



---

# **PLUME**

# **Version 4.8.3**

# **User Manual**

---

**February 2019**

**Ref: ASTRI.UM.763751.ASTR**

**Document issue: 2.0**

Page left intentionally blank

## Legal Aspects

Use of the software and of the present software manual is submitted to a license agreement to be accepted before the software installation on a computer.

All suggestion or error concerning the software or this software manual can be sent to:

### **Astrium**

*For the attention of Mr. C. THEROUDE, Mr. G. CHANTEPERDRIX or Mr. M. LEPILLIEZ*

### **Z.I. du Palays**

**31 rue des Cosmonautes  
31402 TOULOUSE CEDEX 4  
FRANCE**

This document and the information it contains are property of Astrium. It shall not be used for any purpose other than those for which it was supplied. It shall not be reproduced or disclosed (in whole or in part) to any third party without Astrium prior written consent.

© Airbus SAS 2019 - All rights reserved

Page left intentionally blank

# Contents

<b>Introduction.....</b>	<b>1</b>
<b>1 About this Document.....</b>	<b>5</b>
<b>2 About PLUME.....</b>	<b>6</b>
<b>3 Glossary.....</b>	<b>7</b>
<b>4 Typographic Conventions.....</b>	<b>8</b>
<b>General Principles.....</b>	<b>1</b>
<b>1 Overview.....</b>	<b>5</b>
1.1 Introduction .....	5
<b>2 PLUME-Specific SYSTEMA Tabs .....</b>	<b>6</b>
<b>3 PLUME Module and Corresponding Files .....</b>	<b>8</b>
<b>Man-Machine Interface (MMI).....</b>	<b>1</b>
<b>1 Overview.....</b>	<b>5</b>
1.1 Introduction .....	5
1.2 In this Chapter.....	6
<b>2 How to Perform a Plume Impingement Analysis.....</b>	<b>7</b>
<b>3 Using the Interactive MODELER Module.....</b>	<b>8</b>
3.1 PLUME Material Tab.....	9
3.2 Applicative Properties.....	10
Physical.....	10
Transparency .....	11
Computation.....	12
Configuration.....	12
3.3 Importing a SYSEXP File .....	13
<b>4 Interacting with the Meshing .....</b>	<b>14</b>
4.1 Plume Impingement .....	14
Meshing.....	14
Numbering.....	16
4.2 Thrusters specific item .....	17
Meshing.....	17
Numbering.....	17
Plume Properties.....	18
4.3 Group specific item.....	19
Meshing.....	19
Numbering.....	19
Plume Properties.....	20
<b>5 Using the Interactive Kinematics Module.....</b>	<b>21</b>
<b>6 Using the Interactive MISSION Module.....</b>	<b>22</b>

<b>7</b>	<b>Using the Interactive PROCESSING Module .....</b>	<b>24</b>
7.1	Basic Rule .....	25
<b>8</b>	<b>Post-Processing.....</b>	<b>28</b>
8.1	How to Display Results in a 3D View .....	28
8.2	How to Display Results in a 2D Curve View .....	29
<b>Computation methods .....</b>		<b>1</b>
<b>1</b>	<b>General Method: Plume Impingement Computation .....</b>	<b>5</b>
1.1	Overview .....	5
1.2	Thruster definition.....	6
1.3	Group definition .....	6
<b>2</b>	<b>Plume Impingement.....</b>	<b>8</b>
2.1	Theory .....	8
	Introduction .....	8
	Principles.....	8
	Free molecular regime: Knudsen model.....	11
	Transparency .....	12
2.2	Parameters.....	13
2.3	Input / Output.....	14
	Input .....	14
	Output .....	14
<b>3</b>	<b>Content of the result files .....</b>	<b>15</b>
3.1	Impingement information file .....	15
	General information data .....	15
	Ray tracing data .....	15
	“From flow file” data.....	15
3.2	Derivate quantities file .....	17
	Results sorting.....	17
	Maximum quantities .....	17
3.3	Spread sheet report files .....	18
	Created folder structure.....	18
	File organisation .....	19



# Introduction



Page left intentionally blank

## Contents

<b>1</b>	<b>About this Document .....</b>	<b>5</b>
<b>2</b>	<b>About PLUME .....</b>	<b>6</b>
<b>3</b>	<b>Glossary.....</b>	<b>7</b>
<b>4</b>	<b>Typographic Conventions.....</b>	<b>8</b>

Page left intentionally blank

# 1 About this Document

This User Manual is intended for users of the **PLUME** application.

It consists of the following chapters:

- **Introduction**
  - Gives a general description of the manual.
- **General Principles**
  - Gives an overview of the **PLUME** method and functionalities, the different files and module.
- **Man Machine Interface (MMI)**
  - Describes all the elements interfaced with the **PLUME** application.
- **Computation methods**
  - Presents general computation methods used by the module:
    - **Plume impingement**
      - Presents the module describing the calculation of the plume impingement on model meshes: forces and torques, thermal fluxes and mass fluxes.
- **Getting Started with PLUME**
  - Provides step-by-step instructions for getting started with the **PLUME** application.

## 2 About PLUME

**PLUME** is a plug-in application of the **SYSTEMA** environment. This manual makes therefore reference to the modeller, kinematics, mission and processing modules of **SYSTEMA**.


This document presents the **PLUME** software that computes the plume impingement on the spacecraft accounting for the flow fields emitted by one or several thrusters.

This manual contains task-oriented instructions which show you how to use the **PLUME** program. The different computation techniques supported by **PLUME** are also presented.

- **Document issue: 1.0**
- **Software release: PLUME Version 4.5.2**


### 3 Glossary

In this User Manual, some terms have a specific meaning:

Term	Meaning
Local / inherited	 The “local” and “inherited” notions are explained in the <b>SYSTEMA User Guide</b> (Chapter “Geometrical model management”)
Activity	In terms of plume impingement application, the question is to determine whether a surface receives or not particle emitted by thrusters. An inactive surface is considered as not present in the flow field.
Specific Items	The specific items (or virtual nodes) are elements defined in the meshing model, on which results are computed (Groups) or allowing to extend standard geometrical definition (Thrusters).
Referenced shapes	When creating a virtual node, the shapes selected for its definition are called referenced shapes. For a group, this is the list of surfaces on which the user wants to have mean or maximum values. For thrusters, it is the cone by which the flow field is emitted.
Thrusters	A thruster is the surface which will emit the flow field.
Group	A group is a set of surface on which the post-processing of results will be modified.

## 4 Typographic Conventions

The following typographic conventions are used in this manual:

Convention	Description	Example
<i>light blue</i>	Used for the <b>PLUME</b> -related functions	The <b>PLUME</b> application is interfaced with...
<b>Blue</b>	Used for the <b>SYSTEMA</b> -related functions	Within the <b>SYSTEMA</b> processing tab...
	These colours are used in the diagram of Chapter Man-Machine Interface to distinguish the <b>SYSTEMA</b> entities from the <b>PLUME</b> ones.	
<i>Italics</i>	Names of the options related to the current entities or filenames	<b>Fit</b> – <i>Material Filename</i> – <b>Physical properties</b>
Numbered list	Procedure containing the several actions to be performed	<ol style="list-style-type: none"> <li>1. Click the <b>Revert</b> button.</li> <li>2. Enter a new value.</li> </ol>
Bulleted list	Items list	<p>This container will affect the following:</p> <ul style="list-style-type: none"> <li>➤ <b>Forces and torques units</b></li> <li>➤ <b>Thermal fluxes unit</b></li> <li>➤ <b>Mass fluxes unit</b></li> </ul>



**A warning calls attention to information that is essential to run successfully the PLUME application.**



**A caution calls attention to important information which, if not understood, will lead to a conflict or an error.**



*A note provides additional information to emphasize or explain important points in the manual.*



*Cross-reference: prompts the user to refer to a particular section in the manual or to a special document.*

---

\* *Material Filename* is the name of the material which has been loaded during the **SYSTEMA** session.

# **Man-Machine Interface (MMI)**



Page left intentionally blank

# Contents

- 1 Overview..... 5**
  - 1.1 Introduction ..... 5**
  - 1.2 In this Chapter ..... 6**
- 2 How to Perform a Plume Impingement Analysis ..... 7**
- 3 Using the Interactive MODELER Module ..... 8**
  - 3.1 PLUME Material Tab ..... 9**
  - 3.2 Applicative Properties ..... 10**
    - Physical .....10
    - Transparency.....11
    - Computation .....12
    - Configuration .....12
  - 3.3 Importing a SYSEXP File ..... 13**
- 4 Interacting with the Meshing ..... 14**
  - 4.1 Plume Impingement..... 14**
    - Meshing .....14
    - Numbering .....16
  - 4.2 Thrusters specific item ..... 17**
    - Meshing .....17
    - Numbering .....17
    - Plume Properties .....18
  - 4.3 Group specific item ..... 19**
    - Meshing .....19
    - Numbering .....19
    - Plume Properties .....20
- 5 Using the Interactive Kinematics Module ..... 21**
- 6 Using the Interactive MISSION Module ..... 22**
- 7 Using the Interactive PROCESSING Module..... 24**
  - 7.1 Basic Rule..... 25**
- 8 Post-Processing ..... 28**
  - 8.1 How to Display Results in a 3D View ..... 28**
  - 8.2 How to Display Results in a 2D Curve View ..... 29**

Page left intentionally blank

# 1 Overview

## 1.1 Introduction

This chapter describes the general ergonomic principles and basic procedures for using the **PLUME** application into the **SYSTEMA** platform.

The application may be illustrated as follows:

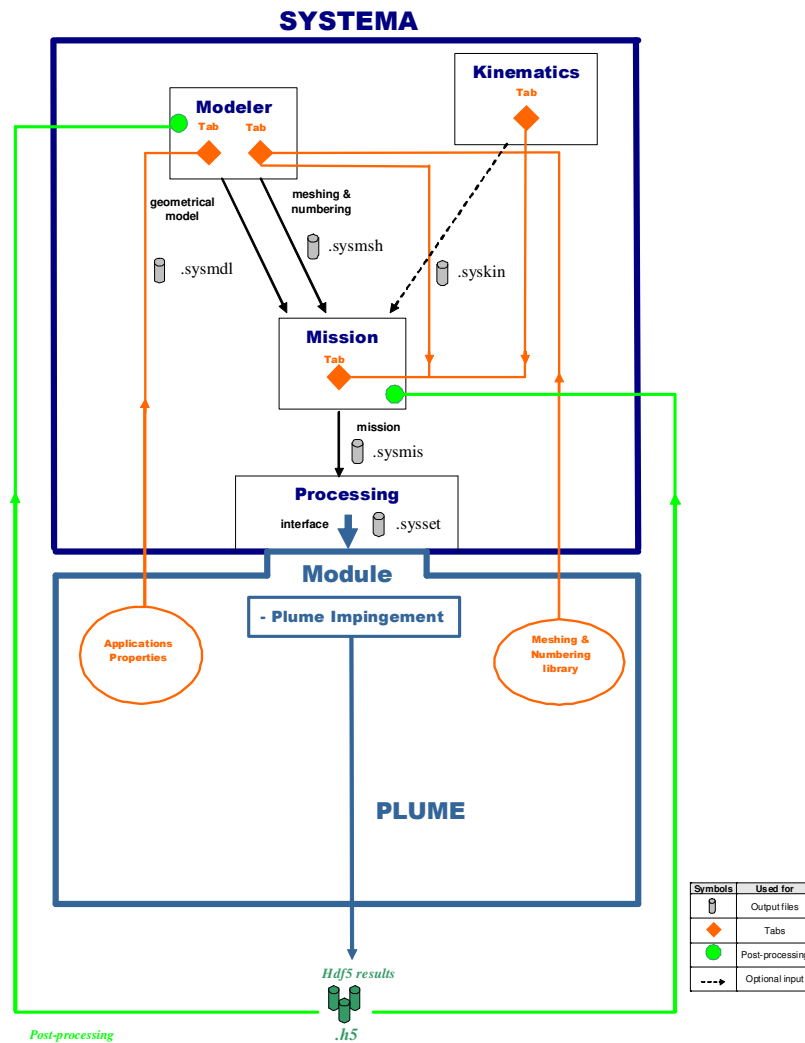


Figure 1 - **PLUME** within **SYSTEMA**

## 1.2 In this Chapter

This chapter covers the following topics:

- **How to perform a plume impingement analysis.**
- **Using the interactive Modeler module.**
- **Meshing and numbering.**
- **Using the interactive Mission module.**
- **Using the interactive Kinematics module.**
- **Using the interactive Processing module.**
- **Post-processing.**

## 2 How to Perform a Plume Impingement Analysis

The **PLUME** application is interfaced with **SYSTEMA**. Consequently, to use the **PLUME** package, the analysis must start with the generation of the geometrical model of the space system.

This first step of the analysis can be performed:

- **Using the interactive 3D MODELER.**

A kinematic can also be designed. This second step can be performed:

- **Using the interactive KINEMATICS module.**

In this case, a mission has also to be designed. This step can be performed:

- **Using the interactive MISSION module.**

The different steps to build a plume impingement analysis are performed:

- **Using the interactive PROCESSING module.**



As shown in the illustration (Figure 1 - **PLUME** within **SYSTEMA**), some **SYSTEMA** modules (Modeler & Processing) are completed by **PLUME** specific tabs.

Within the **SYSTEMA** processing tab, the **PLUME** application is organized into one module, which is based on 1 sub-application and used to manage the **PLUME** entities:

**Plume Impingement:** this module is used to calculate the dynamic effects (forces and torques), thermal effects (heat flux) and contamination (incident and deposited mass).



The above-listed modules are shown on (Figure 10 - **PLUME** modules).

## 3 Using the Interactive MODELER Module

The Modeler tab is used to work on models and meshing. According to the **SYSTEMA** configuration, some applications can be interfaced with **SYSTEMA**. In this case, the elements of the geometrical model have application-dependent properties.

These properties can be consulted or modified using the edition window, via a specific tab (called “**PLUME** tab” in this document). Each application has its own tab.

For a given application, material files may be required to define the application-dependent properties. These files are declared at the root level of the geometrical model.



*Refer to SYSTEMA User Guide (Chapter 4: “Geometrical Model Management”).*

### 3.1 PLUME Material Tab

The user defines material associations via the specific tab of the edition window. **PLUME** is provided with a default reference material. However, the user may want to use its own material database. In the following figure, the user can associate a material file with the application-dependent properties.

#### Example

The Material tab (Figure 2 - **PLUME** material tab) lets the user enter a material filename for the physical properties.

To get access to this window, open the edit window at the root level and unselect “Use default material database”.

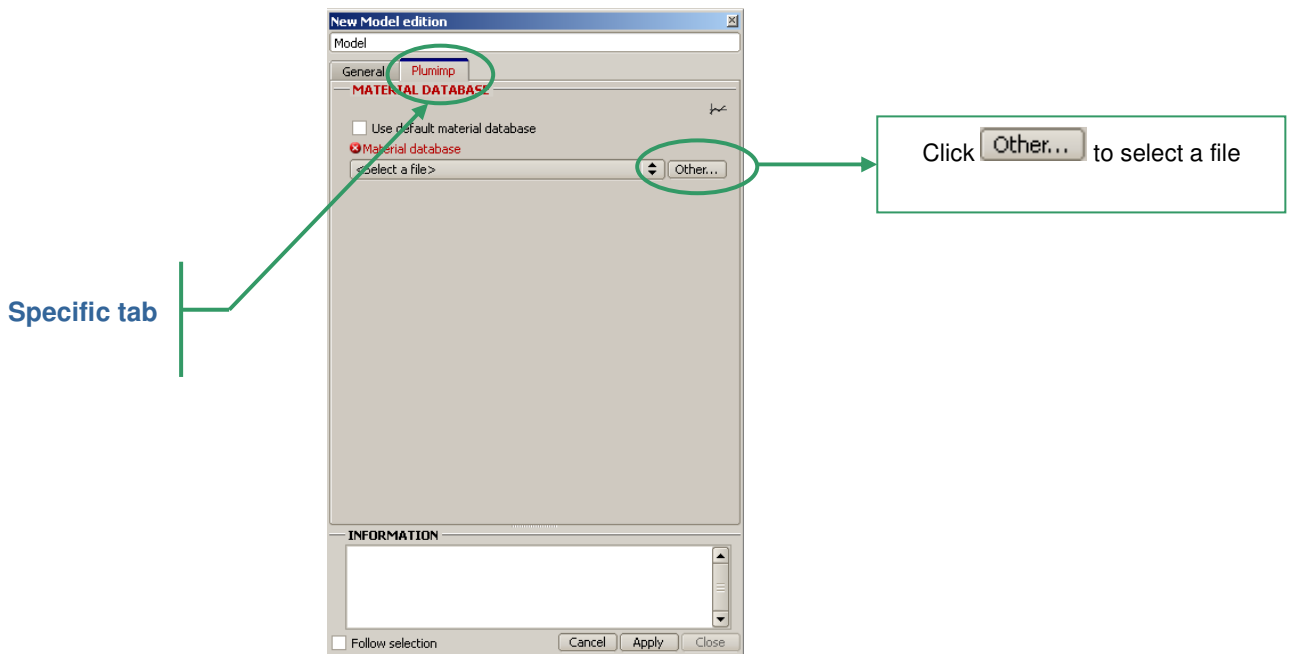


Figure 2 - **PLUME** material tab



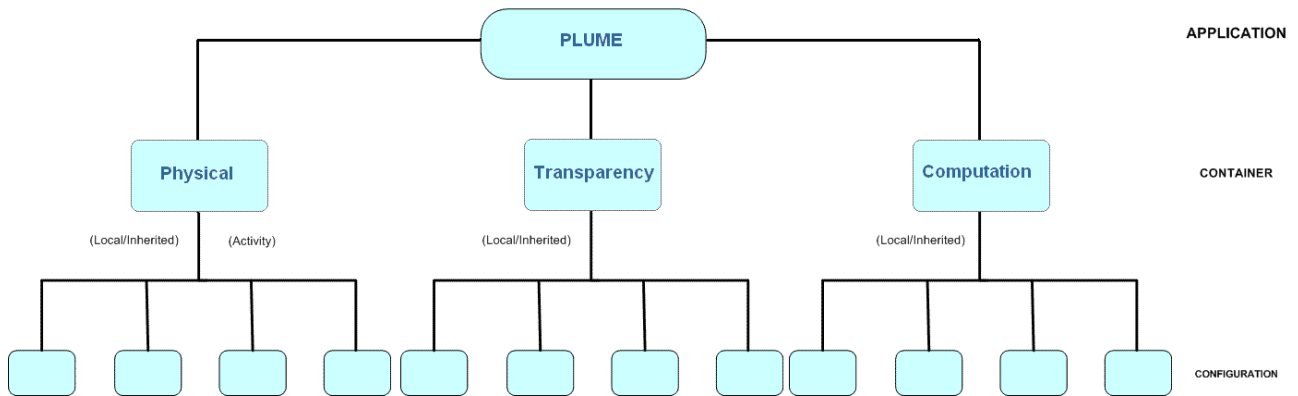
The “.sysmtr” file contains properties of the materials used in the model. A material editor accessible from **material sub-tab** in the **SYSTEMA** modeller tab allows to create or modify the material file. The materials are visible as a green bulleted list.



**PLUME** is delivered with a reference material database. By default, all newly created models will be linked to this reference material database. Use the above procedure to attach or create a user defined material database.



### 3.2 Applicative Properties



There are 3 property clauses called "containers". These containers (PHYSICAL, TRANSPARENCY and COMPUTATION) include applicative properties sub-groups.

Some of these sub-groups may globally be assigned by a material.

#### *Physical*

This container refers to the material settings.

#### Related properties

The properties linked to this container are as follows:

- **Dynamic effect (Absorptivity, Dynamic Specularity, Dynamic Accomodation).**
- **State (Temperature).**
- **Thermal effect (Thermal Specularity, Thermal Accomodation).**

**Example**

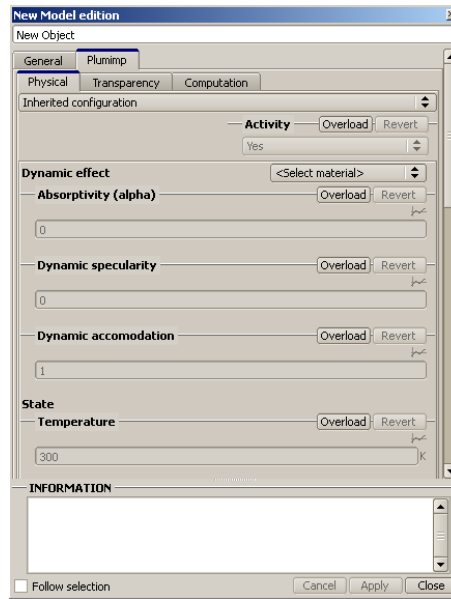


Figure 3 - Object edition

**Activity**

Here, the question is:

- **Is the surface taken into account in the plume impingement calculation?**

If the user wants the surface to be taken into account in the calculation, the Activity has to be set to “yes”. Otherwise, the Activity has to be set to “no”. The activity is thus the answer to the question.

**Transparency**

This container refers to the transparency material settings.

**Related properties**

The properties linked to this container are as follows:

- **Transparency (Phase shift and seven pairs of coefficient/exponent)**

**Activity**

No activity.

## Computation

This container refers to ray tracing algorithm. The Emission property switch allows to enable computation on a surface. When choosing “shadowing” the plume impingement on the surface will not be computed, but the ray-tracing algorithm will consider the surface when computing plume impingement for other surfaces (shadowing effect).

### Related properties

The properties linked to this container are as follows:

- **Ray emission (Emission, ray building method, ray number per mesh or ray density and maximum/minimum number of rays per mesh)**

### Activity

No activity.

## Configuration



Refer to *SYSTEMA User Guide (Chapter 4: “Geometrical Model Management”)*.

The tabs in **PLUME** have inheritance properties which are identical to those of **SYSTEMA**. It is thus possible to redefine a configuration locally (inheritance removed).

### Example

You can enter the value at the higher level and make the next level inherit this value (default behaviour).



*The parameter values can be modified even when in an inheritance configuration.*

In order of priority, a parameter may be affected by:

- **Value overloading (local value).**
- **Material overloading.**
- **Inherited value (value inherited from the father).**
- **Default value.**



*The customization option ‘Overload’ lets the user overload the material’s default value.*



Refer to *SYSTEMA User Guide (Chapter 4: “Geometrical Model Management”)*.

### 3.3 Importing a SYSEXP File

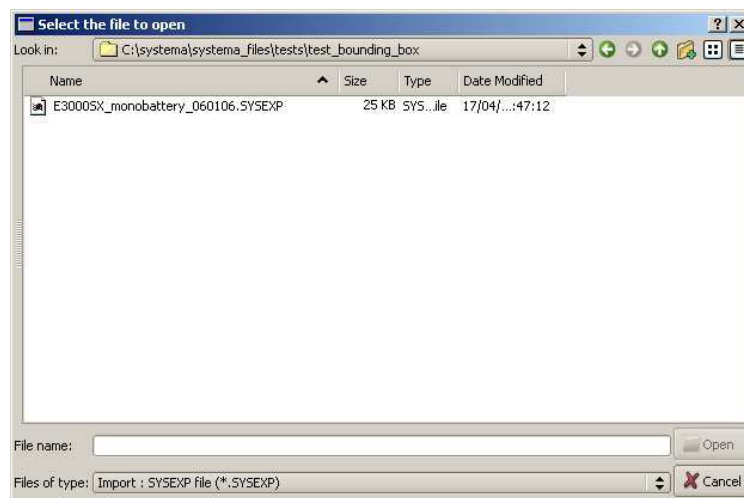
This function allows files (file.SYSEXP) from a previous **PLUME** application (version 3) to be imported and to be compatible with a new **PLUME** application.

To open the existing model:

1. Open the **Modeler** tab in **SYSTEMA**.
2. To open the existing model, click the **File -> Open** menu.

The **Select the file to open** dialog box opens:

#### Example



3. In “Files of type”, select **Import: SYSEXP file (\*.SYSEXP)**.
4. Open the “\*.SYSEXP” file.

The model is displayed in the 3D view in the **PLUME** application.

You can work with the file.

5. Click the **File -> Save** menu to save the file as **.sysmdl** (V4 format).

## 4 Interacting with the Meshing

The entire geometrical model has to be meshed to perform a [plume impingement](#) analysis.

The meshing may also contains additional information. These additional information are stored in specific items, either Group specific items or Thrusters specific items. In this case, the user has to provide **PLUME** properties to the created specific items. The Thrusters specific items allow to define the emission point of the ray tracing, whereas the Group specific items may allow to modify the content of the output files to help the user in obtaining macroscopic results.

### 4.1 Plume Impingement

#### *Meshing*

The meshing is associated with an inheritance property and can be accessed from the tree structure.

- **Application name:**  
**Determines the application for which the meshing is performed (useful in case **PLUME** is not the sole installed application).**



*Refer to SYSTEMA User Guide (Chapter 4: “Geometrical Model Management”).*

The meshing of shapes consists in splitting the surfaces according to a certain number of meshes in each principal direction: this is the “a, b & c” meshing library.

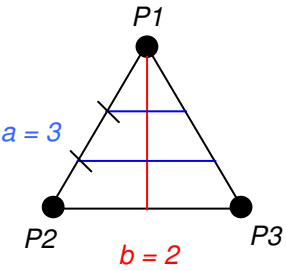
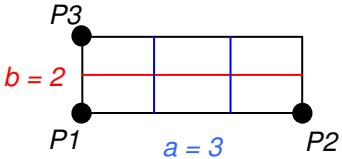
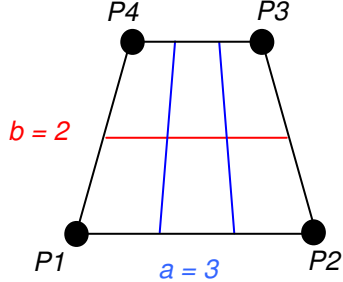
For the **PLUME** application, there is two ways of setting the “a, b & c” parameters: either directly, or by choosing a Length. In this case, the “a, b & c” parameters will be adjusted so that the characteristic size of the meshes correspond to the chosen length. For revolution shapes, a second parameter is available: the maximum angle which allows to define a minimum angular sector to mesh with (“a” parameter).

The three parameters: “a”, “b”, “c” define the number of desired intervals in a specific direction.

- **Parameters “a” & “b”:** refer to all two-dimensional elemental surfaces.
- **Parameter “c”:** refers to three-dimensional surfaces.

The following shapes may be distinguished:

- **Triangle, rectangle, quadrangle**

Triangle	Rectangle	Quadrangle
		

- **Disk, cylinder, cone, parabola, antenna, sphere**

Parameter	Segmentation based on:
a	angular sector

Parameter	Shape	Segmentation based on:
b	disk	radius
	cone, cylinder	height
	sphere, parabola	curvilinear abscissa in iso
	antenna	radius

➤ **Polygons**

- The meshing corresponds to the number of desired intervals in a specific direction. This does not apply to the polygon which is a particular case because it is divided in triangles and, consequently, only the “a” parameter will have an impact as regards the detail of surface triangulation.

➤ **Shape macros and shapes with third dimension**

- Parameters “a” & “b” refer to the basic meshing (see above described categories)
- The “c” parameter completes the meshing in the third dimension.

**Numbering**

It consists in numbering the surfaces for assigning a number to meshes and defining an increment value. Actually, it is an incrementation used for the global numbering of the model or a sub-part of the model to obtain a mesh network with a meaningful numbering system.



**SYSTEMA** provides an automatic numbering sufficient for a **PLUME** calculation.



*Inheritance notion: for a numbering, an incrementation is defined.*

Example

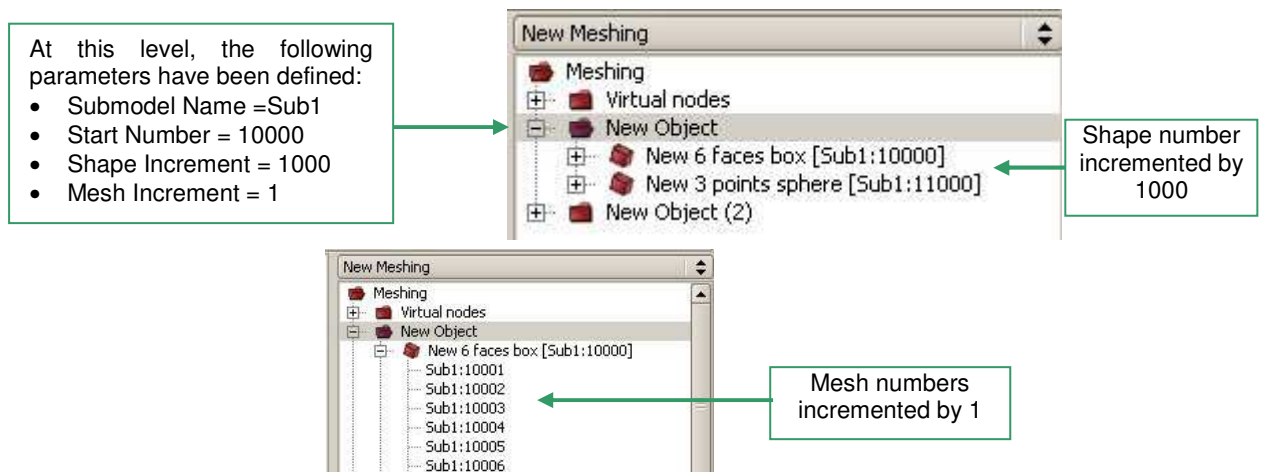


Figure 4 - Inheritance principle for numbering exemple

## 4.2 Thrusters specific item

### Meshing

The Thrusters specific item is the way to define thrusters for the plume impingement analysis.

It is defined in the **meshing model** by one referenced shapes which must be a cone. The emission point will be the center of the cone's base having the larger radius. The orientation of the thruster is deduced from the orientation of the geometrical cone shape.

#### To create a thruster from selection:

1. Open the required meshing model.
2. Select one cone representing a thruster.
3. Via the right click menu, select **Create specific meshing item -> Thruster**.
4. A window opens (see Figure below). In the General Tab, in clicking on Display Browser, the user can check which shape is referenced.

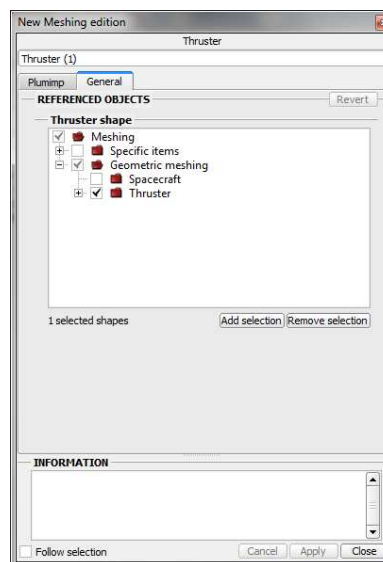


Figure 5 - Thruster referenced shape setting

5. Set the thruster properties in the **Plumimp** tab: Thruster files and Firing (Activity, droplet density, firing duration), see below.

### Numbering

At the thruster creation, a default numbering is performed by **SYSTEMA**.



## Plume Properties

The properties of the Thruster specific item are the following:

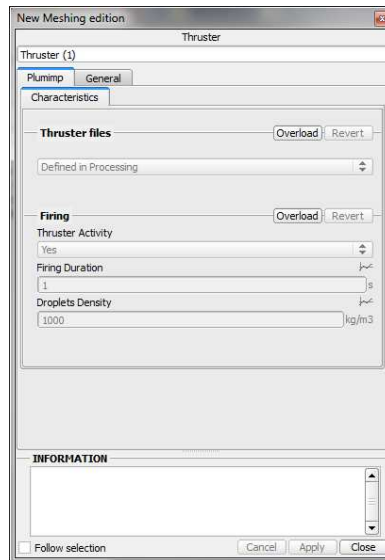


Figure 6 - Thruster **PLUME** properties setting

## 4.3 Group specific item

### Meshing

The group specific item is a way to define a set of shapes on which plume impingement results will be cumulated (mean or max values of some quantities).

It is defined in the **meshing model** by a set of referenced shapes which must be the set of shapes to be accounted for mean or max computation.

#### To create a group from selection:

1. Open the required meshing model.
2. Select one or several shapes of the geometrical model representing the set of shapes of the group to account for.
3. Via the right click menu, select **Create specific meshing item -> group**.
4. A window opens (see Figure below). In the General Tab, it can be checked that the correct shapes are referenced.



Figure 7 - Group referenced shapes setting

5. Set the volume property in the **PLUME** tab, see below.

### Numbering

At the group creation, a default numbering is performed by **SYSTEMA**.

## Plume Properties

The **PLUME** properties of a Group specific item are a set of choice depending on what has to be computed on the set of surfaces.

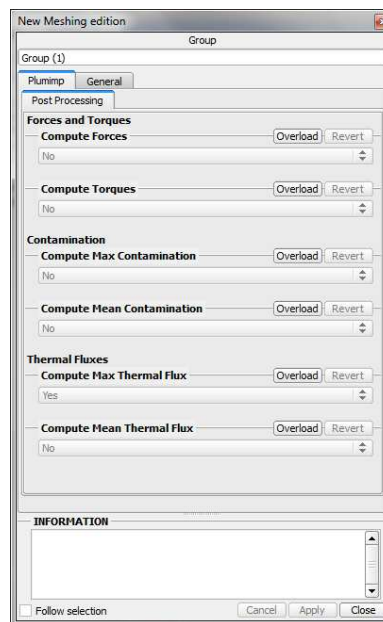


Figure 8 - Group properties settings

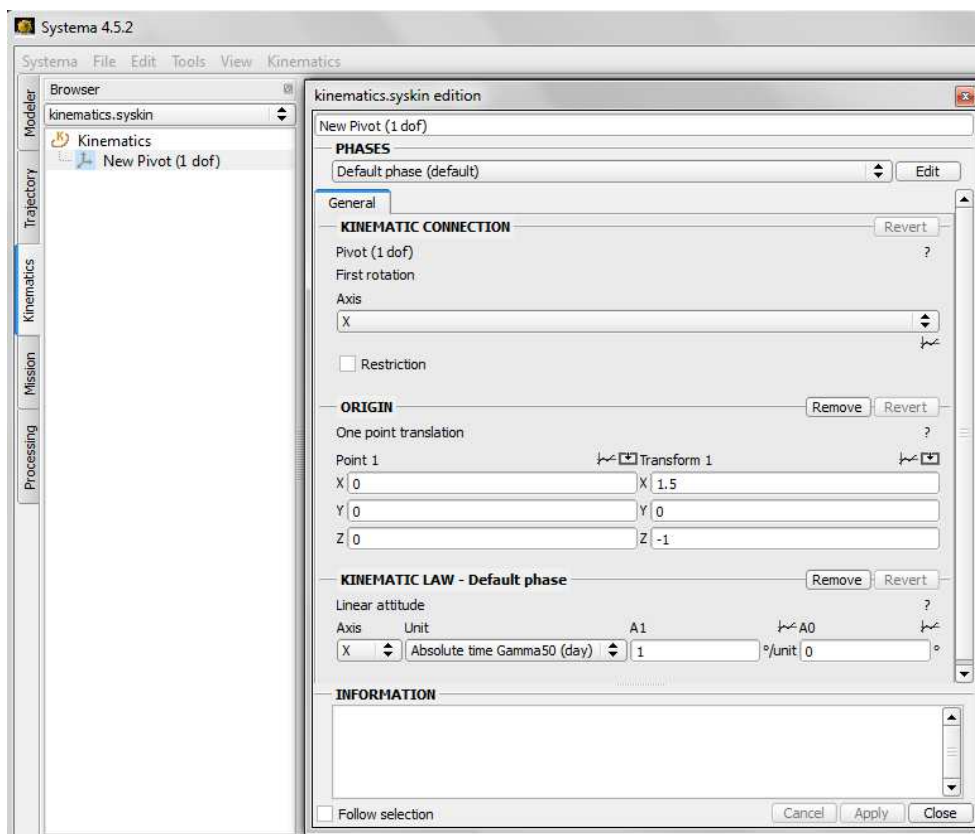
## 5 Using the Interactive Kinematics Module

This Kinematics module allows to define new reference frames with respect to the main reference frame define in the geometrical model. For the **PLUME** application, this allows to simulate rotating solar panel.

### How to create a kinematic:

1. Go to the Kinematics tab
2. Select File -> New in the top menu
3. Following the general rules defined in the **SYSTEMA** user manual, create à new pivot in setting successively:
4. A rotation axis (X, Y or Z)
5. Possibly a transformation to handle the shift between the new reference frame position and the main reference frame. For instance, in the case of a rotating solar panel, a one point translation is sufficient.
6. Choose a Linear attitude and set the corresponding parameters: repeat the rotation axis; choose either “Absolute time Gamma50 (day)” or “Phase elapsed time (sec)” ; a rotating speed (A1) and possibly an initial angle (A0).

### Example

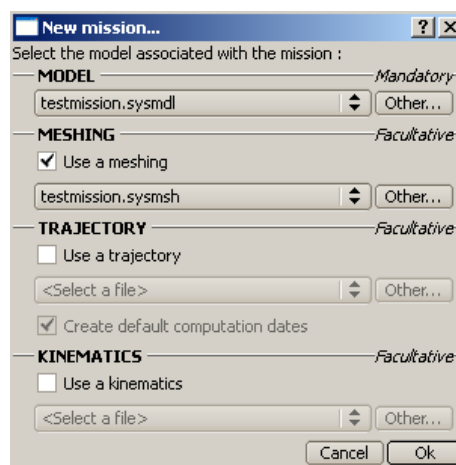


## 6 Using the Interactive MISSION Module

This Mission module is a basic association module (Geometrical model, Meshing model, possibly Kinematics model) making up the application inputs. It is not mandatory to build a mission to perform a plume impingement analysis if there is no kinematics to account for.

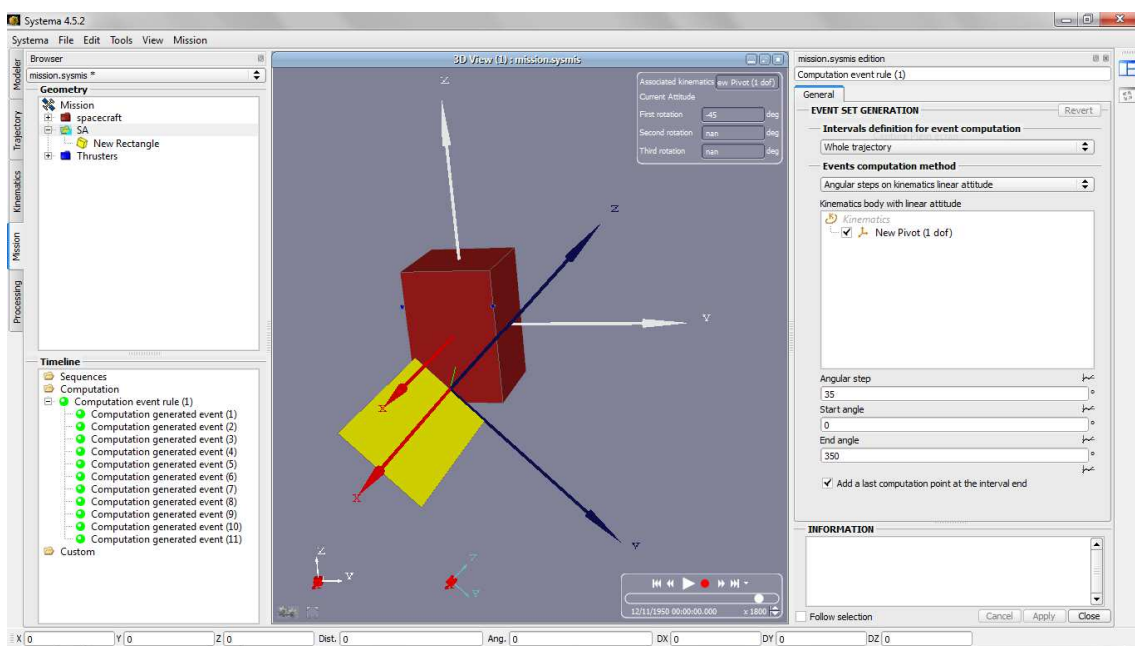
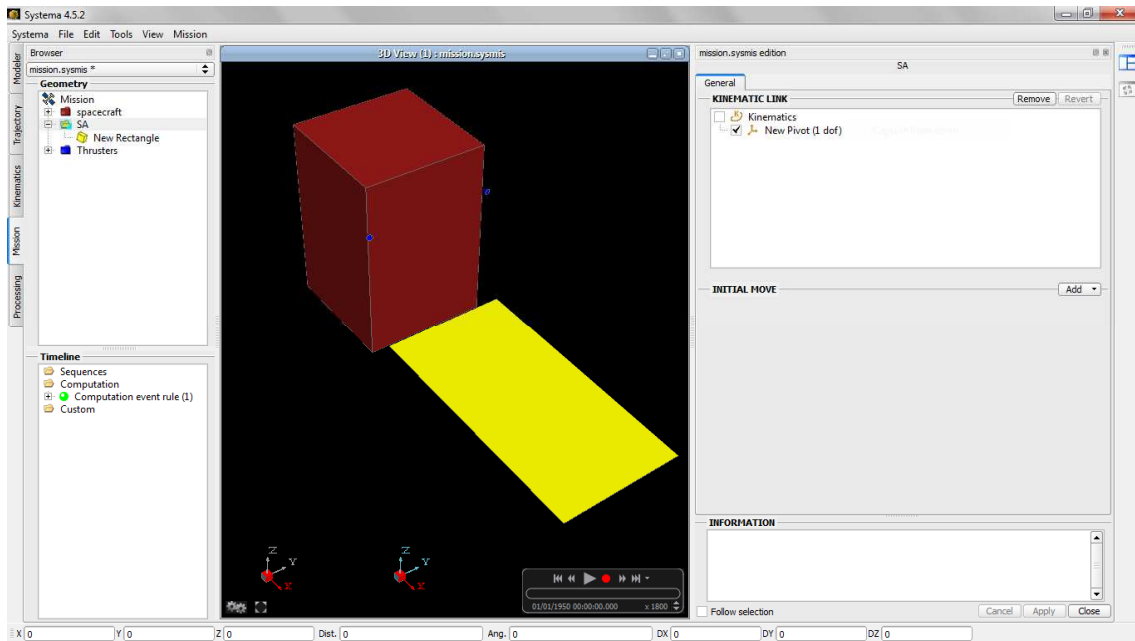
### How to create a mission:

1. Go to the Mission tab
2. File -> New, the following window opens



3. Either click OK if the suggested model, meshing and kinematics are the one defining the desired computation or select the relevant model, meshing and kinematics if necessary. Then click OK
4. Possibly link a kinematic to an object and define the computation points in following the general rules defined in the **SYSTEMA** user manual (see also below example figures).
5. The mission is set and will be available in the processing tab to be attached to a process.

**Example**



## 7 Using the Interactive PROCESSING Module

This module is composed of a sequence of processes, which are specific to the associated application.

A calculation is defined (mission or meshing selected) and performed (run).

The analysis that will be performed using the mission or meshing will output the results based on **Plumimp** module.



Refer to *SYSTEMA 4.5 User Guide (Chapter 8: "Process Management")*.



The modules are not detailed at this stage. In this User Manual, you will find chapters dedicated to the different modules.

### Example

1. Open the **Processing** tab in **SYSTEMA**.
2. To open a new processing, click **File -> New** menu.

Here are the menus proposed in the **SYSTEMA Processing** tab:

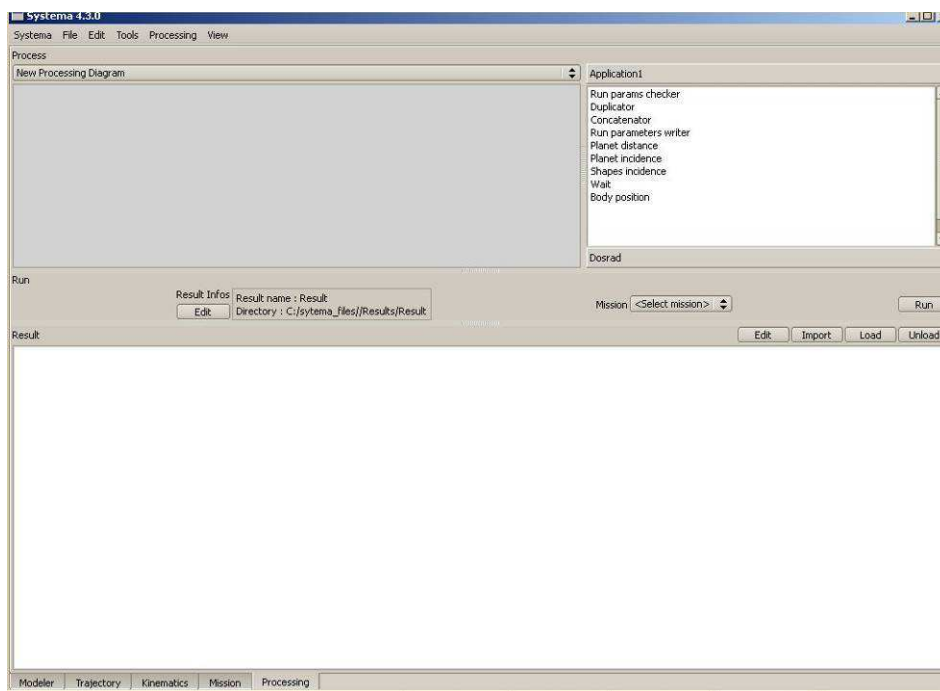


Figure 9 - **SYSTEMA** Processing tab display

## 7.1 Basic Rule

To display the **PLUME** available processes on the process browser:

- ◆ Click **PLUME**.

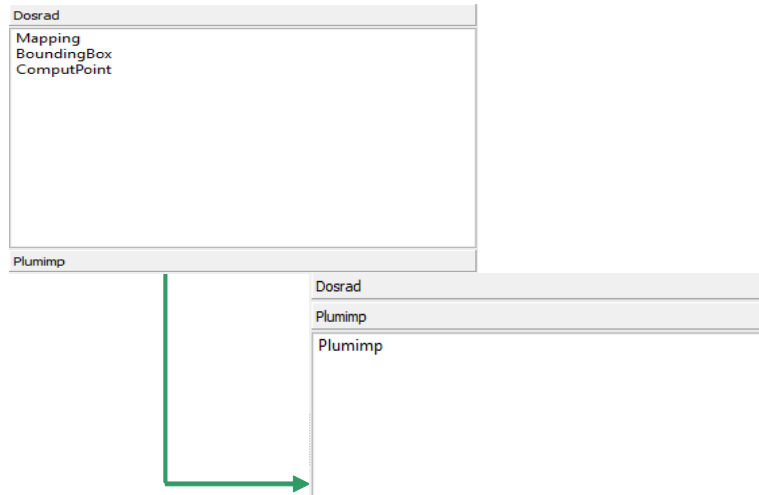


Figure 10 - **PLUME** modules

To display a **PLUME** module:

- ◆ Double click (or Drag and Drop) the selected **PLUME** module (Figure 11 - Plumimp module).

### Example

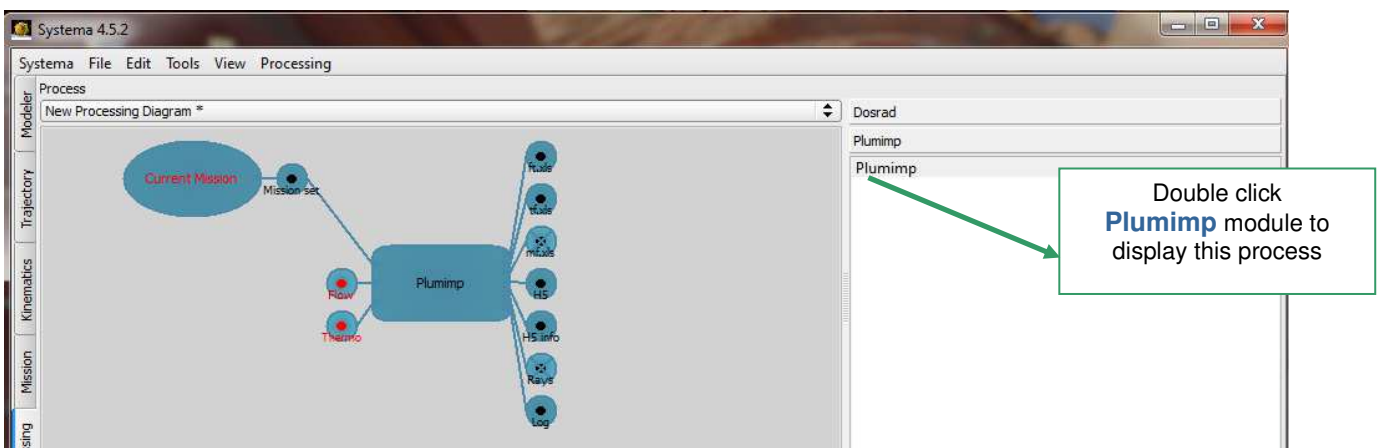


Figure 11 - **Plumimp** module



To display the edit or delete option for one **PLUME** module, as shown on the following window, the user can perform the following actions on the selected **PLUME** module:

- ◆ Double click.
- ◆ Right click.

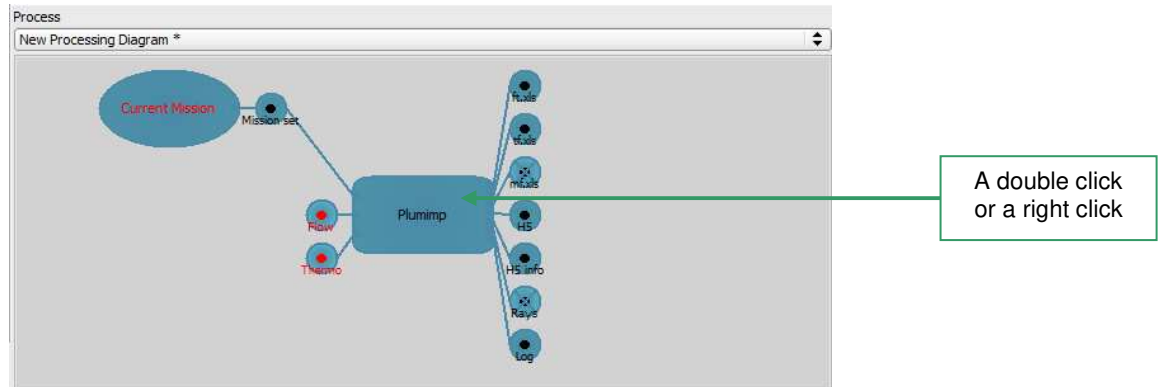


Figure 12 - How to get access to the Edit or Delete option of a **PLUME** module

**Example**

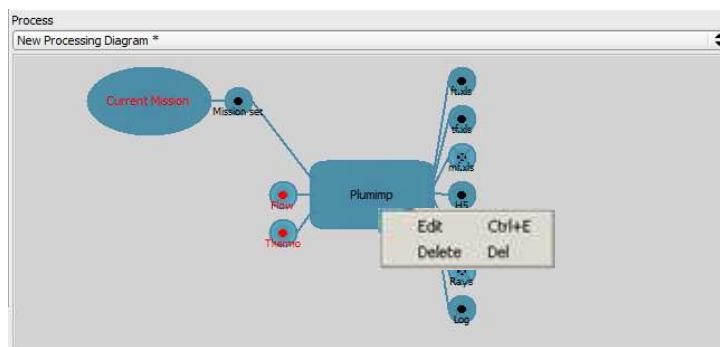


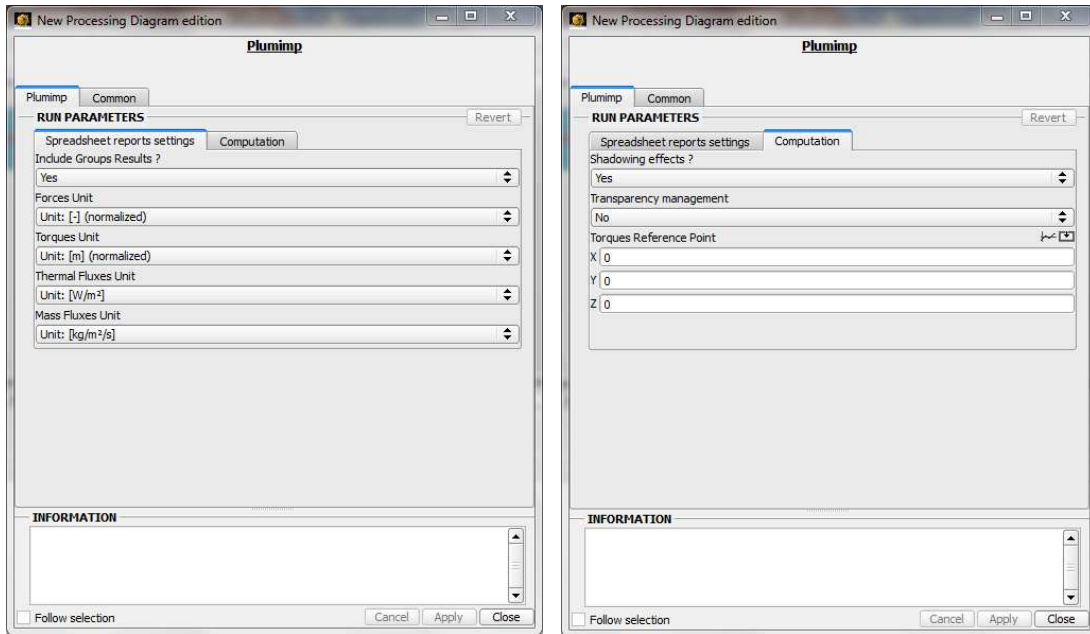
Figure 13 - Edit or delete option of the **PLUME Plumimp** module

- **Using the delete option on any **PLUME** module enables the user to make this module disappear.**
- **As regards the edit option, the **PLUME** module selected has its own run parameters. The run parameters have two tabs:**
  - **Computation:** it allows to select option concerning shadowing and transparency. It is where also the reference point for torques calculation has to be set.

- **Spread sheet report settings:** this allows to tune the results written in the “xls” output files. Note that this settings do not act on the content of the h5 results file

**Example**

**Run parameters available for the Plumimp Module**



## 8 Post-Processing

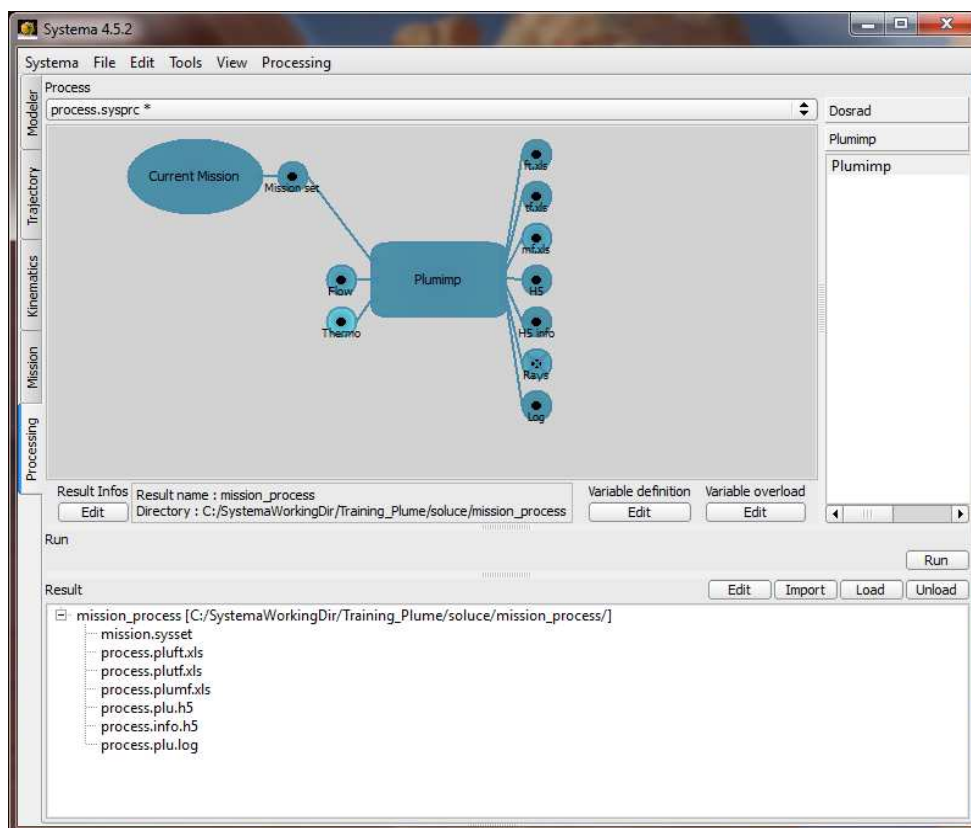


Refer to *SYSTEMA 4.4 User Guide (Chapter 8: “Process Management”, § “Using process result”)*.

The *.h5* files can be used to display the results in the **Modeler** tab.

1. Open the **Processing** tab in *SYSTEMA*.
2. Click **Load** to load the result file (*.sysres*).

### Example

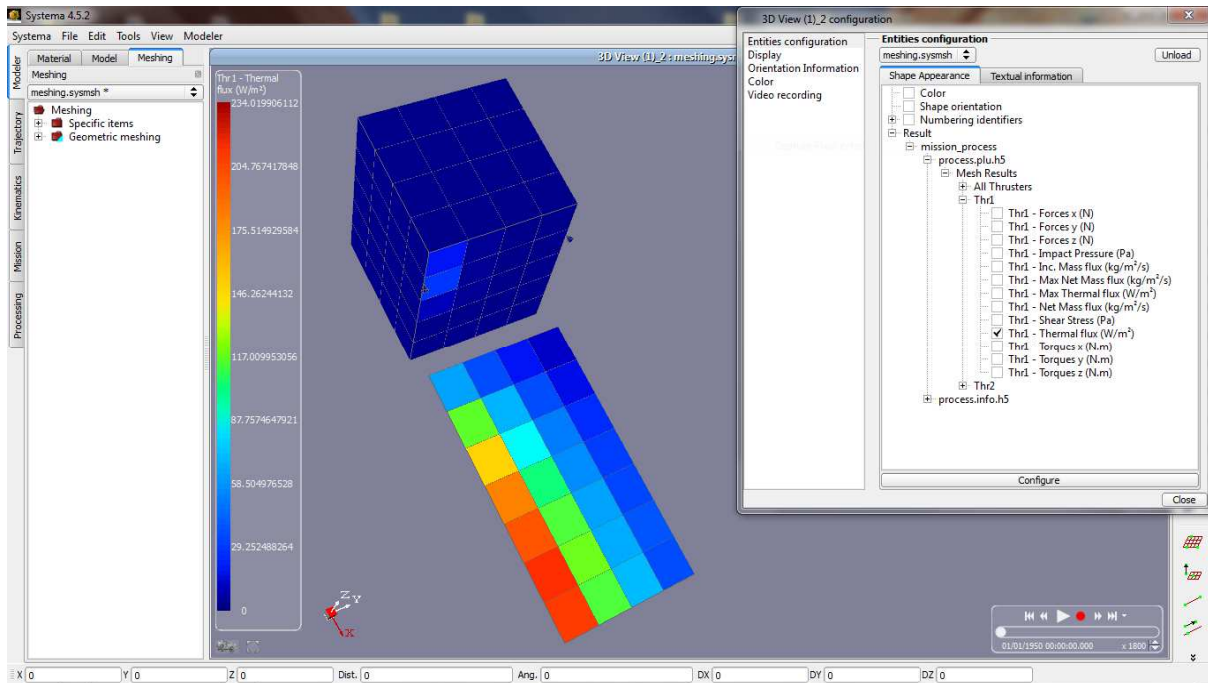


### 8.1 How to Display Results in a 3D View

#### Example

1. Open the **Modeler** tab in *SYSTEMA*.
2. Open the meshing model.
3. In the 3D view, via the right click menu, select **Configuration window**.
4. Select the result file and the property (for example: Thermal Fluxes) you wish to display.
5. Close the configuration window.

In the 3D view, the meshing model is displayed showing the results.

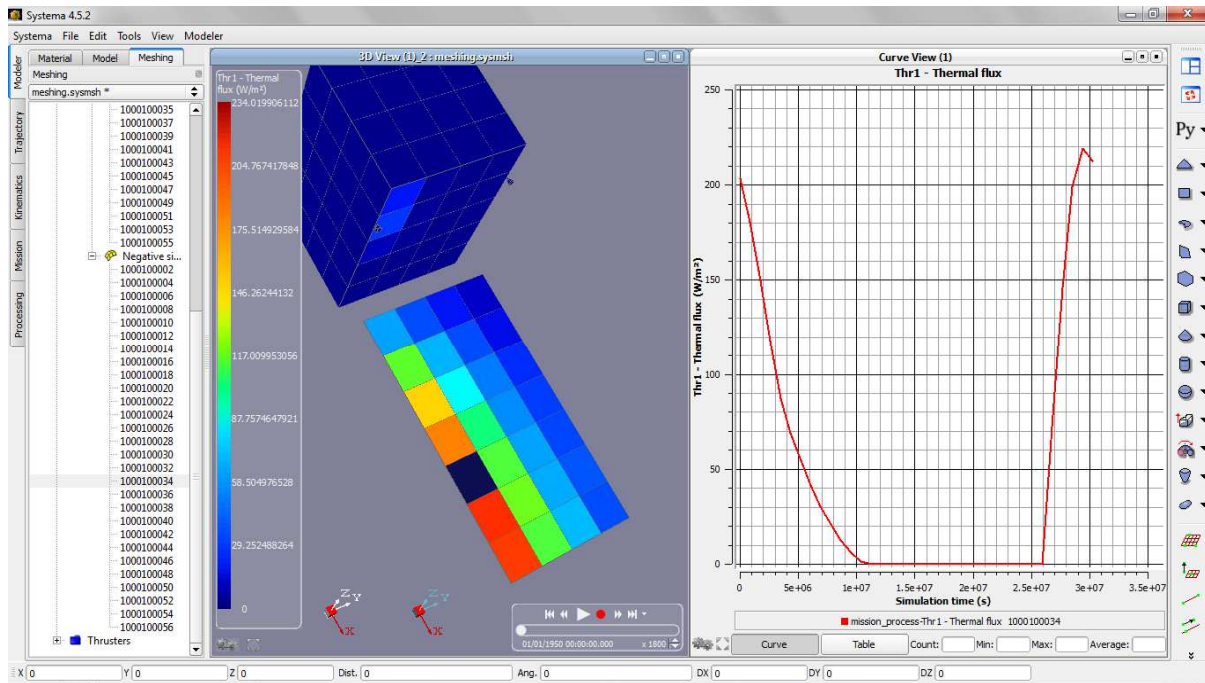


## 8.2 How to Display Results in a 2D Curve View

### Example

1. Open the **Modeler** tab in **SYSTEMA**.
2. Open the meshing model.
3. In the view menu, choose add -> Curve view.
4. Via the right click menu, select **Configuration window** then load the meshing.
5. Select the result file and the property (for example: Thermal Fluxes) you wish to display.
6. Select a node on wish you want to see the quantity evolution according to time
7. Close the configuration window.

In the curve view, the curve is displayed showing the results.



# General Principles

Page left intentionally blank

## Contents

<b>1 Overview.....</b>	<b>5</b>
<b>1.1 Introduction .....</b>	<b>5</b>
<b>2 PLUME-Specific SYSTEMA Tabs .....</b>	<b>6</b>
<b>3 PLUME Module and Corresponding Files .....</b>	<b>8</b>



Page left intentionally blank

# 1 Overview

## 1.1 Introduction

In this chapter the user will find some general principles that he has to know before using the **PLUME** files and modules.

The **PLUME** software predicts the plume effects in terms of dynamic effects, thermal flux and contamination on any spacecraft.

The increasingly complex design of satellites imposes constraints on thruster's implementation used for orbit and attitude control of spacecraft. The impingement of exhaust products of thrusters may present various detrimental effects:

- Forces and torques caused by the impingement of the gas plume producing dynamic perturbations and a loss of thrusters efficiency.
- Undesirable thermal constraints due to the convective heat flux.
- Contamination due to the deposition of vapours or droplets on the surface of sensitive components such as optics and sensors, affecting their performance.
- Erosion caused by the particles and the impingement of unburnt droplets.

The prevention of those effects on sensitive surfaces that might compromise mission objectives and lifetime, interest more and more the satellite design engineers.

To optimise the positioning of thrusters with respect to the performance efficiency and the plume effects, it is essential to have a comprehensive understanding and quantitative characterisation of the pressure, thermal and contamination effects of the exhaust plume of the thrusters at their typical operating conditions.

The software is intended to provide a tool for evaluating plume impingement received at points within spacecraft geometry.

The main functions of **PLUME** are as follows:

- **Geometry modelling and physical properties & meshing.**
- **Physical simulation: plume effects analysis:**
  - **Plume impingement: calculation of the dynamic effects, heat flux and incident mass due to the impingement of the gas ejected by one or several spacecraft thrusters.**

## 2 PLUME-Specific SYSTEMA Tabs

Here the **PLUME** package is fed back to the **SYSTEMA** environment.

The **MODELER** and **PROCESSING** tabs have the same layout.



*A detailed table will be proposed in the relevant MMI related sections to describe each tab.*

### Example

Here are the options proposed in the **Edit** menu of the **SYSTEMA Modeler** tab:

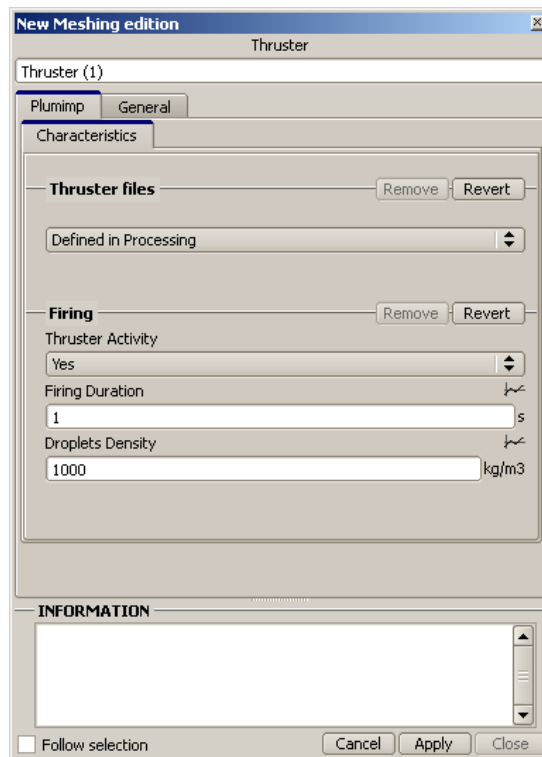

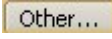


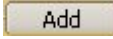

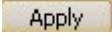


Figure 1 - Options proposed in the **SYSTEMA Modeler** tab

For each tab, the menu consists of both general and specific options.

The standard options are as follows:

**Edit menu (contains options applicable to entity components)**

Option	Used to:
	Enter a specific entity created for <b>PLUME</b>
	Select the file to open
	Remove an entity (property, file, etc...)
	Cancel the display and return to the  option
	<p>Issue a warning or provide information on the processed nodes or the current configuration.</p> <p>The local errors (e.g. a forbidden value) are identified in red. However, some elements cannot be automatically returned in error.</p> <p>If it is not an error to be locally corrected, a warning displays in the lower window. The error is detailed and can be corrected in another area.</p>
	Validate the selected component(s)



The  button is disabled if any error is still present in the information area.

### 3 PLUME Module and Corresponding Files

In this User Manual, the "module" or "PLUME module" terms will be used to refer to the sub-processes managed by the PLUME application, namely *plume impingement*.

The module defines its input files, parameters and output files.

The parameters for the module will be detailed in the chapters dealing with it.

The following tables present the module and corresponding files:

#### INPUT File extension

Module	File extension
<i>Plume Impingement</i>	".sysset" ".flow", ".thermo"

#### OUTPUT File extension

Module	File extension
<i>Plume Impingement</i>	".plu.h5", ".info.h5" ".plu.log" ".pluft.xls", ".plutf.xls", "plumf.xls"



#### **INPUT file extension**

The ".sysset" file refers to a mission defined in the **SYSTEMA V4** platform.

The ".flow" and ".thermo" files contain the flow field emitted by the thrusters and the corresponding thermodynamic tables. These are files produced by the Plumflow application.

#### **OUTPUT file extension**

The ".h5" outputs are HDF5 files used for result interpretation. They can also be loaded into an HDF explorer.

The ".log" file provides general information.

The ".xls" files output by the module are results written under HTML format, readable under an internet browser or a spread sheet report editor.

Page left intentionally blank

# Computation methods

Page left intentionally blank



# Contents

- 1 General Method: Plume Impingement Computation ..... 5**
  - 1.1 Overview ..... 5**
  - 1.2 Thruster definition ..... 6**
  - 1.3 Group definition ..... 6**
- 2 Plume Impingement ..... 8**
  - 2.1 Theory ..... 8**
    - Introduction ..... 8
    - Principles ..... 8
    - Free molecular regime: Knudsen model ..... 11
    - Transparency ..... 12
  - 2.2 Parameters ..... 13**
  - 2.3 Input / Output ..... 14**
    - Input ..... 14
    - Output ..... 14
- 3 Content of the result files ..... 15**
  - 3.1 Impingement information file ..... 15**
    - General information data ..... 15
    - Ray tracing data ..... 15
    - “From flow file” data ..... 15
  - 3.2 Derivate quantities file ..... 17**
    - Results sorting ..... 17
    - Maximum quantities ..... 17
  - 3.3 Spread sheet report files ..... 18**
    - Created folder structure ..... 18
    - File organisation ..... 19

Page left intentionally blank

# 1 General Method: Plume Impingement Computation

## 1.1 Overview

The **PLUME** software evaluates the impact of one or several firing thrusters on the spacecraft geometry.

The analysis is based on an analytical ray-tracing technique. Rays are traced from the thruster of interest to all the meshes of the geometry. Each ray provides information regarding the current mesh and the thruster: incidence angle, distance to the thruster, etc... If another surface is encountered, the current mesh is said to be **shadowed**. Otherwise, the impact parameters as well as the derivate quantities are computed, cumulated (on meshes, on surfaces and possibly on groups) and stored in the corresponding file: in the Impingement information file for the impact parameters and in the Derivate quantities file for the other quantities.

The impact parameters on the **non-shadowed** meshes are calculated as following:

- **For each of the launched rays, the distance and elevation angle with respect to the thruster axis are computed.**

Thanks to these information, the local undisturbed thruster flow and thermodynamic is interpolated in the FLOW and THERMO input files computed by the Plumflow module. The interpolation is made by using the meshed flow field contained in the FLOW input file.

This procedure is approximate, in that it ignores the effects that the presence of the surface could have had on the thruster flow field.

- **For each of the launched rays, from the properties of the surface and the impact parameters, the impact pressure, shear stress, thermal flux and mass flux are computed.**

Then the force, torques, thermal and mass fluxes are computed according to the chosen Impact model (only Knudsen model implemented, corresponding to the free molecular regime assumption). This computation is an integration over an elementary area: for instance the force vector is the integration of the vector formed by the impact pressure (normal contribution) and the shear stress (tangential contribution).

- **The quantities are cumulated on the mesh, on the surface to which the mesh belongs, possibly on the group to which the surface belongs. The quantities are also cumulated on the overall model.**

This accumulation is also an integration process: for instance the force contribution on a surface is the integral of the impact pressure and shear stress over this surface, which is decomposed on the mesh contribution (simple sum).

## 1.2 Thruster definition

The **PLUME** software has to be aware of which cone shapes are supposed to represent the thrusters. This association is done in using the Thruster specific item.

Apart from this emitting shape, a thruster is also defined by a pair of input files: the FLOW file and the THERMO file. The first one provides the flow field on some mesh. This flow field is defined by a set of quantities: density, velocity and pressure of the gas flow. The mandatory quantity to be present in the FLOW file are:

- The density (label "Density")
- The temperature (label "Temperature")
- The velocity norm (label "Velocity Module" or "Velocity module")
- The velocity local angle (label "Velocity Angle" or "Velocity vector")

All the other quantities present in the FLOW file are loaded but not used during computation.

The second file provides the thermodynamic relationship between the temperature and the pressure as well as other gas properties (molar mass, etc...). These two files are defined either at thruster definition level if each thruster has a different flow field or at processing level if a set of thruster file can be shared between thrusters.

These first two information (cone position and flow field) allows to compute the impact parameters, as well as the forces / torques and the thermal flux on every meshes. This also sufficient to compute the incident mass flow rate expressed in  $\text{kg/m}^2/\text{s}$ .

To compute a mass flux expressed in  $\text{kg/m}^2$  (or in  $\mu\text{g/cm}^2$ ), the firing duration is necessary, assuming that the flow field is stationary. Then, to compute a mass deposit expressed in m (or in  $\mu\text{m}$ ), a density is necessary. These two additional parameters have to be set during the thruster definition if the user want to compute mass flux or mass deposit on the surfaces of the spacecraft. The incident mass flow rate (or flux, or deposit) is obtained by multiplying the local incident mass flow rate by the local absorption coefficient.

## 1.3 Group definition

During the computation, the different quantities are cumulated on meshes. Then the mesh results are cumulated on surfaces and finally the surface results are cumulated on the overall spacecraft geometry. The integrated information at spacecraft level is generally sufficient, particularly for forces and torques evaluation.

On the other hand, if the user wants to have integrated quantities on a reduce part of the spacecraft, the surface level may be too fine. Conversely, the overall spacecraft level may be too coarse: an intermediate level of detail could be interesting. That is why the concept of groups has been naturally introduced.

A group is thus a set of surfaces on which integrated quantities are cumulated (by cumulating the integrated quantities on the corresponding surfaces). The user can define which quantities he wants to store: force and torques, thermal flux and/or mass flux and for each of them, he can decide whether to store or not the mean and the maximum of this value.

The corresponding group quantities are stored in the h5 plume impingement file and are available during post-processing for curve display (or textual information). They are also stored in spread sheet report file for direct text access.

One folder is created for each of the thruster present in the computation and the corresponding spread sheet report files are stored inside this specific folder. The unit of the quantities contained in these files are set by the module processing parameters.

## 2 Plume Impingement

### 2.1 Theory

#### *Introduction*

Owing to the great complexity of the 3D flow due to the impingement of a plume on spacecraft geometry, only local methods are applicable in the context of engineering computations. The main features of those methods are:

- Only suited to hypersonic or free molecular flows,
- Only local flow: the impingement at a point ignores the surrounding impingement,
- Only the surfaces in the direct field of view receive the impingement.

The local method depends on the impingement regime, although for the moment, only the free molecular regime is implemented (Knudsen model).

#### *Principles*

The general principle of MATPLIMP computational scheme is presented hereafter.

##### **Modelling**

First, the body is described as a collection of analytical shapes. Each surface is then discretized into simple elements, small enough for the local flow to be considered as uniform. The area and the oriented normal of each element are computed by means of the diagonal cross products.

##### **Local flow computation**

To locate the nozzle exhausting the gas, two points are required:

- The exit section centre,
- A point on the axis.

Because the flow-field file is defined in a reference associated with the nozzle, the coordinates of each element centre are first transformed into the nozzle coordinate system.

The flow-field file generated by PLUMFLOW is ordered in a regular meshing formed by quadrangular cells. For each cell vertex, the following data are supplied:

- $X_p/R^*$ ,  $R_p/R^*$ : coordinates normalized by the throat radius,
- T: static temperature (K),
- $\rho$ : gas density (kg/m<sup>2</sup>),
- V: velocity modules (m/s),
- $\alpha$ : angle between the velocity vector and the plume axis

The computation of the impact parameters is performed as followed

(a) The first step is to know which plume cell contains the element centre. In that view, access tables are created, allowing to limit this searching to some cells only.

(b) The following step is to compute the flow parameters at the point M by a classical bilinear interpolation.

(c) The last step is to compute the unit vector giving the direction of the gas velocity and the tangential unit vector, then the angle  $\theta$  between the velocity and the vector normal to the element.

### **Shading effects**

In the case of complex geometries, some surfaces can occult a part of the analyzed geometry. To take into account these shading effects, the program operates as follows:

- Shadow effects are computed by testing the intersection of the streamline passing through the element centre and all the analytical surfaces making the geometrical model.
- The curvature of the streamlines is neglected (only valid if the model is not too close to the nozzle). That is to say the streamlines are assumed to be straight lines and a ray tracing algorithm is used to determine if there is shadowing or not.

Note: the pressure is supposed to be equal to zero (space vacuum) on the part of the geometry occulted from the plume.

### **Impingement computation**

The equation used to compute the impingement effects are:

- Forces and torques:

Forces and torques are derived from the impingement pressure and the tangential stress:

$$\vec{F} = \sum (P_{imp} \cdot \vec{n} + \tau \cdot \vec{X}_t) a \cdot (1 - Transp)$$

$$\vec{C} = \sum (P_{imp} \cdot \vec{OM} \times \vec{n} + \tau \cdot \vec{OM} \times \vec{X}_t) a \cdot (1 - Transp)$$

Where:

$\vec{F}$ ,  $\vec{C}$  are the resulting forces and torques,

M is the element centre,

$\vec{n}$  is the element unit normal,

$\vec{X}_t$  is the tangential unit vector,

a is the element area,

$P_{imp}$ ,  $\tau$  are the pressure and tangential stress,

O is the reference point for torques computation,

Transp is the transparency (see hereafter).

- Thermal flux:

$$Q_w = \sum Q_{loc}$$

$$Q_{moy} = \frac{Q_w}{\sum a.X_i}$$

Where:

$Q_w$  is the received thermal power,

$Q_{loc}$  is the local thermal flux,

$Q_{moy}$  is the mean thermal flux,

$X_i$  is an indicator of the element visibility.

The computation of  $P_{imp}$ ,  $\tau$  and  $Q_{loc}$  is described hereafter.

### **Impingement regime**

The impingement regime is characterised by the Knudsen number, given by:

$$K_n = \frac{\lambda}{L_{ref}}$$

Where:

$L_{ref}$  is the characteristic length,

$\lambda$  is the mean free path, defined by:

$$\lambda = \frac{\mu\sqrt{\pi}}{\rho\sqrt{2RT}}$$

Where:

$\mu$  is the dynamic viscosity,

$\rho$  is the density,

$R$  is the gas constant,

$T$  is the gas temperature.

$L_{ref}$  is often difficult to define. Generally, it represents the length of the impingement body edges for little bodies or the minimal distance between the body and the nozzle, for objects of great size. The mean free path is stored in the Impingement Information file. It can help the user to know whether the free molecular regime is a good approximation.

The computation method is deduced from the flow regime as follows:

If  $K_n < 0.1$ , the regime is continuum: Newton approximation, **not implemented**

If  $10 > K_n > 0.1$ , the regime is transitional: empirical bridging, **not implemented**

If  $K_n > 10$ , the regime is free molecular: gas kinetic theory, see hereafter.



### **Free molecular regime: Knudsen model**

This model is a three parameters model; it is an improvement of the well-known Maxwell model.

The basis of the model is that the molecules impinging a surface can produce three types of interactions:

- The specular reflection, the  $\tau$  parameter (specular reflection ratio) is the ratio between the specularly reflected mass flux and the reflected mass flux,
- The energy accommodation, the  $\alpha$  parameter (wall accommodation ratio) is the ratio between the energy given to the surface by the diffuse reflected molecules and the energy given to the surface if the molecules were diffusely reflected with the surface temperature. It applies only for the diffuse part of the reflected molecules and can be written as:

$$\alpha = \frac{Q_i - Q_d}{Q_i - Q_w}$$

Where  $Q_i$  is the energy of an incident molecule,  $Q_d$  is the energy of a diffuse reflected molecule and  $Q_w$  the energy of a molecule reflected diffusely at the wall (surface) temperature.

For a mass flux  $\dot{m}$  impinging a surface, the Knudsen model assumes that:

- The fraction  $\tau\dot{m}$  is specularly reflected,
- The fraction  $(1-\tau)\dot{m}$  is diffusely reflected with an energetic accommodation defined by the  $\alpha$  parameter.

The effects that are computed are:

- The impingement pressure:

$$P_{imp} = P_\infty \left\{ A \left[ w(1 + \tau) + \alpha(1 - \tau) \sqrt{\frac{T_w}{T_\infty}} + \frac{2}{3}(1 - \alpha)(1 - \tau) \right] + (1 + \tau) \left( \frac{1 + \operatorname{erf}(w)}{2} \right) \right\}$$

- The heat flux:

$$Q = (1 - \tau_T) \alpha_T P_\infty \sqrt{\frac{RT_\infty}{2}} \left\{ A \left[ S^2 + \frac{1}{RT_\infty} \int_{T_w}^{T_\infty} C_p \cdot dT - \frac{\left(1 - \frac{T_w}{T_\infty}\right)}{2} \right] + w \left( \frac{1 + \operatorname{erf}(w)}{2} \right) \right\}$$

- The tangential stress:

$$\sigma = P_\infty (1 - \tau_p) A S \cos \theta$$

Where:

$$S = \frac{V_\infty}{\sqrt{2RT_\infty}}$$

is the molecular Mach number

$$A = \frac{1}{\sqrt{\pi}} \exp(-w^2) + w(1 + \operatorname{erf}(w))$$

$$w = S \cdot \sin(\theta)$$

### **Transparency**

A transparency *Transp* can be used for certain materials. It is computed with the following expression:

$$Transp = \sum_{i=1}^{NBTERM} z_i \cos^{\exp_i}(\theta - \theta_0)$$

Where:

NBTERM: Number of terms of the polynomial

$z_i$ : Coefficient of the  $i^{\text{th}}$  term

$\exp_i$ : Exponent of the  $i^{\text{th}}$  term

$\theta$ : Incident angle wrt the normal to the surface

$\theta_0$ : Phase angle

## 2.2 Parameters

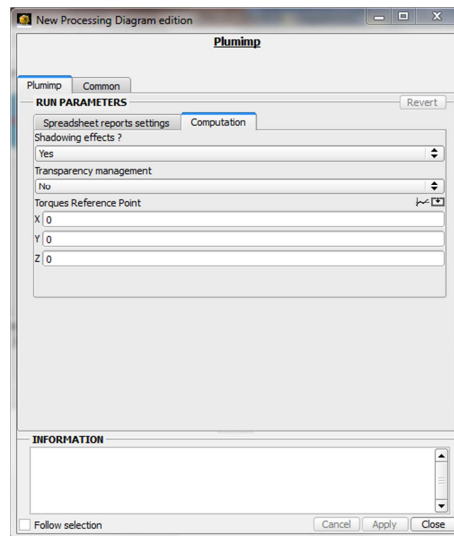
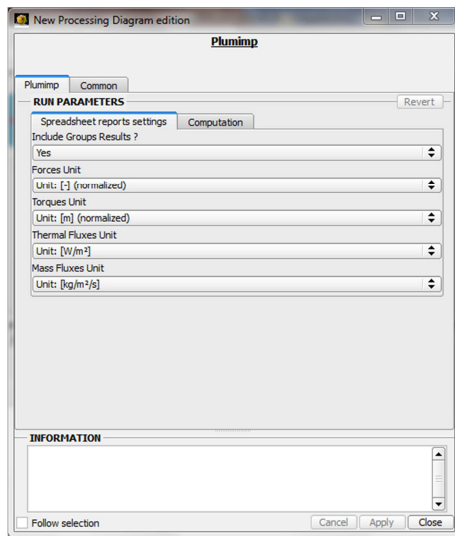
This section gives an overview of the windows in this module.



*The parameters set in the different fields correspond to the default values.*

**Spread sheet report settings**

**Computation settings**



## 2.3 Input / Output



In this **PLUME** User Manual, refer to the General Principles in Chapter 2 (PLUME Modules) for the “INPUT” and “OUTPUT” tables of the different modules and corresponding files.

### Input

- **.SYSSET file**
  - Mission synthesis file for the **PLUME** application. This file is mandatory and is created by the **SYSTEMA** “Current Mission” process module.
- **.FLOW and .THERMO file**
  - Description file of the thruster emitted flow field. These files are not mandatory in the case that a FLOW and THERMO file has been attached to each of the defined thruster (in the meshing part).

### Output

- **.PLU.LOG file**
  - General information file gathering messages written during the computation. If the computation does not go to an end, an error message or a warning may be found in this file. This file cannot be deactivated.
- **.PLU.H5 file, .INFO.H5 file**
  - Used to display the impact parameters (interpolated in the flow file) and the derivate quantities. These files cannot be deactivated.
- **.ft.xls (forces-torques), .tf.xls (thermal flux) files**
  - Files containing integrated quantities under HTML format. These files can be opened with standard spread sheet report editor. These files cannot be deactivated.
- **.mf.xls (mass flux) file**
  - File containing integrated quantities under HTML format. This file can be opened with standard spread sheet report editor. This file can be deactivated.

## 3 Content of the result files

### 3.1 Impingement information file

This file contains data coming from the ray tracing results and from the interpolation in the flow field file. The data in this file can be used to finely understand the results. It can be used in 3D Views (either in the meshing or in the mission tab) for result display as coloured or textual information **on meshes**. It can also be used in Curve Views to display time dependant results **on meshes**.

#### *General information data*

This part contains the number of rays that have been fired as well as the area of the meshes.

#### *Ray tracing data*

This part is sorted by Thruster. For each thruster, the following information are available:

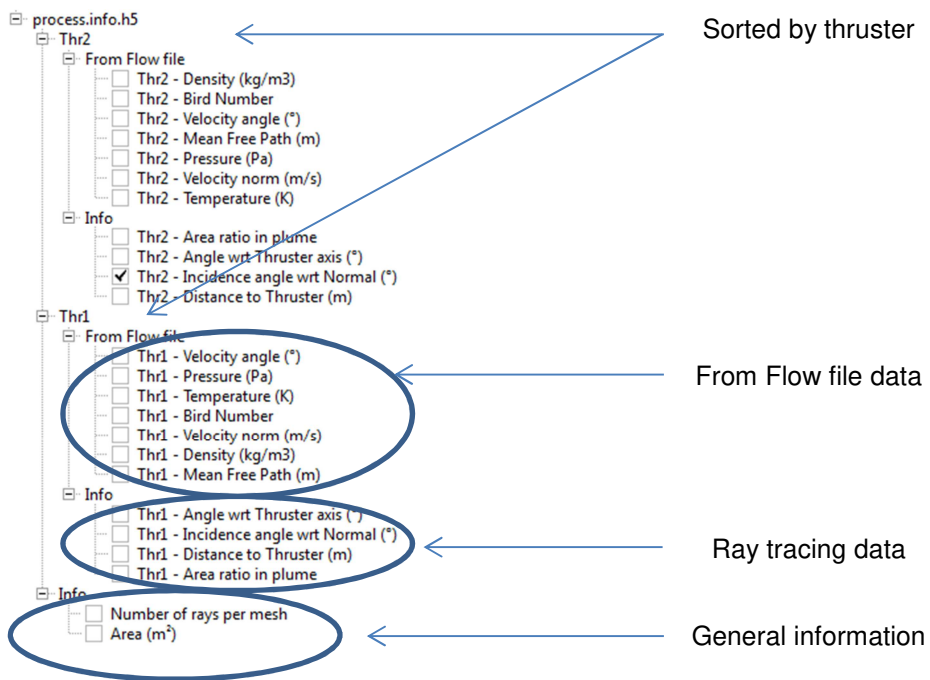
- Area ratio in the plume: if only one ray has been fired has been launched, the value is simply 0 or 1, depending if the center of the mesh is shadowed or not; if several rays have been fired, the value will be between 0 and 1.
- Angle with respect to the Thruster axis: angle between the normal to the mesh and the thruster axis
- Incidence angle with respect to the normal: angle between the local velocity vector and the normal to the mesh
- Distance to thruster.

#### *“From flow file” data*

This part is also sorted by Thruster. All the quantities available here are interpolated in the FLOW file, or computed from the FLOW and THERMO file. For each thruster, the following information are available:

- Directly interpolated: Density, Velocity norm, Velocity angle, Temperature, Bird Number (if present in the FLOW file, otherwise, zero)
- Computed: Pressure (with standard perfect gas formula involving the density, the temperature and the gas constant), Mean Free Path.

The following figure gives an example of the content of this file as seen in the graphical user interface.



### 3.2 Derivate quantities file

This file contains cumulated data over meshes, surfaces and possibly groups. The surfaces and groups results parts are only available for “textual information”, curve view or table view display. The positive and negative side of the surfaces / groups are also split. The meshes results part is available for every kind of display. The negative and positive side are not split because the negative and positive side of a mesh are distinct meshes.

#### Results sorting

First sorted by kind of result (surfaces, meshes)

Then sorted by thruster, then by side (for surfaces and groups). An additional thruster is available, called “All Thruster”, allowing to have information cumulated as if all thrusters were firing at the same time.

Available data for post-processing

#### Maximum quantities

- On meshes, the Max Thermal Flux and Max Net Mass flux are computed as the maximum value obtained among the fired rays on each mesh. For instance, in the case where the user has fired only one ray, the Thermal Flux and the Max Thermal Flux value are the same. All thruster computation is done by taking into account the maximum contribution of all thrusters on each mesh cell (i.e. the sum of the most energetic ray from each thruster on a given cell).
- On surfaces (or groups), the maximum quantities are obtained as the maximum value obtained among the maximum values on the meshes composing the surface (or group). For the all thrusters group, the value given is from the mesh cell with the maximum computed flux, computed as explained above (i.e. the sum of the sum of the most energetic ray from each thruster on a given cell).

### 3.3 Spread sheet report files

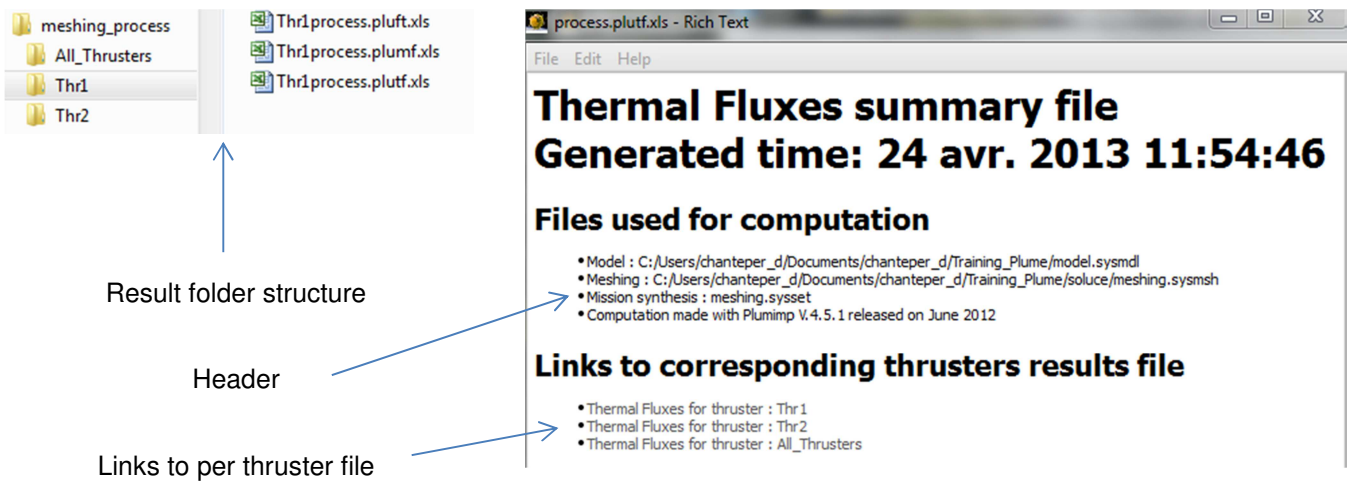
These files contain cumulated data on the overall geometry and possibly on groups. They are intended to give access to the user to synthetic data. One set of data is available for each thruster and for the All Thruster:

- Forces and torques file, extension .ft.xls, always generated
- Thermal Flux file, extension .tf.xls, always generated
- Mass Flux file, extension .mf.xls, generated on demand

#### Created folder structure

The files are placed in a subfolder of the computation folder. The name of this subfolder is the thruster name.

In addition, a file containing links to all specific per thruster file is created directly inside the computation folder.





## File organisation

Header  
  
Overall  
summed  
results  
  
Per group  
results

**Thr1process.plutfx.xls - Rich Text**

File Edit Help

**Thermal fluxes for thruster Thr1**  
**Generated time: 24 avr. 2013 11:54:45**

**Files**

- Model : C:\Users\chanteper\_d\Documents\chanteper\_d\Training\_PLume\model.sysmdl
- Meshing : C:\Users\chanteper\_d\Documents\chanteper\_d\Training\_PLume\solve\meshing.sysmsh
- Mission synthesis : meshing.sysset

**Model Summed Results for thruster Thr1**

**Thermal Fluxes**

Mean Thermal Flux [kW/m <sup>2</sup> ]	Max Thermal Flux [kW/m <sup>2</sup> ]
1.70556E-02	2.15432E-01

**Groups Results for thruster Thr1**

**Thermal fluxes for group Solar Array**

Pos. Mean Thermal Flux [kW/m <sup>2</sup> ]	Pos. Max Thermal Flux [kW/m <sup>2</sup> ]	Neg. Mean Thermal Flux [kW/m <sup>2</sup> ]	Neg. Max Thermal Flux [kW/m <sup>2</sup> ]	Total Mean Thermal Flux [kW/m <sup>2</sup> ]	Total Max Thermal Flux [kW/m <sup>2</sup> ]
0.00000E+00	0.00000E+00	8.24002E-02	2.15432E-01	8.24002E-02	2.15432E-01

Page left intentionally blank