

TRUST Generic Guide V1.9.2

Support team: trust@cea.fr

Link to: [Project Reference Manual](#)

June 13, 2023

Contents

1	Introduction	2
1.1	Before TRUST: a modular software named Trio_U	2
1.2	Short history	3
1.3	Data file	3
1.3.1	Data file example: base blocks	4
1.3.2	Basic rules	6
1.3.3	Objects notion	6
1.3.4	Interpretor notion	7
1.3.5	Example	7
1.3.6	Important remarks	7
1.4	Running a data file	8
1.4.1	Sequential calculation	8
1.4.2	Parallel calculation	10
1.5	Visualization	11
2	Important references	12
3	Data setting	13
3.1	Problems	13
3.2	Domain definition	14
3.3	Mesh	14
3.3.1	Allowed meshes	14
3.3.2	Import a mesh file	14
3.3.3	Quickly create a mesh	15
3.3.4	Transform mesh within the data file	16
3.3.5	Test your mesh	16
3.4	Discretization	17
3.5	Time schemes	18
3.5.1	Some available time schemes	18
3.5.2	Calculation stopping condition	19
3.6	Medium/Type of fluide	19
3.7	Add gravity	19
3.8	Objects association and discretization	20
3.8.1	Association	20
3.8.2	Discretization	20
4	Problem definition	21
4.1	Set of equations	21
4.1.1	Incompressible problems	21
4.1.2	Quasi-compressible problem	23
4.1.3	Conduction problem	24
4.1.4	Coupled problems	25
4.1.5	Other problems	26

4.2	Pressure solvers	26
4.3	Convection	26
4.4	Diffusion	27
4.5	Initial conditions	27
4.6	Boundary conditions	28
4.7	Turbulence models (in TrioCFD)	28
4.8	Source terms	28
4.9	Post-processing	29
4.9.1	Field names	29
4.9.2	Post-processing blocks	32
4.9.3	Post-process location	34
5	End of the data file	35
5.1	Solve	35
5.2	Stop a running calculation	35
5.3	Save	35
5.4	Resume	36
6	Post-processing	37
6.1	Output files	37
6.2	Tools	38
7	Parallel calculation	39
7.1	Basic notions	39
7.2	Performances	39
7.3	Partitioning	39
7.3.1	The different blocks	40
7.3.2	Partitionning: "Assisted" method	40
7.3.3	TRUST available partitioning tools	41
7.3.4	Overlapping width value	42
7.4	Running a parallel calculation	43
7.4.1	On a PC	43
7.4.2	On a cluster	43
7.5	Visualization	43
7.6	Useful information	44
7.6.1	Modify the mesh	44
7.6.2	Modify calculation parameters	44

Chapter 1

Introduction

This document constitutes the generic guide for **TRUST** software and its **Baltik** projects.

TRUST is a thermohydraulic software package for CFD simulations for incompressible monophasic flow.

You can create new project based on **TRUST** platform. Theses projects are named "**BALTIK**" projects.

The currently available modules include a VDF calculation module "Finite Difference Volume", a VEF calculation module "Finite Element Volume" and a PolyMAC module "series of Marker-and-Cell (MAC) schemes that can handle any type of mesh (non-conform, non-orthogonal, poly-hedral types, ...)". For further details, you can see <https://cea-trust-platform.github.io/classes/discretizations/>

The VDF and VEF validated modules are designed to process the 2D or 3D flow of Newtonian, incompressible, weakly expandable fluids the density of which is a function of a local temperature and concentration values (Boussinesq approximation).

1.1 Before TRUST: a modular software named Trio_U

TRUST was born from the cutting in two pieces of **Trio_U** software. **Trio_U** was a software brick based on the **Kernel** brick (which contains the equations, space discretizations, numerical schemes, parallelism...) and used by other CEA applications (cf Figure 1.1).

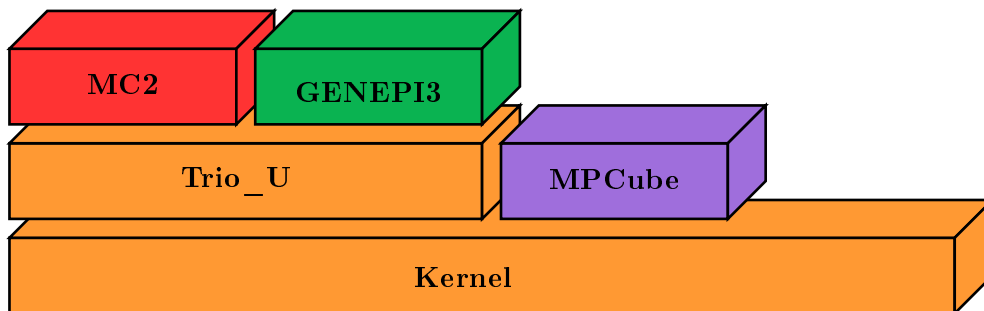


Figure 1.1: Trio_U: brick software

We could create new projects based on Kernel brick or **Trio_U** brick. Theses projects were named "**BALTIK**" projects: "**B**uild an **A**pplication **L**inked to **T**riO_U **K**ernel".

In 2015, **Trio_U** was divided in two parts: **TRUST** and **TrioCFD**.

- **TRUST** is a new platform, its name means: "**TR**io_U Software for **Th**ermohydraulics",

- **TrioCFD** is a BALTIK project based on **TRUST**, which contains the following models: FT, Radiation, LES, zoom...

Here is the structure of **TRUST** platform (cf Figure 1.2):

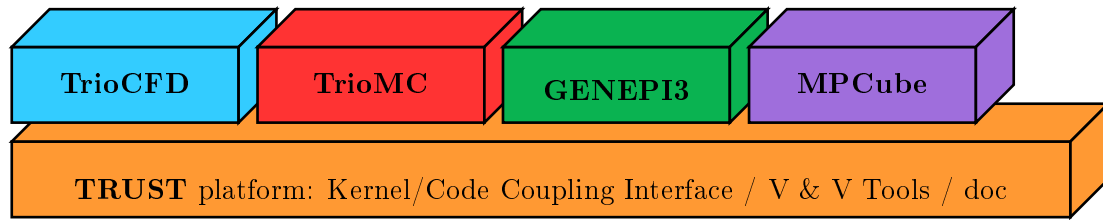


Figure 1.2: TRUST platform & its BALTIKs

Note that: **Trio_U** = **TRUST** + **TrioCFD**.

1.2 Short history

TRUST is now developed at the CEA/DES/ISAS/DM2S/SGLS service. The project starts in 1994 and improved versions were built ever since:

- 1994: start of the project Trio_U
- 01/1997: v1.0 - VDF only
- 06/1998: v1.1 - VEF version
- 04/2000: v1.2 - parallel version
- 07/2001: v1.3 - radiation model, in TrioCFD since v1.7
- 11/2002: v1.4 - new LES turbulence models, in TrioCFD since v1.8
- 02/2006: v1.5 - VDF/VEF Front Tracking, in TrioCFD since v1.7
- 10/2009: v1.6 - data structure revamped
- 06/2015: v1.7 - cut into **TRUST** and **TrioCFD** + switch to open source
- 11/2019: v1.8 - Turbulence features are moved from **TRUST** to **TrioCFD** + PolyMAC discretization
- 06/2022: v1.9 - Pb_Multiphase in **TRUST** + PolyMAC V2 discretization + Pb_HEM in **TrioCFD**

1.3 Data file

To launch a calculation with **TRUST**, you need to write a "data file" which is an input file for **TRUST** and will contain all the information about your simulation. Data files are written following some rules as shown below. But their language is not a programming language, users can't make loops or switch...

Note that:

- lines between `# ... #` and `/* ... */` are comments,
- in that document, words in **bold** are **TRUST** keywords, you can highlight them in your file editor with the command line (details in section 1.4):
`> trust -config gedit|vim|emacs`

- braces "{ }" are elements that **TRUST** reads and interprets, so don't forget them and put space before and after them,
- elements between bracket "[]" are optional.

1.3.1 Data file example: base blocks

Here is the template of a basic sequential data file:

```
# Dimension 2D or 3D #
```

```
Dimension 2
```

```
# Problem definition #
```

```
Pb_hydraulique my_problem
```

```
# Domain definition #
```

```
Domaine my_domain
```

```
# Mesh #
```

```
Read_file my_mesh.geo ;
```

```
# For parallel calculation only! #
```

```
# For the first run: partitioning step #
```

```
# Partition my_domain
```

```
{
```

```
    Partition_tool partitioner_name { option1 option2 ... }
```

```
    Larg_joint 2
```

```
    zones_name DOM
```

```
    ...
```

```
}
```

```
End #
```

```
# For parallel calculation only! #
```

```
# For the second run: read of the sub-domains #
```

```
# Scatter DOM.Zones my_domain #
```

```
# Discretization on hexa or tetra mesh #
```

```
VDF my_discretization
```

```
# Time scheme explicit or implicit #
```

```
Scheme_euler_explicit my_scheme
```

```
Read my_scheme
```

```
{
```

```
    # Initial time #
```

```
    # Time step #
```

```
    # Output criteria #
```

```
    # Stop Criteria #
```

```
}
```

```
# Association between the different objects #
```

```
Associate my_problem my_domain
```

```
Associate my_problem my_scheme
```

```
# Discretization of the problem #
```

```
Discretize my_problem my_discretization
```

```

# New domains for post-treatment #
# By default each boundary condition of the domain is already extracted
  with names such as "my_dom"_boundaries_"my_BC" #
Domaine plane
extraire_surface
{
    domaine plane
    probleme my_probleme
    condition_elements (x>0.5)
    condition_faces (1)
}

```

```

# Problem description #
Read my_problem
{

```

```

    # hydraulic problem #
    Navier_Stokes_standard
    {
        # Physical characteristics of medium #
        Fluide_Incompressible
        {
            ...
            # Gravity vector definition #
            gravity Uniform_field 2 0 -9.81
        }
        # Choice of the pressure matrix solver #
        Solveur_Pression solver { ... }
        # Diffusion operator #
        Diffusion { ... }
        # Convection operator #
        Convection { ... }
        # Sources #
        Sources { ... }
        # Initial conditions #
        Initial_conditions { ... }
        # Boundary conditions #
        Boundary_conditions { ... }
    }
}

```

```

# Post_processing description #
/* To know domains that can be treated directly, search in .err
   output file: "Creating a surface domain named" */
/* To know fields that can be treated directly, search in .err
   output file: "Reading of fields to be postprocessed" */
Post_processing
{
    # Definition of new fields #
    Definition_Champs { ... }

    # Probes #
    Probes { ... }

    # Fields #
    # format default value: lml #
    # select 'lata' for VisIt tool or 'MED' for Salomé #
    format lata
    fields dt_post 1. { ... }

    # Statistical fields #
    Statistiques dt_post 1. { ... }
}

```

```

# Saving and restarting process #
[sauvegarde binaire datafile.sauv]
[resume_last_time binaire datafile.sauv]

```

```

# End of the problem description block #
}

```

```

# The problem is solved with #
Solve my_problem

```

```

# Not necessary keyword to finish #
End

```

1.3.2 Basic rules

There is no line concept in **TRUST**.

Data files uses blocks. They may be defined using the braces:

```

{
    a block
}

```

1.3.3 Objects notion

Objects are created in the data set as follows:

```
Type identificateur
```

- *Type*: must be a type of object recognised by **TRUST**, correspond to the C++ classes. The list of recognised types is given in the file `hierarchie.dump`.

- *identificateur*: the identifier of the object type *Type* created, correspond to an instance of the C++ class *Type*. **TRUST** exits in error if the identifier has already been used.

There are several object types. Physical objects, for example:

- A **Fluide_incompressible** (incompressible_Fluid) object. This type of object is defined by its physical characteristics (its dynamic viscosity μ (keyword **mu**), its density ρ (keyword **rho**), etc...),
- A **Domaine**.

More abstract object types also exist:

- A **VDF**, **VEF** or **PolyMAC** according to the discretization type,
- A **Scheme_euler_explicit** to indicate the time scheme type, here it's a first-order explicit Euler scheme,
- A **Solveur_pression** to denote the pressure system solver type,
- A **Uniform_field** to define, for example, the gravity field.

1.3.4 Interpretor notion

Interprete (interpretor) type objects are then used to handle the created objects with the following syntax:

Type_interprete argument

- *Type_interprete*: any type derived from the **Interprete** (Interpretor) type recognised by **TRUST**. In this manual, they are written in **bold**. You can highlight them in your file editor with the command (details in section 1.4):
> trust -config nedit|vim|emacs
- *argument*: an argument may comprise one or several object identifiers and/or one or several data blocks.

Interpretors allow some operations to be carried out on objects.

Currently available general interpretors include **Read**, **Read_file**, **Ecrire** (Write), **Ecrire_fichier** (Write_file), **Associate**.

1.3.5 Example

A data set to write Ok on screen:

```
Nom a_name      # Creation of an object type. Name identifier a_name #
Read a_name Ok  # Allocates the string "Ok" to a_name #
Ecrire a_name    # Write a_name on screen #
```

1.3.6 Important remarks

1. To insert comments in the data set, use **# .. #** (or **/* ... */**), the character **#** must always be enclosed by blanks.
2. The comma separates items in a list (a comma must be enclosed with spaces or a new line).
3. Interpretor keywords are recognised indiscriminately whether they are written in lower and/or upper case.
4. **On the contrary, object names (identifiers) are recognised differently if they are written in upper or lower case.**
5. In the following description, items (keywords or values) enclosed by [and] are optional.

1.4 Running a data file

To use **TRUST**, your shell must be "bash". So ensure you are in the right shell:

```
> echo $0
/bin/bash
```

To run your data file, you must initialize the TRUST environment using the following command:

```
> source $my_path_to_TRUST_installation/env_TRUST.sh
TRUST vX.Y.Z support : trust@cea.fr
Loading personal configuration /$path_to_my_home_directory/.perso_TRUST.env
```

1.4.1 Sequential calculation

You can run your sequential calculation:

```
> cd $my_test_directory
> trust [-evol] my_data_file
```

where "trust" command call the "trust" script. You can have the list of its options with:

```
> trust -help
```

or

```
> trust -h
```

Here is a panel of available options:

Usage: trust [option] datafile [nb_cpus] [1>file.out] [2>file.err]

Where option may be:

-help -h	: List options.
-baltik [baltik_name]	: Instanciate an empty Baltik project.
-index	: Access to the TRUST ressource index.
-doc	: Access to the TRUST manual (Generic Guide).
-html	: Access to the doxygen documentation.
-config nedit vim emacs gedit	: Configure nedit or vim or emacs or gedit with TRUST keywords.
-edit	: Edit datafile.
-xcheck	: Check the datafile's keywords with xdata.
-xdata	: Check and run the datafile's keywords with xdata.
-partition	: Partition the mesh to prepare a parallel calculation (Creation of
-mesh	: Visualize the mesh(es) contained in the data file.
-eclipse-trust	: Generate Eclipse configuration files to import TRUST sources.
-eclipse-baltik	: Generate Eclipse configuration files to import BALTIK sources (T
-probes	: Monitor the TRUST calculation only.
-evol	: Monitor the TRUST calculation (GUI).
-prm	: Write a prm file (deprecated).
-jupyter	: Create basic jupyter notebook file.
-clean	: Clean the current directory from all the generated files by TRUS
-search keywords	: Know the list of test cases from the data bases which contain ke
-copy	: Copy the test case datafile from the TRUST database under the pr
-check -ctest all testcase list	: Check ctest the non regression of all the test case
-check -ctest function class class::method	: Check ctest the non regression of a list of tests o
-gdb	: Run under gdb debugger.
-valgrind	: Run under valgrind.
-valgrind_strict	: Run under valgrind with no suppressions.
-callgrind	: Run callgrind tool (profiling) from valgrind.

-massif	: Run massif tool (heap profile) from valgrind.
-heaptrack	: Run heaptrack (heap profile). Better than massif.
-advisor	: Run advisor tool (vectorization).
-vtune	: Run vtune tool (profiling).
-perf	: Run perf tool (profiling).
-trace	: Run tracealyzer tool (MPI profiling).
-create_sub_file	: Create a submission file only.
-prod	: Create a submission file and submit the job on the main production queue.
-bigmem	: Create a submission file and submit the job on the big memory production queue.
-queue queue	: Create a submission file with the specified queue and submit the job.
-c ncpus	: Use ncpus CPUs allocated per task for a parallel calculation.
datafile -help_trust	: Print options of TRUST_EXECUTABLE [CASE[.data]] [options].
-convert_data datafile	: Convert a data file to the new 1.9.1 syntax (milieu, interfaces, ...).

1.4.2 Parallel calculation

To run a parallel calculation, you must do two runs:

- the first one, to partition and create your 'n' sub-domains (two methods: "By hand" method below and "Assisted" method cf parts 7.3.1 & 7.3.2),
- the second one, to read your 'n' sub-domains and run the calculation on 'n' processors.

We will explain here how to do such work:

- **Partitioning: "By hand" method**

You have to make two data files:

- *BuildMeshes.data* and
- *Calculation.data*.

The *BuildMeshes.data* file only contains the same information as the beginning of the sequential data file and partitioning information. This file will create the sub-domains (cf .Zones files).

<i>BuildMeshes.data</i>
<pre>Dimension 2 Domaine my_domain # BEGIN MESH # Read_file my_mesh.geo ; # END MESH # # BEGIN PARTITION # Partition my_domain { Partition_tool partitioner_name { option1 option2 ... } Larg_joint 2 zones_name DOM ... } End # END PARTITION #</pre>

Run the *BuildMeshes.data* with **TRUST**:

```
> trust BuildMeshes
```

You may have obtained files named *DOM_000n.Zones* which contains the 'n' sub-domains.

- **Read the sub-domains**

The *Calculation.data* file contains the domain definition, the block which will read the sub-domains and the problem definition (as in sequential calculation).

<i>Calculation.data</i>
<pre> Dimension 2 Domaine <i>my_domain</i> Pb_Hydraulique <i>my_problem</i> # BEGIN SCATTER # Scatter <i>DOM.Zones my_domain</i> # END SCATTER # VDF <i>my_discretization</i> Scheme_euler_explicit <i>my_scheme</i> Read <i>my_scheme</i> { ... } Associate <i>my_problem my_domain</i> Associate <i>my_problem my_scheme</i> Discretize <i>my_problem my_discretization</i> Read <i>my_problem</i> { Fluide_Incompressible { ... } ... } Solve <i>my_problem</i> End </pre>

Run the *Calculation.data* file with **TRUST**:

```
> trust Calculation procs_number
```

This will read your *DOM_000n.Zones* files. You can see the documentation of the "**scatter**" keyword in [this part of the Project Reference Manual](#).

For more information, you can see this [exercise in the TRUST tutorial](#).

1.5 Visualization

To learn how to use the "**-evol**" option, you can see the first exercise of the **TRUST** tutorial: [Flow around an obstacle](#).

Chapter 2

Important references

For details and practices, see:

- [the **TRUST** tutorial](#),
- The **TRUST** & **TrioCFD** user slides,
- The **TRUST** & **TrioCFD** development slides,
- [the Project Reference Manual](#).

Other references:

- Methodology for incompressible single phase flow (Models_Equations_TRUST.pdf),
- Trio_U code validation data base & best practice guidelines (Best_Practice_TRUST.pdf),
- Organisation of TrioCFD validation data base (HowTo_Validation.pdf),
- To access **TRUST** test case list and most documentation:

 `> trust -index`
- To access **TrioCFD** documentation (See how to load TrioCFD environment in the User tutorial):

 `> triocfd -index`

For further information, you can see the **TRUST/TrioCFD** website: <https://triocfd.cea.fr>.

Remember that you can attend to:

- a user training session (2 days),
- a developer training session (2 days)

provided by the support team. To request session, send an email to trust@cea.fr.

Chapter 3

Data setting

We will now explain how to fill a data file. First you must specify some basic information like the dimension of your domain, its name, the problem type... To define the problem dimension, we use the following keyword:

Dimension 2

or

Dimension 3

3.1 Problems

You have to define the problem type that you wish to solve.

Pb_*type my_problem*

Here are some of the available problem types:

- for incompressible flow: **Pb_**[Thermo]**hydraulique**[_Concentration][_Turbulent],
- for quasi-compressible flow: **Pb_Thermohydraulique**[_Turbulent]**_QC**,
- for weakly compressible flow: **Pb_Thermohydraulique**[_Turbulent]**_WC**,
- for solid: **Pb_Conduction**,
- you can find all [problem types](#) in the Reference Manual.

where:

- **hydraulique**: means that we will solve Navier-Stokes equations without energy equation,
- Thermo: means that we will solve Navier-Stokes equations with energy equation,
- Concentration: that we will solve multiple constituent transportation equations,
- Turbulent: that we will simulate a turbulent flow and specify a turbulent model (RANS or LES). (Since version v1.8.0, Turbulence models are in TrioCFD and not anymore in TRUST).
- **Conduction**: resolution of the heat equation,
- **QC**: Navier Stokes equations with energy equation for quasi-compressible fluid under low Mach approach,
- **WC**: Navier Stokes equations with energy equation for weakly compressible fluid under low Mach approach.

3.2 Domain definition

To define the domain, you must name it. This is done thanks to the following block:

```
Domaine my_domain
```

Then you must add your mesh to your simulation.

3.3 Mesh

Notice the presence of the tags:

```
# BEGIN MESH #  
...  
# END MESH #
```

in the data file of section 1.3.1. This is useful for parallel calculation (see section 7) if well placed in datafile.

3.3.1 Allowed meshes

TRUST allows:

- quadrangular or triangular undeformed meshing for 2D cases (Figure 3.1),



Figure 3.1: 2D allowed elements

- hexahedral or tetrahedral undeformed meshing for 3D cases (Figure 3.2).

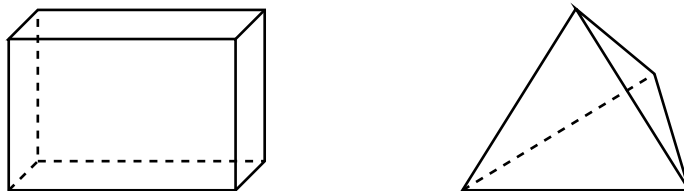


Figure 3.2: 3D allowed elements

Non-standard and hybrid meshing are (partially, since not all physical models in TrioCFD are yet available with this discretization) supported thanks to PolyMAC discretization! (cf Figure 3.3)

3.3.2 Import a mesh file

If your mesh was generated with an external tool like [Salomé](#) (open source software), [ICEM](#) (commercial software), [Gmsh](#) (open source software, included in **TRUST** package) or [Cast3M](#) (CEA software), then you must use one of the following keywords into your data file:

- [Read_MED](#) for a MED file from [Salomé](#), [Gmsh](#),... ,
- [Read_File](#) for a binary mesh file from [ICEM](#),

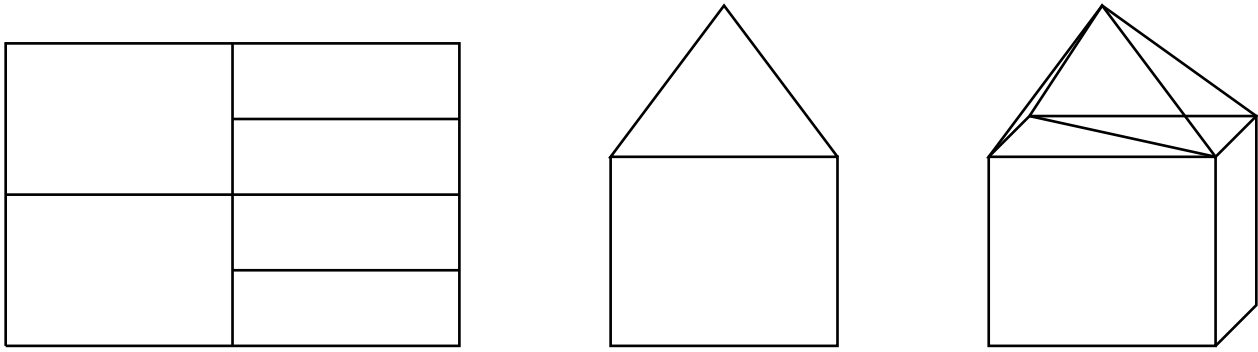


Figure 3.3: Partially supported meshes

- for another format, see the [Project Reference Manual](#).

If you want to learn how to build a mesh with Salomé or Gmsh and read it with **TRUST**, you can look at the exercises of the **TRUST** tutorial: [here](#) for Salomé and [here](#) for Gmsh.

3.3.3 Quickly create a mesh

Here is an example of a simple geometry (of non complex channel type) using the internal tool of **TRUST**:

```
Mailler my_domain
{
    /* Define the domain with one cavity */
    /* cavity 1m*2m with 5*22 cells */
    Pave box
    {
        Origine 0. 0.
        Longueurs 1 2
        /* Cartesian grid */
        Nombre_de_Noeuds 6 23
        /* Uniform mesh */
        Facteurs 1. 1.
    }
    {
        /* Definition and names of boundary conditions */
        bord Inlet    X = 0. 0. <= Y <= 2.
        bord Outlet   X = 1. 0. <= Y <= 2.
        bord Upper    Y = 2. 0. <= X <= 1.
        bord Lower    Y = 0. 0. <= X <= 1.
    }
}
```

To use this mesh in your data file, you just have to add the previous block in your data file or save it in a file named for example "*my_mesh.geo*" and add the line:

```
Read_file my_mesh.geo ;
```

Do not forget the semicolon at the end of the line!

3.3.4 Transform mesh within the data file

You can also make transformations on your mesh after the "**Mailler**" or "**Read_***" command, using the following keywords:

- **Trianguler** to triangulate your 2D cells and create an unstructured mesh.
- **Tetraedriser** to tetrahedralise 3D cells and create an unstructured mesh.
- **Raffiner_anisotrope**/**Raffiner_isotrope** to triangulate/tetrahedralise elements of an unstructured mesh.
- **ExtrudeBord** to generate an extruded mesh from a boundary of a tetrahedral or an hexahedral mesh. **Note** that ExtrudeBord in VEF generates 3 or 14 tetrahedra from extruded prisms.
- **RegroupeBord** to build a new boundary with several boundaries of the domain.
- **Transformer** to transform the coordinates of the geometry.
- for other commands, see the section [interprete](#) of the Project Reference Manual.

Note that theses mesh modifications work on all mesh types (i.e. also for *.geo or *.bin or *.med files).

3.3.5 Test your mesh

The keyword **Discretiser_domaine** is useful to discretize the domain (faces will be created) without defining a problem. Indeed, you can create a minimal data file, post-process your mesh in lata format (for example) and visualize it with VisIt.

Note that you must name all the boundaries!

Here is an example of this kind of data file:

my_data_file.data
<pre>dimension 3 Domaine my_domain Mailler my_domain { Pave box { Origine 0. 0. 0. Longueurs 1 2 1 Nombre_de_Noeuds 6 23 6 Facteurs 1. 1. 1. } { bord Inlet X = 0. 0. <= Y <= 2. 0. <= Z <= 1. bord Outlet X = 1. 0. <= Y <= 2. 0. <= Z <= 1. bord Upper Y = 2. 0. <= X <= 1. 0. <= Z <= 1. bord Lower Y = 0. 0. <= X <= 1. 0. <= Z <= 1. bord Front Z = 0. 0. <= X <= 1. 0. <= Y <= 2. bord Back Z = 1. 0. <= X <= 1. 0. <= Y <= 2. } } discretiser_domaine my_domain postraiter_domaine { domaine my_domain fichier file format lata } End</pre>

To use it, launch in a bash terminal:

```
# if not already done
> source $my_path_to_TRUST_installation/env_TRUST.sh
# then
> trust my_data_file
> visit -o file.lata &
```

To see how to use VisIt, look at the first **TRUST** tutorial exercise: [Flow around an obstacle](#).

If you want to learn how to make a mesh with Salomé or Gmsh and read it with **TRUST**, you can look at the exercises of the **TRUST** tutorial: [here](#) for Salomé and [here](#) for Gmsh.

3.4 Discretization

You have to specify the discretization type which can be **VDF**, **EF** or **VEFPreP1B** or **PolyMAC**.

In **VDF** discretization, the locations of the unknowns are drawn in the Figure 3.4.

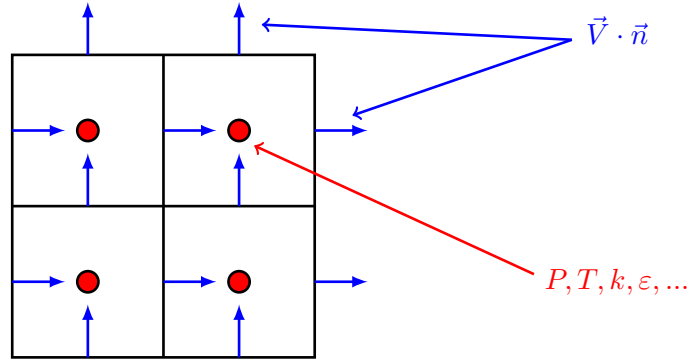


Figure 3.4: VDF unknown locations

For **VEFPreP1B**, the locations of the unknowns are drawn in the Figure 3.5.

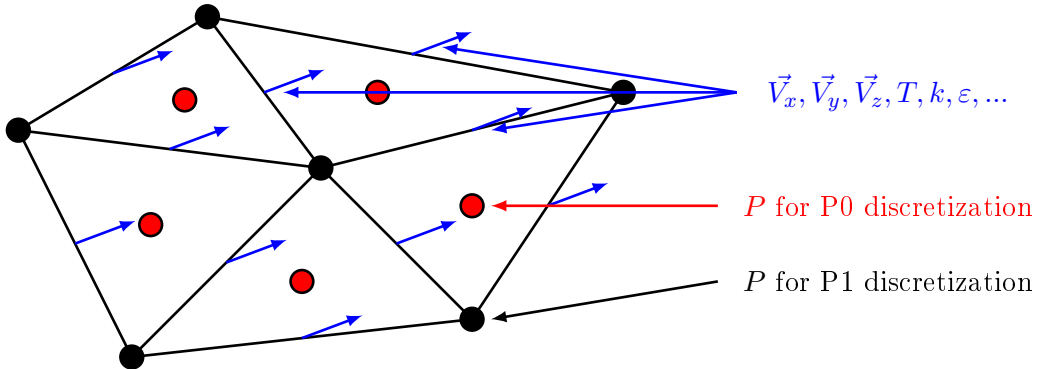


Figure 3.5: VEF unknown locations in 2D

In 3D for the pressure, we can also use the P0+P1+Pa discretization for flow with a strong source term and a low velocity field. In this case P0+P1 pressure gradient has trouble to match the source term so we use P0+P1+Pa discretization (cf Figure 3.6).

To specify the wanted discretization, you have to add the following block to your data file:

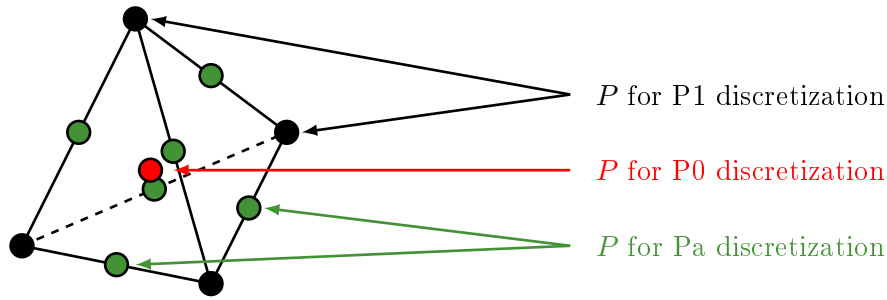


Figure 3.6: VEF pressure location in 3D

```
Discretization_type my_discretization
[Read my_discretization { ... }]
```

You can add parameters to your discretization with the optional keyword **Read** (see **VEFPreP1B discretization**).

On the [TRUST website](#) and [TrioCFD website](#), you can find information about:

- **VDF** discretization in the [PhD thesis of A. Chatelain](#),
- **VEFPreP1B** discretization (Crouzet-Raviart elements) in the [PhD thesis of T. Fortin](#) and [PhD thesis of S. Heib](#).
- **PolyMAC** discretization in the [TRUST website](#).

3.5 Time schemes

Now you can choose your time scheme to solve your problem. For this you must specify the time scheme type wanted and give it a name. then you have to specify its parameters by filling the associated "Read" block.

```
Scheme_type my_time_scheme
Read my_time_scheme { ... }
```

3.5.1 Some available time schemes

Here are some available types of explicit schemes:

- **Scheme_Euler_explicit**,
- **Schema_Adams_Bashforth_order_2**,
- **Runge_Kutta_ordre_3**,

And also some available types of implicit schemes:

- **Scheme_Euler_implicit**,
- **Schema_Adams_Moulton_order_3**.

For other schemes, see [this section](#) of the Reference Manual.

Note that you can implicit diffusion term by activating the **diffusion_implicite** keyword in your explicit time scheme.

3.5.2 Calculation stopping condition

You must specify at least one stopping condition for your simulation. It can be:

- the final time: **tmax**
- the maximal allowed cpu time: **tcpumax**
- the number of time step: **nb_pas_dt_max**
- the convergency treshold: **seuil_statio**

Note that if the time step reaches the minimal time step **dt_min**, **TRUST** will stop the calculation.

If you want to stop properly your running calculation (i.e. with all saves), you may use the *my_data_file.stop* file (cf section 5.2). When the simulation is running, you can see the "0" value in that file.

To stop it, put a "1" instead of the "0", save the file and at the next iteration the calculation will stop properly.

When you don't change anything in that file, at the end of the calculation, you can see that it is written **"Finished correctly"**.

3.6 Medium/Type of fluide

To specify the medium or fluid, you must add the following block.

Fluid_type { ... }

Fluid_type can be one of the following:

- **Fluide_incompressible**
- **Fluide_quasi_compressible**
- **Fluide_Weakly_Compressible**
- **Solide**
- for other types and more information see [Project Reference Manual](#).

Since TRUST v1.9.1, the medium should be read in the begining of the problem definition (before equations). If you want to solve a coupled problem, each medium should be read in the corresponding problem.

3.7 Add gravity

If needed, you can add a gravity term to your simulation. This is done by adding a uniform field, in the medium block since V1.9.1. For example in 2D:

gravity Uniform_field 2 0 -9.81

3.8 Objects association and discretization

3.8.1 Association

Until now, we have created some objects, now we must associate them together. For this, we must use the **Associate** interpreter:

```
# Association between the different objects #  
Associate my_problem my_domain  
Associate my_problem my_time_scheme
```

3.8.2 Discretization

Then you must discretize your domain using the **Discretize** interpreter:

```
Discretize my_problem my_discretization
```

The problem *my_problem* is discretized according to the *my_discretization* discretization.

IMPORTANT: A number of objects must be already associated (a domain, time scheme, ...) prior to invoking the **Discretize** keyword.

Note that when the discretization step succeeds, the mesh is validated by the code.

Chapter 4

Problem definition

4.1 Set of equations

Depending on your choosed problem type, you will have a different set of equations.

4.1.1 Incompressible problems

TRUST solves Navier-Stokes equations with/without the heat equation for an incompressible fluid:

$$\left\{ \begin{array}{l} \nabla \cdot \vec{u} = 0 \\ \frac{\partial \vec{u}}{\partial t} + \nabla \cdot (\vec{u} \otimes \vec{u}) = \nabla \cdot (\nu \nabla \vec{u}) - \nabla P^* \\ \frac{\partial T}{\partial t} + \vec{u} \nabla T = \nabla \cdot (\alpha \nabla T) + \frac{Q}{\rho C_p} \end{array} \right.$$

where: $P^* = \frac{P}{\rho} + gz$, Q is the heat source term, and:

- ρ : density,
- μ : dynamic viscosity,
- $\nu = \frac{\mu}{\rho}$: kinematic viscosity,
- $\vec{g} = gz$: gravity vector in cartesian coordinates,
- $\alpha = \frac{\lambda}{\rho C_p}$: thermal diffusivity.
- C_p : specific heat capacity at constant pressure,
- λ : thermal conductivity,

Note that **red** terms are convective terms and **blue** terms are diffusive terms.

```

Pb_Thermo_hydraulique_Concentration_Turbulent my_problem
...
Read my_problem
{
    # Define medium and its properties + gravity if any #
    Fluide_incompressible { ... }
    # Navier Stokes equations with/without turbulent model #
    Navier_Stokes_Turbulent_Standard
    {
        Solveur_Pression my_solver { ... }
        Diffusion { ... }
        Convection { ... }
        Initial_conditions { ... }
        Boundary_conditions { ... }
        Modele_turbulence modele { ... }
        Sources { ... }
        ...
    }
    # Energy equation with/without turbulent model #
    Convection_Diffusion_Temperature_Turbulent
    {
        Diffusion { ... }
        Convection { ... }
        Initial_conditions { ... }
        Boundary_conditions { ... }
        Modele_turbulence Prandtl { ... }
        Sources { ... }
        ...
    }
    # Constituent transportation equations with/without turbulent model #
    Convection_Diffusion_Concentration_Turbulent
    {
        Diffusion { ... }
        Convection { ... }
        Initial_conditions { ... }
        Boundary_conditions { ... }
        Modele_turbulence Schmidt { ... }
        Sources { ... }
        ...
    }
}

```

For documentation, see:

Thermo	hydraulique	Concentration	Turbulent	Reference Manual
	Pb_hydraulique			doc
	Pb_hydraulique	_Concentration		doc
	Pb_hydraulique		_Turbulent	TrioCFD Reference Manual
	Pb_hydraulique	_Concentration	_Turbulent	TrioCFD Reference Manual
Pb_Thermo	hydraulique			doc
Pb_Thermo	hydraulique	_Concentration		doc

Pb_Thermo	hydraulique		Turbulent	TrioCFD Reference Manual
Pb_Thermo	hydraulique	Concentration	Turbulent	TrioCFD Reference Manual

4.1.2 Quasi-compressible problem

TRUST solves Navier-Stokes equations with/without heat equation for quasi-compressible fluid:

$$\left\{ \begin{array}{l} \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0 \\ \frac{\partial \rho \vec{u}}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u}) = \nabla \cdot (\mu \nabla \vec{u}) - \nabla P - \rho \vec{g} \\ \rho C_p \left(\frac{\partial T}{\partial t} + \vec{u} \nabla T \right) = \nabla \cdot (\lambda \nabla T) + \frac{dP_0}{dt} + Q \end{array} \right.$$

where: $P_0 = \rho RT$, Q is a heat source term, and:

- ρ : density,
- μ : dynamic viscosity,
- $\vec{g} = gz$: gravity vector in cartesian coordinates,
- C_p : specific heat capacity at constant pressure,
- λ : thermal conductivity.

Note that red terms are convective terms and blue terms are diffusive terms.

```

Pb_Thermohydraulique_Turbulent_QC my_problem
...
Read my_problem
{
    # Define medium and its properties + gravity if any #
    Fluide_Quasi_compressible { ... }

    # Navier Stokes equations for quasi-compressible fluid under #
    # low Mach numbers with/without turbulent model #
    Navier_Stokes_Turbulent_QC
    {
        Solveur_Pression my_solver { ... }
        Diffusion { ... }
        Convection { ... }
        Initial_conditions { ... }
        Boundary_conditions { ... }
        Modele_turbulence modele { ... }
        Sources { ... }
        ...
    }
    # Energy equation for quasi-compressible fluid under low Mach #
    # numbers with/without turbulent model #
    Convection_Diffusion_Chaleur_Turbulent_QC
    {
        Diffusion { ... }
        Convection { ... }
        Initial_conditions { ... }
        Boundary_conditions { ... }
        Modele_turbulence Prandtl { ... }
        Sources { ... }
        ...
    }
}

```

For more information on QC problem, go [there](#) and for turbulent QC problem see TrioCFD Reference Manual.

4.1.3 Conduction problem

For this kind of problems, **TRUST** solves the heat equation:

$$\rho C_p \frac{\partial T}{\partial t} = \nabla \cdot (\lambda \nabla T) + Q$$

where:

- ρ : density,
- C_p : specific heat capacity at constant pressure,
- λ : thermal conductivity,
- Q is a heat source term.

Note that term in [blue](#) is the diffusive term.

In your data file, you will have:

```

Pb_Conduction my_problem
...
Read my_problem
{
    # Define medium and its properties #
    Solide { ... }

    # Resolution of the heat equation #
    Conduction
    {
        Diffusion { ... }
        Convection { ... }
        Initial_conditions { ... }
        Boundary_conditions { ... }
        Sources { ... }
        ...
    }
}

```

For more information, see the [Project Reference Manual](#).

4.1.4 Coupled problems

With **TRUST**, we can couple problems. We will explain here the method for two problems but you can couple as many problems as you want.

To couple two problems, we define two problems *my_problem_1* and *my_problem_2* each one associated to a separate domain *my_domain_1* and *my_domain_2*, and to a separate medium *my_medium_1* and *my_medium_2* (associated or not to the gravity).

```

Dimension 2

Pb_ThermoHydraulique_Turbulent my_problem_1
Pb_ThermoHydraulique_Turbulent my_problem_2

Domaine my_domain_1
Read_file my_mesh_1.geo ;

Domaine my_domain_2
Read_file my_mesh_2.geo ;
Associate my_problem_1 my_domain_1

Associate my_problem_2 my_domain_2

```

Then we define a coupled problem associated to a single time scheme like for example:

```

Probleme_Couple my_coupled_problem

VEFPreP1B my_discretization

Scheme_euler_explicit my_scheme
Read my_scheme { ... }

Associate my_coupled_problem my_problem_1
Associate my_coupled_problem my_problem_2
Associate my_coupled_problem my_scheme

```

Then we discretize and solve everything:

```
Discretize my_coupled_problem my_discretization
Read my_problem_1
{
    Fluide_Incompressible { ... }
    ...
}
Read my_problem_2
{
    Fluide_Incompressible { ... }
    ...
}

Solve my_coupled_problem
End
```

You can see the documentation of this kind of problem in the [Project Reference Manual](#).

4.1.5 Other problems

TRUST & TrioCFD can also solve the following types of problems (Turbulence models in TrioCFD):

- [Resolution of NAVIER STOKES/energy/multiple constituent transportation equations, with the additional passive scalar equations.](#)

4.2 Pressure solvers

Then you may indicate the choice of pressure solver (cf [Project Reference Manual](#)) using the following syntax:

```
Solveur_pression my_solver { ... }
```

The *my_solver* may be:

- **GCP**,
- **Petsc** *Petsc_solver_name*,
- **Cholesky**,
- **Gmres**,
- **Gen**,
- **Optimal**.

Reminder: in CFD, a separate solver is used to solve the pressure. For more details, you can have a look at the section "Time and space schemes" of the **TRUST** & **TrioCFD** user slides.

4.3 Convection

There is no default convective scheme so you must choose one [convection scheme](#):

```
convection { convective_scheme }
```

You can use the following convective scheme, following the recommendations of the user training session (cf section "Time and space schemes" of the **TRUST** & **TrioCFD** user slides and the section "Recommendations for schemes") following your discretization type:

- [Amont](#)
- [Muscl](#)
- [EF_stab](#)
- for more, see the [Project Reference Manual](#).

Note that there is no default convective scheme and if you don't want convection in your problem, you may use:

```
convection { negligible }
```

4.4 Diffusion

The diffusion term is more or less a Laplacien operator and is thus always discretized by a second order centered difference scheme. So you just need to do this:

```
diffusion { }
```

Note If you don't want diffusion in your problem, you may use:

```
diffusion { negligible }
```

4.5 Initial conditions

For each equation, you **must** set initial conditions:

```
initial_conditions { ... }
```

To see the syntax of each available initial condition: cf [Project Reference Manual](#). Here are the most used initial conditions:

- **Vitesse** field_type *bloc_lecture_champ*
- **Temperature** field_type *bloc_lecture_champ*
- **K_eps** field_type *bloc_lecture_champ*

We list here some "field_type":

- **Uniform_Field**: for a uniform field,
- **Champ_Fonc_Med**: to read a data field in a MED-format file .med at a specified time,
- **Champ_Fonc_txyz**: for a field which depends on time and space,
- **Champ_Fonc_Fonction_txyz**: for a field which is a function of another field and time and/or space coordinates,
- **Champ_Fonc_Reprise**: to read a data field in a saved file (.xyz or .sauv) at a specified time.
- refer to the [Project Reference Manual](#).

4.6 Boundary conditions

Then you may specify your boundary conditions like:

```
boundary_conditions { ... }
```

It is important to specify here that **TRUST will not accept any boundary conditions by default.**

You can find help for boundary conditions in the [Project Reference Manual](#). Here is a list of the most used boundary conditions:

- [Bord Frontiere_ouverte_vitesse_imposee](#) boundary_field_type *bloc_lecture_champ*
- [Bord Frontiere_ouverte_pression_imposee](#) boundary_field_type *bloc_lecture_champ*
- [Bord Paroi_fixe](#)
- [Bord Symetrie](#)
- [Bord Periodique](#)
- [Bord Frontiere_ouverte_temperature_imposee](#) boundary_field_type *bloc_lecture_champ*
- [Bord Frontiere_ouverte T_ext](#) boundary_field_type *bloc_lecture_champ*
- [Bord Paroi_adiabatique](#)
- [Bord Paroi_flux_impose](#) boundary_field_type *bloc_lecture_champ*
- for more, see the [Project Reference Manual](#).

To choose your "boundary_field_type" parameters, refer to the [Project Reference Manual](#).

4.7 Turbulence models (in TrioCFD)

User can add a turbulence model to his simulation using the keyword:

```
Modele_turbulence my_model { ... }
```

where *my_model* can be:

- [Longueur_Melange](#): RANS model based on mixing length modelling,
- [Sous_maille](#): LES model which uses a structure sub-grid function model,
- [K_epsilon](#): for RANS turbulence model ($k-\varepsilon$),
- for more, see the [Project Reference Manual](#).

4.8 Source terms

To introduce a source term into an equation, add the following line into the block defining the equation. The list of source keyword is described below.

```
Sources { source_keyword }
```

To introduce several source terms into the same equation, the blocks corresponding to the various terms need to be separated by a comma:

```
Sources { source_keyword1 , source_keyword2 , ... }
```

- [Perte_Charge_Reguliere](#) type_perte_charge bloc_definition_pertes_charges
- [Perte_Charge_Singuliere](#) **KX** | **KY** | **KZ** coefficient_value { ... }

- **Canal_perio** { ... }
- **Boussinesq_temperature** { ... } : $\rho(T) = \rho(T_0)(1 - \beta_{th}(T - T_0))$
- **Boussinesq_concentration** { ... }
- **Puissance_thermique** field_type bloc_lecture_champ
- [documentation for hydraulic source terms and for scalar source terms.](#)

4.9 Post-processing

Before post-processing fields, during a run, **TRUST** creates several files which contain information about the calculation, the convergence, fluxes, balances... (see part 6.1 for more information).

Several keywords can be used to create a post-processing block, into a problem. First, you can create a single post-processing task (**Post_processing** keyword). Generally, in this block, results will be printed with a specified format at a specified time period.

```
Post_processing
{
    Postraitement_definition
}
```

But you can also create a list of post-processings with **Post_processings** keyword (named with Post_name1, Post_name2, etc...), in order to print results into several formats or with different time periods, or into different results files:

```
Post_processings
{
    Post_name1 { Postraitement_definition }
    Post_name2 { Postraitement_definition }
    ...
}
```

4.9.1 Field names

• Existing & predefined fields

You can post-process predefined fields and already existing fields. Here is a list of post-processable fields, but it is not the only ones.

Physical values	Keyword for field_name	Unit
Velocity	Vitesse or Velocity	$m.s^{-1}$
Velocity residual	Vitesse_residu	$m.s^{-2}$
Kinetic energy per elements ($0.5\rho u_i ^2$)	Energie_cinetique_elem	$kg.m^{-1}.s^{-2}$
Total kinetic energy ($\frac{\sum_{i=1}^{nb_elem} 0.5\rho u_i ^2 vol_i}{\sum_{i=1}^{nb_elem} vol_i}$)	Energie_cinetique_totale	$kg.m^{-1}.s^{-2}$
Vorticity	Vorticite	s^{-1}
Pressure in incompressible flow ($P/\rho + gz$) For Front Tracking probleme ($P + \rho gz$)	Pression ¹	$Pa.m^3.kg^{-1}$ or Pa
... continued on next page ...		

¹The post-processed pressure is the pressure divided by the fluid's density ($P/\rho + gz$) on incompressible laminar calculation. For turbulent, pressure is $P/\rho + gz + 2/3 * k$ cause the turbulent kinetic energy is in the pressure gradient.

Physical values	Keyword for field_name	Unit
Pressure in incompressible flow ($P + \rho gz$)	Pression_pa or Pressure	Pa
Pressure in compressible flow	Pression	Pa
Hydrostatic pressure (ρgz)	Pression_hydrostatique	Pa
Totale pressure (when quasi compressible model is used)=Pth+P	Pression_tot	Pa
Pressure gradient ($\nabla(P/\rho + gz)$)	Gradient_pression	$m.s^{-2}$
Velocity gradient	gradient_vitesse	s^{-1}
Temperature	Temperature	$^{\circ}C$ or K
Temperature residual	Temperature_residu	$^{\circ}C.s^{-1}$ or $K.s^{-1}$
Phase temperature of a two phases flow	Temperature_EquationName	$^{\circ}C$ or K
Mass transfer rate between two phases	Temperature_mpoint	$kg.m^{-2}.s^{-1}$
Temperature variance	Variance_Temperature	K^2
Temperature dissipation rate	Taux_Dissipation_Temperature	$K^2.s^{-1}$
Temperature gradient	Gradient_temperature	$K.m^{-1}$
Heat exchange coefficient	H_echange_Tref ²	$W.m^{-2}.K^{-1}$
Turbulent heat flux	Flux_Chaleur_Turbulente	$m.K.s^{-1}$
Turbulent viscosity	Viscosite_turbulente	$m^2.s^{-1}$
Turbulent dynamic viscosity (when quasi compressible model is used)	Viscosite_dynamique_turbulente	$kg.m.s^{-1}$
Turbulent kinetic energy	K	$m^2.s^{-2}$
Turbulent dissipation rate	Eps	$m^3.s^{-1}$
Turbulent quantities K and Epsilon	K_Eps	$(m^2.s^{-2}, m^3.s^{-1})$
Residuals of turbulent quantities K and Epsilon residuals	K_Eps_residu	$(m^2.s^{-3}, m^3.s^{-2})$
Constituent concentration	Concentration	
Constituent concentration residual	Concentration_residu	
Component velocity along X	VitesseX	$m.s^{-1}$
Component velocity along Y	VitesseY	$m.s^{-1}$
Component velocity along Z	VitesseZ	$m.s^{-1}$
Mass balance on each cell	Divergence_U	$m^3.s^{-1}$
Irradiancy	Irradiance	$W.m^{-2}$
Q-criteria	Critere_Q	s^{-1}
Distance to the wall $Y^+ = yU/\nu$ (only computed on boundaries of wall type)	Y_plus	dimensionless
Friction velocity	U_star	$m.s^{-1}$
Void fraction	alpha	dimensionless
Cell volumes	Volume_maille	m^3
Chemical potential	Potentiel_Chimique_Generalise	
Source term in non Galilean referential	Acceleration_terme_source	$m.s^{-2}$
Stability time steps	Pas_de_temps	S
Listing of boundary fluxes	Flux_bords	cf each *.out file
Volumetric porosity	Porosite_volumique	dimensionless
Distance to the wall	Distance_Paroi ³	m
Volumic thermal power	Puissance_volumique	$W.m^{-3}$
Local shear strain rate defined as		

... continued on next page ...

²Tref indicates the value of a reference temperature and must be specified by the user. For example, H_echange_293 is the keyword to use for Tref=293K.

³distance_paroi is a field which can be used only if the mixing length model (see 2.15.1.2) is used in the data file.

Physical values	Keyword for field_name	Unit
$\sqrt{(2S_{ij}S_{ij})}$	Taux_cisaillement	s^{-1}
Cell Courant number (VDF only)	Courant_maille	dimensionless
Cell Reynolds number (VDF only)	Reynolds_maille	dimensionless
Viscous force	viscous_force	$kg.m^2.s^{-1}$
Pressure force	pressure_force	$kg.m^2.s^{-1}$
Total force	total_force	$kg.m^2.s^{-1}$
Viscous force along X	viscous_force_x	$kg.m^2.s^{-1}$
Viscous force along Y	viscous_force_y	$kg.m^2.s^{-1}$
Viscous force along Z	viscous_force_z	$kg.m^2.s^{-1}$
Pressure force along X	pressure_force_x	$kg.m^2.s^{-1}$
Pressure force along Y	pressure_force_y	$kg.m^2.s^{-1}$
Pressure force along Z	pressure_force_z	$kg.m^2.s^{-1}$
Total force along X	total_force_x	$kg.m^2.s^{-1}$
Total force along Y	total_force_y	$kg.m^2.s^{-1}$
Total force along Z	total_force_z	$kg.m^2.s^{-1}$

Remark: Physical properties (conductivity, diffusivity,...) can also be post-processed.

The name of the fields and components available for post-processing is displayed in the error file after the following message: "Reading of fields to be postprocessed". Of course, this list depends of the problem being solved.

For more information, you can see the [Project Reference Manual](#).

- **Creating new fields**

The **Definition_champs** keyword is used to create new or more complex fields for advanced post-processing.

Definition_champs { *field_name_post* *field_type* { ... } }

field_name_post is the name of the new created field and *field_type* is one of the following possible type:

- **refChamp**
- **Reduction_0D** using for example the **min**, **max** or **somme** methods.
- **Transformation**
- for details and other keywords, see the [Project Reference Manual](#).

Note that you can combine several *field_type* keywords to create your field and then use your new fields to create other ones.

Here is an example of new field named *max_temperature*:

```

Read my_problem {
  ...
  Postraitement {
    Definition_champs {
      # Creation of a 0D field: maximal temperature of the domain #
      max_temperature Reduction_0D {
        methode max
        source refChamp { Pb_champ my_problem temperature }
      }
    }
    Probes {
      # Print max(temperature) into the datafile_TMAX.son file #
      tmax max_temperature periode 0.01 point 1 0. 0.
    }
    Champs dt_post 1.0 { ... }
  }
}

```

You can find other examples in the **TRUST & TrioCFD** user slides in the section "Post processing description".

4.9.2 Post-processing blocks

There are three methods to post-process in **TRUST**: using probes, fields or making statistics.

- **Probes**

Probes refer to sensors that allow a value or several points of the domain to be monitored over time. The probes are a set of points defined:

- one by one: **Points** keyword or
- by a set of points evenly distributed over a straight segment: **Segment** keyword or
- arranged according to a layout: **Plan** keyword or
- arranged according to a parallelepiped: **Volume** keyword.

Here is an example of 2D **Probes** block:

```

Probes {
  pressure_probe [loc] pressure Periode 0.5 Points 3 1. 0. 1. 1. 1. 2.
  velocity_probe [loc] velocity Periode 0.5 Segment 10 1. 0. 1. 4.
}

```

where the use of "loc" option allow to specify the wanted location of the probes. The available values are "**grav**" for gravity center of the element, "**nodes**" for faces and "**som**" for vertices. There is not default location. If the point does not coincide with a calculation node, the value is extrapolated linearly according to neighbouring node values.

For complete syntax, see the [Project Reference Manual](#), also for [all options](#).

- **Fields**

This keyword allows to post-process fields on the whole domain, specifying the name of the backup file, its format, the post-processing time step and the name (and location) of the post-processed fields.

Here is an example of **Fields** block:

```

Fichier results
Format lata
Fields dt_post 1. {
    velocity [faces] [som] [elem]
    pressure [elem] [som]
    temperature [elem] [som]
}

```

where "**faces**" , "**elem**" and "**som**" are keywords allowed to specify the location of the field.

Note that when you don't specify the location of the field, the default value is "**som**" for values at the vertices. So fields are post-processed at the vertices of the mesh.

To visualize your post-processed fields, you can use open source softwares like: [VisIt](#) (included in **TRUST** package) for lata and med files, or for med files: [Salomé](#) or [Paraview](#).

You can see the [complete syntax](#) and [all options](#) in the [Project Reference Manual](#).

- **Statistics**

Using this keyword, you will compute statistics on your unknowns. You must specify the begining and ending time for the statistics, the post-processing time step, the statistic method, the name (and location) of your post-processed field.

Here is an example of **Statistiques** block:

```

Statistiques dt_post 0.1 {
    t_deb 1. t_fin 5.
    moyenne velocity [faces] [elem] [som]
    ecart_type pressure [elem] [som]
    correlation pressure velocity [elem] [som]
}

```

This block will write at every **dt_post** the average of the velocity $\overline{V(t)}$:

$$\overline{V(t)} = \begin{cases} 0 & , \text{ for } t \leq t_{deb} \\ \frac{1}{t-t_{deb}} \int_{t_{deb}}^t V(t) dt & , \text{ for } t_{deb} < t \leq t_{fin} \\ \frac{1}{t_{fin}-t_{deb}} \int_{t_{deb}}^{t_{fin}} V(t) dt & , \text{ for } t > t_{fin} \end{cases}$$

the standard deviation of the pressure $\langle P(t) \rangle$:

$$\langle P(t) \rangle = \begin{cases} 0 & , \text{ for } t \leq t_{deb} \\ \frac{1}{t-t_{deb}} \sqrt{\int_{t_{deb}}^t [P(t) - \overline{P(t)}]^2 dt} & , \text{ for } t_{deb} < t \leq t_{fin} \\ \frac{1}{t_{fin}-t_{deb}} \sqrt{\int_{t_{deb}}^{t_{fin}} [P(t) - \overline{P(t)}]^2 dt} & , \text{ for } t > t_{fin} \end{cases}$$

and correlation between the pressure and the velocity $\langle P(t).V(t) \rangle$ like:

$$\langle P(t).V(t) \rangle = \begin{cases} 0 & , \text{ for } t \leq t_{deb} \\ \frac{1}{t-t_{deb}} \int_{t_{deb}}^t [P(t) - \overline{P(t)}] \cdot [V(t) - \overline{V(t)}] dt & , \text{ for } t_{deb} < t \leq t_{fin} \\ \frac{1}{t_{fin}-t_{deb}} \int_{t_{deb}}^{t_{fin}} [P(t) - \overline{P(t)}] \cdot [V(t) - \overline{V(t)}] dt & , \text{ for } t > t_{fin} \end{cases}$$

Remark: Statistical fields can be plotted with probes with the keyword "operator_field_name" like for example: Moyenne_Vitesse or Ecart_Type_Pression or Correlation_Vitesse_Vitesse. For that, it is mandatory to have the statistical calculation of this fields defined with the keyword **Statistiques**.

For the complete syntax, see the Project Reference Manual [here](#), and for all options see the [Project Reference Manual](#).

4.9.3 Post-process location

You can use location keywords to specify where you want to post-process your fields in order to avoid interpolations on your post-processed fields.

For the **VDF** discretization, you can see the Figure 3.4 and here is a table with a reminder of the computation location of the fields in the **VDF** discretization:

Names	Keyword	Location
Pressure	pressure	element gravity center
Velocity	velocity	faces center
Temperature	temperature	element gravity center
Density ρ	masse_volumique	element gravity center
Kinematic viscosity ν	viscosite_cinematique	element gravity center
Dynamic viscosity μ	viscosite_dynamique	element gravity center
K	k	element gravity center
eps	eps	element gravity center
y^+	y_plus	element gravity center
u^*	u_star	faces center
Turbulent viscosity	viscosite_turbulente	element gravity center

For the **VEFPreP1B** discretization, you can see the Figure 3.5 and 3.6. Here is a table with a reminder of the computation location of the fields in **VEFPreP1B** discretization:

Names	Keyword	Location
Pressure	pressure	element gravity center vertices
Velocity	velocity	faces center
Temperature	temperature	faces center
Density ρ	masse_volumique	element gravity center
Kinematic viscosity ν	viscosite_cinematique	element gravity center
Dynamic viscosity μ	viscosite_dynamique	element gravity center
K	k	faces center
eps	eps	faces center
y^+	y_plus	element gravity center
u^*	u_star	faces center
Turbulent viscosity	viscosite_turbulente	element gravity center

Be careful, if you are in P0+P1 discretization (default option) and you post-process the pressure field at the element (or at the vertices), you will have an **interpolation** because the field is computed at the element **and** at the vertices.

Chapter 5

End of the data file

5.1 Solve

Now that you have finished to specify all your computation parameters, you may add the **"Solve"** keyword at the end of your data file, in order to solve your problem. You may also add the **"End"** keyword to specify the end of your data file.

```
Solve my_problem
[End]
```

For more details, see the [Project Reference Manual](#).

You can see methods to run your data file in section [1.4](#).

5.2 Stop a running calculation

Your calculation will automatically stop if it has reached:

- the end of the calculation time,
- the maximal allowed cpu time,
- the maximal number of iterations or
- the threshold of convergence.

You may use the *my_data_file.stop* file, if you want to stop properly your running calculation (i.e. with all saves).

When the simulation is running, you can see the **"0"** value in that file. To stop it, put a **"1"** instead of the **"0"** and at the next iteration the calculation will stop properly. When you don't change anything to that file, at the end of the calculation, you can see that it is written **"Finished correctly"**.

5.3 Save

TRUST makes automatic backups during the calculation. The unknowns (velocity, temperature,...) are saved in:

- one **.xyz** file, happening:
 - at the end of the calculation,
 - but, user may disable it with the specific keyword **"EcritureLectureSpecial 0"** added just before the **"Solve"** keyword.

- one (or several in case of parallel calculation) **.sauv** files, happening:
 - at the start of the calculation,
 - at the end of the calculation,
 - each 23 hours of CPU, to change it, uses **"periode_sauvegarde_securite_en_heure"** keyword (default value 23 hours),
 - user may also specify a time physical period with **"dt_sauv"** keyword,
 - periodically using **"tcpumax"** keyword for which calculation stops after the specified time (default value 10^{30}), use it for calculation on CCRT/TGCC and CINES clusters for example.

By default, the name for the **.sauv** files is **"filename_problemname.sauv"** for sequential calculation, **"filename_problemname_000n.sauv"** for parallel calculation (one per process). The format of these files is binary and the files are completed during successive saves.

You can change the behaviour using the following keywords just before the **solve** instruction:

sauvegarde	binaire xyz	<i>filename.sauv filename.xyz</i>
-------------------	--------------------	-----------------------------------

with **"xyz"**: the **.xyz** file is written instead of the **.sauv** files.

Note that, you can use **"sauvegarde_simple"** instead of **"sauvegarde"** where the **.sauv** or **.xyz** file is deleted before saves, in order to keep disk space:

sauvegarde_simple	binaire xyz	<i>filename.sauv filename.xyz</i>
--------------------------	--------------------	-----------------------------------

For more details, see the [Project Reference Manual](#).

5.4 Resume

To resume your calculation, you may:

- change your initial time, the new initial time will be the real final calculation time of the previous calculation (cf **.err** file),
- change your final calculation time to the new wanted value and
- add the following block just before the **"Solve"** keyword:

reprise	binaire xyz	<i>filename.sauv filename.xyz</i>
----------------	--------------------	-----------------------------------

You can resume your calculation:

- from **.sauv** file(s) (one file per process): you can only resume the calculation with the **same number of equations on the same number of processes**,
- or from a **.xyz** file: here you can resume your calculation by **changing the number of equations solved** and/or with a **different number of processes**.

Instead of **"reprise"** keyword, you can use **"resume_last_time"** where **tinit** is automatically set to the last time of saved files (but you may change **tmax**):

resume_last_time	binaire xyz	<i>filename.sauv filename.xyz</i>
-------------------------	--------------------	-----------------------------------

For examples, see the [TRUST tutorial](#) and the [Project Reference Manual](#).

Note that you can run a calculation with initial condition read into a save file (**.xyz** or **.sauv**) from a previous calculation using **Champ_Fonc_reprise** or read a into a MED file with **Champ_Fonc_MED**.

Chapter 6

Post-processing

6.1 Output files

After running, you will find different files in your directory. Here is a short explanation of what you will find in each type of file depending on its extension.

Even if you don't post-process anything, you will have output files which are listed here:

File	Contents
<i>my_data_file.dt_ev</i>	Time steps, facsec, equation residuals
<i>my_data_file.stop</i>	Stop file ('0', '1' or 'Finished correctly')
<i>my_data_file.log</i>	Journal logging
<i>my_data_file.TU</i>	CPU performances
<i>my_data_file_detail.TU</i>	Statistics of execution
<i>my_data_file_problem_name.sauv</i> or <i>.xyz</i> or <i>specified_name.sauv</i> or <i>.xyz</i>	Saving 2D/3D results for resume (binary files)

and the listing of boundary fluxes where:

- *my_data_file_Contrainte_visqueuse.out* correspond to the friction drag exerted by the fluid,
- *my_data_file_Convection_qdm.out* contains the momentum flow rate,
- *my_data_file_Debit.out* is the volumetric flow rate,
- *my_data_file_Force_pression.out* correspond to the pressure drag exerted by the fluid.

If you add post-processings in your data files, you will find:

File	Contents
<i>my_data_file.sons</i>	1D probes list
<i>my_data_file_probe_name.son</i>	1D results with probes
<i>my_data_file_probe_name.plan</i>	3D results with probes
<i>my_data_file.lml</i> (default format) <i>my_data_file.lata</i> (with all *.lata.* files) <i>my_data_file.med</i> or <i>specified_name.lml</i> or <i>.lata</i> or <i>.med</i>	2D/3D results

The screen outputs are automatically redirected in *my_data_file.out* and *my_data_file.err* files if you run a parallel calculation or if you use the "-evol" option of the "trust" script. Else you can redirect them with the following command:

```
# if not already done
> source $my_path_to_TRUST_installation/env_TRUST.sh
# then
> trust my_data_file 1>file_out.out 2>file_err.err
```

In the .out file, you will find the listing of physical infos with mass balance and in the .err file, the listing of warnings, errors and domain infos.

6.2 Tools

To open your 3D results in **lata** format, you can use [VisIt](#) which is an open source software included in **TRUST** package. For that you may "source" **TRUST** environment and launch VisIt:

```
# if not already done
> source $my_path_to_TRUST_installation/env_TRUST.sh
# then
> visit -o my_data_file.lata &
```

To learn how to use it, you can do the **TRUST** tutorial exercise [Flow around an obstacle](#).

To open your 3D results in **med** format, you can also use [VisIt](#) or [Salomé](#) or [Paraview](#).

Here are some actions that you can perform when your simulation is finished:

- To visualize the positions of your probes in function of the 2D/3D mesh, you can open your .son files at the same time of the .lata file in VisIt.
- If you need more probes, you can create them with VisIt (if you have post-processed the good fields) or with MEDCoupling.
- You can use the option "**-evol**" of the trust script, like:

```
trust -evol my_data_file
```

and access to the probes or open VisIt for 2D/3D visualizations via this tool.

Chapter 7

Parallel calculation

TRUST is a platform which allows to make parallel calculation following some basic rules:

- **Single Program, Multiple Data** model: tasks are split up and run simultaneously on multiple processors with different input in order to obtain results faster,
- messages exchange by **Message Passing Interface**,
- from a PC to a massively parallel computer, with shared or distributed memory.

7.1 Basic notions

To make a parallel calculation, you have to partition your domain. Each sub-domain will be treated by one processor. In order to have good performances, ensure you that:

- sub-domains have almost the same number of cells,
- joint lengths (boundaries between sub-domains) are minimal,

7.2 Performances

You have to choose a number of processors which is in agreement with the number of cells. So, you can evaluate your speed-up (sequential time/parallel time which must be as close as possible of the number of processors) or efficiency ($=1/\text{SpeedUp}$) to choose the right number of processors.

From our experience, for good performance with **TRUST**, each processor has to treat between 20000 and 30000 cells.

7.3 Partitioning

To run a parallel calculation, you must:

- make some modifications on your *my_data_file.data* file,
- do two runs:
 - the first one, to partitioning and create your 'n' sub-domains (two methods will be presented),
 - the second one, to read your 'n' sub-domains and run the calculation on 'n' processors.

7.3.1 The different blocks

Different blocks appear in the data file.

- **Modifications on the mesh block**

First you may add the tags "# BEGIN MESH #" and "# END MESH #" before and after your mesh block, for example:

```
# BEGIN MESH #
Read_file my_mesh.geo ;
[Trianguler_h my_domain]
# END MESH #
```

You can refer to section 3.3 to have more information.

- **Adding a partitioning block**

You may now add the partitioning block which contains the cutting instruction, after your mesh block:

```
# BEGIN PARTITION
Partition my_domain
{
    Partition_tool partitioner_name { option1 option2 ... }
    Larg_joint 2
    zones_name DOM
    ...
}
End
END PARTITION #
```

Where *partitioner_name* is the name of the chosen partitioner, one of **METIS**, **Tranche**, **Sous_Zones**, **Partition** or **Fichier_Decoupage** (cf section 7.3.3).

The "**Larg_joint**" keyword allows to specify the overlapping width value. You can see the documentation of [this part in the Project Reference Manual](#).

Note the "End" before the last line. It will be useful for the cutting step.

This block will make the partitioning of your domain into the specified number of sub-domains during the partitioning run.

- **Adding a block to read the sub-domains**

At last, you will add a block which will be activated during the parallel calculation and will allow to read the sub-domains:

```
# BEGIN SCATTER
Scatter DOM.Zones my_domain
END SCATTER #
```

7.3.2 Partitionning: "Assisted" method

Here we will use the "trust -partition datafile" command line to make the partitioning step. We consider that you have correctly add the "#" in your *my_data_file.data* file with the partitioning block and cutting block.

Be careful with the hashtags "#", they are interpreted by the script!

To automatically perform the partitioning step and obtain the parallel data file, you have to run:

```
> trust -partition my_data_file [parts_number]
```

Note that here `parts_number` is the number of sub-domains created but it is also the number of processors which will be used.

This command creates:

- a `SEQ_my_data_file.data` file which is a backup file of `my_data_file.data` the sequential data file,
- a `DEC_my_data_file.data` file which is the first data file to be run to make the partitioning. It is immediately run by the command line **trust -partition datafile** to create a partition, located in the `DOM_000n.Zones` files.

Note that the code stops reading this file at the "**End**" keyword just before the "**# END PARTITION #**" block.

- a `PAR_my_data_file.data` file which is the data file for the parallel calculation. It reads the `DOM_000n.Zones` files through the instruction "**Scatter**".

Note that the meshing and cut of the mesh are commented here.

To see your sub-domains, you can run:

```
> trust -mesh PAR_my_data_file
```

For more information, you can go to see this [exercise](#) in the **TRUST** tutorial.

7.3.3 TRUST available partitioning tools

In **TRUST**, you can make partitioning with:

- the external partitioning library "**METIS**" (open source). It is a general algorithm that will generate a partition of the domain (cf [Project Reference Manual](#)),

```
partition_tool Metis {  
    nb_parts N  
    [use_weights]  
    [pmetis | kmetis]  
    [nb_essais N]  
}
```

- internal **TRUST** partitioning tool: **Tranche** which makes parts by cutting the domain following x, y and/or z directions.

```
partition_tool Tranche {  
    tranches nx ny [nz]  
}
```

- [other internal partitioning tools](#).

The Figure 7.1 is an example of what you can obtain by cutting a 1m x 1m square, divided in three parts using **METIS** and the same square divided in three slices following the x direction with **Tranche**.

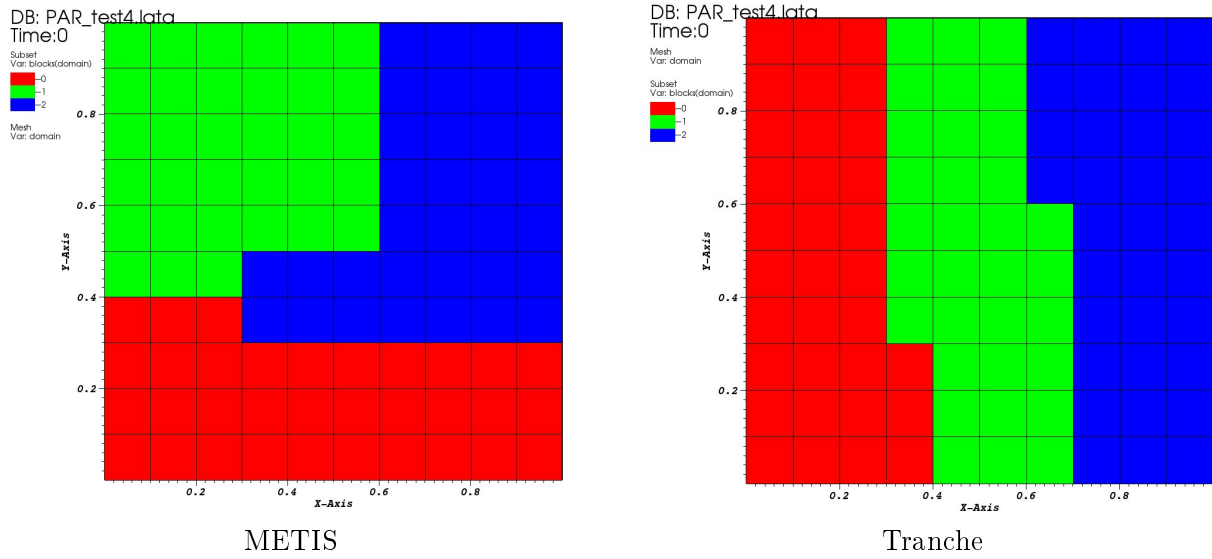


Figure 7.1: Partitioning tools

You can see the documentation of [this part in the Project Reference Manual](#).

7.3.4 Overlapping width value

To make the partitioning, you will have to specify the overlapping width value. This value corresponds to the thickness of the virtual ghost zone (data known by one processor though not owned by it) i.e. the number of vertices or elements on the remote sub-domain known by the local sub-domain (cf Figure 7.2).

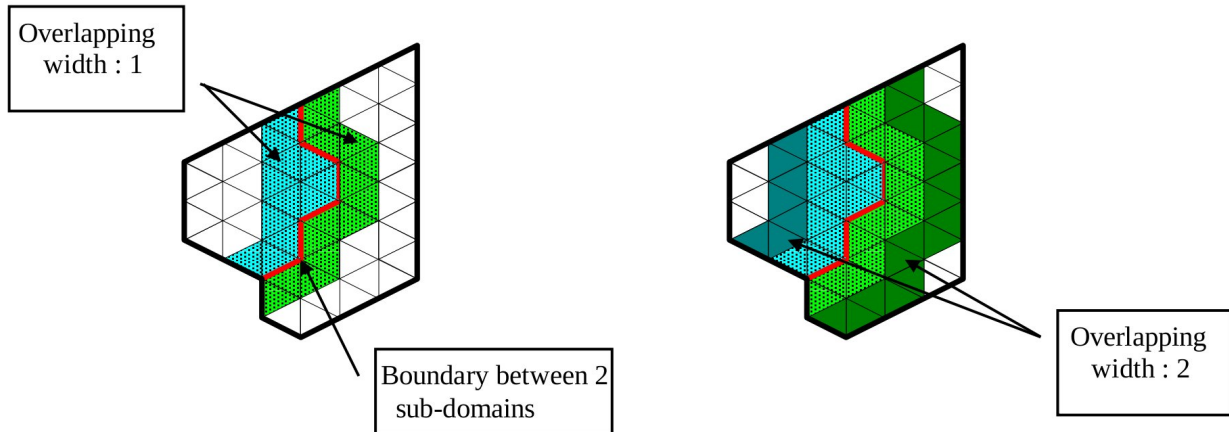


Figure 7.2: Overlapping width

This value depends on the space discretization scheme orders:

- 1 if 1st or 2nd order,
- 2 if 3rd or 4th order.

Note that in general, you will use "2"!

7.4 Running a parallel calculation

7.4.1 On a PC

To launch the calculation, you have to run the calculation by the usual command completed by the number of processors needed:

```
> trust my_parallel_data_file procs_number
```

and *procs_number* is the number of processors used. In fact it is the same as the number of sub-domains.

You can see the **TRUST** & **TrioCFD** user slides in the "Parallel calculation" section for more information and those two exercises of the **TRUST** tutorial: [exercise 1](#) and [exercise 2](#).

7.4.2 On a cluster

You must submit your job in a queue system. For this, you must have a submission file. **TRUST** can create a submission file for you **on clusters on which the support team has done installations**. To create this file, run:

```
> trust -create_sub_file my_parallel_data_file
```

You obtain a file named "**sub_file**", you can open it and verify/change values (for example the name of the job, the name of the exe, ...).

Then you must submit your calculation with:

```
> sbatch sub_file
```

or

```
> ccc_msub sub_file
```

following the queue system of the cluster.

You can see the **TRUST** & **TrioCFD** user slides in the "Parallel calculation" section for more information and [this exercise of the TRUST tutorial](#).

7.5 Visualization

To visualize your probes, you can use the CurvePlot tool, with the command line:

```
> trust -evol my_parallel_data_file
```

or use Gnuplot or any software which reads values in columns in a file.

There are three ways to visualize your parallel results with VisIt:

- HPCDrive or Nice DCV on CCRT/TGCC clusters: opens a deported graphic session on dedicated nodes with more memory (on TGCC cluster: [HPCDrive](#)),
- local mode: copy your results from the cluster to your local computer and open it with a local parallel version of VisIt with:

```
> visit -np 4 &
```

You can have a look at the **TRUST** & **TrioCFD** user slides in the "Parallel calculation description" section.

7.6 Useful information

7.6.1 Modify the mesh

If you want to modify your mesh, you have two possibilities:

- modify the *my_data_file.data* file and run:

```
> trust -partition my_data_file [parts_number]
```

Be carefull it will erase the *SEQ_my_data_file.data*, *DEC_my_data_file.data* and *PAR_my_data_file.data* files and creates new ones.

Then it will run the new *DEC_my_data_file.data* file which gives your new *DOM_000n.Zones* files.

- modify the meshing part of file *DEC_my_data_file.data* and run it with:

```
> trust DEC_my_data_file
```

Then run the parallel calculation normally, on the new *DOM_000n.Zones* files.

```
> trust PAR_my_data_file procs_number
```

7.6.2 Modify calculation parameters

If you want to modify the calculation parameters, you can modify:

- the file *my_data_file.data* and run:

```
> trust -partition data_file_name [parts_number]
```

But it will erase the *SEQ_my_data_file.data*, *DEC_my_data_file.data* and *PAR_my_data_file.data* files and create new ones.

Then it will run the new *DEC_my_data_file.data* file which gives your new *DOM_000n.Zones* files.

Note that in that case, you don't need to re-create the mesh so you can use the second point below:

- modify the *PAR_my_data_file.data* file without running "trust -partition datafile" command line.

Then run the *PAR_my_data_file.data* file with:

```
> trust PAR_my_data_file procs_number
```

Note that if after a certain time, you want to reopen an old case and understand what you did in it without any doubts, you may create two files by hands:

- one "BuildMeshes.data" file only for the mesh and the cut of the mesh, and
- one "calculation.data" file for the parallel calculation.

You will run it like:

```
> trust BuildMeshes
```

```
> trust calculation processors_number
```

For this point, you can have a look at [this exercise of the TRUST tutorial](#).