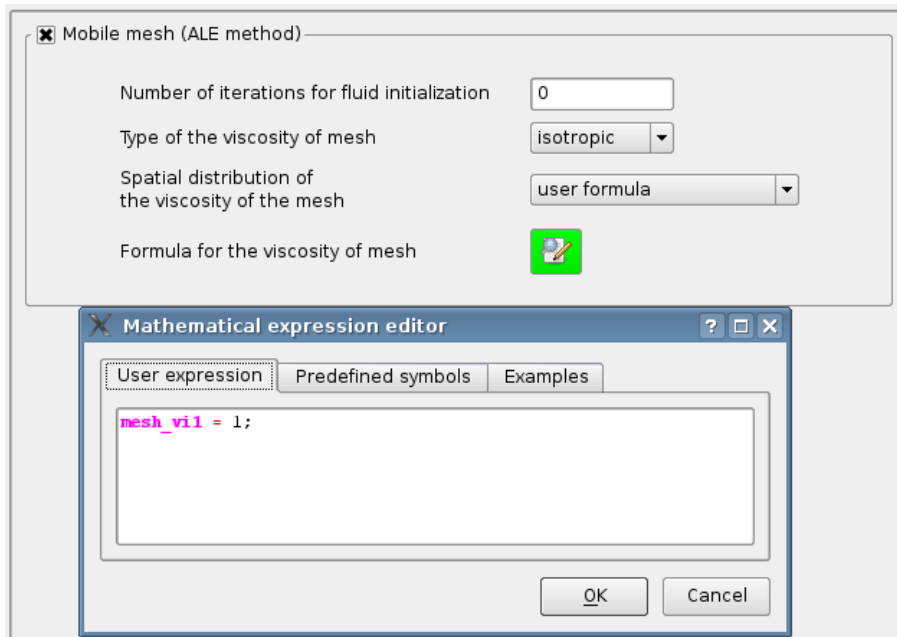


Figure 57: Thermophysical models - mobile mesh (ALE method)

The following paragraphs are relevant if the GUI is not used.

#### SUBROUTINE USALIN

*Subroutine called at the start.* This subroutine completes `usini1`.

`usalin` allows to set option for the ale module, and in particular to active the ale module

#### SUBROUTINE USSTR1

`usstr1` allows to specify for the structure module the following pieces of information:

- number of structure (`nbstru`).
- initial value of displacement, velocity and acceleration (`xstr0`, `xstreq` and `vstr0`).

Below is a list of the different variables that might be modified:

- `nbstru`  
the number of structures
- `idfstr(i)`  
index of the structure, where I is the index of the face

- **xstr0(i,k)**  
initial position of a structure, where **i** is the dimension of space and **k** the index of the structure
- **xstreq(i,k)**  
position of balance of a structure, where **i** is the dimension of space and **k** the index of the structure
- **vstr0(i,k)**  
initial velocity of a structure, where **i** is the dimension of space and **k** the index of the structure

### 8.11.2 Boundary conditions of velocity mesh

The boundary conditions can be managed with the Graphical User Interface (GUI) or with the subroutine **usalcl** (called at each time step). In the GUI, when the item “Mobile mesh” is activated the item “Fluid structure interaction” appears under the heading “Boundary conditions”. Two types of Fluid-structure coupling are offered. The first one is internal, using a simplified structure model and the second is external with *Code\_Aster*, see figs. 58 and 59.

#### SUBROUTINE USALCL

When the GUI is not used, the use of **usalcl** is mandatory to run a calculation using the ale module just as it is in **usini1**. The way of using it is the same as the way of using **usclim** in the framework of standard calculations, that is to say a loop on the boundary faces marked out by their colour (or more generally by a property of their family), where the type of boundary condition of velocity mesh for each variable are defined.

The main numerical variables are described below.

**ialtyb(nfabor)** [ia]: In the ale module, the user defines the velocity mesh from the colour of the boundary faces, or more generally from their properties (colours, groups, ...), from the boundary conditions defined in **usclim**, or even from their coordinates. To do so, the array **ialtyb(nfabor)** gives for each face **ifac** the velocity mesh boundary condition types marked out by the key words **ivimpo**, **igliss**, **ibfixe**

- If **ialtyb=ivimpo**: imposed velocity.

→ In the case where all the nodes of a face have a imposed displacement, it is not necessary to fill the tables with boundary conditions velocity mesh for this face, they will be erased. In the other case, the value of the Dirichlet must be given in **rcodcl(ifac,ivar,1)** for every value of **ivar** (**iuma**, **ivma** and **iwma**) The other boxes of **rcodcl** and **icodcl** are completed automatically.

The tangential velocity mesh is taken like a tape speed under the boundary conditions of wall for the fluid, except if wall velocity was specified by the user in the interface or **usclim** (in which case it is this speed which is considered).

- if **ialtyb(nfac) = ibfixe**: fixed wall

→ the velocity is null.

- if **ialtyb(nfac) = igliss**: sliding wall

→ the tangential velocity is not used.

### 8.11.3 Modification of the viscosity

The user subroutine `usvima` is used along the ALE (Arbitrary Lagrangian Eulerian Method) module, it fills mesh viscosity arrays. It is called at each time step. The user can modify mesh viscosity values to prevent cells and nodes from huge displacements in awkward areas, such as boundary layer for example. If `iortvm = 0`, the mesh viscosity modelling is considered as isotropic and therefore only the `viscmx` array needs to be filled. If `iortvm = 1`, mesh viscosity modeling is orthotropic therefore all arrays `viscmx`, `viscmx`, and `viscmz` need to be filled. Note that `viscmx`, `viscmx` and `viscmz` arrays are initialized at the first time step with the value 1.

### 8.11.4 Fluid - Structure internal coupling

In the subroutine `usstru` the user provides the parameters of two other subroutines. `usstr1` is called at the beginning of the calculation. It is used to define and initialise the internal structures where Fluid-Structure coupling occurs. For each boundary face `ifac`, `idfstr(ifac)` is the number of the structure the face belongs to (if `idfstr(ifac) = 0`, the face `ifac` doesn't belong to any structure). When using internal coupling, structure number necessarily needs to be positive. The number of "internal" structures is automatically defined with the maximum value of the `idfstr` table, meaning that internal structure numbers must be defined sequentially with positive values, beginning with integer value '1'.

For each internal structure one can define here:

- an initial velocity `vstr0`
- an initial displacement `xstr0` (i.e. `xstr0` is the value of the displacement `xstr` compared to the initial mesh at time  $t = 0$ )
- a displacement compared to equilibrium `xstreq` (i.e. `xstreq` is the initial displacement of the internal structure compared to its position at equilibrium; at each time step  $t$  and for a displacement `xstr(t)`, the associated internal structure will undergo a force  $-k * ((t) + XSTREQ)$  due to the spring).

`xstr0` and `vstr0` are initialised with the value 0. When starting a calculation using ALE, or re-starting a calculation with ALE, based on a first calculation without ALE, an initial iteration 0 is automatically performed in order to take initial arrays `xstr0`, `vstr0` and `xstreq` into account. In any other case, add the following expression '`italin=1`' in subroutine `usalin`, so that the code can deal with the arrays `xstr0`, `vstr0` and `xstreq`.

When `ihistr` is set to 1, the code writes in the output the history of the displacement, of the structural velocity, of the structural acceleration and of the fluid force. The value of structural history output step is the same as the one for standard variables `nthist`.

The second subroutine, `usstr2`, is called at each iteration. One defines in this subroutine structural parameters (considered as potentially time dependent): i.e., mass `m xmstru`, friction coefficients `c xcstru`, and stiffness `k xkstru`. `forstr` array gives fluid stresses acting on each internal structure. Moreover it's possible to take external forces (gravity for example) into account, too.

- . `xstr` array indicates the displacement of the structure compared to its position in initial mesh,
- . `xstr0` array gives the displacement of the structures in initial mesh compared to structural equilibrium,
- . `vstr` array stands for structural velocity.

`xstr`, `xstr0` and `vstr` are DATA tables that can be used to define arrays Mass, Friction and Stiffness. Those are not to be modified.

The 3D structural equation that is solved is the following one :

$$\underline{\underline{m}}.\partial_{tt}\underline{x} + \underline{\underline{c}}.\partial_t\underline{x} + \underline{\underline{k}}.(\underline{x} + \underline{x}_0) = \underline{f}, \quad (3)$$

where  $\underline{x}$  stands for the structural displacement compared to initial mesh position  $\underline{x}_{str}$ ,  $\underline{x}_0$  represents the displacement of the structure in initial mesh compared to equilibrium. Note that  $\underline{\underline{m}}$ ,  $\underline{\underline{c}}$ , and  $\underline{\underline{k}}$  are 3x3 matrices. Equation (3) is solved using a Newmark HHT algorithm. Note that the time step used to solve this equation,  $\underline{dt}_{str}$ , can be different from the one of fluid calculations. The user is free to define  $\underline{dt}_{str}$  array. At the beginning of the calculation  $\underline{dt}_{str}$  is initialised to the value of  $\underline{dt}_{cel}$  (fluid time step).

## 8.12 Management of the structure property

The use of `usstr2` is mandatory to run a calculation using the ale module with a structure module. It is called at each time step.

For each structure, the system that will be solved is:

$$M.\underline{\underline{x}}'' + C.\underline{\underline{x}}'' + K.(\underline{x} - \underline{x}_0) = 0 \quad (4)$$

where

- $M$  is the mass structure (`xmstru`).
- $C$  is the dumping coefficient of the structure (`xcstru`).
- $K$  is the spring constant or force constant of the structure (`xkstru`).
- $\underline{x}_0$  is the initial position

Below is a list of the different variables that might be modified:

- `xmstru(i,j,k)`  
the mass structure of the structure, where `i,j` is the array of mass structure and `k` the index of the structure.
- `xcstru(i,j,k)`  
dumping coefficient of the structure, where `i,j` is the array of dumping coefficient and `k` the index of the structure.
- `xkstru(i,j,k)`  
spring constant of the structure, where `i,j` is the array of spring constant and `k` the index of the structure.
- `forstr(i,k)`  
force vector of the structure, where `i` is the force vector and `k` the index of the structure.

## 8.13 Management of the Atmospheric module

### 8.13.1 Initialisation of the variables

The initialisation can be done in the Graphical User Interface (GUI) or in the subroutine `usativ` (called only during the calculation initialisation). Under the heading “Thermophysical models”, when in the item “Calculation features” one of the atmospheric flow model is selected, it activates an item under the same heading: “Atmospheric flows” where the path leading to a file containing meteorological data must be specified, see fig. 60. In addition is the atmospheric flow model chosen is the “dry atmosphere”,

EDF R&D	<b>Code_Saturne version 2.1.3 practical user's guide</b>	Code_Saturne documentation Page 134/205
---------	--	---

an option appear the item “Time step” under the heading “Numerical parameters” plus an additional variable “PotTemp” in the table of the “Equation parameters” item.

When the GUI is not used, `usativ` allows to initialise or modify (in case of a restarted calculation) the calculation variables and the values of the time step. It plays a similar role as `usiniv` for the additional variables introduced with the air-cooling module. The quantities that can be initialised here in user-selected zones are:

- the air velocity with the array `rtp(iel,iu)` (with `iv` and `iw` for the other components),
- the air temperature with the array `rtp(iel,isca(ihumid))`,
- turbulent quantities depending on the turbulent model selected.

The example provided in the user file performs the initialisation of the variables from meteorological profiles using the interpolation routine `intprf`.

### 8.13.2 Non standard options

The subroutine `usati1` initialises non-standard parameters for atmospheric calculations. These parameters are for instance:

- `imeteo`,
- `irovar` for each phase,
- `ivivar` for each phase.

### 8.13.3 Management of the boundary conditions

The user subroutine `usatl1` allows to define the boundary conditions of the variables unknown by `usclim`. It is called at each time step. Boundary conditions are applied to mesh faces selected using the subroutine `'getfbr'` for instance. For each type of boundary condition, these faces are grouped as physical zones characterised by an arbitrary number `izone` chosen by the user. If a boundary condition is retrieved from a meteorological profile, the variable `iprofm(izone)` of the zone must be set to 1. Examples are provided in `usatl1`.

## 8.14 Cooling tower modelling

### 8.14.1 Parameters

*Subroutine called only during calculation initialisation? OR AT EACH ITERATION?.*

The subroutine `uscti1` contains calculation parameters such as:

- temperature parameters,
- the number of exchange zones at various locations,
- the air properties.

### 8.14.2 Initialisation of the variables

The subroutine `usctiv` allows to initialise or modify (in case of a restarted calculation) the calculation variables and the values of the time step. It is called only during the calculation initialisation. It plays a similar role as `usiniv` for the additional variables introduced with the air-cooling module. The quantities that can be initialised here in user-selected zones are:



- the air temperature by filling the array `rtp(iel,isca(ihumid))`,
- the air humidity by filling the array `rtp(iel,isca(ityp4))`,
- the air velocity by filling the array `rtp(iel,iu)` (with `iv` and `iw` for the other components),

where `iel` can be an element found in a list returned by the routine '`getcel`'.

### 8.14.3 Definition of the exchange zones

The subroutine `usctdz` is used to define the exchange zones of a cooling tower. The user provides the following parameters:

- `imzech`: its value is related to the model used:
  - 0: no model is used,
  - 1: Merkel model is used,
  - 2: Poppe model is used,
- 10 exchange zone parameters.

These arguments are passed to the subroutine '`defct`' along with a geometrical selection criterion.

### 8.14.4 Management of the boundary conditions

The subroutine `usctcl`, called at each time step, allows to define the boundary conditions of the variables unknown by `usclim`. Boundary conditions are applied to mesh faces selected using the subroutine `getfbr` for instance. For each type of boundary condition, these faces are grouped as physical zones characterised by an arbitrary number `izone` chosen by the user. The list of boundary conditions offered in this module is given below:


- Dirichlet,
- flux density (velocities, pressure, scalar),
- sliding wall (velocity),
- friction (velocity),
- roughness (velocity),
- free inlet/outlet (velocity),
- symmetry.

Internal coupling with a simplified structure model    External coupling with Code\_Aster

Internal coupling

Maximum number of sub-iterations for implicit coupling with internal structures

Relative precision for implicit coupling with internal structures

Advanced options 

Structures definition

Structure number	Label	Location
------------------	-------	----------

Initial position

X  m    Y  m    Z  m


Position of equilibrium


X  m    Y  m    Z  m


Initial velocity

V<sub>x</sub>  m/s    V<sub>y</sub>  m/s    V<sub>z</sub>  m/s

Characteristics of the structure

Mass matrix 

Damping matrix 

Stiffness matrix 


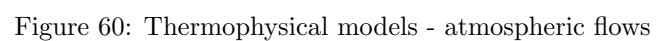
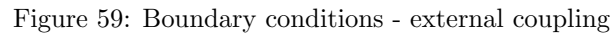
Force applied to the structure 

Figure 58: Boundary conditions - internal coupling



## 9 Key word list

The key words are classified under headings. For each key word of the Kernel of *Code\_Saturne*, the following data are given:

Variable name	Type	Allowed values	[Default]	O/C	Level
	Description	Potential dependences			

- **Variable name:** Name of the variable containing the key word.
- **Type:** a (Array), i (Integer), r (Real number), c (Character string).
- **Allowed values:** list or range of allowed values.
- **Default:** value defined by the code before any user modification (every key word has one). In some cases, a non-allowed value is given (generally  $-999$  or  $-10^{12}$ ), to force the user to specify a value. If he does not do it, the code may:
  - automatically use a recommended value (for instance, automatical choice of the variables for which chronological records will be generated).
  - stop, if the key word is essential (for instance, value of the time step).
- **O/C:** Optional/Compulsory
  - O: optional key word, whose default value may be enough.
  - C: key word which must imperatively be specified (for instance, the time step).
- **Level:** L1, L2 or L3
  - L1 (level 1): the users will have to modify it in the framework of standard applications. The L1 key words are written in bold.
  - L2 (level 2): the users may have to modify it in the framework of advanced applications. The L2 key word are all optional.
  - L3 (level 3): the developers may have to modify it ; it keeps its default value in any other case. The L3 key word are all optional.
- **Description:** key word description, with its potential dependences.

The L1 key words can be modified through the Graphical Use Interface or in the `usini1` subroutine. L2 and L3 key words can only be modified through the `usini1` subroutine, even if they do not appear in the version proposed as example it the `SRC/REFERENCE/base` directory. It is however recommended not to modify the key words which do not belong to the L1 level.

The alphabetical key word list is displayed in the index, in the end of this report.

### NOTES

- The notation “d” refers to a double precision real. For instance, 1.8d-2 means 0.018.
- The notation “**grand**” (which can be used in the code) corresponds to  $10^{12}$ .

## 9.1 Input-output

### NOTES

- Two different files can have neither the same unit number nor the same name.

### 9.1.1 "Calculation" files

#### GENERAL

<b>impstp</b>	i	strictly positive integer	[12]	O	L3
		unit of the calculation interactive stop file always useful (because of the interactive character)			
<b>ficstp</b>	c	string of 6 characters	[ficstp]	O	L3
		name of the calculation interactive stop file (see p.17) always useful (because of the interactive characteristic)			

#### 1D WALL THERMAL MODULE

<b>ficmt1</b>	c	string of 13 characters	[t1damo]	O	L3
		name of the upstream restart file for the 1D wall thermal module. useful if and only if <b>isuit1</b> = 1 and <b>nfpt1d</b> >0			
<b>ficvt1</b>	c	string of 13 characters	[t1dava]	O	L3
		name of the upstream restart file for the 1D wall thermal module useful if and only if <b>nfpt1d</b> >0			

#### VORTEX METHOD FOR LES

<b>impmvo</b>	i	strictly positive integer	[impmvo]	O	L3
		unit of the upstream restart file for the vortex method useful if and only if <b>isuivo</b> = 1 and <b>ivrtex</b> =1			
<b>ficmvo</b>	c	string of 13 characters	[voramo]	O	L3
		name of the upstream restart file for the vortex method This is always a text file (this file has a different structure from the other restart files) useful if and only if <b>isuivo</b> = 1 and <b>ivrtex</b> =1			
<b>impvvo</b>	i	strictly positive integer	[impvvo]	O	L3
		unit of the downstream restart file for the vortex method useful if and only if <b>ivrtex</b> =1			
<b>ficvvo</b>	c	string of 13 characters	[vorava]	O	L3
		name of the upstream restart file for the vortex method This is always a text file (this file has a different structure from the other restart files) useful if and only if <b>ivrtex</b> =1			
<b>impdvo</b>	i	strictly positive integer	[impdvo]	O	L3
		unit of the <b>ficvor</b> data files for the vortex method. These files are text files. Their number and names are specified by the user in the <b>usvort</b> subroutine. (Although it corresponds to an "upstream" data file, <b>impdvo</b> is initialized to 20 because, in case of multiple vortex entries, it is opened at the same time as the <b>ficmvo</b>			

upstream restart file, which already uses unit 11)  
useful if and only if `ivrtex=1`

#### RADIATION

<code>ficamr</code>	c	string of 13 characters name of the radiation upstream restart file. useful if and only if <code>isuid = 1</code>	[rayamo]	O	L3
<code>ficavr</code>	c	string of 13 characters name of the radiation downstream restart file always useful in case of radiation modeling	[rayava]	O	L3

#### THERMOCHEMISTRY

<code>impfpp</code>	i	strictly positive integer unit of the thermochemical data file useful in case of gas or pulverised coal combustion or electric arc	[25]	O	L3
<code>ficfpp</code>	c	string of 6 characters name of the thermochemical data file. The launch script is designed to copy the user specified thermochemical data file in the temporary execution directory under the name <code>dp.tch</code> , for <i>Code_Saturne</i> to open it properly. Should the value of <code>ficfpp</code> be changed, the launch script would have to be adapted. useful in case of gas or pulverised coal combustion	[dp.tch]	O	L3
<code>impjnf</code>	i	strictly positive integer unit of the JANAF data file useful in case of gas or pulverised coal combustion	[impfpp]	O	L3
<code>ficjnf</code>	c	string of 5 characters name of the JANAF data file. The launch script is designed to copy the user specified JANAF data file in the temporary execution directory under the name <code>JANAF</code> , for <i>Code_Saturne</i> to open it properly. Should the value of <code>ficjnf</code> be changed, the launch script would have to be adapted. useful in case of gas or pulverised coal combustion	[JANAF]	O	L3

#### LAGRANGIAN

<code>ficaml</code>	c	string of 6 characters name of the upstream restart file in case of Lagrangian modeling. useful if and only if <code>isuila = 1</code>	[lagamo]	O	L3
<code>ficmls</code>	c	string of 13 characters name of the upstream restart file for the statistics in case of Lagrangian modeling. useful if and only if <code>isuist = 1</code>	[lasamo]	O	L3

ficavl	c	string of 13 characters	[lagava]	O	L3
		name of the downstream restart file in case of Lagrangian modeling always useful in case of Lagrangian modeling			
ficvls	c	string of 6 characters	[lasava]	O	L3
		name of the downstream restart file for the statistics in case of Lagrangian modeling useful in case of Lagrangian modeling with statistics			
impla1	i	strictly positive integer	[50]	O	L3
		unit of a file specific to Lagrangian modeling useful in case of Lagrangian modeling			
impla2	i	strictly positive integer	[51]	O	L3
		unit of a file specific to Lagrangian modeling useful in case of Lagrangian modeling			
impla3	i	strictly positive integer	[52]	O	L3
		unit of a file specific to Lagrangian modeling useful in case of Lagrangian modeling			
impla4	i	strictly positive integer	[53]	O	L3
		unit of a file specific to Lagrangian modeling useful in case of Lagrangian modeling			
impla5	ia	strictly positive integer	[54 to 68]	O	L3
		units of files specific Lagrangian modeling, 15-dimension array useful in case of Lagrangian modeling			

### 9.1.2 Post-processing for *EnSight* or other tools

#### NOTES

- The format depends on the user choices.
- The post-processing files, directly generated by the Kernel through the FVM library, can be of the following formats: *EnSight Gold*, *MED\_fichier* or *CGNS*. The use of the two latter formats depends on the installation of the corresponding external libraries.
- For each quantity (problem unknown, preselected numerical variable or preselected physical parameter), the user specifies if a post-processing output is wanted. The output frequency can be set.

ichrvl	i	0 or 1	[1]	O	L3
		indicates whether post-processing outputs are wanted (=1) or not (=0) on the 3D volume mesh always useful			
ichrbo	i	0 or 1	[0]	O	L2
		indicates whether post-processing outputs are wanted (=1) or not (=0) on the 2D boundary mesh always useful			
ichrsy	i	0 or 1	[0]	O	L2
		indicates whether post-processing outputs are wanted (=1) or not (=0) on the 2D			

boundary mesh patches coupled with the SYRTHES conjugate heat transfer code  
always useful

**ichrmd**      i      0, 1, 2, 10, 11 or 12      [0]      O      L2

indicates whether the post-processing geometry varies with time:

- = 0: time independent
- = 1: deforming or moving mesh
- = 2: changing vertex coordinates and topology
- = 10: time independent base, with time dependent nodal displacement field
- = 11: deforming or moving mesh, plus nodal displacement field
- = 12: changing vertex coordinates and topology, plus nodal displacement field

**fntchr**      c      string of less than 32 characters      [Ensignt Gold]      O      L1

name of the output format, among the following:

- “Ensignt Gold”
- “MED.fichier” (if available)
- “CGNS” (if available)

**optchr**      c      string of less than 96 characters      [binary]      O      L2

options associated to the selected output format. The string is given as a series of key words, separated by a comma (and optional spaces). The key words are among the following:

- *text* for a text format (for *EnSight*)
- *binary* for a binary format (default choice)
- *big-endian* to force outputs to be in *big-endian* mode; this can be useful when using *ParaView*, which uses this mode by default.
- *discard\_polygons* to prevent from exporting faces with more than four edges (which may not be recognised by some post-processing tools); such faces will therefore appear as “holes” in the post-processing mesh.
- *discard\_polyhedra* to prevent from exporting elements which are neither tetrahedra, prisms, pyramids nor hexahedra (which may not be recognised by some post-processing tools); such elements will therefore appear as “holes” in the post-processing mesh
- *divide\_polygons* to divide faces with more than four edges into triangles, so that any post-processing tool can recognise them
- *divide\_polyhedra* to divide elements which are neither tetrahedra, prisms, pyramids nor hexahedra into simpler elements (tetrahedra and pyramids), so that any post-processing tool can recognise them
- *split\_tensors* to export the components of a tensor variable as a series of independent variables (always the case for now)

**ntchr**      i      -1 or strictly positive integer      [-1]      O      L1

output period for the post-processing

- = -1: only at the end of the calculation
- > 0: period (every **ntchr** time step)

always useful

**ichrvr**      ia      -999, 0 or 1      [-999]      O      L1

for each quantity defined at the cell centers (physical or numerical variable), indicator of whether it should be post-processed or not

- = -999: not initialised. By default, the post-processed quantities are the unknowns (pressure, velocity,  $k$ ,  $\varepsilon$ ,  $R_{ij}$ ,  $\omega$ ,  $\varphi$ ,  $\bar{f}$ , scalars), density, turbulent viscosity and the time step if is not uniform



= 0: not post-processed

= 1: post-processed

useful if and only if the variable is defined at the cell centers: calculation variable, physical property (time step, density, viscosity, specific heat) or turbulent viscosity if  $iturb \geq 10$

**ipstdv**      i      integer  $\geq 1$ : see below      [ipstyp\*ipstcl\*ipstft] O      L1

indicates the data to post-process on the boundary mesh (the boundary mesh must have been activated with **ichrbo=1**). The value of **ipstdv** is the product of the following integers, depending on the variables that should be post-processed:

**ipstyp**:  $y^+$  at the boundary

**ipstcl**: value of the variables at the boundary (using the boundary conditions but without reconstruction)

**ipstft**: thermal flux at the boundary ( $W m^{-2}$ ), if a thermal scalar has been defined (**iscalt**)

For instance, with **ipstdv=ipstyp\*ipstcl**,  $y^+$  and the variables will be post-processed at the boundaries.

With **ipstdv=1**, none of these data are post-processed at the boundaries.

always useful if **ichrbo=1**

### 9.1.3 Chronological records of the variables on specific points

#### STANDARD USE THROUGH INTERFACE OR **USINI1**

For each quantity (problem unknown, preselected numerical variable or preselected physical parameter), the user indicates whether chronological records should be generated, the output period and the position of the probes. The code produces chronological records at the cell centers located closest to the geometric points defined by the user by means of their coordinates. For each quantity, the number of probes and their index-numbers must be specified (it is not mandatory to generate all the variables at all the probes).

**ncapt**      i      positive or null integer      [0]      O      L1

total number of probes (limited to **ncaptm=100**)

always useful

**xyzcap**      ra      real numbers      [0.0]      O      L1

3D-coordinates of the probes

the coordinates are written: **xyzcap(i, j)**, with  $i = 1, 2$  or  $3$  and  $j \leq ncapt$

useful if and only if **ncapt > 0**

**ihisvr**      ia      -999, -1 or positive or null integer      [-999]      O      L1

number **ihisvr(n, 1)** and index-numbers **ihisvr(n, j>1)** of the record probes to be used for each variable, *i.e.* calculation variable or physical property defined at the cell centers. With **ihisvr(n, 1)=-999** or **-1**, **ihisvr(n, j>1)** is useless.

- **ihisvr(n, 1)**: number of record probes to use for the variable N
  - = -999: by default: chronological records are generated on all the probes if N is one of the main variables (pressure, velocity, turbulence, scalars), the local time step or the turbulent viscosity. For the other quantities, no chronological record is generated.
  - = -1: chronological records are produced on all the probes
  - = 0: no chronological record on any probe
  - > 0: chronological record on **ihisvr(n, 1)** probes to be specified with **ihisvr(n, j>1)**

always useful, must be inferior or equal to **ncapt**

- **ihisvr(n, j>1)**: index-numbers of the probes used for the variable *n*  
(with  $j \leq \text{ihisvr}(n, 1) + 1$ )

= -999: by default: if **ihisvr(n, 1)  $\neq$  -999**, the code stops. Otherwise, refer to the description of the case **ihisvr(n, 1)=-999**

useful if and only if **ihisvr(n, 1) > 0**

The condition **ihisvr(n, j)  $\leq$  ncapt** must be respected.

For an easier use, it is recommended to simply specify **ihisvr(n,1)=-1** for all the interesting variables.

**imphis**      ia      strictly positive integer      [30 and 31]      O      L3  
working units for the production of chronological record files by the Kernel  
useful if and only if chronological files are produced (*i.e.* there is *n* for which **ihisvr(n, 1)  $\neq$  0**)

**emphis**      c      string of less than 80 characters      [./]      O      L3  
directory in which the potential chronological record files generated by the Kernel will be written (path related to the execution directory)  
it is recommended to keep the default value and, if necessary, to modify the launch script to copy the files in the alternate destination directory  
useful if and only if chronological record files are generated (*i.e.* there is *n* for which **ihisvr(n, 1)  $\neq$  0**)

**exthis**      c      string of less than 80 characters      [hst]      O      L3  
extension of the chronological record files  
useful if and only if chronological record files are generated (*i.e.* there is *n* for which **ihisvr(n, 1)  $\neq$  0**)

**nthist**      i      -1 or strictly positive integer      [1 or -1]      O      L1  
output period of the chronological record files  
= -1: no output  
> 0: period (every **nthist** time step)  
The default value is -1 if there is no chronological record file to generate (if there is no probe, **ncapt** = 0, or if **ihisvr(n, 1)=0** for all the variables) and 1 otherwise  
If chronological records are generated, it is usually wise to keep the default value **nthist=1**, in order to avoid missing any high frequency evolution (unless the total number of time steps is much too big)  
useful if and only if chronological record files are generated (*i.e.* there are probes (**ncapt>0**) there is *n* for which **ihisvr(n, 1)  $\neq$  0**)

**nthsav**      i      -1 or positive or null integer      [0]      O      L3  
saving period the chronological record files (they are first stored in a temporary file and then saved every **nthsav** time step)  
= 0: by default (4 times during a calculation)  
= -1: saving at the end of the calculation  
> 0: period (every **nthsav** time step)  
During the calculation, the user can read the chronological record files in the execution directory when they have been saved, *i.e.* at the first time step, at the tenth time step and when the time step number is a multiple of **nthsav** (multiple of (**ntmabs-ntpabs**)/4 if **nthsav=0**)

*Note: using the **ficstp** file allows to update the value of **ntmabs**. Hence, if the calculation is at the time step *n*, the saving of the chronological record files can be forced by changing **ntmabs** to **ntpabs+4(n+1)** using **ficstp**; after the files have been saved,*

`ntmabs` can be reset to its original value, still using `ficstp`.

useful if and only if chronological record files are generated (*i.e.* there are probes (`ncapt`>0) there is `n` for which `ihisvr(n, 1) ≠ 0`)

#### NON-STANDARD USE THROUGH `USHIST`

(see p.90)

<code>impush</code>	<code>ia</code>	strictly positive integer units of the user chronological record files useful if and only if the subroutine <code>ushist</code> is used	[33 to 32+ <code>nushmx</code> =49]	O	L3
<code>ficush</code>	<code>ca</code>	strings of 13 characters names of the user chronological record files. In the case of a non-parallel calculation, the suffix applied the file name is a three digit number: <code>ush001</code> , <code>ush002</code> , <code>ush003</code> ... In the case of a parallel-running calculation, the processor index-number is added to the suffix. For instance, for a calculation running on two processors: <code>ush001.n_0001</code> , <code>ush002.n_0001</code> , <code>ush003.n_0001</code> ... and <code>ush001.n_0002</code> , <code>ush002.n_0002</code> , <code>ush003.n_0002</code> ... The opening, closing, format and location of these files must be managed by the user. useful if and only if the subroutine <code>ushist</code> is used	[ <code>ush*</code> or <code>ush*.n_*</code> ]	O	L2

### 9.1.4 Time averages

The code allows the calculation of time averages of the type  $\langle f_1 * f_2 \dots * f_n \rangle$ . The variables  $f_i$  (defined at the cell centers) which may be taken into account are the followings:

- the solved calculation variables (velocity, pressure ...),
- the auxiliary variables from the array `propce` (density and physical properties when they are variable in space).

The averages are treated like auxiliary variables defined at the cell centers and stored in the `propce` array. The standard post-processing actions may therefore be activated, like the writing in the listing or the output of result files (EnSight, MED, ...). However, if the user wants to manipulate the averages in a more advanced way, it is recommended to refer first to the user subroutines `usproj` and `usvpst` which provide examples. Indeed, the `propce` array does not contain the time averages directly, but only the cumulated value of the product  $f_1 * f_2 \dots * f_n$  of the selected variables  $f_i$ . The division by the cumulated duration is done only before the writing of the results. See also page 36.

To calculate  $p$  time averages of the type  $\langle f_1 * f_2 \dots * f_{n(imom)} \rangle$ , the user must:

- make sure that  $p \leq \text{nbmomx}$  (do not overstep the maximum number of averages),
- make sure that  $n(imom) \leq \text{ndgmox}$  for every average `imom` (do not overstep the maximum degree, *i.e.* the maximum number of variables which may compose an average),
- define every average `imom` ( $1 \leq imom \leq p$ , without skipping any index-number) by marking out the  $n(imom)$  variables which form it by means of the array `idfmom(ii, imom)` (with  $1 \leq ii \leq n(imom)$ ),
- define for each average `imom` the time step number at which the calculation of the cumulated value must begin, by means of the array `ntdmom(imom)`.

The total number of averages ( $p = \text{nbmomt}$ ) is automatically determined by the code from the values of `idtmom`. The user must not specify it.

<b>idfmom</b>	<p>ia      0, <math>\pm</math> variable index-number      [0]      O      L1</p> <p>Index-number of the variables composing a time average of the type <math>\langle f_1 * f_2 \dots * f_n \rangle</math>. For every time average <b>imom</b> to calculate:</p> <ul style="list-style-type: none"> <li>- if <b>idfmom(ii,imom)</b> is positive, it refers to the index-number of a solved variable (stored in the array <b>rtp</b>), like for instance a velocity component (<b>iu</b>, <b>iv</b>, <b>iw</b>) or the pressure (<b>ipr</b>)</li> <li>- if <b>idfmom(ii,imom)</b> is negative, it refers to the index-number of an auxiliary variable (stored in <b>propce</b>), like for instance the density (<b>idfmom(ii,imom)=-irom</b>)</li> </ul> <p>useful if and only if the user wants to calculate time averages</p>
<b>ntdmom</b>	<p>ia      integer      [-1]      O      L1</p> <p>For every average <b>imom</b> to calculate, absolute time step number at which the calculation should begin. The value -1 means "never". Every strictly negative value (in particular -1) will be considered an error and cause the calculation to stop (because the user is supposed to want to calculate the averages he has defined)</p> <p>useful if and only if the user wants to calculate time averages</p>
<b>imoold</b>	<p>ia      -2, <math>1 \leq \text{integer} \leq \text{jbmomt}</math>      [-2]      O      L1</p> <p>Correspondence table of the averages in the case of a calculation restart. In this case, for every average <b>imom</b> in the current calculation (<math>1 \leq \text{imom} \leq \text{nbmomx}</math>), <b>imoold(imom)</b> gives the index-number of the corresponding average in the previous calculation (in which <b>jbmomt</b> averages were calculated).</p> <ul style="list-style-type: none"> <li>- if <b>imoold(imom) = -2</b>, the user lets the code automatically determine the correspondence. By default, the average <b>ii</b> in the current calculation will correspond to the average <b>ii</b> in the previous calculation, if it existed. Otherwise, <b>ii</b> will be a new average.</li> <li>- if <b>imoold(imom) = -1</b>, the average is reset to zero.</li> <li>- if <b>imoold(imom) = kk</b>, the average <b>imom</b> will correspond to the average <b>kk=imoold(imom)</b> in the previous calculation.</li> </ul> <p>useful if and only if the user wants to calculate averages. Allows to add or suppress some averages, to reset them, to change their order, ...</p> <p><i>Warning: if the calculation is not a restart, imoold must not be specified (its value must remain -2)</i></p>

### 9.1.5 Others

<b>impusr</b>	<p>ia      strictly positive integer      [70 to 69+nusrmx=79]      O      L3</p> <p>unit numbers for potential user specified files</p> <p>useful if and only if the user needs files (therefore always useful, by security)</p>
<b>ficusr</b>	<p>ca      string of 13 characters      [usrf* or usrf*.n*]      O      L1</p> <p>name of the potential user specified files. In the case of a non-parallel calculation, the suffix applied to the file name is a two digit number: from <b>usrf01</b> to <b>usrf10</b>. In the case of a parallel-running calculation, the four digit processor index-number is added to the suffix. For instance, for a calculation running on two processors: from <b>usrf01.n.0001</b> to <b>usrf10.n.0001</b> and from <b>usrf01.n.0002</b> to <b>usrf10.n.0002</b>. The opening, closing, format and location of these files must be managed by the user.</p> <p>useful if and only if the user needs files (therefore always useful, by security)</p>
<b>ilisvr</b>	<p>ia      -999, 1 or 0      [-999]      O      L1</p> <p>for every quantity (variable, physical or numerical property ...), indicator concerning</p>

the writing in the execution report file

= -999: automatically converted into 1 if the concerned quantity is one of the main variables (pressure, velocity, turbulence, scalar), the density, the time step if `idtvar`  $\neq$  0 or the turbulent viscosity. Otherwise converted into 0.

= 1: writing in the execution listing.

= 0: no writing.

always useful

**iwarni**      ia      integer      [0]      O      L1  
**iwarni**(**ivar**) characterises the level of detail of the outputs for the variable **ivar** (from 1 to **nvar**). The quantity of information increases with its value. Impose the value 0 or 1 for a reasonable listing size. Impose the value 2 to get a maximum quantity of information, in case of problem during the execution.  
always useful

**nomvar**      ca      string of less than 80 characters      [“”]      O      L1  
name of the variables (unknowns, physical properties ...): used in the execution listing, in the post-processing files, etc.  
“”: not initialised (the code chooses the manes by default)  
It is recommended not to define variable names of more than 8 characters, to get a clear execution listing (some advanced writing levels take into account only the first 8 characters).  
always useful

**ntlist**      i      -1 or strictly positive integer      [1]      O      L1  
writing period in the execution report file  
= -1: no writing  
> 0: period (every **ntlist** time step)  
The value of **ntlist** must be adapted according to the number of iterations carried out in the calculation. Keeping **ntlist** to 1 will indeed provide a maximum volume of information, but if the number of time steps is too large, the execution report file might become too big and unusable (problems with disk space, memory problems while opening the file with a text editor, problems finding the desired information in the file, ...).  
always useful

**ntsuit**      i      -1, 0 or positive or null integer      [0]      O      L3  
saving period of the restart files  
= -2: no restart at all  
= -1: only at the end of the calculation  
= 0: by default (four times during the calculation)  
> 0: period  
always useful

## 9.2 Numerical options

### 9.2.1 Calculation management

**iecaux**      i      0 or 1      [1]      O      L2  
indicates the writing (=1) or not (=0) of the auxiliary calculation restart file  
always useful

EDF R&D	<i>Code_Saturne</i> version 2.1.3 practical user's guide	<i>Code_Saturne</i> documentation Page 148/205
<b>ileaux</b>	i      0 or 1      [1] indicates the reading (=1) or not (=0) of the auxiliary calculation restart file useful only in the case of a calculation restart	O      L2
<b>inpdto</b>	i      0 or 1      [0] indicates the calculation mode: 1 for a zero time step control calculation, <i>i.e.</i> without solving the transport equations, and 0 for a standard calculation. In case of a calculation using the control mode ( <b>inpdto</b> =1), when the calculation is not a restart, the equations are not solved, but the physical properties and the boundary conditions are calculated. When the calculation is a restart, the physical properties and the boundary conditions are those read from the restart file (note: in the case of a second-order time scheme, the mass flow is modified as if a normal time step was realised: the mass flow generated in an potential post-processing is therefore not the mass flow read from the restart file). In the control mode ( <b>inpdto</b> =1), the variable <b>ntmabs</b> is not used. In the standard mode ( <b>inpdto</b> =0), the code solves the equations at least once, even if <b>ntmabs</b> =0. always useful	O      L1
<b>isuite</b>	i      0 or 1      [0] indicator of a calculation restart (=1) or not (=0) always useful. This value is set automatically by the code, depending on whether a restart directory is present, and should not be modified by the user	C      L1
<b>ntcabs</b>	i      integer      [ntpabs] current time step number always useful <b>ntcabs</b> is initialised and updated automatically by the code, its value is not to be modified by the user	O      L3
<b>ntmabs</b>	i      integer > <b>ntpabs</b> [10] number of the last time step after which the calculation stops. It is an absolute number: for the restart calculations, <b>ntmabs</b> takes into account the number of time steps of the previous calculations. For instance, after a first calculation of 3 time steps, a restart file of 2 time steps is realised by setting <b>ntmabs</b> =3+2=5 always useful	C      L1
<b>ntpabs</b>	i      integer      [0, read] number of the last time step in the previous calculation. In the case of a restart calculation, <b>ntpabs</b> is read from the restart file. Otherwise it is initialised to 0 always useful <b>ntpabs</b> is initialised automatically by the code, its value is not to be modified by the user	O      L3
<b>tmarus</b>	r      -1 or strictly positive real      [-1] margin in seconds on the remaining CPU time which is necessary to allow the calculation to stop automatically and write all the required results (for the machines having a queue manager) = -1: calculated automatically > 0: margin defined by the user always useful, but the default value should not be changed unless absolutely necessary.	O      L3

<b>ttcabs</b>	<p>r positive or null real number [ttpabs] O L3</p> <p>physical simulation time at the current time step. For the restart calculations, ttcabs takes into account the physical time of the previous calculations.</p> <p>If the time step is uniform (<code>idtvar=0</code> or <code>1</code>), <b>ttcabs</b> increases of <code>dt</code> (value of the time step) at each iteration. If the time step is non-uniform (<code>idtvar=2</code>), <b>ttcabs</b> increases of <code>dtref</code> at each time step.</p> <p>always useful</p> <p><b>ttcabs</b> is initialised and updated automatically by the code, its value is not to be modified by the user</p>
<b>ttpabs</b>	<p>r positive or null real number [0, read] O L3</p> <p>simulation physical time at the last time step of the previous calculation. In the case of a restart calculation, <b>ttpabs</b> is read from the restart file. Otherwise it is initialised to 0.</p> <p>always useful</p> <p><b>ttcabs</b> is initialised automatically by the code, its value is not to be modified by the user</p>

## 9.2.2 Scalar unknowns

<b>iscold</b>	<p>ia -999, <math>1 \leq \text{integer} \leq \text{jscal}</math> [-999] O L1</p> <p>correspondence table of the scalars in the case of a calculation restart. For a calculation restart with <code>nscal</code> scalars, <code>iscold(iscal)</code> gives, for every scalar <code>iscal</code> of the current calculation (<math>1 \leq \text{iscal} \leq \text{nscal}</math>), the index-number of the corresponding scalar in the previous calculation (in which <code>jscal</code> scalars were taken into account).</p> <p><code>iscold(iscal) = -999</code>: the code automatically determines the correspondence. By default, the following rules are applied:</p> <ul style="list-style-type: none"> <li>- the user scalar <code>ii</code> of the current calculation is initialised by the the user scalar <code>ii</code> of the previous calculation, if this scalar existed already (otherwise, <code>ii</code> is a new scalar).</li> <li>- the particular physics scalar <code>jj</code> is initialised by the particular physics scalar <code>jj</code> of the previous calculation if this scalar existed already (otherwise, <code>jj</code> is a new scalar).</li> </ul> <p><code>iscold(iscal) = kk</code>: the scalar <code>iscal</code> (user or particular physics scalar) is initialised by the scalar <code>kk=iscold(iscal)</code> of the previous calculation.</p> <p>always useful. Allows to add or remove some scalars, to change the solving order, to change the physics, ...</p>
<b>nscaus</b>	<p>i <math>0 \leq \text{integer} \leq \text{nscmax}</math> [0] O L1</p> <p>number of user scalars solutions of an advection equation</p> <p>always useful</p>
<b>iscavr</b>	<p>ia <math>0, 1 \leq \text{integer} \leq \text{nscal}</math> [0] O L1</p> <p>if the scalar <code>iscal</code> is the average of the square of the fluctuations of a scalar <code>kk</code>, then <code>iscavr(iscal)=kk</code>. Otherwise <code>iscavr(iscal)=0</code>. For <code>iscal</code> and <code>kk</code>, the user can only use index-numbers referring to user scalars (<math>\leq \text{nscaus}</math>).</p> <p>always useful</p>
<b>iscalt</b>	<p>ia -1 or integer <math>&gt; 0</math> [-1] O L1</p> <p><b>iscalt</b> is the index-number of the scalar representing the temperature or the enthalpy. If <code>iscalt=-1</code>, no scalar represents the temperature nor the enthalpy. When a specific</p>



physics module is activated (gas combustion, pulverised coal, electricity or compressible), the user must not modify `iscalt` (the choice is made automatically)<sup>33</sup>.  
useful if and only if `nscal`  $\geq$  1

<b>iscsth</b>	ia	-1, 0, 1, 2 or 3	[-10]	O	L1
type of scalar = -10: not specified. By default, the code chooses <code>iscsth(iscal)=0</code> for the scalars apart from <code>iscalt</code> = -1: temperature in degrees Celsius (use only in case of radiation modeling) = 0: passive scalar = 1: temperature (in Kelvin if the radiation modeling is activated) = 2: enthalpy = 3: total energy (this value is automatically chosen by the code when using the compressible module, it must never be used otherwise and must never be specified by the user) useful if and only if <code>nscal</code> $\geq$ 1. The distinction between <code>iscsth(iscal) = -1</code> or <code>1</code> (respectively degrees Celsius or Kelvin) is useful only in case of radiation modeling. For calculations without radiation modeling, use <code>iscsth(iscal)=1</code> for the temperature. When a particular physics module is activated (gas combustion, pulverised coal, electricity or compressible), the user must not modify <code>iscsth</code> (the choice is made automatically: the solved variable is the enthalpy or the total energy). It is also reminded that, in the case of a coupling with SYRTHES, the solved thermal variable should be the temperature ( <code>iscsth(iscal)=1</code> or <code>-1</code> ). More precisely, everything is designed in the code to allow for the running of a calculation coupled with SYRTHES with the enthalpy as thermal variable (the correspondence and conversion is then specified by the user in the subroutine <code>usthht</code> ). However this case has never been used in practice and has therefore not been tested. With the compressible model, it is possible to carry out calculations coupled with SYRTHES, although the thermal scalar represents the total energy and not the temperature.					
<b>iclvfl</b>	ia	-1, 0, 1 or 2	[-1]	O	L3
for every scalar <code>iscal</code> representing the average of the square of the fluctuations of another scalar <code>ii=iscavr(iscal)</code> (noted $f$ ), indicator of the clipping method = -1: no clipping because the scalar does not represent the average of the square of the fluctuations of another scalar = 0: clipping to 0 for lower values = 1: clipping to 0 for lower values and to $(f - f_{min})(f_{max} - f)$ for higher values, where $f$ is the associated scalar, $f_{min}$ and $f_{max}$ its minimum and maximum values specified by the user ( <i>i.e.</i> <code>scamin(ii)</code> and <code>scamax(ii)</code> ) = 2: clipping to <code>max(0,scamin(iscal))</code> for lower values and to <code>scamax(iscal)</code> for higher values. <code>scamin</code> and <code>scamax</code> are limits specified by the user useful for the scalars <code>iscal</code> for which <code>iscavr(iscal)&gt;0</code> .					
<b>itbrrb</b>	i	0 or 1	[0]	O	L3
Reconstruction (=1) or not (=0) of the temperature, enthalpy or total energy value in the boundary cells. Useful in the case of coupling with SYRTHES and with radiation.					
<b>icpsyr</b>	ia	-999,0,1	[-999]	O	L3
For each scalar <code>iscal</code> , <code>icpsyr(iscal)</code> indicates if it is coupled with SYRTHES (=1) or not (=0). There can be only one coupled scalar per calculation. =-999: by default					

<sup>33</sup>in the case of the compressible module, `iscalt` does not correspond to the temperature nor enthalpy but to the total energy



- `icpsyr(iscal)=1` for the thermal scalar `iscal=(iscalt)` when a coupling with SYRTHES has been specified in the Interface or the launch script
  - `icpsyr(iscal)=0` otherwise
    - = 0: the scalar `iscal` is not coupled with SYRTHES
    - = 1: the scalar `iscal` is coupled with SYRTHES
- useful in case of coupling with SYRTHES

## 9.2.3 Definition of the equations

<code>istat</code>	ia	0 or 1	[1 or 0]	O	L2
for each unknown <code>ivar</code> to calculate, indicates if non-stationary terms are present ( <code>istat(ivar)=1</code> ) or not (0) in the matrices. By default, <code>istat</code> is set to 0 for the pressure (variable <code>ivar=ipr</code> ) or $\bar{f}$ in v2f modeling (variable <code>ivar=ifb</code> ) and set to 1 for the other unknowns. useful for all the unknowns					
<code>iconv</code>	ia	0 or 1	[1 or 0]	O	L2
for each unknown <code>ivar</code> to calculate, indicates if the convection is taken into account ( <code>iconv(ivar)=1</code> ) or not (0). By default, <code>iconv</code> is set to 0 for the pressure (variable <code>ivar=ipr</code> ) or $\bar{f}$ in v2f modeling (variable <code>ivar=ifb</code> ) and set to 1 for the other unknowns. useful for all the unknowns					
<code>idiff</code>	ia	0 or 1	[1]	O	L2
for each unknown <code>ivar</code> to calculate, indicates if the diffusion is taken into account ( <code>idiff(ivar)=1</code> ) or not (0) useful for all the unknowns					
<code>idifft</code>	ia	0 or 1	[1]	O	L3
for each unknown <code>ivar</code> to calculate, when diffusion is taken into account ( <code>idiff(ivar)=1</code> ), <code>idifft(ivar)</code> indicates if the turbulent diffusion is taken into account ( <code>idifft(ivar)=1</code> ) or not (0) useful for all the unknowns					
<code>idircl</code>	ia	0 or 1	[1 or 0]	O	L3
for each unknown <code>ivar</code> to calculate, indicates whether the diagonal of the matrix should be slightly shifted ( <code>idircl(ivar)=1</code> ) or not (0) if there is no Dirichlet boundary condition and if <code>istat=0</code> . Indeed, in such a case, the matrix for the general advection/diffusion equation is singular. A slight shift in the diagonal will make it invertible again. By default, <code>idircl</code> is set to 1 for all the unknowns, except $\bar{f}$ in v2f modeling, since its equation contains another diagonal term that ensures the regularity of the matrix. useful for all the unknowns					
<code>ivisse</code>	ia	0 or 1	[1]	O	L3
indicates whether the source terms in transposed gradient and velocity divergence should be taken into account in the momentum equation. In the compressible module, these terms also account for the volume viscosity (cf. <code>viscv0</code> and <code>iviscv</code> ): $\partial_i [(\kappa - 2/3 (\mu + \mu_t)) \partial_k U_k] + \partial_j [(\mu + \mu_t) \partial_i U_j]$ = 0: not taken into account = 1: taken into account always useful					

## 9.2.4 Definition of the time advancement

<b>idtvar</b>	i	-1, 0, 1, 2	[0]	O	L1
type of time step = 0: constant in time and spatially uniform = 1: variable in time and spatially uniform = 2: variable in time and in space = -1: steady-state algorithm If the numerical scheme is a second-order in time, only the option 0 is allowed. always useful					
<b>iptlro</b>	i	0 or 1	[0]	O	L2
when density gradients and gravity are present, a local thermal time step can be calculated, based on the Brunt-Vaissala frequency. In numerical simulations, it is usually wise for the time step to be lower than this limit, otherwise numerical instabilities may appear <b>iptlro</b> indicates whether the time step should be limited to the local thermal time step (=1) or not (=0) when <b>iptlro</b> =1, the listing shows the number of cells where the time step has been clipped due to the thermal criterium, as well as the maximum ratio between the time step and the maximum thermal time step. If <b>idtvar</b> =0, since the time step is fixed and cannot be clipped, this ratio can be larger than 1 <sup>34</sup> . When <b>idtvar</b> >0, this ratio will be smaller than 1, except if the constraint <b>dtmin</b> has prevented the code from reaching a sufficiently low value for <b>dt</b> useful when density gradients and gravity are present					
<b>cdtvar</b>	ra	strictly positive real number	[1]	O	L1
multiplicative factor applied to the time step for each scalar Hence, the time step used when solving the evolution equation for the variable is the time step used for the dynamic equations (velocity/pressure) multiplied by <b>cdtvar</b> . The size of the array <b>cdtvar</b> is <b>nvar</b> . For instance, the multiplicative coefficient applied to the scalar 2 is <b>cdtvar(isca(2))</b> . Yet, the value of <b>cdtvar</b> for the velocity components and the pressure is not used. Also, although it is possible to change the value of <b>cdtvar</b> for the turbulent variables, it is highly unrecommended useful if and only if <b>nscal</b> ≥ 1					
<b>coumax</b>	r	strictly positive real number	[1]	O	L1
target local or maximum Courant number in case of non-constant time step useful if <b>idtvar</b> ≠ 0					
<b>foumax</b>	r	strictly positive real number	[10]	O	L1
target local or maximum Fourier number in case of non-constant time step useful if <b>idtvar</b> ≠ 0					
<b>dtref</b>	r	strictly positive real number	[-grand*10]	C	L1
reference time step always useful. It is the time step value used in the case of a calculation run with a uniform and constant time step, <i>i.e.</i> <b>idtvar</b> =0 (restart calculation or not). It is the value used to initialise the time step in the case of an initial calculation run with a non-constant time step ( <b>idtvar</b> =1 or 2). It is also the value used to initialise the time step in the case					

<sup>34</sup>it is then the user's choice to decide whether he should diminish DTREF or not

of a restart calculation in which the type of time step has been changed (for instance, `idtvar=1` in the new calculation and `idtvar=0` or `2` in the previous calculation): see `usini`

<b>dtmin</b>	r	positive or null real number	[0.1*dtref]	O	L2
lower limit for the calculated time step when non-constant time step is activated useful if <code>idtvar</code> $\neq$ 0					
<b>dtmax</b>	r	strictly positive real number	[1000*dtref]	O	L2
upper limit for the calculated time step when non-constant time step is activated useful if <code>idtvar</code> $\neq$ 0					
<b>varrdt</b>	r	strictly positive real number	[0.1]	O	L3
maximum allowed relative increase in the calculated time step value between two successive time steps (to ensure stability, any decrease in the time step is immediate and without limit) useful if <code>idtvar</code> $\neq$ 0					
<b>relxst</b>	r	$0 < \text{real} \leq 1$	[0.9]	O	L2
relaxation coefficient for the steady algorithm ( <code>relaxp(iphas)=1</code> : no relaxation) useful if <code>idtvar=-1</code>					
<b>relaxv</b>	ra	$0 \leq \text{real} \leq 1$	[0.7 or 1.]	O	L3
for each variable <code>ivar</code> , relaxation coefficient of the variable. This relaxation parameter is only useful for the pressure with the unsteady algorithm (so as to improve the convergence in case of meshes of insufficient quality or and for some of the turbulent models ( <code>iturb(iphas) = 20, 21, 50</code> or <code>60</code> and <code>ikecou(iphas)=0</code> ; if <code>ikecou(iphas)=1</code> , <code>relaxv(ivar)</code> is not used, whatever its value may be). Default values are 0.7 for turbulent variables and 1. for pressure. It also stores the value of the relaxation coefficient when using the steady algorithm, deduced from the value of <code>relxst</code> (defaulting to <code>relaxv(ivar eq 1. - relxst)</code> ) useful only for the pressure and for turbulent variables if and only if ( $k - \varepsilon$ , $v2f$ or $k - \omega$ models without coupling) with the unsteady algorithm always useful with the steady algorithm					

#### NON-CONSTANT TIME STEP

The calculation of the time step uses a reference time step DTREF (at the calculation beginning). Later, every time step, the time step value is calculated by taking into account the different existing limits, in the following order:

- `coumax`, `foumax`: the more restrictive limit between both is used (in the compressible module, the acoustic limitation is added),
- `varrdt`: progressive increase and immediate decrease in the time step,
- `iptlro`: limitation by the thermal time step,
- `dtmax` and `dtmin`: clipping of the time step to the maximum, then to the minimum limit.

## 9.2.5 Turbulence

<b>iturb</b>	ia	0, 10, 20, 21, 30, 31, 40, 41, 42, 50, 60, [7099]	O	L1
indicator of the turbulence model <code>iturb</code> = -999: not initialised. This value is not allowed and must be modified by the				

user

- = 0: laminar
- = 10: mixing length (not validated)
- = 20:  $k - \varepsilon$
- = 21:  $k - \varepsilon$  with linear production (Laurence & Guimet)
- = 30:  $R_{ij} - \varepsilon$  “standard” LRR (Launder, Reece & Rodi)
- = 31:  $R_{ij} - \varepsilon$  SSG (Speziale, Sarkar & Gatski)
- = 40: LES (Smagorinsky model)
- = 41: LES (dynamic model)
- = 42: LES (WALE model)
- = 50: v2-f,  $\varphi$ -model version
- = 60:  $k - \omega$ , SST version
- = 70: Spalart-Allmaras

always useful

The  $k - \varepsilon$  (standard and linear production) and  $R_{ij} - \varepsilon$  (LRR and SSG) turbulence models implemented in *Code\_Saturne* are “High-Reynolds” models. It is therefore necessary to make sure that the thickness of the first cell neighboring the wall is larger than the thickness of the viscous sublayer (at the wall,  $y^+ > 2.5$  is required as a minimum, and preferably between 30 and 100)<sup>35</sup>. If the mesh does not respect this condition, the results may be biased (particularly if thermal processes are involved). Using scalable wall-functions (cf. key word **ideuch**) may help avoiding this problem.

The v2-f model is a “Low-Reynolds” model, it is therefore necessary to make sure that the thickness of the first cell neighboring the wall is smaller than the thickness of the viscous sublayer ( $y^+ < 1$ ).

The  $k - \omega$  SST model provides correct results whatever the thickness of the first cell. Yet, it requires the knowledge of the distance to the wall in every cell of the calculation domain. The user may refer to the key word **icdpar** for more details about the potential limitations.

The  $k - \varepsilon$  model with linear production allows to correct the known flaw of the standard  $k - \varepsilon$  model which overestimates the turbulence level in case of strong velocity gradients (stopping point).

With LES, the wall functions are usually not greatly adapted. It is generally more advisable (if possible) to refine the mesh towards the wall so that the first cell is in the viscous sublayer, where the boundary conditions are simple natural no-slip conditions.

Concerning the LES model, the user may refer to the subroutine **ussmag** for complements about the dynamic model. Its usage and the interpretation of its results require particular attention. In addition, the user must pay further attention when using the dynamic model with the least squares method based on a partial extended neighborhood (**imrga**=3). Indeed, the results may be degraded if the user does not implement his own way of averaging the dynamic constant in **ussmag** (*i.e.* if the user keeps the local average based on the extended neighborhood).

**ideuch**      ia      0, 1 or 2      [0 or 1]      O      L2

indicates the type of wall function is used for the velocity boundary conditions on a frictional wall.

- = 0: one-scale model
- = 1: two-scale model
- = 2: scalable wall function

**ideuch** is initialised to 0 for **iturb**=0, 10, 40 or 41 (laminar, mixing length, LES).

**ideuch** is initialised to 1 for **iturb**=20, 21, 30, 31 or 60 ( $k - \varepsilon$ ,  $R_{ij} - \varepsilon$  LRR,  $R_{ij} - \varepsilon$  SSG and  $k - \omega$  SST models).

The v2f model (**iturb**=50) is not designed to use wall functions (the mesh must be “low Reynolds”).

The value **ideuch**=1 is not compatible with **iturb**=0, 10, 40 or 41 (laminar, mixing length and LES).

Concerning the  $k - \varepsilon$  and  $R_{ij} - \varepsilon$  models, the two-scales model is usually at least as

<sup>35</sup>While creating the mesh,  $y^+ = \frac{yu^*}{\nu}$  is generally unknown. It can be roughly estimated as  $\frac{yU}{10\nu}$ , where  $U$  is the characteristic velocity,  $\nu$  is the kinematic viscosity of the fluid and  $y$  is the mid-height of the first cell near the wall.

satisfactory as the one-scale model.

The scalable wall function allows to virtually “shift” the wall when necessary in order to be always in a logarithmic layer. It is used to make up for the problems related to the use of High-Reynolds models on very refined meshes.

useful if **iturb** is different from 50

<b>ilogpo</b>	ia	0 or 1	[1]	O	L3
type of wall function used for the velocity: power law ( <b>ilogpo</b> =0) or logarithmic law ( <b>ilogpo</b> =1) always useful					
<b>ypluli</b>	ra	real number > 0	[1/xkappa, 10.88]	O	L3
limit value of $y^+$ for the viscous sublayer <b>ypluli</b> depends on the chosen wall function: it is initialised to 10.88 for the scalable wall function ( <b>ideuch</b> =2), otherwise it is initialised to $1/\kappa \approx 2,38$ In LES, <b>ypluli</b> is taken by default to be 10.88 always useful					

#### $k - \varepsilon$ , $k - \varepsilon$ WITH LINEAR PRODUCTION, v2-F AND $k - \omega$ SST

<b>igrake</b>	ia	0 or 1	[1]	O	L1
indicates if the terms related to gravity in the equations of $k$ and $\varepsilon$ or $\omega$ are taken into account ( <b>igrake</b> =1) or not (0) useful if and only if <b>iturb</b> = 20, 21, 50 or 60, ( <b>gx</b> , <b>gy</b> , <b>gz</b> ) $\neq$ (0,0,0) and the density is not uniform					
<b>igrhok</b>	ia	0 or 1	[0]	O	L2
indicates if the term $\frac{2}{3}\text{grad } \rho k$ is taken into account ( <b>igrhok</b> =1) or not (0) in the velocity equation useful if and only if <b>iturb</b> = 20, 21, 50 or 60. This term may generate non-physical velocities at the wall. When it is not explicitly taken into account, it is implicitly included into the pressure.					
<b>ikecou</b>	ia	0 or 1	[0 or 1]	O	L3
indicates if the coupling of the source terms of $k$ and $\varepsilon$ or $k$ and $\omega$ is taken into account ( <b>ikecou</b> =1) or not (0) if <b>ikecou</b> =0 in $k - \varepsilon$ model, the term in $\varepsilon$ in the equation of $k$ is made implicit <b>ikecou</b> is initialised to 0 if <b>iturb</b> = 21 or 60, and to 1 if <b>iturb</b> = 20 <b>ikecou</b> =1 is forbidden when using the v2f model ( <b>iturb</b> =50) useful if and only if <b>iturb</b> = 20, 21 or 60 ( $k - \varepsilon$ and $k - \omega$ models)					
<b>iclkep</b>	ia	0 or 1	[0]	O	L3
indicates the clipping method used for $k$ and $\varepsilon$ , for the $k - \varepsilon$ and v2f models = 0: clipping in absolute value = 1: clipping from physical relations useful if and only if <b>iturb</b> = 20, 21 or 50 ( $k - \varepsilon$ and v2f models). The results obtained with the method corresponding to <b>iclkep</b> =1 showed in some cases a substantial sensitivity to the values of the length scale <b>amax</b> . The option <b>iclkep</b> =1 is therefore not recommended, and, if chosen, must be used cautiously.					

$R_{ij} - \varepsilon$  (LRR AND SSG)

<b>iclp<sub>ptr</sub></b>	ia      0 or 1      [0]      O      L3 indicates if $R_{ij}$ is made partially implicit ( <b>iclp<sub>ptr</sub></b> =1) or not (0) in the wall boundary conditions. useful if and only if <b>iturb</b> = 30 or 31 ( $R_{ij} - \varepsilon$ model)
<b>icls<sub>yr</sub></b>	ia      0 or 1      [0]      O      L3 indicates if $R_{ij}$ is made partially implicit ( <b>icls<sub>yr</sub></b> =1) or not (0) in the symmetry boundary conditions. useful if and only if <b>iturb</b> = 30 or 31 ( $R_{ij} - \varepsilon$ model)
<b>idif<sub>re</sub></b>	ia      0 or 1      [1]      O      L3 complete ( <b>idif<sub>re</sub></b> =1) or simplified (0) taking into account of the diagonals of the diffusion tensors of $R_{ij}$ and $\varepsilon$ , for the LLR model. useful if and only if <b>iturb</b> = 30 (LLR $R_{ij} - \varepsilon$ model)
<b>igrari</b>	ia      0 or 1      [1]      O      L1 indicates if the terms related to gravity are taken into account ( <b>igrari</b> =1) or not (0) in the equations of $R_{ij} - \varepsilon$ . useful if and only if <b>iturb</b> = 30 or 31 and ( <b>gx</b> , <b>gy</b> , <b>gz</b> ) $\neq$ (0,0,0) ( $R_{ij} - \varepsilon$ model with gravity) and the density is not uniform
<b>irij<sub>ec</sub></b>	ia      0 or 1      [0]      O      L2 indicates if the wall echo terms in $R_{ij} - \varepsilon$ LRR model are taken into account ( <b>irij<sub>ec</sub></b> =1) or not (0). useful if and only if <b>iturb</b> = 30 ( $R_{ij} - \varepsilon$ LRR). It is not recommended to take these terms into account: they have an influence only near the walls, their expression is hardly justifiable according to some authors and, in the configurations studied with <i>Code_Saturne</i> , they did not bring any improvement in the results. In addition, their use induces an increase in the calculation time. The wall echo terms imply the calculation of the distance to the wall for every cell in the domain. See <b>icdpar</b> for potential restrictions due to this.
<b>irij<sub>nu</sub></b>	ia      0 or 1      [0]      O      L3 addition ( <b>irij<sub>nu</sub></b> =1) or not (0) of a turbulent viscosity in the matrix of the incremental system solved for the velocity in $R_{ij} - \varepsilon$ models. The goal is to improve the stability of the calculation. The usefulness of <b>irij<sub>nu</sub></b> =1 has however not been clearly demonstrated. Since the system is solved in incremental form, this extra turbulent viscosity does not change the final solution for steady flows. However, for unsteady flows, the parameter <b>nsw<sub>rsm</sub></b> should be increased. useful if and only if <b>iturb</b> = 30 or 31 ( $R_{ij} - \varepsilon$ model).
<b>irij<sub>rb</sub></b>	ia      0 or 1      [0]      O      L3 reconstruction ( <b>irij<sub>rb</sub></b> =1) or not (0) of the boundary conditions at the walls for $R_{ij}$ and $\varepsilon$ . useful if and only if <b>iturb</b> = 30 or 31 ( $R_{ij} - \varepsilon$ model)

LES

<b>ivrtex</b>	i	0 or 1	[0]	O	L1	<p>activates (=1) or not (=0) the generation of synthetic turbulence at the different inlet boundaries with the LES model (generation of unsteady synthetic eddies)</p> <p>useful if <b>iturb</b>=40 or 41</p> <p>this key word requires the completion of the routine <b>usvort</b></p>
<b>isuivo</b>	i	0 or 1	[isuite]	O	L1	<p>for the vortex method, indicates whether the synthetic vortices at the inlet should be initialised (=0) or read from the restart file <b>ficmvo</b>.</p> <p>useful if <b>iturb</b>=40 or 41 and <b>ivrtex</b>=1</p>
<b>idries</b>	ia	0 or 1	[0,1]	O	L2	<p><b>idries</b> activates (1) or not (0) the van Driest wall-damping for the Smagorinsky constant (the Smagorinsky constant is multiplied by the damping function <math>1 - e^{-y^+/cdries}</math>, where <math>y^+</math> designates the adimensional distance to the nearest wall). The default value is 1 for the Smagorinsky model and 0 for the dynamic model.</p> <p>the van Driest wall-damping requires the knowledge of the distance to the nearest wall for each cell in the domain. Refer to key word <b>icdpar</b> for potential limitations</p> <p>useful if and only if <b>iturb</b> = 40 or 41</p>
<b>cdries</b>	ra	real number > 0	[26]	O	L3	<p><b>cdries</b> is the constant appearing in the van Driest damping function applied to the Smagorinsky constant: <math>1 - e^{-y^+/cdries}</math></p> <p>useful if and only if <b>iturb</b> = 40 or 41</p>
<b>csmago</b>	ra	real number > 0	[0.065]	O	L2	<p><b>csmago</b> is the Smagorinsky constant used in the Smagorinsky model for LES</p> <p>the sub-grid scale viscosity is calculated by <math>\mu_{sg} = \rho C_{smago}^2 \bar{\Delta}^2 \sqrt{2\bar{S}_{ij}\bar{S}_{ij}}</math> where <math>\bar{\Delta}</math> is the width of the filter and <math>\bar{S}_{ij}</math> the filtered strain rate</p> <p>useful if and only if <b>iturb</b> = 40</p>
<b>smagmx</b>	ra	real number > 0	[10*csmago]	O	L3	<p><b>smagmx</b><sup>2</sup> is the maximum allowed value for the variable <math>C</math> appearing in the LES dynamic model (the “square” comes from the fact that the variable of the dynamic model corresponds to the square of the constant of the Smagorinsky model). Any larger value yielded by the calculation procedure of the dynamic model will be clipped to <b>smagmx</b><sup>2</sup></p> <p>useful if and only if <b>iturb</b> = 41</p>
<b>xlesfl</b>	ra	real number > 0	[2]	O	L3	<p><b>xlesfl</b> is a constant used to define, for each cell <math>\Omega_i</math>, the width of the (implicit) filter: <math>\bar{\Delta} = xlesfl(ales *  \Omega_i )^{bles}</math></p> <p>useful if and only if <b>iturb</b> = 40 or 41</p>
<b>ales</b>	ra	real number > 0	[1]	O	L3	<p><b>ales</b> is a constant used to define, for each cell <math>\Omega_i</math>, the width of the (implicit) filter: <math>\bar{\Delta} = xlesfl(ales *  \Omega_i )^{bles}</math></p> <p>useful if and only if <b>iturb</b> = 40 or 41</p>
<b>bles</b>	ra	real number > 0	[1/3]	O	L3	<p><b>bles</b> is a constant used to define, for each cell <math>\Omega_i</math>, the width of the (implicit) filter:</p>

$\bar{\Delta} = xlesfl(ales * |\Omega_i|)^{bles}$   
useful if and only if **iturb** = 40 or 41

**xlesfd**      **ra**      real number > 0      [1.5]      O      L3  
**xlesfd** is the constant used to define, for each cell  $\Omega_i$ , the width of the explicit filter used in the framework of the LES dynamic model:  
 $\widetilde{\Delta} = xlesfd \bar{\Delta}$   
useful if and only if **iturb** = 41

## 9.2.6 Time scheme

By default, the standard time scheme is a first-order. A second-order scheme is activated automatically with LES modeling. On the other hand, when “specific physics” (gas combustion, pulverised coal, compressible module) are activated, the second-order scheme is not allowed.

In the current version, the second-order time scheme is not compatible with the estimators (**iescal**), the velocity-pressure coupling (**ipucou**), the modeling of hydrostatic pressure (**icalhy** and **iphydr**) and the time- or space-variable time step (**idtvar**).

Also, in the case of a rotation periodicity, a proper second-order is not ensured for the velocity, but calculations remain possible.

It is recommended to keep the default values of the variables listed below. Hence, in standard cases, the user does not need to specify these options.

**ischtp**      **ia**      1 or 2      [1 or 2]      O      L2  
**ischtp** indicates the order of the activated time scheme (this indicator allows the code to automatically complete the other indicators related to the time scheme)  
= 1: first-order  
= 2: second-order  
when **ischtp**=2, the physical properties are by default not second-order. It is possible to modify this by means of the following indicators.  
due to specific coupling between certain variables, the source terms in the turbulence equations (except convection and diffusion) cannot be second order, except with the  $R_{ij}$  models (cf. key word **isto2t**)  
by default, **ischtp** is initialised to 2 with the LES model and 1 otherwise  
always useful

**istmpf**      **ia**      0, 1 or 2      [0 or 1]      O      L3  
**istmpf** specifies the time scheme activated for the mass flow. The chosen value for **istmpf** will automatically determine the value given to the variable **thetf1**  
= 0: “explicit” first-order: the mass flow calculated at the previous time step (“n”) is used in the convective terms of all the equations (momentum, turbulence and scalars)  
= 1: “standard” first-order: the mass flow calculated at the previous time step (“n”) is used in the convective terms of the momentum equation, and the updated mass flow (time “n+1”) is used in the equations of turbulence and scalars  
= 2: second-order: the mass flow used in the momentum equations is extrapolated at “n+**thetf1**” (=n+1/2) from the values at the two former time steps (Adams Bashforth); the mass flow used in the equations for turbulence and scalars is interpolated at time “n+**thetf1**” (=n+1/2) from the values at the former time step and at the newly calculated “n+1” time step.  
by default, **istmpf**=2 is used in the case of a second-order time scheme (if **ischtp**=2) and **istmpf**=1 otherwise  
always useful



<b>isno2t</b>	<p>ia      0, 1 or 2      [0 or 1]      O      L3</p> <p><b>isno2t</b> specifies the time scheme activated for the source terms of the momentum equation, apart from convection and diffusion (for instance: head loss, transposed gradient, ...).</p> <p>= 0: "standard" first-order: the terms which are linear functions of the solved variable are implicit and the others are explicit</p> <p>= 1: second-order: the terms of the form <math>S_i\phi</math> which are linear functions of the solved variable <math>\phi</math> are expressed as second-order terms by interpolation (according to the formula <math>(S_i\phi)^{n+\theta} = S_i^n[(1-\theta)\phi^n + \theta\phi^{n+1}]</math>, <math>\theta</math> being given by the value of <b>thetav</b> associated with the variable <math>\phi</math>); the other terms <math>S_e</math> are expressed as second-order terms by extrapolation (according to the formula <math>(S_e)^{n+\theta} = [(1+\theta)S_e^n - \theta S_e^{n-1}]</math>, <math>\theta</math> being given by the value of <b>thetsn</b>=0.5)</p> <p>= 2: the linear terms <math>S_i\phi</math> are treated in the same way as when <b>isno2t</b>=1; the other terms <math>S_e</math> are extrapolated according to the same formula as when <b>isno2t</b>=1, but with <math>\theta</math>=<b>thetsn</b>=1</p> <p>by default, <b>isno2t</b> is initialised to 1 (second-order) when the selected time scheme is second-order (<b>ischtp</b>=2), otherwise to 0.</p> <p>always useful</p>
<b>isto2t</b>	<p>ia      0, 1 or 2      [0]      O      L3</p> <p><b>isto2t</b> specifies the time scheme activated for the source terms of the turbulence equations (related to <math>k</math>, <math>R_{ij}</math>, <math>\varepsilon</math>, <math>\omega</math>, <math>\varphi</math>, <math>\bar{f}</math>), apart from convection and diffusion.</p> <p>= 0: "standard" first-order: the terms which are linear functions of the solved variable are implicit and the others are explicit</p> <p>= 1: second-order: the terms of the form <math>S_i\phi</math> which are linear functions of the solved variable <math>\phi</math> are expressed as second-order terms by interpolation (according to the formula <math>(S_i\phi)^{n+\theta} = S_i^n[(1-\theta)\phi^n + \theta\phi^{n+1}]</math>, <math>\theta</math> being given by the value of <b>thetav</b> associated with the variable <math>\phi</math>); the other terms <math>S_e</math> are expressed as second-order terms by extrapolation (according to the formula <math>(S_e)^{n+\theta} = [(1+\theta)S_e^n - \theta S_e^{n-1}]</math>, <math>\theta</math> being given by the value of <b>thetst</b>=0.5)</p> <p>= 2: the linear terms <math>S_i\phi</math> are treated in the same way as when <b>isto2t</b>=1; the other terms <math>S_e</math> are extrapolated according to the same formula as when <b>isto2t</b>=1, but with <math>\theta</math>=<b>thetst</b>=1</p> <p>due to certain specific couplings between the turbulence equations, <b>isto2t</b> is allowed the value 1 or 2 only for the <math>R_{ij}</math> models (<b>iturb</b>=30 or 31); hence, it is always initialised to 0.</p> <p>always useful</p>
<b>isso2t</b>	<p>ia      0, 1 or 2      [0 or 1]      O      L3</p> <p>for each scalar <b>iscal</b>, <b>isso2t(iscal)</b> specifies the time scheme activated for the source terms of the equation for the scalar, apart from convection and diffusion (for instance: variance production, user-specified terms, ...).</p> <p>= 0: "standard" first-order: the terms which are linear functions of the solved variable are implicit and the others are explicit</p> <p>= 1: second-order: the terms of the form <math>S_i\phi</math> which are linear functions of the solved variable <math>\phi</math> are expressed as second-order terms by interpolation (according to the formula <math>(S_i\phi)^{n+\theta} = S_i^n[(1-\theta)\phi^n + \theta\phi^{n+1}]</math>, <math>\theta</math> being given by the value of <b>thetav</b> associated with the variable <math>\phi</math>); the other terms <math>S_e</math> are expressed as second-order terms by extrapolation (according to the formula <math>(S_e)^{n+\theta} = [(1+\theta)S_e^n - \theta S_e^{n-1}]</math>, <math>\theta</math> being given by the value of <b>thetss(iscal)</b>=0.5)</p> <p>= 2: the linear terms <math>S_i\phi</math> are treated in the same way as when <b>isso2t</b>=1; the other terms <math>S_e</math> are extrapolated according to the same formula as when <b>isso2t</b>=1, but with <math>\theta</math>=<b>thetss(iscal)</b>=1</p> <p>by default, <b>isso2t(iscal)</b> is initialised to 1 (second-order) when the selected time</p>

scheme is second-order (`ischtp=2`), otherwise to 0.  
always useful

<b>iroext</b>	ia      0, 1 or 2      [0]      O      L3
<b>iroext</b> specifies the time scheme activated for the physical property $\phi$ “density”. = 0: “standard” first-order: the value calculated at the beginning of the current time step (from the variables known at the end of the previous time step) is used = 1: second-order: the physical property $\phi$ is extrapolated according to the formula $\phi^{n+\theta} = [(1 + \theta)\phi^n - \theta\phi^{n-1}]$ , $\theta$ being given by the value of <code>thetro=0.5</code> = 2: first-order: the physical property $\phi$ is extrapolated at $n + 1$ according to the same formula as when <code>iroext=1</code> but with $\theta=\text{thetro}=1$ always useful	
<b>iviext</b>	ia      0, 1 or 2      [0]      O      L3
<b>iviext</b> specifies the time scheme activated for the physical property $\phi$ “total viscosity” (molecular+turbulent or sub-grid viscosities). = 0: ”standard” first-order: the value calculated at the beginning of the current time step (from the variables known at the end of the previous time step) is used = 1: second-order: the physical property $\phi$ is extrapolated according to the formula $\phi^{n+\theta} = [(1 + \theta)\phi^n - \theta\phi^{n-1}]$ , $\theta$ being given by the value of <code>thetvi=0.5</code> = 2: first-order: the physical property $\phi$ is extrapolated at $n + 1$ according to the same formula as when <code>iviext=1</code> , but with $\theta=\text{thetvi}=1$ always useful	
<b>icpext</b>	ia      0, 1 or 2      [0]      O      L3
<b>icpext</b> specifies the time scheme activated for the physical property $\phi$ “specific heat”. = 0: ”standard” first-order: the value calculated at the beginning of the current time step (from the variables known at the end of the previous time step) is used = 1: second-order: the physical property $\phi$ is extrapolated according to the formula $\phi^{n+\theta} = [(1 + \theta)\phi^n - \theta\phi^{n-1}]$ , $\theta$ being given by the value of <code>thetcp=0.5</code> = 2: first-order: the physical property $\phi$ is extrapolated at $n + 1$ according to the same formula as when <code>icpext=1</code> , but with $\theta=\text{thetcp}=1$ always useful	
<b>ivsext</b>	ia      0, 1 or 2      [0]      O      L3
for each scalar <code>iscal</code> , <code>ivsext(iscal)</code> specifies the time scheme activated for the physical property $\phi$ “diffusivity”. = 0: ”standard” first-order: the value calculated at the beginning of the current time step (from the variables known at the end of the previous time step) is used = 1: second-order: the physical property $\phi$ is extrapolated according to the formula $\phi^{n+\theta} = [(1 + \theta)\phi^n - \theta\phi^{n-1}]$ , $\theta$ being given by the value of <code>thetvs(iscal)=0.5</code> = 2: first-order: the physical property $\phi$ is extrapolated at $n + 1$ according to the same formula as when <code>ivsext=1</code> , but with $\theta=\text{thetvs(iscal)}=1$ always useful	
<b>thetav</b>	ra $0 \leq \text{real} \leq 1$ [1 or 0.5]      O      L3
for each variable <code>ivar</code> , <code>thetav(ivar)</code> is the value of $\theta$ used to express at the second-order the terms of convection, diffusion and the source terms which are linear functions of the solved variable (according to the formula $\phi^{n+\theta} = (1 - \theta)\phi^n + \theta\phi^{n+1}$ ). Generally,	

only the values 1 and 0.5 are used. The user is not allowed to modify this variable.

= 1: first-order

= 0.5: second-order

Concerning the pressure, the value of **thetav** is always 1. Concerning the other variables, the value **thetav**=0.5 is used when the second-order time scheme is activated by **ischtp**=2 (standard value for LES calculations), otherwise **thetav** is set to 1. always useful

**thetfl**      ra       $0 \leq \text{real} \leq 1$       [0 or 0.5]      O      L3  
**thetfl** is the value of  $\theta$  used to interpolate the convective fluxes of the variables when a second-order time scheme has been activated for the mass flow (see **istmpf**) generally, only the value 0.5 is used. The user is not allowed to modify this variable.  
= 0.0: “explicit” first-order (corresponds to **istmpf**=0 or 1)  
= 0.5: second-order (corresponds to **istmpf**=2). The mass flux will be interpolated according to the formula  $Q^{n+\theta} = \frac{1}{2-\theta}Q^{n+1} + \frac{1-\theta}{2-\theta}Q^{n+1-\theta}$ .  
always useful

**thetsn**      ra       $0 \leq \text{real} \leq 1$       [0, 0.5 or 1]      O      L3  
**thetsn** is the value of  $\theta$  used to extrapolate the non linear explicit source terms  $S_e$  of the momentum equation, when the source term extrapolation has been activated (see **isno2t**), following the formula  
 $(S_e)^{n+\theta} = (1 + \theta)S_e^n - \theta S_e^{n-1}$   
the value of  $\theta$ =**thetsn** is deduced from the value chosen for **isno2t**. Generally, only the value 0.5 is used. The user is not allowed to modify this variable.  
= 0: first-order (unused, corresponds to **isno2t**=0)  
= 0.5: second-order (used when **isno2t**=1)  
= 1: first-order (used when **isno2t**=2)  
always useful

**thetst**      ra       $0 \leq \text{real} \leq 1$       [0, 0.5 or 1]      O      L3  
**thetst** is the value of  $\theta$  used to extrapolate the non linear explicit source terms  $S_e$  of the turbulence equations, when the source term extrapolation has been activated (see **isto2t**), following the formula  
 $(S_e)^{n+\theta} = (1 + \theta)S_e^n - \theta S_e^{n-1}$   
the value of  $\theta$ =**thetsn** is deduced from the value chosen for **isto2t**. Generally, only the value 0.5 is used. The user is not allowed to modify this variable.  
= 0: first-order (unused, corresponds to **isto2t**=0)  
= 0.5: second-order (used when **isto2t**=1)  
= 1: first-order (used when **isto2t**=2)  
always useful

**thetss**      ra       $0 \leq \text{real} \leq 1$       [0, 0.5 or 1]      O      L3  
for each scalar **iscal**, **thetss(iscal)** is the value of  $\theta$  used to extrapolate the non linear explicit source terms  $S_e$  of the scalar equation, when the source term extrapolation has been activated (see **isso2t**), following the formula  
 $(S_e)^{n+\theta} = (1 + \theta)S_e^n - \theta S_e^{n-1}$   
the value of  $\theta$ =**thetss(iscal)** is deduced from the value chosen for **isso2t(iscal)**. Generally, only the value 0.5 is used. The user is not allowed to modify this variable.  
= 0: first-order (unused, corresponds to **isso2t(iscal)**=0)  
= 0.5: second-order (used when **isso2t(iscal)**=1)  
= 1: first-order (used when **isso2t(iscal)**=2)  
useful if **nscal**>1

<b>thetro</b>	ra $0 \leq \text{real} \leq 1$ [0, 0.5 or 1]      O      L3 <b>thetro</b> is the value of $\theta$ used to extrapolate the physical property $\phi$ “density” when the extrapolation has been activated (see <b>iroext</b> ), according to the formula $\phi^{n+\theta} = (1 + \theta)\phi^n - \theta\phi^{n-1}$ the value of $\theta=\text{thetro}$ is deduced from the value chosen for <b>iroext</b> . Generally, only the value 0.5 is used. The user is not allowed to modify this variable. = 0: first-order (unused, corresponds to <b>iroext</b> =0) = 0.5: second-order (corresponds to <b>iroext</b> =1) = 1: first-order (corresponds to <b>iroext</b> =2) always useful
<b>thetvi</b>	ra $0 \leq \text{real} \leq 1$ [0, 0.5 or 1]      O      L3 <b>thetvi</b> is the value of $\theta$ used to extrapolate the physical property $\phi$ “total viscosity” when the extrapolation has been activated (see <b>iviext</b> ), according to the formula $\phi^{n+\theta} = (1 + \theta)\phi^n - \theta\phi^{n-1}$ the value of $\theta=\text{thetvi}$ is deduced from the value chosen for <b>iviext</b> . Generally, only the value 0.5 is used. The user is not allowed to modify this variable. = 0: first-order (unused, corresponds to <b>iviext</b> =0) = 0.5: second-order (corresponds to <b>iviext</b> =1) = 1: first-order (corresponds to <b>iviext</b> =2) always useful
<b>thetcp</b>	ra $0 \leq \text{real} \leq 1$ [0, 0.5 or 1]      O      L3 <b>thetcp</b> is the value of $\theta$ used to extrapolate the physical property $\phi$ “specific heat” when the extrapolation has been activated (see <b>icpext</b> ), according to the formula $\phi^{n+\theta} = (1 + \theta)\phi^n - \theta\phi^{n-1}$ the value of $\theta=\text{thetcp}$ is deduced from the value chosen for <b>icpext</b> . Generally, only the value 0.5 is used. The user is not allowed to modify this variable. = 0: first-order (unused, corresponds to <b>icpext</b> =0) = 0.5: second-order (corresponds to <b>icpext</b> =1) = 1: first-order (corresponds to <b>icpext</b> =2) always useful
<b>thetvs</b>	ra $0 \leq \text{real} \leq 1$ [0, 0.5 or 1]      O      L3 for each scalar <b>iscal</b> , <b>thetvs(iscal)</b> is the value of $\theta$ used to extrapolate the physical property $\phi$ “diffusivity” when the extrapolation has been activated (see <b>ivsext</b> ), according to the formula $\phi^{n+\theta} = (1 + \theta)\phi^n - \theta\phi^{n-1}$ the value of $\theta=\text{thetvs(iscal)}$ is deduced from the value chosen for <b>ivsext(iscal)</b> . Generally, only the value 0.5 is used. The user is not allowed to modify this variable. = 0: first-order (unused, corresponds to <b>ivsext(iscal)</b> =0) = 0.5: second-order (corresponds to <b>ivsext(iscal)</b> =1) = 1: first-order (corresponds to <b>ivsext(iscal)</b> =2) useful if <b>nscal</b> >1

## 9.2.7 Gradient reconstruction

<b>imrgra</b>	i      0, 1, 2, 3 or 4      [0]      O      L2 indicates the type of gradient reconstruction (one method for all the variables) = 0: iterative reconstruction of the non-orthogonalities = 1: least squares method based on the first neighbor cells (cells which share a face with the treated cell) = 2: least squares method based on the extended neighborhood (cells which
---------------	---

share a node with the treated cell)

= 3: least squares method based on a partial extended neighborhood (all first neighbors plus the extended neighborhood cells that are connected to a face where the non-orthogonality angle is larger than parameter **anamax**)

= 4: iterative reconstruction with initialisation using the least squares method (first neighbors)

if **imrga** fails due to probable mesh quality problems, it is usually effective to use **imrga=3**. Moreover, **imrga=3** is usually faster than **imrga=0** (but with less feedback on its use).

it should be noted that **imrga=1, 2 or 3** automatically triggers a gradient limitation procedure. See **imligr**.

useful if and only if there is **n** so that **nswrgr(n) > 1**

<b>nswrgr</b>	ia	positive integer	[100]	O	L3	for each unknown <b>ivar</b> , <b>nswrgr(ivar) ≤ 1</b> indicates that the gradients are not reconstructed if <b>imrga = 0 or 4</b> , <b>nswrgr(ivar)</b> is the number of iterations for the gradient reconstruction if <b>imrga = 1, 2 or 3</b> , <b>nswrgr(ivar) &gt; 1</b> indicates that the gradients are reconstructed (but the method is not iterative, so any value larger than 1 for <b>nswrgr</b> yields the same result) useful for all the unknowns
<b>epsrgr</b>	ra	real number > 0	[10 <sup>-5</sup> ]	O	L3	for each unknown <b>ivar</b> , relative precision for the iterative gradient reconstruction: <b>epsrgr(ivar)</b> useful for all the unknowns when <b>imrga = 0 or 4</b>
<b>imligr</b>	ia	-1, 0 or 1	[-1 or 1]	O	L3	for each unknown <b>ivar</b> , indicates the type of gradient limitation: <b>imligr(ivar)</b> =-1: no limitation = 0: based on the neighbors = 1: superior order for all the unknowns, <b>imligr</b> is initialised to -1 if <b>imrga=0 or 4</b> and to 1 if <b>imrga = 1, 2 or 3</b> useful for all the unknowns
<b>climgr</b>	ra	real number > 0	[1.5]	O	L3	for each unknown <b>ivar</b> , factor of gradient limitation: <b>climgr(ivar)</b> (high value means little limitation) useful for all the unknowns <b>ivar</b> for which <b>imligr(ivar) ≠ -1</b>
<b>extrag</b>	ra	0, 0.5 or 1	[0]	O	L3	for the variable "pressure" <b>ivar=ipr</b> , extrapolation coefficient of the gradients at the boundaries. It affects only the Neumann conditions. The only possible values of <b>extrag(ipr)</b> are: = 0: homogeneous Neumann calculated at first-order = 0.5: improved homogeneous Neumann, calculated at second-order in the case of an orthogonal mesh and at first-order otherwise = 1: gradient extrapolation (gradient at the boundary face equal to the gradient in the neighbor cell), calculated at second-order in the case of an orthogonal mesh and at first-order otherwise <b>extrag</b> often allows to correct the non-physical velocities that appear on horizontal walls when density is variable and there is gravity. It is strongly advised to keep



<b>imgr</b>	ia	0 or 1	[0]	O	L3	for each unknown <b>ivar</b> , indicates the use ( <b>imgr(ivar)=1</b> ) or not ( <b>=0</b> ) of the algebraic multigrid method for the solution of the linear systems <b>imgr(ivar)</b> can be set independently for every variable always useful. Generally, its use is designed for the variable “pressure” in case of meshes with strongly stretched cells. It is recommended not to modify <b>imgr</b>
<b>ncegrm</b>	i	integer > 0	[30]	O	L3	for the multigrid method, maximum number of cells on the coarsest grid useful if and only if <b>imgr(ivar) = 1</b> for at least one variable <b>ivar</b>
<b>ncymax</b>	ia	integer > 0	[100]	O	L3	for each unknown <b>ivar</b> , <b>ncymax(ivar)</b> is the maximum number of cycles when using the multigrid method. useful if and only if <b>imgr(ivar) = 1</b>
<b>ngrmax</b>	i	$1 \leq \text{integer} \leq \text{ngrmmx}$	[ngrmmx]	O	L3	when using the multigrid method, maximum number of grid levels useful if and only if <b>imgr(ivar) = 1</b> for at least one variable <b>ivar</b>
<b>ncymax</b>	ia	integer > 0	[10]	O	L3	for each unknown <b>ivar</b> , <b>ncymax(ivar)</b> is the maximum number of multigrid cycles. useful if and only if <b>imgr(ivar) = 1</b>
<b>nitmgf</b>	ia	integer > 0	[10]	O	L3	for each unknown <b>ivar</b> , <b>nitmgf(ivar)</b> is the maximum number of iterations on all grids except for the coarsest when the multigrid method is used; the resolution on the coarsest grid uses <b>nitmax</b> . useful if and only if <b>imgr(ivar) = 1</b>

#### WARNING

The algebraic multigrid method has only been tested for the “pressure” variable (**imgr(ipr)=1**).

### 9.2.9 Convective scheme

<b>blencv</b>	ra	$0 \leq \text{real} \leq 1$	[0 or 1]	O	L1	for each unknown <b>ivar</b> to calculate, <b>blencv(ivar)</b> indicates the proportion of second-order convective scheme (0 corresponds to an “upwind” first-order scheme) ; in case of LES calculation, a second-order scheme is recommended and activated by default ( <b>blencv=1</b> ) useful for all the unknowns <b>ivar</b> for which <b>iconv(ivar) = 1</b>
<b>ischcv</b>	ia	0 or 1	[1]	O	L2	for each unknown <b>ivar</b> to calculate, <b>ischcv(ivar)</b> indicates the type of second-order convective scheme = 0: Second Order Linear Upwind = 1: Centered useful for all the unknowns <b>ivar</b> which are convected ( <b>iconv(ivar)=1</b> ) and for which a second-order scheme is used ( <b>blencv(ivar) &gt; 0</b> )



**isstpc**      **ia**      0 or 1      [0]      O      L2

for each unknown **ivar** to calculate, **isstpc(ivar)** indicates whether a “slope test” should be used to switch from a second-order to an “upwind” convective scheme under certain conditions, to ensure stability.

    = 0: “slope test” activated for the considered unknown  
    = 1: “slope test” deactivated for the considered unknown

useful for all the unknowns **ivar** which are convected (**iconv(ivar)=1**) and for which a second-order scheme is used (**blencv(ivar) > 0**).

the use of the “slope test” stabilises the calculation but may bring the order in space to decrease quickly.

### 9.2.10 Pressure-continuity step

**iprco**      **i**      0 or 1      [1]      O      L3

indicates if the pressure-continuity step is taken into account (1) or not (0)  
always useful

**arak**      **ra**       $0 < \text{real} \leq 1$       [1]      O      L3

**arak** is the Arakawa coefficient before the Rhie& Chow filter  
always useful

**irevmc**      **ia**      0, 1 or 2      [0]      O      L3

method used to update the velocity after the pressure correction:

- standard gradient of pressure increment (**irevmc=0**)
- least squares on the pressure increment (**irevmc=1**)
- “rt0” *i.e.* least squares on the updated mass flux (**irevmc=2**)

the method **irevmc=2** is generally not recommended  
always useful

**iphydr**      **i**      0 or 1      [0]      O      L2

method for taking into account the balance between the pressure gradient and the source terms (gravity and head losses): by extension it will be referenced as “taking into account of the hydrostatic pressure”

    = 0: standard algorithm  
    = 1: improved algorithm

always useful

When the density effects are important, the choice of **iphydr=1** allows to improve the interpolation of the pressure and correct the non-physical velocities which may appear in highly stratified areas or near horizontal walls (thus avoiding the use of **extrag** if the non-physical velocities are due only to gravity effects).

The improved algorithm also allows to eradicate the velocity oscillations which tend to appear at the frontiers of areas with high head losses.

In the case of a stratified flow, the calculation cost is higher when the improved algorithm is used (about 30% depending on the case) because the hydrostatic pressure must be recalculated at the outlet boundary conditions: see **icalhy**.

On meshes of insufficient quality, in order to improve the convergence, it may be useful to increase the number of iterations for the reconstruction of the pressure right-hand member, *i.e.* **nswrsm(ipr)**.

If head losses are present just along an outlet boundary, it is necessary to specify **icalhy=0** in order to deactivate the recalculation of the hydrostatic pressure at the boundary, which may otherwise cause instabilities.



**icalhy**      **i**      0 or 1      [0 or 1]      **O**      **L3**  
 activates the calculation of hydrostatic pressure boundary conditions at outlet boundaries  
     = 0: no calculation of the hydrostatic pressure at the outlet boundary  
     = 1: calculation of the hydrostatic pressure at the outlet boundary  
 always useful  
 This option is automatically specified depending on the choice of **iphydr** and the value of gravity (**icalhy**=1 if **iphydr**=1 and gravity is different from 0; otherwise **icalhy**=0). The activation of this option generates an additional calculation cost (about 30% depending on the case).  
 If head losses are present just along an outlet boundary, it is necessary to specify **icalhy**=0 in order to deactivate the recalculation of the hydrostatic pressure at the boundary, which may otherwise cause instabilities

### 9.2.11 Error estimators for Navier-Stokes

There are currently **nestmx**=4 types of local estimators provided at every time step, with two possible definitions for each<sup>38</sup>. These scalars indicate the areas (cells) in which some error types may be important. They are stored in the array **propce** containing the properties at the cells (see **iestim**). For each estimator, the code writes the minimum and maximum values in the listing and generates post-processing outputs along with the other variables.

The additional memory cost is about one real number per cell and per estimator. The additional calculation cost is variable. For instance, on a simple test case, the total estimator **iestot** generates an additional cost of 15 to 20 % on the CPU time<sup>39</sup>; the cost of the three others may be neglected. If the user wants to avoid the calculation of the estimators during the computation, it is possible to run a calculation without estimators first, and then activate them on a restart of one or two time steps.

It is recommended to use the estimators only for visual and qualitative analysis. Also, their use is compatible neither with a second-order time scheme nor with a calculation with a frozen velocity field.

**iest = iespre: prediction** (default name: EsPre). After the velocity prediction step (yielding  $\underline{u}^*$ ), the estimator  $\eta_{i,k}^{pred}(\underline{u}^*)$ , local variable calculated at every cell  $\Omega_i$ , is created from  $\underline{\mathcal{R}}^{pred}(\underline{u}^*)$ , which represents the residual of the equation solved during this step:

$$\begin{aligned} \underline{\mathcal{R}}^{pred}(\underline{u}^*) &= \rho^n \frac{\underline{u}^* - \underline{u}^n}{\Delta t} + \rho^n \underline{u}^n \cdot \underline{grad}(\underline{u}^*) - \text{div} \left( (\mu + \mu_t)^n \underline{grad}(\underline{u}^*) \right) + \underline{grad}(P^n) \\ &- \text{rest of the right-hand member } (\underline{u}^n, P^n, \text{other variables}^n) \end{aligned}$$

By definition:

$$\eta_{i,k}^{pred}(\underline{u}^*) = |\Omega_i|^{(k-2)/2} \|\underline{\mathcal{R}}^{pred}(\underline{u}^*)\|_{\mathbb{L}^2(\Omega_i)}$$

- The first family,  $k = 1$ , suppresses the volume  $|\Omega_i|$  which intrinsically appears with the norm  $\mathbb{L}^2(\Omega_i)$ .

- The second family,  $k = 2$ , exactly represents the norm  $\mathbb{L}^2(\Omega_i)$ . The size of the cell therefore appears in its calculation and induces a weighting effect.

$\eta_{i,k}^{pred}(\underline{u}^*)$  is ideally equal to zero when the reconstruction methods are perfect and the associated system is solved exactly.

**iest = iesder: drift** (default name: EsDer). The estimator  $\eta_{i,k}^{der}(\underline{u}^{n+1})$  is based on the following quantity (intrinsic to the code):

$$\begin{aligned} \eta_{i,k}^{der}(\underline{u}^{n+1}) &= |\Omega_i|^{(k-2)/2} \|\text{div}(\text{corrected mass flow after the pressure step}) - \Gamma\|_{\mathbb{L}^2(\Omega_i)} \\ &= |\Omega_i|^{(1-k)/2} \|\text{div}(\text{corrected mass flow after the pressure step}) - \Gamma\| \end{aligned} \quad (5)$$

<sup>38</sup>choice made by the user

<sup>39</sup>indeed, all the first-order in space differential terms have to be recalculated at the time  $t^{n+1}$

Ideally, it is equal to zero when the Poisson equation related to the pressure is solved exactly.

**iest = iescor: correction** (default name: EsCor). The estimator  $\eta_{i,k}^{corr}(\underline{u}^{n+1})$  comes directly from the mass flow calculated with the updated velocity field:

$$\eta_{i,k}^{corr}(\underline{u}^{n+1}) = |\Omega_i|^{\delta_{2,k}} |div(\rho^n \underline{u}^{n+1}) - \Gamma|$$

The velocities  $\underline{u}^{n+1}$  are taken at the cell centers, the divergence is calculated after projection on the faces.

$\delta_{2,k}$  represents the Kronecker symbol.

- The first family,  $k = 1$ , is the absolute raw value of the divergence of the mass flow minus the mass source term.

- The second family,  $k = 2$ , represents a physical property and allows to evaluate the difference in  $kg.s^{-1}$ .

Ideally, it is equal to zero when the Poisson equation is solved exactly and the projection from the mass flux at the faces to the velocity at the cell centers is made in a set of functions with null divergence.

**iest = iestot: total** (default name: EsTot). The estimator  $\eta_{i,k}^{tot}(\underline{u}^{n+1})$ , local variable calculated at every cell  $\Omega_i$ , is based on the quantity  $\underline{\mathcal{R}}^{tot}(\underline{u}^{n+1})$ , which represents the residual of the equation using the updated values of  $\underline{u}$  and  $P$ :

$$\begin{aligned} \underline{\mathcal{R}}^{tot}(\underline{u}^{n+1}) &= \rho^n \frac{\underline{u}^{n+1} - \underline{u}^n}{\Delta t} + \rho^n \underline{u}^{n+1} \cdot \underline{grad}(\underline{u}^{n+1}) - div \left( (\mu + \mu_t)^n \underline{grad}(\underline{u}^{n+1}) \right) + \underline{grad}(P^{n+1}) \\ &- \text{rest of the right-hand member } (\underline{u}^{n+1}, P^{n+1}, \text{other variables}^n) \end{aligned}$$

By definition:

$$\eta_{i,k}^{tot}(\underline{u}^{n+1}) = |\Omega_i|^{(k-2)/2} \|\underline{\mathcal{R}}^{tot}(\underline{u}^{n+1})\|_{L^2(\Omega_i)}$$

The mass flux in the convective term is recalculated from  $\underline{u}^{n+1}$  expressed at the cell centers (and not taken from the updated mass flow at the faces).

As for the prediction estimator:

- The first family,  $k = 1$ , suppresses the volume  $|\Omega_i|$  which intrinsically appears with the norm  $L^2(\Omega_i)$ .

- The second family,  $k = 2$ , exactly represents the norm  $L^2(\Omega_i)$ . The size of the cell therefore appears in its calculation and induces a weighting effect.

The estimators are evaluated depending on the values of **iescal**.

<b>iescal</b>	ia	0, 1 or 2	[0]	O	L1
---------------	----	-----------	-----	---	----

**iescal(iest)** indicates the calculation mode for the error estimator **iest** (**iespre**, **iesder**, **iescor** or **iestot**), for the Navier-Stokes equation:  
**iescal** = 0: estimator not calculated,  
**iescal** = 1: the estimator  $\eta_{i,1}^*$  is calculated, without contribution of the volume,  
**iescal** = 2: the estimator  $\eta_{i,2}^*$  is calculated, with contribution of the volume ("norm  $L^2$ "), except for **iescor**, for which  $|\Omega_i| \eta_{i,1}^{corr}$  is calculated.

The name of the estimators appearing in the listing and the post-processing is made up of the default name (given before), followed by the value of **iescal**. For instance, EsPre2 is the estimator **iespre** calculated with **iescal**=2.

always useful

## 9.2.12 Calculation of the distance to the wall

<b>icdpar</b>	i	-1, 1, -2 or 2	[-1]	O	L2
---------------	---	----------------	------	---	----

specifies the method used to calculate the distance to the wall  $y$  and the adimensional

distance  $y^+$  for all the cells of the calculation domain (when necessary):

= 1: standard algorithm (based on a Poisson equation for  $y$  and convection equation for  $y^+$ ), with reading of the distance to the wall from the restart file if possible

=-1: standard algorithm (based on a Poisson equation for  $y$  and convection equation for  $y^+$ ), with systematic recalculation of the distance to the wall in case of calculation restart

= 2: former algorithm (based on geometrical considerations), with reading of the distance to the wall from the restart file if possible

=-2: former algorithm (based on geometrical considerations) with systematic recalculation of the distance to the wall in case of calculation restart

In case of restart calculation, if the position of the walls haven't changed, reading the distance to the wall from the restart file can save a fair amount of CPU time.

Useful in  $R_{ij} - \varepsilon$  model with wall echo (`iturb=30` and `irijec=1`), in LES with van Driest damping (`iturb=40` and `idries=1`) and in  $k - \omega$  SST (`iturb=60`).

By default, `icdpar` is initialised to -1, in case there has been a change in the definition of the boundary conditions between two computations (change in the number or the positions of the walls). Yet, with the  $k - \omega$  SST model, the distance to the wall is needed to calculate the turbulent viscosity, which is done before the calculation of the distance to the wall. Hence, when this model is used (and only in that case), `icdpar` is set to 1 by default, to ensure total continuity of the calculation at restart.

**As a consequence, with the  $k - \omega$  SST model, if the number and positions of the walls are changed at a calculation restart, it is mandatory for the user to set `icdpar` explicitly to -1**, otherwise the distance to the wall used will not correspond to the actual position of the walls.

The former algorithm is not compatible with parallelism nor periodicity. Also, whatever the value chosen for `icdpar`, the calculation of the distance to the wall is made at the most once for all at the beginning of the calculation. It is therefore not compatible with moving walls. Please contact the development team if you need to override this limitation.

The following options are related to `icdpar=1` or -1. The options of level 2 are described first. Some options are used only in the case of the calculation of the adimensional distance to the wall  $y^+$  (LES model with van Driest damping). Most of these key words are simple copies of the key words for the numerical options of the general equations, with a potentially specific value in the case of the calculation of the distance to the wall.

<code>iwarny</code>	i integer [0] O L2	specifies the level of the output writing concerning the calculation of the distance to the wall with <code>icdpar=1</code> or -1. The higher the value, the more detailed the outputs useful when <code>icdpar=1</code> or -1
<code>ntcmxy</code>	i positive integer [1000] O L2	number of pseudo-time iterations for the calculation of the adimensional distance to the wall $y^+$ useful when <code>icdpar=1</code> or -1 for the calculation of $y^+$
<code>nitmay</code>	i integer > 0 [10000] O L3	maximum number of iterations for the solution of the linear systems useful when <code>icdpar=1</code> or -1
<code>nswrsy</code>	i positive integer [1] O L3	number of iterations for the reconstruction of the right-hand members: corresponds

to **nswrsm**  
useful when **icdpar**=1 or -1

<b>nswrgy</b>	i	positive integer	[100]	O	L3
number of iterations for the gradient reconstruction: corresponds to <b>nswrgr</b> useful when <b>icdpar</b> =1 or -1					
<b>imligy</b>	i	-1, 0 or 1	[-1 or 1]	O	L3
type of gradient limitation: corresponds to <b>imligr</b> useful when <b>icdpar</b> =1 or -1					
<b>ircfly</b>	i	0 or 1	[1]	O	L3
indicates the reconstruction of the convective and diffusive fluxes at the faces: corresponds to <b>ircflu</b> useful when <b>icdpar</b> =1 or -1					
<b>ischcy</b>	i	0 or 1	[1]	O	L3
type of second-order convective scheme: corresponds to <b>ischcv</b> useful when <b>icdpar</b> =1 or -1 for the calculation of $y^+$					
<b>isstpy</b>	i	0 or 1	[0]	O	L3
indicates if a “slope test” should be used for a second-order convective scheme: corresponds to <b>isstpc</b> useful when <b>icdpar</b> =1 or -1 for the calculation of $y^+$					
<b>imgrpy</b>	i	0 or 1	[0]	O	L3
indicates whether the algebraic multigrid method should be used ( <b>imgr(ivar)</b> =1) or not (0): corresponds to <b>imgr</b> useful when <b>icdpar</b> =1 or -1					
<b>blency</b>	r	$0 \leq \text{real} \leq 1$	[0]	O	L3
proportion of second-order convective scheme: corresponds to <b>blencv</b> useful when <b>icdpar</b> =1 or -1 for the calculation of $y^+$					
<b>epsily</b>	r	real number $> 0$	$[10^{-8}]$	O	L3
relative precision for the solution of the linear systems: corresponds to <b>epsilo</b> useful when <b>icdpar</b> =1 or -1					
<b>epsrgy</b>	r	real number $> 0$	$[10^{-5}]$	O	L3
relative precision for the iterative gradient reconstruction: corresponds to <b>epsrgr</b> useful when <b>icdpar</b> =1 or -1					
<b>climgy</b>	r	real number $> 0$	[1.5]	O	L3
limitation factor of the gradients: corresponds to <b>climgr</b> useful when <b>icdpar</b> =1 or -1					
<b>extray</b>	r	0, 0.5 or 1	[0]	O	L3
extrapolation coefficient of the gradients at the boundaries: corresponds to <b>extrag</b> useful when <b>icdpar</b> =1 or -1					

coumxy	r	strictly positive real number	[5000]	O	L3	Target Courant number for the calculation of the adimensional distance to the wall useful when <code>icdpar</code> =1 or -1 for the calculation of $y^+$
epscvy	r	strictly positive real number	[10 <sup>-8</sup> ]	O	L3	relative precision for the convergence of the pseudo-transient regime for the calculation of the adimensional distance to the wall useful when <code>icdpar</code> =1 or -1 for the calculation of $y^+$
yplmxy	r	real number	[200]	O	L3	value of the adimensional distance to the wall above which the calculation of the distance is not necessary (for the damping) useful when <code>icdpar</code> =1 or -1 for the calculation of $y^+$

### 9.2.13 Others

iccvfg	i	0 or 1	[0]	O	L1	indicates whether the dynamic field should be frozen (1) or not (0) in such a case, the values of velocity, pressure and the variables related to the potential turbulence model ( $k$ , $R_{ij}$ , $\varepsilon$ , $\varphi$ , $\bar{f}$ , $\omega$ , turbulent viscosity) are kept constant over time and only the equations for the scalars are solved also, if <code>iccvfg</code> =1, the physical properties modified in <code>usphyv</code> will keep being updated. Beware of non-consistencies if these properties would normally affect the dynamic field (modification of density for instance) useful if and only if <code>nsca1</code> > 0 and the calculation is a restart
ipucou	i	0 or 1	[0]	O	L1	indicates the algorithm for velocity/pressure coupling = 0: standard algorithm = 1: reinforced coupling in case calculation with long time steps always useful (it is seldom advised, but it can prove very useful, for instance, in case of flows with weak convection effects and highly variable viscosity)
isuit1	i	0 or 1	[0]	O	L1	for the 1D wall thermal module, activation (1) or not(0) of the reading of the mesh and of the wall temperature from the <code>ficmt1</code> restart file useful if <code>nfpt1d</code> >0.
imvisf	i	0 or 1	[0]	O	L3	indicates the interpolation method used to project variables from the cell centers to the faces = 0: linear = 1: harmonic always useful
ircflu	ia	0 or 1	[1]	O	L2	for each unknown <code>ivar</code> , <code>ircflu(ivar)</code> indicates whether the convective and diffusive fluxes at the faces should be reconstructed: = 0: no reconstruction = 1: reconstruction

deactivating the reconstruction of the fluxes can have a stabilising effect on the calculation. It is sometimes useful with the  $k-\varepsilon$  model, if the mesh is strongly non-orthogonal in the near-wall region, where the gradients of  $k$  and  $\varepsilon$  are strong. In such a case, setting `ircflu(ik)=0` and `ircflu(iep)=0` will probably help (switching to a first order convective scheme, `blencv=0`, for  $k$  and  $\varepsilon$  might also help in that case)

always useful

<b>nswrsm</b>	ia	positive integer	[1, 2, 5 or 10]	O	L3
for each unknown <code>ivar</code> , <code>nswrsm(ivar)</code> indicates the number of iterations for the reconstruction of the right-hand members of the equations					
with a first-order scheme in time (standard case), the default values are 2 for pressure and 1 for the other variables. With a second-order scheme in time ( <code>ischtp=2</code> ) or LES, the default values are 5 for pressure and 10 for the other variables.					
useful for all the unknowns					
<b>epsrsm</b>	ra	real number > 0	[10 <sup>-8</sup> , 10 <sup>-5</sup> ]	O	L3
for each unknown <code>ivar</code> , relative precision on the reconstruction of the right hand-side. The default value is <code>epsrsm(ivar)=10<sup>-8</sup></code> . This value is set low on purpose. When there are enough iterations on the reconstruction of the right-hand side of the equation, the value may be increased (by default, in case of second-order in time, with <code>nswrsm</code> = 5 or 10, <code>epsrsm</code> is increased to 10 <sup>-5</sup> ).					
always useful					

## 9.3 Numerical, physical and modeling parameters

### 9.3.1 Numeric Parameters

These parameters correspond to numeric reference values in the code. They can be used but shall not be modified (they are defined as `parameter`).

<b>zero</b>	r	0	[0]	O	L3
Parameter containing the value 0					
<b>epzero</b>	r	10 <sup>-12</sup>	[10 <sup>-12</sup> ]	O	L3
“Small” real parameter, used for the comparisons of real numbers (absolute value of the difference lower than <code>epzero</code> )					
<b>pi</b>	r	3.141592653589793	[3.141592653589793]	O	L3
Parameter containing an approximate value of $\pi$					
<b>grand</b>	r	10 <sup>12</sup>	[10 <sup>12</sup> ]	O	L3
“Large” real parameter, generally used by default as a non physical value for the initialisations of variables which have to be modified by the user					
<b>rinfin</b>	r	10 <sup>30</sup>	[10 <sup>30</sup> ]	O	L3
Real parameter used to represent “infinity”					

### 9.3.2 Physical parameters

These parameters correspond to physical reference values in the code. They can be used but shall not be modified (they are defined as `parameter`).

EDF R&D	<i>Code_Saturne</i> version 2.1.3 practical user's guide			<i>Code_Saturne</i> documentation Page 173/205	
<b>tkelvi</b>	r	273.15	[273.15]	O	L3
Temperature in Kelvin corresponding to 0 degrees Celsius.					
<b>tkelvn</b>	r	-273.15	[-273.15]	O	L3
Temperature in degrees Celsius corresponding to 0 Kelvin.					
<b>rr</b>	r	8.31434	[8.31434]	O	L3
Perfect gas constant in $J/mol/K$					
<b>treft</b>	r	25 + tkelvi	[25 + tkelvi]	O	L3
Reference temperature for the specific physics, in $K$					
<b>prefth</b>	r	101325	[101325]	O	L3
Reference pressure for the specific physics, in $Pa$					
<b>volmol</b>	r	22.41.10 <sup>-3</sup>	[22.41.10 <sup>-3</sup> ]	O	L3
Molar volume under normal pressure and temperature conditions (1 atmosphere, 0°C) in $m^{-3}$					
<b>stephn</b>	r	5.6703.10 <sup>-8</sup>	[5.6703.10 <sup>-8</sup> ]	O	L3
Stephan constant for the radiative module $\sigma$ in $W.m^{-2}.K^{-4}$					
<b>permvi</b>	r	1.2566.10 <sup>-6</sup>	[1.2566.10 <sup>-6</sup> ]	O	L3
Vacuum magnetic permeability $\mu_0$ ( $=4\pi.10^{-7}$ ) in $kg.m.A^{-2}.s^{-2}$					
<b>epszer</b>	r	8.854.10 <sup>-12</sup>	[8.854.10 <sup>-12</sup> ]	O	L3
Vacuum permittivity $\varepsilon_0$ in $F.m^{-1}$					

### 9.3.3 Physical variables

<b>gx,gy,gz</b>	r	3 real numbers	[0,0,0]	O	L1
gravity components always useful					
<b>irovar</b>	ia	0 or 1	[-1]	C	L1
<b>irovar</b> =0 indicates that the density is constant. Its value is the reference density <b>ro0</b> . <b>irovar</b> =1 indicates that the density is variable: its variation law must be given in the user subroutine <b>usphyv</b> negative value: not initialised always useful					
<b>ivivar</b>	ia	0 or 1	[-1]	C	L1
<b>ivivar</b> =0 indicates that the molecular dynamic viscosity is constant. Its value is the reference molecular dynamic viscosity <b>viscl0</b> . <b>ivivar</b> =1 indicates that the molecular dynamic viscosity is variable: its variation law must be given in the user subroutine <b>usphyv</b> negative value: not initialised always useful					

EDF R&D	<i>Code_Saturne</i> version 2.1.3 practical user's guide	<i>Code_Saturne</i> documentation Page 174/205
<b>ro0</b>	<p>ra      real number <math>\geq 0</math>      [-grand*10]</p> <p><b>ro0</b> is the reference density negative value: not initialised its value is not used in gas or coal combustion modeling (it will be calculated following the perfect gas law, with <math>P_0</math> and <math>T_0</math>). With the compressible module, it is also not used by the code, but it may be (and often is) referenced by the user in user subroutines; it is therefore better to specify its value. always useful otherwise, even if a law defining the density is given by the user subroutine <b>usphyv</b> or <b>uselph</b> indeed, except with the compressible module, <i>Code_Saturne</i> does not use the total pressure <math>P</math> when solving the Navier-Stokes equation, but a reduced pressure <math>P^* = P - \rho_0 g \cdot (\underline{x} - \underline{x}_0) + P_0^* - P_0</math> where <math>\underline{x}_0</math> is a reference point (see <b>xyzp0</b>) and <math>P_0^*</math> and <math>P_0</math> are reference values (see <b>pred0</b> and <b>p0</b>). Hence, the term <math>-\text{grad } P + \rho g</math> in the equation is treated as <math>-\text{grad } P^* + (\rho - \rho_0)g</math>. The closer <b>ro0</b> is to the value of <math>\rho</math>, the more <math>P^*</math> will tend to represent only the dynamic part of the pressure and the faster and more precise its solution will be. Whatever the value of <b>ro0</b>, both <math>P</math> and <math>P^*</math> appear in the listing and the post-processing outputs. with the compressible module, the calculation is made directly on the total pressure</p>	C      L1
<b>viscl0</b>	<p>ra      real number <math>&gt; 0</math>      [-grand*10]</p> <p><b>viscl0</b> is the reference molecular dynamic viscosity negative value: not initialised always useful, it is the used value unless the user specifies the viscosity in the subroutine <b>usphyv</b></p>	C      L1
<b>srrom</b>	<p>r      <math>0 \leq \text{r��el} &lt; 1</math>      [-grand or 0]</p> <p>With gas combustion, pulverised coal or the electric module, <b>srrom</b> is the sub-relaxation coefficient for the density, following the formula: <math>\rho^{n+1} = \text{srrom } \rho^n + (1 - \text{srrom}) \rho^{n+1}</math> hence, with a zero value, there is no sub-relaxation. With combustion and pulverised coal, <b>srrom</b> is initialised to <b>-grand</b> and the user must specify a proper value through the Interface or the initialisation subroutines (<b>usd3p1</b>, <b>usebu1</b>, <b>uslwc1</b>, <b>uscpi1</b> or <b>uscpi11</b>). With the electric module, <b>srrom</b> is initialised in to 0 and may be modified by the user in <b>useli1</b>. With gas combustion, pulverised coal or electric arc, <b>srrom</b> is automatically used after the second time-step. With Joule effect, the user decides whether or not it will be used in <b>uselph</b> from the coding law giving the density.</p> <p>always useful with gas combustion, pulverized coal or the electric module.</p>	c or O      L1
<b>p0</b>	<p>ra      real number      [1.013e - 5]</p> <p><b>p0</b> is the reference pressure for the total pressure except with the compressible module, the total pressure <math>P</math> is evaluated from the reduced pressure <math>P^*</math> so that <math>P</math> is equal to <b>p0</b> at the reference position <math>\underline{x}_0</math> (given by <b>xyzp0</b>) with the compressible module, the total pressure is solved directly always useful</p>	O      L1
<b>pred0</b>	<p>ra      real number      [0]</p> <p><b>pred0</b> is the reference value for the reduced pressure <math>P^*</math> (see <b>ro0</b>) it is especially used to initialise the reduced pressure and as a reference value for the outlet boundary conditions for an optimised precision in the resolution of <math>P^*</math>, it is wiser to keep <b>pred0</b> to 0</p>	O      L3



with the compressible module, the “pressure” variable appearing in the equations directly represents the total pressure. It is therefore initialised to `p0` and not `pred0` (see `ro0`)

always useful, except with the compressible module

<b>xyzp0</b>	<p>ra      3 real numbers      [0,0,0]      O      L1</p> <p><code>xyzp0(ii)</code> is the <code>ii</code> coordinate (<math>1 \leq ii \leq 3</math>) of the reference point <math>\underline{x}_0</math> for the total pressure when there are no Dirichlet conditions for the pressure (closed domain), <code>xyzp0</code> does not need to be specified (unless the total pressure has a clear physical meaning in the configuration treated)</p> <p>when Dirichlet conditions on the pressure are specified but only through standart outlet conditions (as it is in most configurations), <code>xyzp0</code> does not need to be specified by the user, since it will be set to the coordinates of the reference outlet face (<i>i.e.</i> the code will automatically select a reference outlet boundary face and set <code>xyzp0</code> so that <math>P</math> equals <code>p0</code> at this face). Nonetheless, if <code>xyzp0</code> is pecified by the user, the calculation will remain correct</p> <p>when direct Dirichlet conditions are specified by the user (specific value set on specific boundary faces), it is better to specify the corresponding reference point (<i>i.e.</i> specify where the total pressure is <code>p0</code>). This way, the boundary conditions for the reduced pressure will be close to <code>pred0</code>, ensuring an optimal precision in the resolution. If <code>xyzp0</code> is not specified, the reduced pressure will be shifted, but the calculations will remain correct.</p> <p>with the compressible module, the “pressure” variable appearing in the equations directly represents the total pressure. <code>xyzp0</code> is therefore not used.</p> <p>always useful, except with the compressible module</p>
<b>t0</b>	<p>ra      real number      [0]      O      L1</p> <p><code>t0</code> is the reference temperature</p> <p>useful for the specific physics gas or coal combustion (initialisation of the density), for the electricity modules to initialise the domain temperature and for the comperssible module (initialisations). It must be given in Kelvin.</p>
<b>cp0</b>	<p>ra      real number &gt; 0      [-grand*10]      O      L1</p> <p><code>cp0</code> is the reference specific heat</p> <p>useful if there is <math>1 \leq n \leq nscaus</math><sup>40</sup> so that <code>iscsth(n)=1</code> (there is a scalar “temperature”), unless the user specifies the specific heat in the user subroutine <code>usphyv</code><sup>41</sup> (<code>icp</code> &gt; 0)</p> <p>with the compressible module or coal combustion, <code>cp0</code> is also needed even when there is no user scalar</p>
<b>icp</b>	<p>ia      0 or 1      [0]      O      L1</p> <p>indicates if the specific heat <math>C_p</math> is variable (<code>icp=1</code>) or not (0)</p> <p>When gas or coal combustion is activated, <code>icp</code> is automatically set to 0 (constant <math>C_p</math>). With the electric module, it is automatically set to 1. The user is not allowed to modify these default choices.</p> <p>When <code>icp=1</code> is specified, the code automatically modifies this value to make <code>icp</code> designate the effective index-number of the property “specific heat”. For each cell <code>iel</code>, the value of <math>C_p</math> is then specified by the user in the appropriate subroutine (<code>usphyv</code> for the standard physics) and stored in the array <code>propce(iel,iproc(icp))</code> (see p.77 for specific conditions of use)</p> <p>useful if there is <math>1 \leq N \leq nscal</math> so that <code>iscsth(n)=1</code> (there is a scalar “temperature”) or with the compressible module for non perfect gases</p>

<sup>40</sup>none of the scalars from the specific physics is a temperature

<sup>41</sup>when using the Graphical Interface, `cp0` is also used to calculate the diffusivity of the thermal scalars, based on their conductivity; it is therefore needed, unless the diffusivity is also specified in `usphyv`

EDF R&D	<i>Code_Saturne</i> version 2.1.3 practical user's guide	<i>Code_Saturne</i> documentation Page 176/205
<b>visls0</b>	<p>ra      real number <math>&gt; 0</math>      <math>[-\text{grand}*10]</math>      C      L1</p> <p><b>visls0(j)</b>: reference molecular diffusivity related to the scalar J (<math>\text{kg.m}^{-1}.\text{s}^{-1}</math>) negative value: not initialised useful if <math>1 \leq J \leq \text{nscal}</math>, unless the user specifies the molecular diffusivity in the appropriate user subroutine (<b>usphyv</b> for the standard physics) (<b>ivisls(iscal) &gt; 0</b>) <i>Warning: visls0 corresponds to the diffusivity. For the temperature, it is therefore defined as <math>\lambda/C_p</math> where <math>\lambda</math> and <math>C_p</math> are the conductivity and specific heat. When using the Graphical Interface, <math>\lambda</math> and <math>C_p</math> are specified separately, and visls0 is calculated automatically</i> <i>With the compressible module, visls0 (given in <b>uscfxi2</b>) is directly the thermal conductivity <math>\text{W.m}^{-1}.\text{K}^{-1}</math></i> <i>With gas or coal combustion, the molecular diffusivity of the enthalpy (<math>\text{kg.m}^{-1}.\text{s}^{-1}</math>) must be specified by the user in the variable <b>diftl0</b> (<b>usebu1</b>, <b>usd3p1</b>, <b>uslwc1</b>, <b>uscpi1</b>, <b>uscpl1</b>)</i> <i>With the electric module, for the Joule effect, the diffusivity is specified by the user in <b>uselph</b> (even if it is constant). For the electric arc, it is calculated from the thermochemical data file</i></p>	
<b>ivisls</b>	<p>ia      positive or zero integer      <math>[0]</math>      O      L1</p> <p>indicates if the viscosity related to the scalar <b>iscal</b> is variable (<b>ivisls(iscal)=1</b>) or not (0). The user must specify <b>ivisls</b> only for the user scalars (<b>iscal <math>\leq</math> nscaus</b>). When <b>ivisls(iscal)=1</b> is specified, the code automatically modifies this value to make <b>ivisls(iscal)</b> designate the effective index-number of the property “diffusivity of the scalar <b>iscal</b>”. For each cell <b>iel</b>, the value is then specified by the user in the appropriate subroutine (<b>usphyv</b> for the standard physics) and stored in the array <b>propce(iel,iproc(ivisls))</b> (see p.77 for specific conditions of use) useful if <math>1 \leq n \leq \text{nscal}</math></p>	
<b>diftl0</b>	<p>r      real number <math>&gt; 0</math>      <math>[-\text{grand}]</math>      C      L1</p> <p>molecular diffusivity for the enthalpy (<math>\text{kg.m}^{-1}.\text{s}^{-1}</math>) for gas or coal combustion (the code then automatically sets <b>visls0</b> to <b>diftl0</b> for the scalar representing the enthalpy) always useflu for gas or coal combustion</p>	
<b>scamin</b>	<p>ra      real number      <math>[\text{grand}]</math>      O      L1</p> <p><b>scamin(iscal)</b> is the lower limit value for the scalar <b>iscal</b>. At each time step, in every cell where the calculated value for <b>rtp(iel,isca(iscal))</b> is lower than <b>scamin(iscal)</b>, <b>rtp(iel,isca(iscal))</b> will be reset to <b>scamin(iscal)</b> there is no limitation if <b>scamin(iscal) &gt; scamax(iscal)</b> <b>scamin</b> shall not be specified for non-user scalars (specific physics) or for scalar variances useful if and only if <math>1 \leq \text{iscal} \leq \text{nscaus}</math></p>	
<b>scamax</b>	<p>ra      real number      <math>[-\text{grand}]</math>      O      L1</p> <p><b>scamax(iscal)</b> is the higher limit value for the scalar <b>iscal</b>. At each time step, in every cell where the calculated value for <b>rtp(iel,isca(iscal))</b> is higher than <b>scamax(iscal)</b>, <b>rtp(iel,isca(iscal))</b> will be reset to <b>scamax(iscal)</b> there is no limitation if <b>scamin(iscal) &gt; scamax(iscal)</b> <b>scamax</b> shall not be specified for non-user scalars (specific physics) or for scalar variances useful if and only if <math>1 \leq \text{iscal} \leq \text{nscaus}</math></p>	
<b>sigmas</b>	<p>ra      real number <math>&gt; 0</math>      <math>[1]</math>      O      L2</p>	

**sigmas(iscal)**: turbulent Prandtl (or Schmidt) number for the scalar **iscal**  
useful if and only if  $1 \leq \text{iscal} \leq \text{nscal}$

**rvarfl**      ra      real number  $> 0$       [0.8]      O      L2  
when **iscavr(iscal)** $>0$ , **rvarfl(iscal)** is the coefficient  $R_f$  in the dissipation term  $-\frac{\rho}{R_f} \frac{\varepsilon}{k}$  of the equation concerning the scalar **iscal**, which represents the root mean square of the fluctuations of the scalar **iscavr(iscal)**  
useful if and only if there is  $1 \leq \text{iscal} \leq \text{nscal}$  such as **iscavr(iscal)** $>0$

### 9.3.4 Modeling parameters

**xlomlg**      ra      real number  $> 0$       [-grand\*10]      O      L1  
**xlomlg** is the mixing length  
useful if and only if **iturb**= 10 (mixing length)

**almax**      ra      -grand, real number  $> 0$       [-grand\*10]      O      L2  
**almax** is a characteristic macroscopic length of the domain, used for the initialisation of the turbulence and the potential clipping (with **iclkep**=1)  
negative value: not initialised (the code then uses the cubic root of the domain volume)  
useful if and only if **turb**= 20, 21, 30, 31, 50 or 60 (RANS models)

**uref**      ra      real number  $> 0$       [-grand\*10]      C      L1  
**uref** is the characteristic flow velocity, used for the initialisation of the turbulence  
negative value: not initialised  
useful if and only if **iturb**= 20, 21, 30, 31, 50 or 60 (RANS model) and the turbulence is not initialised somewhere else (restart file or subroutine **usinv**)

#### BASIC CONSTANTS OF THE $k - \varepsilon$ AND THE OTHER RANS MODELS

**xkappa**      r      real number  $> 0$       [0.42]      O      L3  
Kármán constant  
useful if and only if **iturb** $\geq 10$  (mixing length,  $k - \varepsilon$ ,  $R_{ij} - \varepsilon$ , LES, v2f or  $k - \omega$ )

**cstlog**      r      real number  $> 0$       [5.2]      O      L3  
constant of the logarithmic wall function  
useful if and only if **iturb** $\geq 10$  (mixing length,  $k - \varepsilon$ ,  $R_{ij} - \varepsilon$ , LES, v2f or  $k - \omega$ )

**cmu**      r      real number  $> 0$       [0.09]      O      L3  
constant  $C_\mu$  for all the RANS turbulence models except for the v2f model (see **cv2fmu** for the value of  $C_\mu$  in case of v2f modeling)  
useful if and only if **iturb**= 20, 21, 30, 31 or 60 ( $k - \varepsilon$ ,  $R_{ij} - \varepsilon$  or  $k - \omega$ )

**ce1**      r      real number  $> 0$       [1.44]      O      L3  
constant  $C_{\varepsilon 1}$  for all the RANS turbulence models except for the v2f and the  $k - \omega$  models  
useful if and only if **iturb**= 20, 21, 30 or 31 ( $k - \varepsilon$  or  $R_{ij} - \varepsilon$ )

**ce2**      r      real number  $> 0$       [1.92]      O      L3  
constant  $C_{\varepsilon 2}$  for the  $k - \varepsilon$  and  $R_{ij} - \varepsilon$  LRR models  
useful if and only if **iturb**= 20, 21 or 30 ( $k - \varepsilon$  or  $R_{ij} - \varepsilon$  LRR)

<b>ce4</b>	r	real number $> 0$	[1.2]	O	L3
constant $C_{\varepsilon 4}$ for the interfacial term (Lagrangian module) in case of two-way coupling useful in case of Lagrangian modeling, in $k - \varepsilon$ and $R_{ij} - \varepsilon$ with two-way coupling					
<b>sigmak</b>	r	real number $> 0$	[1.0]	O	L3
Prandtl number for $k$ with $k - \varepsilon$ and v2f models useful if and only if <b>iturb</b> =20, 21 or 50 ( $k - \varepsilon$ or v2f)					
<b>sigmae</b>	r	real number $> 0$	[1.3]	O	L3
Prandtl number for $\varepsilon$ useful if and only if <b>iturb</b> = 20, 21, 30, 31 or 50 ( $k - \varepsilon$ , $R_{ij} - \varepsilon$ or v2f)					

#### CONSTANTS SPECIFIC TO THE $R_{ij} - \varepsilon$ LRR MODEL (**iturb**=30)

<b>crij1</b>	r	real number $> 0$	[1.8]	O	L3
constant $C_1$ for the $R_{ij} - \varepsilon$ LRR model useful if and only if <b>iturb</b> =30 ( $R_{ij} - \varepsilon$ LRR)					
<b>crij2</b>	r	real number $> 0$	[0.6]	O	L3
constant $C_2$ for the $R_{ij} - \varepsilon$ LRR model useful if and only if <b>iturb</b> =30 ( $R_{ij} - \varepsilon$ LRR)					
<b>crij3</b>	r	real number $> 0$	[0.55]	O	L3
constant $C_3$ for the $R_{ij} - \varepsilon$ LRR model useful if and only if <b>iturb</b> =30 ( $R_{ij} - \varepsilon$ LRR)					
<b>crijep</b>	r	real number $> 0$	[0.18]	O	L3
constant $C_\varepsilon$ for the $R_{ij} - \varepsilon$ LRR model useful if and only if <b>iturb</b> =30 ( $R_{ij} - \varepsilon$ LRR)					
<b>csrij</b>	r	real number $> 0$	[0.22]	O	L3
constant $C_s$ for the $R_{ij} - \varepsilon$ LRR model useful if and only if <b>iturb</b> =30 ( $R_{ij} - \varepsilon$ LRR)					
<b>crijp1</b>	r	real number $> 0$	[0.5]	O	L3
constant $C'_1$ for the $R_{ij} - \varepsilon$ LRR model, corresponding to the wall echo terms useful if and only if <b>iturb</b> =30 and <b>irijec</b> =1 ( $R_{ij} - \varepsilon$ LRR)					
<b>crijp2</b>	r	real number $> 0$	[0.3]	O	L3
constant $C'_2$ for the $R_{ij} - \varepsilon$ LRR model, corresponding to the wall echo terms useful if and only if <b>iturb</b> =30 and <b>irijec</b> =1 ( $R_{ij} - \varepsilon$ LRR)					

#### CONSTANTS SPECIFIC TO THE $R_{ij} - \varepsilon$ SSG MODEL

<b>cssgs1</b>	r	real number $> 0$	[1.7]	O	L3
constant $C_{s1}$ for the $R_{ij} - \varepsilon$ SSG model useful if and only if <b>iturb</b> =31 ( $R_{ij} - \varepsilon$ SSG)					

<b>cssgs2</b>	r      real number $> 0$ constant $C_{s2}$ for the $R_{ij} - \varepsilon$ SSG model useful if and only if <b>iturb</b> =31 ( $R_{ij} - \varepsilon$ SSG)	[-1.05]	O	L3
<b>cssgr1</b>	r      real number $> 0$ constant $C_{r1}$ for the $R_{ij} - \varepsilon$ SSG model useful if and only if <b>iturb</b> =31 ( $R_{ij} - \varepsilon$ SSG)	[0.9]	O	L3
<b>cssgr2</b>	r      real number $> 0$ constant $C_{r2}$ for the $R_{ij} - \varepsilon$ SSG model useful if and only if <b>iturb</b> =31 ( $R_{ij} - \varepsilon$ SSG)	[0.8]	O	L3
<b>cssgr3</b>	r      real number $> 0$ constant $C_{r3}$ for the $R_{ij} - \varepsilon$ SSG model useful if and only if <b>iturb</b> =31 ( $R_{ij} - \varepsilon$ SSG)	[0.65]	O	L3
<b>cssgr4</b>	r      real number $> 0$ constant $C_{r4}$ for the $R_{ij} - \varepsilon$ SSG model useful if and only if <b>iturb</b> =31 ( $R_{ij} - \varepsilon$ SSG)	[0.625]	O	L3
<b>cssgr5</b>	r      real number $> 0$ constant $C_{r1}$ for the $R_{ij} - \varepsilon$ SSG model useful if and only if <b>iturb</b> =31 ( $R_{ij} - \varepsilon$ SSG)	[0.2]	O	L3
<b>cssge2</b>	r      real number $> 0$ constant $C_{\varepsilon 2}$ for the $R_{ij} - \varepsilon$ SSG model useful if and only if <b>iturb</b> =31 ( $R_{ij} - \varepsilon$ SSG)	[1.83]	O	L3

#### CONSTANTS SPECIFIC TO THE v2f $\varphi$ -MODEL

<b>cv2fa1</b>	r      real number $> 0$ constant $a_1$ for the v2f $\varphi$ -model useful if and only if <b>iturb</b> =50 (v2f $\varphi$ -model)	[0.05]	O	L3
<b>cv2fe2</b>	r      real number $> 0$ constant $C_{\varepsilon 2}$ for the v2f $\varphi$ -model useful if and only if <b>iturb</b> =50 (v2f $\varphi$ -model)	[1.85]	O	L3
<b>cv2fmu</b>	r      real number $> 0$ constant $C_{\mu}$ for the v2f $\varphi$ -model useful if and only if <b>iturb</b> =50 (v2f $\varphi$ -model)	[0.22]	O	L3
<b>cv2fc1</b>	r      real number $> 0$ constant $C_1$ for the v2f $\varphi$ -model useful if and only if <b>iturb</b> =50 (v2f $\varphi$ -model)	[1.4]	O	L3
<b>cv2fc2</b>	r      real number $> 0$ constant $C_2$ for the v2f $\varphi$ -model useful if and only if <b>iturb</b> =50 (v2f $\varphi$ -model)	[0.3]	O	L3

cv2fct	r	real number $> 0$ constant $C_T$ for the v2f $\varphi$ -model useful if and only if <code>iturb=50</code> (v2f $\varphi$ -model)	[6]	O	L3
cv2fc1	r	real number $> 0$ constant $C_L$ for the v2f $\varphi$ -model useful if and only if <code>iturb=50</code> (v2f $\varphi$ -model)	[0.25]	O	L3
cv2fet	r	real number $> 0$ constant $C_\eta$ for the v2f $\varphi$ -model useful if and only if <code>iturb=50</code> (v2f $\varphi$ -model)	[110]	O	L3

#### CONSTANTS SPECIFIC TO THE $k - \omega$ SST MODEL

ckwsk1	r	real number $> 0$ constant $\sigma_{k1}$ for the $k - \omega$ SST model useful if and only if <code>iturb=60</code> ( $k - \omega$ SST)	[1/0.85]	O	L3
ckwsk2	r	real number $> 0$ constant $\sigma_{k2}$ for the $k - \omega$ SST model useful if and only if <code>iturb=60</code> ( $k - \omega$ SST)	[2]	O	L3
cksw1	r	real number $> 0$ constant $\sigma_{\omega1}$ for the $k - \omega$ SST model useful if and only if <code>iturb=60</code> ( $k - \omega$ SST)	[2]	O	L3
cksw2	r	real number $> 0$ constant $\sigma_{\omega2}$ for the $k - \omega$ SST model useful if and only if <code>iturb=60</code> ( $k - \omega$ SST)	[1/0.856]	O	L3
ckwbt1	r	real number $> 0$ constant $\beta_1$ for the $k - \omega$ SST model useful if and only if <code>iturb=60</code> ( $k - \omega$ SST)	[0.075]	O	L3
ckwbt2	r	real number $> 0$ constant $\beta_2$ for the $k - \omega$ SST model useful if and only if <code>iturb=60</code> ( $k - \omega$ SST)	[0.0828]	O	L3
ckwgm1	r	real number $> 0$ constant $\gamma_1$ for the $k - \omega$ SST model useful if and only if <code>iturb=60</code> ( $k - \omega$ SST) <i>Warning: <math>\gamma_1</math> is calculated before the call to <code>usini1</code>. Hence, if <math>\beta_1</math>, <math>C_\mu</math>, <math>\kappa</math> or <math>\sigma_{\omega1}</math> is modified in <code>usini1</code>, CKWGM1 must also be modified in accordance</i>	$[\frac{\beta_1}{C_\mu} - \frac{\kappa^2}{\sqrt{C_\mu \sigma_{\omega1}}}]$	O	L3
ckwgm2	r	real number $> 0$ constant $\gamma_2$ for the $k - \omega$ SST model useful if and only if <code>iturb=60</code> ( $k - \omega$ SST) <i>Warning: <math>\gamma_2</math> is calculated before the call to <code>usini1</code>. Hence, if <math>\beta_2</math>, <math>C_\mu</math>, <math>\kappa</math> or <math>\sigma_{\omega2}</math> is modified in <code>usini1</code>, ckwgm2 must also be modified in accordance</i>	$[\frac{\beta_2}{C_\mu} - \frac{\kappa^2}{\sqrt{C_\mu \sigma_{\omega2}}}]$	O	L3

<b>ckwa1</b>	r	real number $> 0$ constant $a_1$ for the $k - \omega$ SST model useful if and only if <b>iturb</b> =60 ( $k - \omega$ SST)	[0.31]	O	L3
<b>ckwc1</b>	r	real number $> 0$ constant $c_1$ for the $k - \omega$ SST model useful if and only if <b>iturb</b> =60 ( $k - \omega$ SST)	[10]	O	L3

## 9.4 ALE

<b>iale</b>	i	0 or 1 activates (=1) or not (=0), activate the ALE module	[C]	O	L1
<b>nalinf</b>	i	0 or positive integer The number of sub-iterations of initialization of the fluid	[0]	C	L2
<b>nbstr</b>	i	0 or positive integer number of structures	[0]	C	L1
<b>alpnmk</b>	r	real <i>alpha</i> newmark's method	[0]	C	L3
<b>betnmk</b>	r	real <i>beta</i> newmark's method	[-grand]	C	L3
<b>gamnmk</b>	r	real <i>gamma</i> newmark's method	[-grand]	C	L3
<b>nalimx</b>	i	positive integer maximum number of imlicitation iterations of of the structure displacement	[15]	C	L2
<b>epalim</b>	r	positive real Relative precision of implicitation of the structure displacement	[1.10 <sup>-5</sup> ]	C	L2

## 9.5 Thermal radiative transfers: global settings

All the following key words may be modified in the user subroutines **usray\*** (or, for some of them, by through the thermochemical data files). It is however not recommended to modify those which do not belong to level L1.

<b>iirayo</b>	ia	0, 1, 2 <b>iirayo</b> activates ( $> 0$ ) or deactivates ( $=0$ ) the radiation module The different values correspond to the following modelings: = 1 discrete ordinates (standard option for radiation in semi-transparent media) = 2 "P-1" model <i>Warning: the P-1 model allows faster computations, but it may only be applied to media with uniform large optical thickness, such as some cases of pulverised coal combustion</i>	[0]	O	L1
---------------	----	---	-----	---	----

<b>imodak</b>	i	0 or 1	[0]	O	L3	when gas or coal combustion is activated, <b>imodak</b> indicates whether the absorption coefficient shall be calculated “automatically” (=1) or read from the data file (=0) useful if the radiation module is activated
<b>isuir</b>	i	0 or 1	[suite]	C	L1	indicates whether the radiation variables should be initialised (=0) or read from a restart file (=1) useful if and only if the radiation module is activated (in this case, a restart file <i>rayamo</i> must be available)
<b>nfreq</b>	i	strictly positive integer	[1]	O	L1	period of the radiation module the radiation module is called every <b>nfreq</b> time steps (more precisely, every time <b>ntcabs</b> is a multiple of <b>nfreq</b> ). Also, in order to have proper initialisation of the variables, whatever the value of <b>nfreq</b> , the radiation module is called at the first time step of a calculation (restart or not) useful if and only if the radiation module is activated
<b>ndirec</b>	i	32 or 128	[32]	O	L1	number of directions for the angular discretisation of the radiation propagation with the DOM model ( <b>iirayo</b> =1) no other possible value, because of the way the directions are calculated the calculation with 32 directions may break the symmetry of physically axisymmetric cases (but the cost in CPU time is much lower than with 128 directions) useful if and only if the radiation module is activated with the DOM method
<b>xnp1mx</b>	r	real number	[10]	O	L3	with the P-1 model ( <b>iirayo</b> =2), <b>xnp1mx</b> is the percentage of cells of the calculation domain for which it is acceptable that the optical thickness is lower than unity <sup>42</sup> , although it is not to be desired useful if and only if the radiation module is activated with the P-1 method
<b>idiver</b>	i	0, 1 or 2	[2]	C	L1	indicates the method used to calculate the radiative source term: = 0: semi-analytic calculation (compulsory with transparent media) = 1: conservative calculation = 2: semi-analytic calculation corrected in order to be globally conservative useful if and only if the radiation module is activated <i>Note: if the medium is transparent, the choice has no effect on the calculation</i>
<b>iimpar</b>	i	0, 1 or 2	[1]	O	L1	choice of the display level in the listing concerning the calculation of the wall temperatures: = 0: no display = 1: standard = 2: complete useful if and only if the radiation module is activated
<b>iimlum</b>	i	0, 1 or 2	[1]	O	L1	choice of the display level in the listing concerning the solution of the radiative transfer

<sup>42</sup>more precisely, where  $KL$  is lower than 1, where  $K$  is the absorption coefficient of the medium and  $L$  is a characteristic length of the domain



equation:

- = 0: no display
- = 1: standard
- = 2: complete

useful if and only if the radiation module is activated

<b>nbrvaf</b>	ca	string of less than 80 characters [name]	O	L1
name associated for the post-processing to each of the following variables, defined at the boundary faces ( <i>see</i> [6] for more details concerning their definitions):				
nbrvaf(itparp): wall temperature at the boundary faces ( $K$ )				
nbrvaf(iqincp): radiative incident flux density ( $W/m^2$ )				
nbrvaf(ixlamp): thermal conductivity of the boundary faces ( $W/m/K$ )				
nbrvaf(iepap): wall thickness ( $m$ )				
nbrvaf(iepsp): wall emissivity				
nbrvaf(ifnetp): net radiative flux density ( $W/m^2$ )				
nbrvaf(ifconp): convective flux density ( $W/m^2$ )				
nbrvaf(ihconp): convective exchange coefficient ( $W/m^2/K$ )				
The default values are:				
nbrvaf(itparp) = Wall_temp				
nbrvaf(iqincp) = Incident_flux				
nbrvaf(ixlamp) = Th_conductivity				
nbrvaf(iepap) = Thickness				
nbrvaf(iepsp) = Emissivity				
nbrvaf(ifnetp) = Net_flux				
nbrvaf(ifconp) = Convective_flux				
nbrvaf(ihconp) = Convective_exch_coef				
useful if and only if the radiation module is activated				
<b>irayvf</b>	ia	-1 or 1 [-1]	O	L1
activates (=1) or deactivates (=-1) the post-processing for each of the following variables defined at the boundary faces:				
irayvf(itparp): wall temperature at the boundary faces ( $K$ )				
irayvf(iqincp): radiative incident flux density ( $W/m^2$ )				
irayvf(ixlamp): thermal conductivity of the boundary faces ( $W/m/K$ )				
irayvf(iepap): wall thickness ( $m$ )				
irayvf(iepsp): wall emissivity				
irayvf(ifnetp): net radiative flux density ( $W/m^2$ )				
irayvf(ifconp): convective flux density ( $W/m^2$ )				
irayvf(ihconp): convective exchange coefficient ( $W/m^2/K$ )				
useful if and only if the radiation module is activated				
<b>tmin</b>	r	real number positif [0]	O	L3
minimum allowed value for the wall temperatures in Kelvin				
useful if and only if the radiation module is activated				
<b>tmax</b>	r	real number positif [grand + 273.15]	O	L3
maximum allowed value for the wall temperatures in Kelvin				
useful if and only if the radiation module is activated				

## 9.6 Electric module (Joule effect and electric arc): specificities

The electric module is composed of a Joule effect module (`ippmod(ieljou)`) and an electric arc module (`ippmod(ielarc)`).

The Joule effect module is designed to take into account the Joule effect (for instance in glass furnaces) with real or complex potential in the enthalpy equation. The Laplace forces are not taken into account in the impulse momentum equation. Specific boundary conditions can be applied to account for the coupled effect of transformers (offset) in glass furnaces.

The electric arc module is designed to take into account the Joule effect (only with real potential) in the enthalpy equation. The Laplace forces are taken into account in the impulse momentum equation.

The key words used in the global settings are quite few. They are found in the subroutine **usel11** (see the description of this user subroutine §8.8.4).

<b>ielcor</b>	i	0, 1	[0]	O	L1	when <b>ielcor</b> =1, the boundary conditions for the potential will be tuned at each time step in order to reach a user-specified target dissipated power <b>puisim</b> (Joule effect) or a user-specified target current intensity <b>couimp</b> (electric arc) the boundary condition tuning is controlled by the subroutine <b>uselrc</b> always useful
<b>couimp</b>	r	real number $\geq 0$	[0]	O	L1	with the electric arc module, <b>couimp</b> is the target current intensity (A) for the calculations with boundary condition tuning for the potential the target intensity will be reached if the boundary conditions are expressed using the variable <b>dpot</b> or if the initial boundary conditions are multiplied by the variable <b>coejou</b> useful with the electric arc module if <b>ielcor</b> =1
<b>puisim</b>	r	real number $\geq 0$	[0]	O	L1	with the Joule effect module, <b>puisim</b> is the target dissipated power (W) for the calculations with boundary condition tuning for the potential the target power will be reached if the boundary conditions are expressed using the variable <b>dpot</b> or if the initial boundary conditions are multiplied by the variable <b>coejou</b> useful with the Joule effect module if <b>ielcor</b> =1
<b>dpot</b>	r	real number $\geq 0$	[0]	O	L1	<b>dpot</b> is the potential difference (V) which generates the current (and the Joule effect) for the calculations with boundary conditions tuning for the potential. This value is initialised set by the user ( <b>usel11</b> ). It is then automatically tuned depending on the value of dissipated power (Joule effect module) or the intensity of current (electric arc module). In order for the correct power or intensity to be reached, the boundary conditions for the potential must be expressed with <b>dpot</b> ( <b>uselc1</b> ). The tuning can be controlled in <b>uselrc</b> useful if <b>ielcor</b> =1
<b>coejou</b>	r	real number $\geq 0$	[1]	O	L2	only with the Joule effect, <b>coejou</b> can be used if the user does not wish to use <b>dpot</b> ; <b>coejou</b> is the coefficient to be applied to the initial potential difference to reach the target dissipated power. Its value is automatically initialised to 1 and is updated during the calculation. In order for the correct power to be reached, the boundary conditions for the potential must be expressed with <b>coejou</b> ( <b>uselc1</b> ). The tuning can be controlled in <b>uselrc</b> Useful if <b>ielcor</b> =1

## 9.7 Compressible module: specificities

The key words used in the global settings are quite few. They are found in the subroutines **uscfx1** and **uscfx2** (see the description of these user subroutines, §8.7.1).

**icfgrp**      ia      0 or 1      [1]      C      L1  
**icfgrp** indicates if the boundary conditions should take into account (=1) or not (=0) the hydrostatic balance.  
 always useful.  
 In the cases where gravity is predominant, taking into account the hydrostatic pressure allows to get rid of the disturbances which may appear near the horizontal walls when the flow is little convective.  
 Otherwise, when **icfgrp**=0, the pressure condition is calculated from the solution of the unidimensional Euler equations for a perfect gas near a wall, for the variables “normal velocity”, “density” and “pressure”:

Case of an expansion ( $M \leq 0$ ):

$$\begin{cases} P_p = 0 & \text{if } 1 + \frac{\gamma-1}{2}M < 0 \\ P_p = P_i \left(1 + \frac{\gamma-1}{2}M\right)^{\frac{2\gamma}{\gamma-1}} & \text{otherwise} \end{cases}$$

Case of a shock ( $M > 0$ ):

$$P_p = P_i \left(1 + \frac{\gamma(\gamma+1)}{4}M^2 + \gamma M \sqrt{1 + \frac{(\gamma+1)^2}{16}M^2}\right)$$

with  $M = \frac{u_i \cdot n}{c_i}$ , internal Mach number calculated with the variables taken in the cell adjacent to the wall.

**iviscv**      ia      0 or 1      [0]      C      L1  
**iviscv**=0 indicates that the volume viscosity is constant and equal to the reference volume viscosity **viscv0**.  
**iviscv**=1 indicates that the volume viscosity is variable: its variation law must be specified in the user subroutine **uscfpv**.  
 always useful  
 The volume viscosity  $\kappa$  is defined by the formula expressing the stress:

$$\underline{\underline{\sigma}} = -P \underline{\underline{Id}} + \mu(\text{grad } \underline{u} + {}^t\text{grad } \underline{u}) + \left(\kappa - \frac{2}{3}\mu\right) \text{div}(\underline{u}) \underline{\underline{Id}} \quad (6)$$

**viscv0**      ra      real number  $\geq 0$       [0]      O      L1  
**viscv0** is the reference volume viscosity (noted  $\kappa$  in the equation expressing  $\underline{\underline{\sigma}}$  in the paragraph dedicated to **iviscv**)  
 always useful, it is the used value, unless the user specifies the volume viscosity in the user subroutine **uscfpv**

**igrdpp**      i      0 or 1      [0]      O      L3  
 indicates whether the pressure should be updated (=1) or not (=0) after the solution of the acoustic equation  
 always useful

## 9.8 Lagrangian multiphase flows

Most of these key words may be modified in the user subroutines `uslag1`, `uslag2`, `uslabo`, `uslaen`, `uslast` and `uslaed`. It is however strongly recommended not to modify those belonging to the level L3.

First of all, it should be noted that the Lagrangian module is compliant with all the RANS turbulence models and with laminar flows. However, the particule turbulent diffusion is not specially adapted to the second order  $R_{ij} - \varepsilon$  models. The same isotropic model is used as in the  $k - \varepsilon$  models, with  $k$  calculated from the trace of  $R_{ij}$ . Also, two-way coupling is not compatible with the  $k - \omega$  SST model.

### 9.8.1 Global settings

<b>iilag</b>	I	0, 1, 2, 3	[0]	C	L1	activates (>0) or deactivates (=0) the Lagrangian module the different values correspond to the following modelings: = 1 Lagrangian two-phase flow in one-way coupling (no influence of the particles on the continuous phase) = 2 Lagrangian two-phase flow with two-way coupling (influence of the particles on the dynamics of the continuous phase). It must be noted that the two-way coupling is taken into account only for the first eulerian phase. Dynamics, temperature and mass may be coupled independently = 3 Lagrangian two-phase flow on frozen continuous phase. This option can only be used in case of a calculation restart. All the eulerian fields are frozen (including the scalar fields). This option automatically implies <code>iccvfg = 1</code> always useful
<b>isuila</b>	i	0, 1	[0]	C	L1	activation (=1) or not (=0) of a Lagrangian calculation restart. The calculation restart file read when this option is activated ( <code>ficaml</code> ) only contains the data related to the particles (see also <code>isuist</code> ) the global calculation must also be a restart calculation always useful
<b>isuist</b>	i	0, 1	[0]	C	L1	during a Lagrangian calculation restart, indicates whether the particle statistics (volume and boundary) and two-way coupling terms are to be read from a restart file (=1) or reinitialised (=0). The file to be read is <code>ficmls</code> useful if <code>isuila = 1</code>
<b>nbpmax</b>	i	positive or null integer	[1000]	C	L1	maximum number of particles allowed simultaneously in the calculation domain. It must be reminded that the required memory evolves accordingly
<b>nbpart</b>	i	positive or null integer	[0]	O	L3	number of particles treated during one Lagrangian time step <b>nbpart</b> must always be lower than <b>nbpmax</b> always useful, but initialised and updated without intervention of the user
<b>nvls</b>	i	integer between 0 and 10	[0]	O	L2	number of additional variables related to the particles the additional variables can be accessed in the arrays <code>ettp</code> and <code>ettpa</code> by means of

the pointer `jvls`: `ettp(nbpt,jvls(ii))` and `ettpa(nbpt,jvls(ii))` (`nbpt` is the index-number of the treated particle, and `ii` an integer between 1 and `nvls`)

<b>isttio</b>	i	0, 1	[0]	C	L1
<p>indicates the steady (=1) or unsteady (=0) state of the continuous phase flow in particular, <b>isttio</b> = 1 is needed in order to:</p> <ul style="list-style-type: none"> <li>calculate stationary statistics in the volume or at the boundaries (starting respectively from the Lagrangian iterations <b>nstist</b> and <b>nstbor</b>)</li> <li>calculate time-averaged two-way coupling source terms (from the Lagrangian iteration <b>nstits</b>)</li> </ul> <p>useful if <b>iilagr</b>=1 or <b>iilagr</b>=2 (if <b>iilagr</b>=3, then <b>isttio</b>=1 automatically)</p>					
<b>injcon</b>	i	0, 1	[0]	O	L1
<p>activates (=1) or not (=0) the continuous injection of particles this option allows to inject particles continuously during the duration of the Lagrangian time step <b>dtp</b> rather than only once at the beginning of the Lagrangian iteration. It helps avoiding the fractioning of the particle cloud close to the injection areas</p>					
<b>iroule</b>	i	0, 1	[0]	O	L1
<p>activates (=1) or not (=0) of the particle cloning/fusion technique (option also called “Russian roulette”) when <b>iroule</b> = 1, the importance function must be specified <i>via</i> the array <b>croule</b> in the user subroutine <b>uslaru</b></p>					
<b>isuivi</b>	i	0, 1	[0 or 1]	O	L2
<p>specifies if a particle should be followed (=1) or will disappear from the domain (=0) after an interaction with a boundary:</p> <ul style="list-style-type: none"> <li>= 0: the particle must not be followed in the calculation domain after an interaction between its trajectory and a boundary face, for instance entry (<b>ientrl</b>), outlet (<b>isortl</b>), definitive deposition on a wall (<b>idepo1</b>, <b>idepo2</b>)</li> <li>= 1: the particle must still be followed in the calculation domain after an interaction between its trajectory and a boundary face, for instance rebound (<b>irebol</b>), deposition with potential resuspension (<b>idepo3</b>)</li> </ul> <p>the value of <b>isuivi</b> (<b>isuivi</b> = 0 or <b>isuivi</b> = 1) for a type of interaction can be defined as a function of the particle behaviour or properties. It is for example the default case for the fouling interaction type (<b>iencrl</b>) always useful</p>					
<b>ttclag</b>	r	positive real number	[0]	O	L3
<p>physical time of the Lagrangian simulation always useful</p>					
<b>iplas</b>	i	integer > 0	[1]	O	L3
<p>absolute iteration number (including the restarts) in the Lagrangian module (<i>i.e.</i> Lagrangian time step number) always useful</p>					

## 9.8.2 Specific physics models associated with the particles

<b>iphyla</b>	i	0, 1, 2	[0]	C	L1
<p>activates (&gt;0) or deactivates (=0) the physical models associated to the particles:</p>					

= 1: allows to associate with the particles evolution equations on their temperature (in degrees Celsius), their diameter and their mass

= 2: the particles are pulverised coal particles. Evolution equations on temperature (in degree Celsius), mass of reactive coal, mass of char and diameter of the shrinking core are associated with the particles. This option is available only if the continuous phase represents a pulverised coal flame  
always useful

<b>idpvar</b>	i	0, 1	[0]	O	L1	activation (=1) or not (=0) of an evolution equation on the particle diameter useful if <b>iphyla</b> = 1
<b>itpvar</b>	i	0, 1	[0]	O	L1	activation (=1) or not (=0) of an evolution equation on the particle temperature (in degrees Celsius) useful if <b>iphyla</b> = 1 and if there is a thermal scalar associated with the continuous phase
<b>impvar</b>	i	0, 1	[0]	O	L1	activation (=1) or not (=0) of an evolution equation on the particle mass useful if <b>si iphyla</b> = 1
<b>tpart</b>	r	real number > <b>tkelvn</b>	[700]	O	L1	initialisation temperature (in degree Celsius) for the particles already present in the calculation domain when an evolution equation on the particle temperature is activated during a calculation ( <b>iphyla</b> = 1 and <b>itpvar</b> = 1) useful if <b>isuila</b> = 1 and <b>itpvar</b> = 0 in the previous calculation
<b>cppart</b>	r	positive real number	[5200]	O	L1	initialisation value for the specific heat ( $J.kg^{-1}.K^{-1}$ ) of the particles already present in the calculation domain when an evolution equation on the particle temperature is activated during a calculation ( <b>iphyla</b> = 1 and <b>itpvar</b> = 1) useful if <b>isuila</b> = 1 and <b>itpvar</b> = 0 in the previous calculation
<b>iencra</b>	i	0, 1	[0]	O	L1	activates (=1) or not (=0) the option of coal particle fouling. It then is necessary to specify the domain boundaries on which fouling may take place. useful if <b>iphyla</b> = 2
<b>tprenc</b>	r	real number > <b>tkelvn</b>	[600]	O	L1	limit temperature (in degree Celsius) below which the coal particles do not cause any fouling (if the fouling model is activated) useful if <b>iphyla</b> = 2 and <b>iencra</b> = 1
<b>visref</b>	r	positive real number	[10000]	O	L1	ash critical viscosity in $kg.m^{-1}.s^{-1}$ , in the fouling model <sup>43</sup> useful if <b>iphyla</b> = 2 and <b>iencra</b> = 1

<sup>43</sup>J.D. Watt et T. Fereday (*J.Inst.Fuel*, Vol.42-p99)

### 9.8.3 Options for two-way coupling

<b>nstits</b>	i	strictly positive integer	[1]	O	L1	number of absolute Lagrangian iterations (including the restarts) after which a time-average of the two-way coupling source terms is calculated indeed, if the flow is steady ( <b>isttio</b> =1), the average quantities that appear in the two-way coupling source terms can be calculated over different time steps, in order to get a better precision if the number of absolute Lagrangian iterations is strictly inferior to <b>nstits</b> , the code considers that the flow has not yet reached its steady state (transition period) and the averages appearing in the source terms are reinitialised at each time step, as it is the case for unsteady flows ( <b>isttio</b> =0) useful if <b>iilag</b> = 2 and <b>isttio</b> = 1
<b>ltsdyn</b>	i	0, 1	[0]	O	L1	activation (=1) or not (=0) of the two-way coupling on the dynamics of the continuous phase useful if <b>iilag</b> = 2 and <b>iccvfg</b> = 0
<b>ltsmas</b>	i	0, 1	[0]	O	L1	activation (=1) or not (=0) of the two-way coupling on the mass useful if <b>iilag</b> = 2, <b>iphyla</b> = 1 and <b>impvar</b> = 1
<b>ltsthe</b>	i	0, 1	[0]	O	L1	if <b>iphyla</b> = 1 and <b>itpvar</b> = 1, <b>ltsthe</b> activates (=1) or not (=0) the two-way coupling on temperature if <b>iphyla</b> = 2, <b>ltsthe</b> activates (=1) or not (=0) the two-way coupling on the eulerian variables related to pulverised coal combustion useful if <b>iilag</b> = 2

### 9.8.4 Numerical modeling

<b>nordre</b>	i	1, 2	[2]	O	L2	order of integration for the stochastic differential equations = 1 integration using a first-order scheme = 2 integration using a second-order scheme always useful
<b>ilapoi</b>	i	0, 1	[0]	O	L3	activation (=1) or not (=0) of the solution of a Poisson's equation for the correction of the particle instantaneous velocities (in order to obtain a null divergence) this option is not validated and reserved to the development team. Do not change the default value
<b>idistu</b>	i	0, 1	[1]	O	L3	activation (=1) or not (=0) of the particle turbulent dispersion the turbulent dispersion is compatible only with the RANS turbulent models ( $k - \varepsilon$ , $R_{ij} - \varepsilon$ , $v2f$ or $k - \omega$ ) ( <b>iturb</b> =20, 21, 30, 31, 50 or 60) always useful

<b>idiff1</b>	i	0, 1	[0]	O	L3	<b>idiff1=1</b> suppresses the crossing trajectory effect, making turbulent dispersion for the particles identical to the turbulent diffusion of fluid particles useful if <b>idistu=1</b>
<b>modcpl</b>	i	positive integer	[0]	O	L1	activates (>0) or not (=0) the complete turbulent dispersion model when <b>modcpl</b> is strictly positive, its value is interpreted as the absolute Lagrangian time step number (including restarts) after which the complete model is applied since the complete model uses volume statistics, <b>modcpl</b> must either be 0 or be larger than <b>idstnt</b> useful if <b>istala = 1</b>
<b>idirla</b>	i	1, 2, 3	[1]	O	L1	<i>x, y</i> or <i>z</i> direction of the complete model it corresponds to the main directions of the flow useful if <b>modcpl &gt; 0</b>

### 9.8.5 Volume statistics

<b>istala</b>	i	0, 1	[0]	C	L1	activation (=1) or not (=0) of the calculation of the volume statistics related to the dispersed phase if <b>istala = 1</b> , the calculation of the statistics is activated starting from the absolute iteration (including the restarts) <b>idstnt</b> by default, the statistics are not stationary (reset to zero at every Lagrangian iteration). But if <b>isttio=1</b> , since the flow is steady, the statistics will be averaged over the different time steps the statistics represent the significant results on the particle cloud always useful
<b>seuil</b>	r	positive real number	[0]	O	L1	every cell of the calculation domain contains a certain quantity of particles, representing a certain statistical weight (sum of the statistical weights of all the particles present in the cell). <b>seuil</b> is the limit statistical weight value, below which the contribution of the cell in term of statistical weight is not taken into account in the volume statistics (for the complete turbulent dispersion model, in the Poisson's equation used to correct the mean velocities or in the listing and post-processing outputs) useful if <b>istala = 1</b>
<b>idstnt</b>	i	strictly positive integer	[1]	C	L1	absolute Lagrangian iteration number (including the restarts) after which the calculation of the volume statistics is activated useful if <b>istala = 1</b>
<b>nstist</b>	i	integer $\geq$ <b>idstnt</b>	[ <b>idstnt</b> ]	O	L1	absolute Lagrangian iteration number (including the restarts) after which the volume statistics are cumulated over time (they are then said to be stationary) if the absolute Lagrangian iteration number is lower than <b>nstist</b> , or if the flow is unsteady ( <b>isttio=0</b> ), the statistics are reset to zero at every Lagrangian iteration (the volume statistics are then said to be non-stationary) useful if <b>istala=1</b> and <b>isttio=1</b>



<b>nomlag</b>	ca	string of less than 50 characters	[VarLagXXXX]	O	L1	name of the volumetric statistics, displayed in the listing and the post-processing files. The default value is given above, with "XXXX" representing a four digit number (for instance 0001, 0011 ...) useful if <b>istala</b> = 1 <i>Warning: this name is also used to reference information in the restart file (<b>isuist</b> = 1). If the name of a variable is changed between two calculations, it will not be possible to read its value from the restart file</i>
<b>nvlst</b>	i	$0 \leq \text{integer} \leq \text{nussta}=20$	[0]	O	L1	number of additional user volume statistics the additional statistics (or their cumulated value in the stationary case) can be accessed in the array <b>statis</b> by means of the pointer <b>ilvu</b> : <b>statis(iel,ilvu(ii))</b> ( <b>iel</b> is the cell index-number and <b>ii</b> an integer between 1 and <b>nvlst</b> ) useful if <b>istala</b> = 1
<b>npst</b>	i	positive integer	[0]	O	L3	number of iterations during which stationary volume statistics have been cumulated useful if <b>istala</b> =1, <b>isttio</b> =1 and if <b>nstist</b> is inferior or equal to the current Lagrangian iteration <b>npst</b> is initialised and updated automatically by the code, its value is not to be modified by the user
<b>npstt</b>	i	positive integer	[0]	O	L3	number of iterations during which volume statistics have been calculated (the potential iterations during which non-stationary statistics have been calculated are counted in <b>npstt</b> ) useful if <b>istala</b> =1 <b>npstt</b> is initialised and updated automatically by the code, its value is not to be modified by the user
<b>tstat</b>	r	positive real number	[dtp]	O	L3	if the volume statistics are calculated in a stationary way, <b>tstat</b> represents the physical time during which the statistics have been cumulated if the volume statistics are calculated in a non-stationary way, then <b>tstat</b> = <b>dtp</b> (it is the Lagrangian time step, because the statistics are reset to zero at every iteration) useful if <b>istala</b> =1 <b>tstat</b> is initialised and updated automatically by the code, its value is not to be modified by the user

## 9.8.6 Display of trajectories and particle movements

<b>iensi1</b>	i	0, 1	[0]	O	L1	activation (=1) or not (=0) of the post-processing in trajectory mode this option generates files allowing to display the trajectory of some pre-selected particles in the <i>EnSight6</i> format always useful <i>Warning: this option very expensive with regards to CPU time and may generate very large files</i>
<b>iensi2</b>	i	0, 1	[0]	O	L1	activation (=1) or not (=0) of the post-processing in movement mode

This option generates files allowing to display the movement of some pre-selected particles in the *EnSight6* format  
always useful  
*Warning: this option very expensive with regards to CPU time and may generate very large files*

<b>nbvis</b>	i	positive integer	[nliste]	O	L1	number of particles selected for post-processing display in trajectory or movement mode nbvis must be lower than nbpmax and nliste (set to 500 in lagpar.h and not to be modified) useful if iensi1 = 1 or iensi2 = 1
<b>nvisla</b>	i	strictly positive integer	[1]	O	L1	output period for the post-processing in trajectory or movement mode may be useful to diminish the size of the post-processing files useful if iensi1 = 1 or iensi2 = 1
<b>liste</b>	ia	positive integers	[between 1 and 500]	O	L1	contains the index-numbers of the particles selected for the display in trajectory or movement mode useful if iensi1 = 1 or iensi2 = 1
<b>ivisv1</b>	i	0, 1	[0]	O	L1	associates (=1) or not (=0) the variable “velocity of the locally undisturbed fluid flow field” with the display in trajectory or movement mode useful if iensi1 = 1 or iensi2 = 1
<b>ivisv2</b>	i	0, 1	[0]	O	L1	associates (=1) or not (=0) the variable “particle velocity” with the display in trajectory or movement mode useful if iensi1 = 1 or iensi2 = 1
<b>ivistp</b>	i	0, 1	[0]	O	L1	associates (=1) or not (=0) the variable “residence time” with the display in trajectory or movement mode useful if iensi1 = 1 or iensi2 = 1
<b>ivisdm</b>	i	0, 1	[0]	O	L1	associates (=1) or not (=0) the variable “particle diameter” with the display in trajectory or movement mode useful if iensi1 = 1 or iensi2 = 1
<b>iviste</b>	i	0, 1	[0]	O	L1	associates (=1) or not (=0) the variable “particle temperature” with the display in trajectory or movement mode useful if iensi1 = 1 or iensi2 = 1
<b>ivismp</b>	i	0, 1	[0]	O	L1	associates (=1) or not (=0) the variable “particle mass” with the display in trajectory

or movement mode  
useful if `iensi1 = 1` or `iensi2 = 1`

<b>ivishp</b>	i	0, 1	[0]	O	L1	associates (=1) or not (=0) the variable “temperature of the coal particles” with the display in trajectory or movement mode useful if <code>iensi1 = 1</code> or <code>iensi2 = 1</code> , if and only if <code>iphyla = 2</code>
<b>ivisdK</b>	i	0, 1	[0]	O	L1	associates (=1) or not (=0) the variable “shrinking core diameter of the coal particles” with the display in trajectory or movement mode useful if <code>iensi1 = 1</code> or <code>iensi2 = 1</code> , if and only if <code>iphyla = 2</code>
<b>ivisch</b>	i	0, 1	[0]	O	L1	associates (=1) or not (=0) the variable “mass of reactive coal of the coal particles” with the display in trajectory or movement mode useful if <code>iensi1 = 1</code> or <code>iensi2 = 1</code> , if and only if <code>iphyla = 2</code>
<b>ivisck</b>	i	0, 1	[0]	O	L1	associates (=1) or not (=0) the variable “mass of char of the coal particles” with the display in trajectory or movement mode useful if <code>iensi1 = 1</code> or <code>iensi2 = 1</code> , if and only if <code>iphyla = 2</code>

### 9.8.7 Display of the particle/boundary interactions and the statistics at the boundaries

<b>iensi3</b>	i	0, 1	[0]	C	L1	activation (=1) or not (=0) of the recording of the particle/boundary interactions in <b>parbor</b> , and of the calculation of the statistics at the corresponding boundaries, for post-processing ( <i>EnSight6</i> format) By default, the statistics are non-stationary (reset to zero at every Lagrangian iteration). They may be stationary if <code>isttio=1</code> ( <i>i.e.</i> calculation of a cumulated value over time, and then calculation of an average over time or over the number of interactions with the boundary) always useful
<b>nstbor</b>	i	strictly positive integer	[1]	O	L1	number of absolute Lagrangian iterations (including the restarts) after which the statistics at the boundaries are considered stationary and are averaged (over time or over the number of interactions) If the number of absolute Lagrangian iterations is lower than <b>nstbor</b> , or if <code>isttio=0</code> , the statistics are reset to zero at every Lagrangian iteration (non-stationary statistics) useful if <code>iensi3=1</code> and <code>isttio=1</code>
<b>seuilf</b>	r	positive real number	[0]	O	L1	every boundary face of the mesh undergoes a certain number of interactions with particles, expressed in term of statistical weight (sum of the statistical weights of all the particles which have interacted with the boundary face). <b>seuilf</b> is the limit statistical weight value, below which the contribution of the face is not taken into account in the statistics at the boundaries for post-processing useful if <code>iensi3=1</code>

<b>inbrbd</b>	i	0, 1	[1]	O	L1	activation (=1) or not (=0) of the recording of the number of particle/boundary interactions, and of the calculation of the associated boundary statistics. <b>inbrd</b> = 1 is a compulsory condition to use the particulate average <b>imoybr</b> = 2 the selection of the type of interactions that are to be recorded is specified in the subroutine <b>uslabo</b> useful if <b>iensi3</b> =1
<b>iflmbd</b>	i	0, 1	[0]	O	L1	activation (=1) or not (=0) of the recording of the particulate mass flow related to the particle/boundary interactions, and of the calculation of the associated boundary statistics the selection of the type of interactions that are to be recorded is specified in the subroutine <b>uslabo</b> <b>inbrd</b> = 1 is a compulsory condition to use <b>iflmbd</b> =1 useful if <b>iensi3</b> =1 and <b>inbrbd</b> =1
<b>iangbd</b>	i	0, 1	[0]	O	L1	activation (=1) or not (=0) of the recording of the angle between a particle trajectory and a boundary face involved in a particle/boundary interaction, and of the calculation of the associated boundary statistics the selection of the type of interactions that are to be recorded is specified in the subroutine <b>uslabo</b> useful if <b>iensi3</b> =1
<b>ivitbd</b>	i	0, 1	[0]	O	L1	activation (=1) or not (=0) of the recording of the velocity of a particle involved in a particle/boundary interaction, and of the calculation of the associated boundary statistics the selection of the type of interactions that are to be recorded is specified in the subroutine <b>uslabo</b> useful if <b>iensi3</b> =1
<b>iencbd</b>	i	0, 1	[0]	O	L1	activation (=1) or not (=0) of the recording of the mass of coal particles stuck to the wall due to fouling, on the boundary faces of the <b>iencrl</b> interaction type useful if <b>iensi3</b> =1, <b>iphyla</b> =2, <b>iencra</b> =1, and if there is at least one boundary face of the <b>iencrl</b> interaction type
<b>nusbor</b>	i	positive integer	[0]	O	L1	number additional user data to record for the calculation of additional boundary statistics in <b>parbor</b> useful if <b>iensi3</b> =1
<b>nombrd</b>	ca	string of less than 50 characters	[see <b>uslag1</b> ]	O	L1	name of the boundary statistics, displayed in the listing and the post-processing files useful if <b>iensi3</b> =1 <i>Warning: this name is also used to reference information in the restart file (<b>isui</b>st =1). If the name of a variable is changed between two calculations, it will not be possible to read its value from the restart file</i>
<b>imoybr</b>	ia	0, 1, 2	[0 , 1 or 2]	O	L1	the recordings in <b>parbor</b> at every particle/boundary interaction are cumulated values

(possibly reset to zero at every iteration in the non-stationary case). They must therefore be divided by a quantity to get boundary statistics. The user can choose between two average types:

= 0: no average is applied to the recorded cumulated values

= 1: a time-average is calculated. The cumulated value is divided by the physical duration in the case of stationary averages (**isttio**=1). The cumulated value is divided by the value of the last time step in the case of non-stationary averages (**isttio**=0), and also in the case of stationary averages while the absolute Lagrangian iteration number is inferior to **nstbor**

= 2: a particulate average is calculated. The cumulated value is divided by the number of particle/boundary interactions (in terms of statistical weight) recorded in **parbor(nfabor,inbr)**. This average can only be calculated when **inbrbd**=1. The average is calculated if the number of interactions (in statistical weight) of the considered boundary face is strictly higher than **seuilf**, otherwise the average at the face is set to zero

only the cumulated value is recorded in the restart file

useful if **iensi3**=1

<b>npstf</b>	i	positive integer	[0]	O	L3
number of iterations during which stationary boundary statistics have been cumulated useful if <b>iensi3</b> =1, <b>isttio</b> =1 and <b>nstbor</b> inferior or equal to the current Lagrangian iteration					
<b>npstf</b> is initialised and updated automatically by the code, its value is not to be modified by the user					
<b>npstft</b>	i	positive integer	[0]	O	L3
number of iterations during which boundary statistics have been calculated (the potential iterations during which non-stationary statistics have been calculated are counted in <b>npstft</b> ) useful if <b>iensi3</b> =1 <b>npstft</b> is initialised and updated automatically by the code, its value is not to be modified by the user					
<b>tstatp</b>	r	positive real number	[dtp]	O	L3
if the recording of the boundary statistics is stationary, <b>tstatp</b> contains the cumulated physical duration of the recording of the boundary statistics if the recording of the boundary statistics is non-stationary, then <b>tstatp</b> = <b>dtp</b> (it is the Lagrangian time step, because the statistics are reset to zero at every time step) useful if <b>iensi3</b> =1					

## 10 Bibliography

- [1] F. ARCHAMBEAU, N. MÉCHITOUA, M. SAKIZ,  
*Code\_Saturne: a Finite Volume Code for the Computation of Turbulent Incompressible Flows*,  
Industrial Applications, International Journal on Finite Volumes, Vol. 1, 2004.
- [2] F. ARCHAMBEAU, *et al.*,  
*Note de validation de Code\_Saturne version 1.1.0*,  
EDF Report HI-83/04/003/A, 2004 (in french).
- [3] S. BENHAMADOUCHE,  
*Modélisation de sous-maille pour la LES - Validation avec la Turbulence Homogène Isotrope (THI)*  
*dans une version de développement de Code\_Saturne*,  
EDF Report HI-83/01/033/A, 2001 (in french).
- [4] M. BOUCKER, F. ARCHAMBEAU, N. MÉCHITOUA,  
*Quelques éléments concernant la structure informatique du Solveur Commun - Version 1.0\_init0*,  
Compte-rendu express EDF I81-00-8, 2000 (in french).
- [5] M. BOUCKER, J.D. MATTÉI,  
*Proposition de modification des conditions aux limites de paroi turbulente pour le Solveur Commun*  
*dans le cadre du modèle  $k - \epsilon$  standard*,  
EDF Report HI-81/00/019/A, 2000 (in french).
- [6] A. DOUCE, N. MÉCHITOUA,  
*Mise en œuvre dans Code\_Saturne des physiques particulières. Tome3 : Transfert thermique radiatif*  
*en milieu gris semi-transparent*,  
EDF Report HI-81/02/019/A, 2002 (in french).
- [7] A. DOUCE,  
*Physiques particulières dans Code\_Saturne 1.1, Tome 5 : modélisation stochastique lagrangienne*  
*d'écoulements turbulents diphasiques polydispersés*,  
EDF Report, HI-81/04/03/A, 2005 (in french).
- [8] A. ESCAICH, P. PLION, *Mise en œuvre dans Code\_Saturne des modélisations physiques particulières.*  
*Tome 1 : Combustion en phase gaz*,  
EDF Report, HI-81/02/03/A, 2002 (in french).
- [9] A. ESCAICH, *Mise en œuvre dans Code\_Saturne des modélisations physiques particulières. Tome 2 :*  
*Combustion du charbon pulvérisé*,  
EDF Report, HI-81/02/09/A, 2002 (in french).
- [10] N. MÉCHITOUA, F. ARCHAMBEAU,  
*Prototype de solveur volumes finis co-localisé sur maillage non-structuré pour les équations de*  
*Navier-Stokes 3D incompressibles et dilatables avec turbulence et scalaire passif*,  
EDF Report HE-41/98/010/B, 1998 (in french).
- [11] Code\_Saturne DOCUMENTATION,  
*Code\_Saturne 2.1.3 Theory and Programmer's guide*,  
on line with the release of Code\_Saturne 2.1.3 ([info.cs theory](#)).
- [12] M. SAKIZ, VALIDATION TEAM,  
*Validation de Code\_Saturne version 1.2 : note de synthèse*,  
EDF Report H-I83-2006-00818-FR, 2006 (in french).
- [13] M. TAGORTI., S. DAL-SECCO, A. DOUCE, N. MÉCHITOUA,  
*Physiques particulières dans Code\_Saturne, tome 4 : le modèle P-1 pour la modélisation des trans-*  
*ferts thermiques radiatifs en milieu gris semi-transparent*,  
EDF Report HI-81/03/017/A, 2003 (in french).

EDF R&D	<b><i>Code_Saturne</i> version 2.1.3 practical user's guide</b>	<i>Code_Saturne</i> documentation Page 197/ <a href="#">205</a>
---------	---	---

- [14] *Code\_Saturne* DOCUMENTATION,  
*Code\_Saturne version 2.1.3 tutorial*, on line with the release of *Code\_Saturne* 2.1.3 ([info\\_cs](#)  
[tutorial](#)).

# Index of the main variables and keywords

## – Symbols –

)	92
iortvm	35
isvhb	36
isvtb	37
isympa	37
ttclag	187
coejou	47
dpot	47
dteom	36
icdpar	46
icdtmo	36
icodcl	68
idtmom	36
iep	33
ifb	33
ik	32
iomg	33
iphi	33
ipr	32
ir11	33
ir12	33
ir13	33
ir22	33
ir23	33
ir33	33
iscapp	33
iscavr	33
isca	33
itypfb	68
iu	32
iv	32
iw	32
rcodcl	68

## – A –

alch	95
a2ch	95
ahetch	95
ales	79, 157
almax	177
alpnmk	181
anomax	164
arak	166
atcoel	95
atgaze	93
auxl	45

## – B –

betnmk	181
blencv	165
blency	170
bles	79, 157

## – C –

cch	95
cck	95
cdgfac	31
cdgfb	31
cdries	157
cdtvar	152
ce1	177
ce2	177
ce4	178
cebu	105
ckabs1	95
ckabsg	94
ckupdc	38, 82
ckwa1	181
ckwbt1	180
ckwbt2	180
ckwc1	181
ckwgm1	180
ckwgm2	180
ckwsk1	180
ckwsk2	180
cksw1	180
cksw2	180
climgr	163
climgy	170
cmu	177
coefa	37
coefb	37
coejou	184
compog	93
couimp	126, 184
coumax	152
coumxy	171
cp0	175
cp2ch	95
cpashc	95
cpgd1	44
cpgd2	44
cpght	44
cpart	188
crij1	178
crij2	178
crij3	178
crijep	178
crijp1	178
crijp2	178
croule	45, 187
csmago	79, 157
csrij	178
cssge2	179
cssgr1	179



cssgr2	179
cssgr3	179
cssgr4	179
cssgr5	179
cssgs1	178
cssgs2	179
cstlog	177
cv2fa1	179
cv2fc1	179
cv2fc2	179
cv2fcl	180
cv2fct	180
cv2fe2	179
cv2fet	180
cv2fmu	177, 179

– D –

diam20	95
diftl0	105, 176
diipb	31
dijpf	31
dispar	38
distbr	32
distch	102
divukw	39
dofij	32
dpot	184
dt	38
dtmax	153
dtmin	153
dtp	187, 191, 195
dtpt1d	112
dtref	152

– E –

e1ch	95
e2ch	95
ehetch	95
ehgazg	94
emphis	144
epalim	181
eppt1d	38, 112
epscvy	171
epsilo	164
epsily	170
epsrgr	163
epsrgy	170
epsrsm	172
epszer	173
epzero	172
ettp	43, 186
ettpa	43, 186
exthis	144
extrag	163
extray	170

– F –

ficaml	140
ficamr	140
ficavl	141
ficavr	140
ficfpp	140
ficjnf	140
ficmls	140
ficmt1	139
ficmvo	139
ficstp	139
ficush	145
ficusr	146
ficvls	141
ficvt1	139
ficvvo	139
fment	101
fntchr	142
foumax	152
fs(1)	94

– G –

gamnmk	181
gradpr	44
gradvf	44
grand	172
gx,gy,gz	173

– H –

h0ashc	95
hbord	36
hch	95
hck	95
hept1d	112

– I –

iale	181
ialtyb	131
iangbd	194
ibfixe	131
ically	167
icapt	90
iccvfg	171
icdpar	154, 168
icdtmo	36
icelbr	32
icepdc	38
icepdp	82
icetsm	38, 84
icfgrp	185
ichrbo	141
ichrmd	142
ichrsy	141
ichrvl	141
ichrvr	102, 142
ickabs	104

iclkep	155	ientcp	101
iclpnr	156	ientfu	101
iclrtp	37	ientgb	101
iclsyr	156	ientgf	101
iclt1d	112	ientox	101
iclvfl	150	ientre	70, 101
iclvor	75	ientrl	117
icmome	36	iescal	168
icocel	43	iescor	35, 168
icod3p	91	iesder	35, 167
icoebu	91	iespre	35, 167
icoef	37	iestim	35, 167
icoeff	37	iestot	35, 168
icolwc	91	if1m	99, 104
icompf	92	if2m	99, 104
iconv	151	if3m	99, 104
icour	35	if3p2m	104
icp	34, 175	if4p2m	99
icp3pl	92	if4pm	104
icpa	34	ifabor	31
icpext	160	ifacel	31
icpl3c	92	ifapat	37
icpsyr	150	ifinty	74
idebty	74	iflmbd	194
idepo1	117	ifluaa	35
idepo2	117	ifluma	35
idepo3	117	ifm	99, 103
ideuch	154	ifmcel	38
idfmom	146	ifmfbr	37
idiam2	104	ifour	35
idiff	151	ifp2m	99, 104
idiffl	190	ifp3m	99
idifft	151	ifpt1d	38
idifre	156	ifrlag	116
idircl	151	igfuel	93
idirla	190	igliss	131
idist	31	igmdch	104
idistu	189	igmdv1	104
idiver	182	igmdv2	104
idpvar	188	igmhet	104
idries	157	igoxy	93
idstnt	190	igrake	155
idtvar	152	igrari	156
iecaux	45, 147	igrdpp	185
iefjou	128	igrhok	155
ielarc	92, 183	ih2	99, 104
ielcor	184	ihisvr	143
ieljou	92, 183	ihm	98, 103, 104, 128
iencbd	194	iilagr	186
iencra	188	iimlum	182
iencrl	117	iimpar	182
iensi1	191	iindef	70
iensi2	191	iirayo	181
iensi3	193	iisymp	37
ientat	101	ikecou	155

ilapla(i)	128	iprfml	37
ilapoi	189	iprtot	35
ileaux	45, 148	ipstcl	143
ilisvr	102, 146	ipstdv	143
ilogpo	155	ipstft	143
ilvu	191	ipstyp	143
imgr	165	iptlro	152
imgrpy	170	ipucou	171
imligr	163	iqimp	101
imligy	170	irayvf	183
immel	104	ircflu	171
imodak	182	ircfly	170
imoold	146	irebol	117
imoybr	194	irepvo	74
impdvo	139	iresol	164
impfpp	140	irevmc	166
imphis	144	irijec	156
impjnf	140	irijnu	156
impla1	141	irjrb	156
impla2	141	iroext	160
impla3	141	irom	34
impla4	141	irom2	104
impla5	141	iroma	34
impmvo	139	iroule	187
impstp	139	irovar	173
impush	145	iscalt	33, 149
impusr	146	iscavr	149
impvar	188	ischcv	165
impvvo	139	ischcy	170
imrga	162	ischtp	158
imvisf	171	iscold	149
inbrbd	194	iscsth	150
indep	44	ismago	35
indjon	93	isno2t	159
injcon	187	isolib	70
inp	99, 104	isortl	117
inpdt0	148	isso2t	159
iochet	95	isstpc	166
iparoi	70	isstpy	170
iparug	70	istala	190
ipci	95	istat	151
iphydr	166	istmpf	158
iphyla	187	isto2t	159
iplas	187	isttio	187
ipnfac	31	isuila	186
ipnfbr	31	isuid	182
ipoti	128	isuiet	186
ipotr	128	isuit1	171
ipotva	128	isuite	148
ippmod	91	isuivi	121, 187
ipppro	103	isuivo	157
ipprob	34	isymet	70
ipproc	34, 103	it3m	104
ipprof	34	it4m	104
ipreo	166	itbrrb	150

itemp .....	103, 128	iy(1) .....	103
itemp1 .....	104	iy(2) .....	103
itemp2 .....	104	iy(3) .....	104
itepa .....	43	iy1(1) .....	104
itpvar .....	188	iy1(2) .....	104
itrifb .....	37, 74	iy1(3) .....	104
itsnsa .....	35	iy1(4) .....	104
itssca .....	35	iy1(5) .....	105
itstua .....	35	iy1(6) .....	105
iturb .....	153	iy1(7) .....	105
itycel .....	43	izone .....	101
itypfb .....	37		
itypsm .....	38, 84	– J –	
iu .....	101	jbord1 .....	117
iusclb .....	117	jvls .....	187
iuslag .....	118		
iusncl .....	116	– K –	
iusvis .....	120	kabse .....	93
iv .....	101		
iviext .....	160	– L –	
ivimpo .....	131	liste .....	192
ivisch .....	193	lnfac .....	29
ivisck .....	193	lnfbr .....	29
iviscl .....	34	lnnod .....	42
ivisct .....	34	ltsdyn .....	189
ivisev .....	185	ltsmas .....	189
ivisdk .....	193	ltsthe .....	189
ivisdm .....	192		
ivishp .....	193	– M –	
ivisla .....	34	modcpl .....	190
ivisls .....	35, 176		
ivisma .....	35	– N –	
ivismp .....	192	nalimx .....	181
ivissa .....	35	nalinf .....	181
ivisse .....	151	nato .....	93, 95
ivista .....	34	nbmomt .....	30
iviste .....	192	nbmomx .....	30
ivistp .....	192	nbpart .....	186
ivisv1 .....	192	nbpmax .....	42, 186
ivisv2 .....	192	nbrvaf .....	183
ivitbd .....	194	nbstr .....	181
ivivar .....	173	nbvis .....	192
ivrtex .....	46, 157	ncapt .....	143
ivsext .....	160	ncegrm .....	165
iw .....	101	ncel .....	29
iwarni .....	147	ncelbr .....	29
iwarny .....	169	ncelet .....	29
ix2 .....	104	ncepdc .....	38, 82
ixch .....	99, 104	ncepdp .....	82
ixck .....	99, 104	ncesmp .....	84
ixkabe .....	96	ncetsm .....	38, 84
iy1ch .....	95	ncharb .....	95, 118
iy2ch .....	95	ncharm .....	92, 118
iycoel .....	128	nclacp .....	30, 95
iygfm .....	98, 103	nclagm .....	116
		nclepm .....	30

[illegible]

– Q –

qimp	101
qimpat	101
qimpcp	102

– R –

ra	39
rcodel	101
rcpt1d	112
relaxv	153
relxst	153
rgpt1d	112
rho0ch	95
rinfin	172
ro0	174
rr	173
rtp	32, 98
rtpa	32
ruslag	119
rvarfl	177

– S –

s2kw	38
scamax	176
scamin	176
seuil	190
seuilf	193
sigmae	178
sigmak	178
sigmas	176
smacel	38, 84
smagmx	157
srrom	105, 174
statis	44, 191
stephn	173
stoeg	93
surfac	31
surfan	32
surfbn	32
surfbo	31

– T –

t0	175
tbord	37
tepa	43
tept1d	112
th	94
thetav	160
thetcp	162
thetfl	161
thetro	162
thetsn	161
thetss	161
thetst	161
thetvi	162
thetvs	162

timpat	102
timpcp	102
tinful	101
tinoxy	101
tkelvi	173
tkelvn	173
tkent	101
tmarus	148
tmax	93, 95, 183
tmin	93, 95, 183
tpart	188
tppt1d	112
tprenc	118, 188
trefth	173
tslagr	45
tstat	123, 191
tstatp	195
ttcabs	149
ttpabs	149

– U –

uetbor	37
uref	177

– V –

vagaus	45
varrdt	153
viscl0	174
viscv0	185
visls0	176
visref	118, 188
vitflu	44, 121
vitpar	44, 121
volmol	173
volume	31

– W –

wmolat	93, 95
wmolg	94

– X –

xashch	95
xco2	94
xh2o	94
xkabe	96
xkabel	96
xkappa	177
xlesfd	158
xlesfl	79, 157
xlmt1d	112
xlomlg	177
xnp1mx	182
xyzcap	143
xyzcen	31
xyznod	31
xyzp0	175

– **Y** –

y1ch .....	<a href="#">95</a>
y2ch .....	<a href="#">95</a>
yplmxy .....	<a href="#">171</a>
yplpar .....	<a href="#">38</a>
ypluli .....	<a href="#">155</a>

– **Z** –

zero .....	<a href="#">172</a>
------------	---------------------