Title

DROPLET

APPLICATION MANUAL

	Name and Function	Date	Signature
Prepared by	C. THEROUDE	23/09/2009	
Verified by	P. CHEOUX-DAMAS	23/09/2009	
Approved by			
Authorized by			
Application authorized by			

Document type	Nb WBS	Keywords

NbCars 17044
NbWords 3506
FileName Droplet_AM_V2.doc

SUMMARY

This document is the application manual of the DROPLET module. This module is dedicated to the computation of the droplets or particles flow field inside and outside the thruster. It is integrated in the PLUMFLOW procedure.

Chapter 3 presents the DROPLET module, chapter 4 describes in detail the input and output of the module and chapter 5 gives an example of application.

DOCUMENT CHANGE LOG

Issue/ Revision	Date	Modification Nb	Modified pages	Observations
00/00	09/03/98	1	All	First version
02/00	23/09/09	2	All	Formatting

PAGE ISSUE RECORD

Issue of this document comprises the following pages at the issue shown

Page	Issue/ Rev.	Page	Issue/ Rev.	Page	Issue/ Rev.	Page	Issue/ Rev.	Page	Issue/ Rev.	Page	Issue/ Rev.
All	**00/00**										
All	**02/00**										

DROPLET V3.2 – Application Manual

This Manual contains task-oriented instructions that show you how to use the Droplet module.

Document issue : 2.0

Software version : Droplet Version 3.2

The information contained in this document is subject to change without notice and should not be construed as a commitment by EADS ASTRIUM SAS. EADS ASTRIUM SAS assumes no responsibility for any errors that may appear in this document.

The software described is licensed for installation according to the following conditions :

- The software shall not be distributed to third parties,
- Modification of the software is not permitted.

All errors and suggestions concerning the software or this documentation should be sent to :

C. Theroude at :

EADS ASTRIUM

31 avenue des Cosmonautes

Z.I. du Palays

31402 TOULOUSE CEDEX 4

France

Page left intentionally blank

TABLE OF CONTENTS

1 REFERENCE DOCUMENTS

[RD1] "Spécification d'intégration : PLUMFLOW, PLUME, CONTAMINE dans ESABASE". P. Chèoux-Damas, C. Theroude. Doc. MMS : S413/RT/17.94. 12/01/95.

[RD2] "PLUME V3.2 - Interface files definition". P. Chèoux-Damas. Doc. MMS : S413/RT/41.97. 24/10/97.

2 INTRODUCTION

The objective of the DROPLET module is to compute the flow of droplets or particles inside and outside a thruster. This is particularly important for the contamination analysis where most of the contaminants deposited at ambient temperature are droplets. This module allows to compute the droplets / particles expansion from an experimental distribution measured outside the thruster. It allows also to perform the simulation of the droplets / particles flow inside the thruster.

The DROPLET module is integrated inside the PLUMFLOW module and is interfaced with the NAVIER, MATFLOW, TRAJET and CONTAMINE modules.

This document is the application manual of the DROPLET software. Chapter 3 presents the DROPLET module, chapter 4 describes in detail the input and output of the module and chapter 5 gives an example of application.

3 DROPLET PRESENTATION

The DROPLET module computes the droplets or particles flow field inside and outside the thruster. The gaseous flow field is previously computed (by NAVIER or MATFLOW) and is assumed not to be modified by the droplets / particles.

For the computation of the droplets / particles in the core of the plume, two methods can be used :

1. The droplets / particles are injected at the thruster throat (or inside the chamber) with a uniform distribution. The droplets / particles are then propagated in the gaseous flow field taking into account the drag force.
2. The droplets / particles are initialised at the thruster exit from a distribution given by the user (typically resulting from experimental measurements), and are propagated using a point-source expansion.

The user has also the capability to define a distribution of droplets at the thruster lip. That allows to evaluate the droplets distribution in the thruster backflow (due to the film cooling).

The droplets / particles are distributed among different groups characterised by their size (and also by their flow rate, temperature and velocity).

4 DROPLET INPUT/OUTPUT

4.1 DROPLET ARCHITECTURE

The input files of the DROPLET software are the following :

- The file definition (PLUMFLOW.SYSINPUT),
- The run parameters file (.DRI),
- The flow field file (.FLOW) generated by the NAVIER or MATFLOW modules,
- The thermodynamic file (.THERMO) generated by ODE.

These files are presented at the figure Figure 4.1-1 and detailed in the paragraph 4.2.

All these files are formatted in ASCII to allow the portability on different operating systems (HP, SUN, ULTRIX, etc).

The output files of the DROPLET software are :

- The listing file (.DRO) resuming the run parameters and the progress of the run,
- The flow field file (.FLOW) containing the characteristics of the gaseous flow field previously calculated and the characteristics of the droplets / particles flow field calculated by DROPLET. This file can be visualised using the TRAJET module.

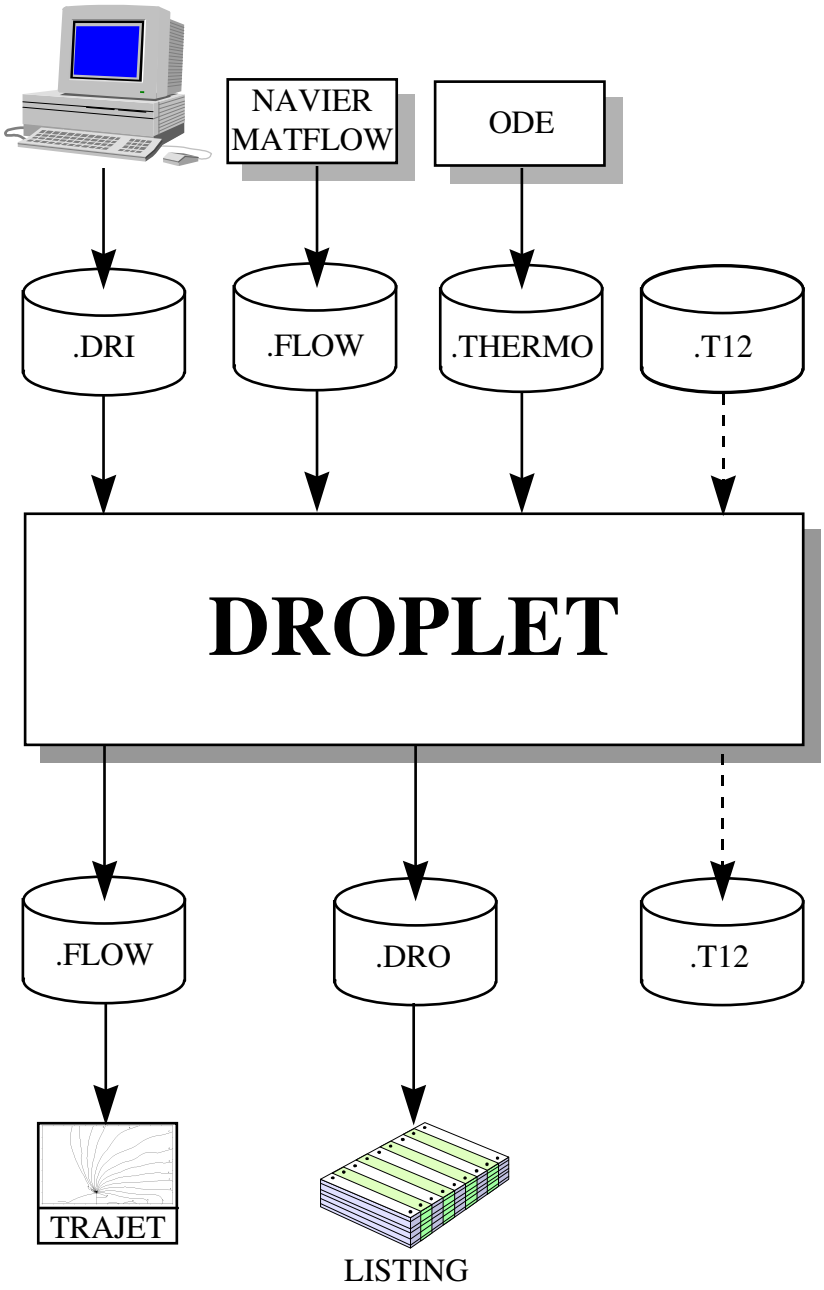

Figure 4.1-1 : Architecture of the DROPLET module

4.2 DROPLET INPUT FILES DESCRIPTION

4.2.1 The definition file : PLUMFLOW.SYSINPUT

This file contains only the name of the thruster (e.g. NAME). This name allows DROPLET to open the different files (NAME.DRI, NAME.FLOW, etc) at the beginning of the DROPLET run. This file is created by the PLUMFLOW framework and has not to be created by the user excepted if he wants to run DROPLET outside the PLUMFLOW framework.

4.2.2 The run parameters file : .DRI

This file contains the run parameters of the DROPLET software. It is characterised by its extension (.DRI). The structure of the run parameters file is based on a set of namelist.

The first namelist, called $COMMAND is used to define the main run parameters.

The namelist called $MESH is used to define the mesh parameters of DROPLET.

The namelist called $PARTICLE is used to define the general characteristics of the groups of droplets / particles.

The namelists called $GROUP define the angular distribution of the different groups of droplets / particles.

The namelist called $LIP_FLOW defines the distribution of droplets / particles at the lip of the thruster.

The contents of the file is described hereafter.

Format of the file :

```
$COMMAND
NAVIER      MATFLOW
IC          ILIM
$END
$MESH
NPTI        NPTJ
RGI         RGJ
ZIT
$END
$PARTICLE
NBG
DIAM(I)     RHOP(I)     DEB(I)      CPP(I)            (I = 1,NBGR)
$END
$GROUP
NPR
ALPHA (J)          RHO(J)                           (J = 1,NPR)
NPV
BETA(K)            VIT(K)                           (K = 1,NPV)
TEMP
$END
...
$GROUP
...
$END
$LIP_FLOW
NBGLIP
DLIP(I)            AMINLIP(I)        AMAXLIP(I)              (I = 1,NBGLIP)
VITLIP(I)          TEMPLIP(I)        DIAMLIP(I)              (I = 1,NBGLIP)
RHOPLIP(I)         DEBLIP(I)         CPPLIP(I)               (I = 1,NBGLIP)
$END
```

$COMMAND part

NAVIER : Defines if DROPLET shall perform the simulation inside the thruster using a .FLOW file computed by the NAVIER module.

Type : boolean

Default : .FALSE.

Range : .FALSE. : DROPLET does not use the gaseous flow field inside the thruster.

.TRUE. : DROPLET uses the gaseous flow field inside the thruster. If MATFLOW = .FALSE. a complete simulation of the droplets flow field is performed inside and outside the thruster. If MATFLOW = .TRUE. the program uses the results of the .T12 file to perform the simulation outside the thruster.

MATFLOW : Defines if DROPLET shall perform the simulation by a source-point expansion method.

Type : boolean

Default : .TRUE.

Range :

.FALSE. : DROPLET does not perform the simulation using a source-point expansion method.

.TRUE. : DROPLET performs the simulation using a source-point expansion method. If NAVIER = .FALSE. DROPLET performs the simulation using the angular droplets / particles distribution defined in the .DRI file. If NAVIER = .TRUE. the program uses the results of the .T12 file to perform the simulation outside the thruster.

IC : Defines the start line ordinate of the .FLOW mesh where the particles are injected.

Type : integer

Default : 25

$MESH part

NPTI : Number of points along Z

Type : integer

Default : 80

NPTJ : Number of points along R (number of streamlines).

Type : integer

Default : 20

RGI : Geometrical progression of the mesh in the Z direction

Type : real

Default : 1.1

RGJ : Geometrical progression of the mesh in the R direction

Type : real

Default : 0.7

ZIT : Defines the time step of integration. If ZIT increases, the time step decreases.

Type : real

Default : 3.

$PARTICLE part

NBG : Number of groups of droplets / particles inside the thruster.

Type : integer

Default : 1

DIAM(I) : Diameter of the I^{th} group of droplets / particles inside the thruster.

Type : real

Unit : m

Default : $1.\ 10^{-6}$

RHOP(I) : Density of the Ith group of droplets / particles inside the thruster.

Type : real

Unit : kg/m^3

Default : 1 000.

DEB(I) : Mass flow rate of the Ith group of droplets / particles inside the thruster.

Type : real

Unit : kg/s

Default : 1. 10^{-5}

CPP(I) : Heat capacity of the Ith group of droplets / particles inside the thruster.

Type : real

Unit : J.kg^{-1}.K

Default : 1 000.

$GROUP part

This section is repeated for each group of droplets / particles inside the thruster (NBG times). It is used only if NAVIER = .FALSE. and MATFLOW = .TRUE.

NPR : Number of angles where the relative flow rate is specified.

Type : real

Default : 2

ALPHA(J) : Value of the Jth angle.

Type : real

Unit : degree

Default : ALPHA(1) = 0.

ALPHA(2) = 20.

RHO(J) : Droplets / particles flow rate ratio for the J^{th} angle.

Type : real

Default : RHO(1) = 1.

RHO(2) = 1.

NPV : Number of angles where the velocity rate is specified.

Type : integer

Default : 2

BETA(K) : Value of the K^{th} angle.

Type : real

Unit : degree

Default : BETA(1) = 0.

BETA(2) = 20.

VIT(K) : Droplets / particles velocity for the K^{th} angle.

Type : real

Unit : $m.s^{-1}$

Default : VIT(1) = 2 000.

VIT(2) = 2 000.

TEMP : Droplets / particles temperature.

Type : real

Unit : K

Default : 1 000.

$LIP_FLOW part

NBGLIP : Number of droplets / particles groups for the lip injection.

Type : integer

Default : 0

DLIP(I) : Diameter of the thruster lip for the Ith group to be injected at the lip.

Type : real

Unit : Adimensioned by the thruster throat radius.

AMINLIP(I) : Minimal angle of droplets / particles injection at the lip for the Ith group.

Type : real

Unit : degree

AMAXLIP(I) : Maximal angle of droplets / particles injection at the lip for the Ith group.

Type : real

Unit : degree

VITLIP(I) : Initial droplets / particles velocity of the Ith group.

Type : real

Unit : m.s^{-1}

DIAMLIP(I) : Diameter of the Ith group of droplets / particles.

Type : real

Unit : m

RHOPLIP(I) : Density of the Ith group of droplets / particles.

Type : real

Unit : kg/m^3

DEBLIP(I) : Mass flow rate of the Ith group of droplets / particles.

Type : real

Unit : kg/s

CPPLIP(I) : Heat capacity of the Ith group of droplets / particles.

Type : real

Unit : J.kg^{-1}.K

4.2.3 The flow field file : .FLOW

This file contains the characteristics (density, temperature, velocity) of the gaseous flow field. If the flow field has been computed by NAVIER, it contains the flow field inside and in the vicinity of the thruster. If it has been computed by DROPLET, it contains the flow field outside the thruster. The complete description of the .FLOW file is presented in the [RD2].

4.2.4 The thermodynamic file : .THERMO

This file contains the thermodynamic characteristics (C_p, γ, molar mass, viscosity, Prandtl number) of the gaseous flow field. It is created by the ODE module. The complete description of the .THERMO file is presented in the [RD2].

4.3 THE DROPLET OUTPUT FILES

4.3.1 The listing file : .DRO

This file is a listing file containing a summary of the input parameters and some information concerning the progress of the run.

An example of such a file is given hereafter. It contains :

- The description of the run parameters and in particular the properties of the groups of droplets / particles,
- The results of the computation and the mass flow rate of each streamlines.

```
TITLE OF MATFLOW INPUT FILE :   NAME OF THRUSTER :mbb10_new
---------------------------


NAMELIST COMMAND
----------------
COMPUTATION OF DROPLET MODULE AFTER :  NAVIER : F  MATFLOW : T


NAMELIST MESH
-------------
NUMBER OF PARTICLE STREAMLINES :            NPTJ =20
NUMBER OF COMPUTATION POINT :               NPTI =80


NAMELIST PARTICLE
-----------------
NUMBER OF PARTICLE GROUPS IN THE NOZZLE :   NBG =  2

DATA FOR THE PARTICLE GROUP : 1
DIAMETER :                    0.5000E-05  m
DENSITY  :                    0.1000E+04  Kg/m3
MASS FLOW RATE :              0.1000E-04  Kg/s
CALORIFIC CAPACITY :          0.1000E+04  J/K/Kg

DATA FOR THE PARTICLE GROUP : 2
DIAMETER :                    0.1000E-03  m
DENSITY  :                    0.1000E+04  Kg/m3
MASS FLOW RATE :              0.1000E-04  Kg/s
CALORIFIC CAPACITY :          0.1000E+04  J/K/Kg


NUMBER OF INJECTION PARTICLE GROUPS :   NBGLIP = 1

DATA FOR THE PARTICLE GROUP : 3
MINIMAL ANGLE :               0.2000E+02  0
MAXIMAL ANGLE :               0.1200E+03  0
AVERAGE VELOCITY :            0.5000E+03  m/s
AVERAGE TEMPERATURE :         0.1000E+04  °C
DIAMETER :                    0.1000E-03  m
DENSITY  :                    0.1000E+04  Kg/m3
MASS FLOW RATE :              0.1000E-04  Kg/s
CALORIFIC CAPACITY :          0.1000E+04  J/Kg/K


RESULT OF THE COMPUTATION
-------------------------

MASS FLOW RATE FOR THE PARTICLE GROUPE :    1
TOTAL MASS FLOW RATE : 0.1001E-04  Kg/s

PARTICULE STREAMLINES NUMBER :  1  MASS FLOW =  0.3786E-07  Kg/s
PARTICULE STREAMLINES NUMBER :  2  MASS FLOW =  0.1124E-06  Kg/s

PARTICULE STREAMLINES NUMBER : 20  MASS FLOW =  0.7025E-06  Kg/s


MASS FLOW RATE FOR THE PARTICLE GROUPE :    3
TOTAL MASS FLOW RATE : 0.1000E-04  Kg/s

PARTICULE STREAMLINES NUMBER :  1  MASS FLOW =  0.2680E-06  Kg/s

PARTICULE STREAMLINES NUMBER : 19  MASS FLOW =  0.5439E-06  Kg/s


COMPUTATION OF DROPLET MODULE HAS BEEN COMPLETED
```

Extract of the .DRO file

4.3.2 The flow field file : .FLOW

This file contains the characteristics (density, temperature, velocity) of the gaseous flow field as provided by the NAVIER or MATFLOW modules. In addition, the characteristics of the particles / droplets flow field computed by DROPLET are included in the .FLOW file. It allows to perform the visualisation of the flow field using the TRAJET module.

4.3.3 The droplets distribution file : .T12

This file contains the characteristics of the droplet flow (density, temperature, velocity). It is created by the DROPLET module when NAVIER = .TRUE. and MATFLOW = .FALSE.. It is also used by DROPLET when NAVIER = .TRUE. and MATFLOW = .TRUE. to perform a source-point expansion.

5 HOW TO USE DROPLET

The goal of this chapter is to present the use of the DROPLET module on a real case and to give to the user some advises.

5.1 APPLICATION CASE

In this paragraph, a complete case of DROPLET application is presented. The calculation presented below is a source-point expansion of the droplets computed from an experimental angular distribution. The droplets angular distribution used as input was measured by Trinks (see Figure 5.1-1).

Erreur ! Liaison incorrecte.

Figure 5.1-1 : Droplets angular distribution

These curves allow to determine the droplets angular distribution and the mass flow rate. In this calculation we use two groups of droplets with the following characteristics.

Group 1			
Droplets characteristics		Angular distribution	
Diameter	5 µm	Angle	Flow rate ratio
Temperature	2 000 K	0°	1.
Velocity	3 000 $m.s^{-1}$	2°	0.96
Flow rate	2. 10^{-5} $kg.s^{-1}$	5°	0.8
		7°	0.63
		10°	0.47
		15°	0.1

Group 2			
Droplets characteristics		Angular distribution	
Diameter	100 µm	Angle	Flow rate ratio
Temperature	2 000 K	0°	1.
Velocity	2 000 $m.s^{-1}$	0.5°	0.78
Flow rate	7.56 10^{-4} $kg.s^{-1}$	1°	0.56
		2°	0.24
		5°	0.073
		6°	0.037

5.1.1 External input file

To perform this application case, the user shall have in the current directory :

- A mbb10.FLOW file computed by the MATFLOW module,
- A mbb10.THERMO file computed by ODE.

5.1.2 Generation of the .DRI file

The DROPLET input file can be easily generated by clicking on ***Edit input file*** and then on ***DROPLET.*** The parameter NAVIER is equal to F and MATFLOW is equal to T, in order to perform a source-point expansion outside the thruster.

The DROPLET input file is given below. In addition to the two main groups, a specific group for the flow lip has been added in order to model the wall film expulsion.

mbb10.DRI

```
$COMMAND
 NAVIER  = F,
 MATFLOW = T,
 IC = 25,
 ILIM = 0
 $END
 $MESH
 NPTI    = 80,
 NPTJ    = 20,
 RGI     =  1.1,
 RGJ     = 0.7,
 ZIT     =  3.
 $END
 $PARTICLE
 NBG     =      2,
 DIAM    = 5.e-6,100.e-6,
 RHOP    = 1000.,1000.,
 DEB     = 2.e-5,7.56e-4,
 CPP     = 1000.,1000.,
 $END
 $GROUP
 NPR     = 6,
 ALPHA   = 0., 2.,   5.,  7.,   10.,  15.,
 RHO     = 1., 0.96, 0.8, 0.63, 0.47, 0.1,
 NPV     = 2,
 BETA    = 0.,15.
 VIT     = 3000.,3000.
 TEMP    = 2000.
 $END
 $GROUP
 NPR     = 6,
 ALPHA   = 0., 0.5,  1.,   2.,   5.,    6.,
 RHO     = 1., 0.78, 0.56, 0.24, 0.073, 0.037,
 NPV     = 2,
 BETA    = 0.,6.
 VIT     = 2000.,2000.
 TEMP    = 2000.
 $END
 $LIP_FLOW
 NBGLIP     =     1,
 DLIP       = 0.1,
 AMINLIP    = 20.,
 AMAXLIP    = 120.,
 VITLIP     = 500.,
 TEMPLIP    = 1000.,
 DIAMLIP    = 1.e-4,
 RHOPLIP    = 1000.,
 DEBLIP     = 1.e-6,
 CPPLIP     = 1350.
 $END
```

5.1.3 Execution of DROPLET

Once the .DRI file has been created, the user can easily run DROPLET by clicking on ***DROPLET*** and then on ***OK***.

After execution, the user can visualise the DROPLET output file (.DRO) by clicking on ***Edit output file*** and on ***OK***.

5.1.4 Visualisation of the results

The results can be displayed using the TRAJET module. The droplets density of the group 1 is given at the Figure 5.1-2, the droplets density of the group 2 is given at the Figure 5.1-3 and the droplets density of the group 3 is given at the Figure 5.1-4.

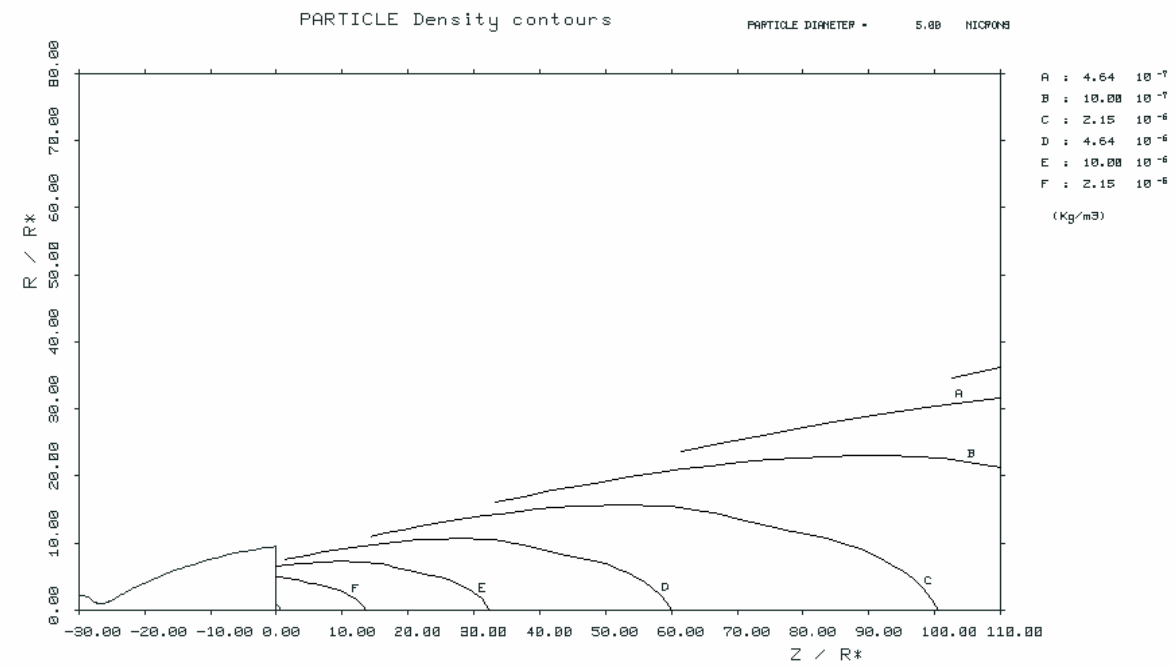

Figure 5.1-2 : Droplets density of the group 1

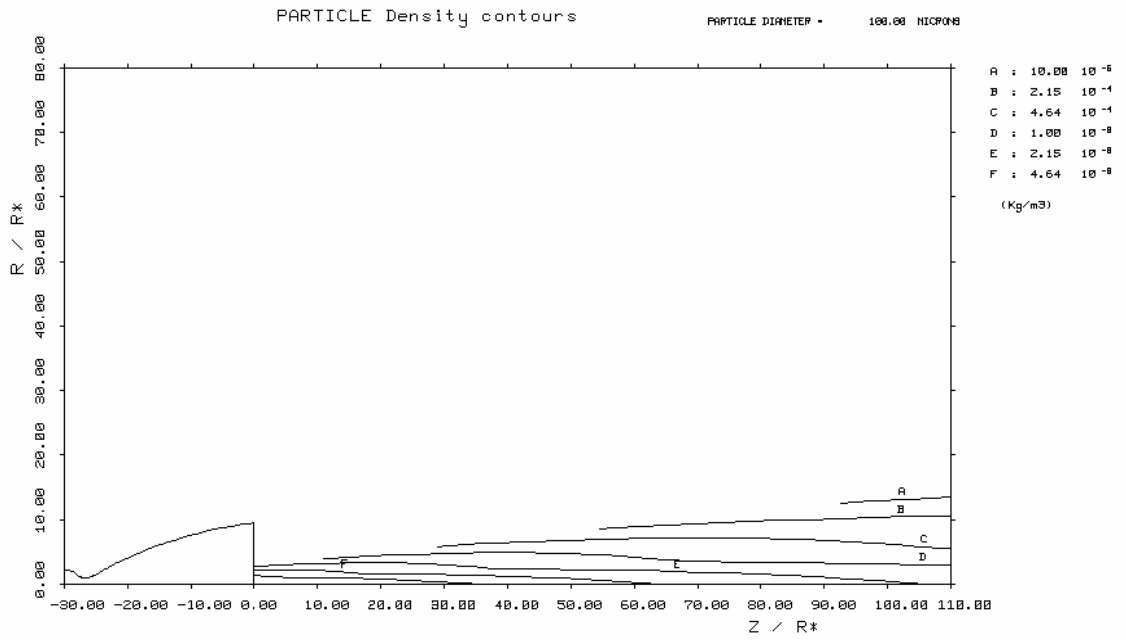

Figure 5.1-3 : Droplets density of the group 2

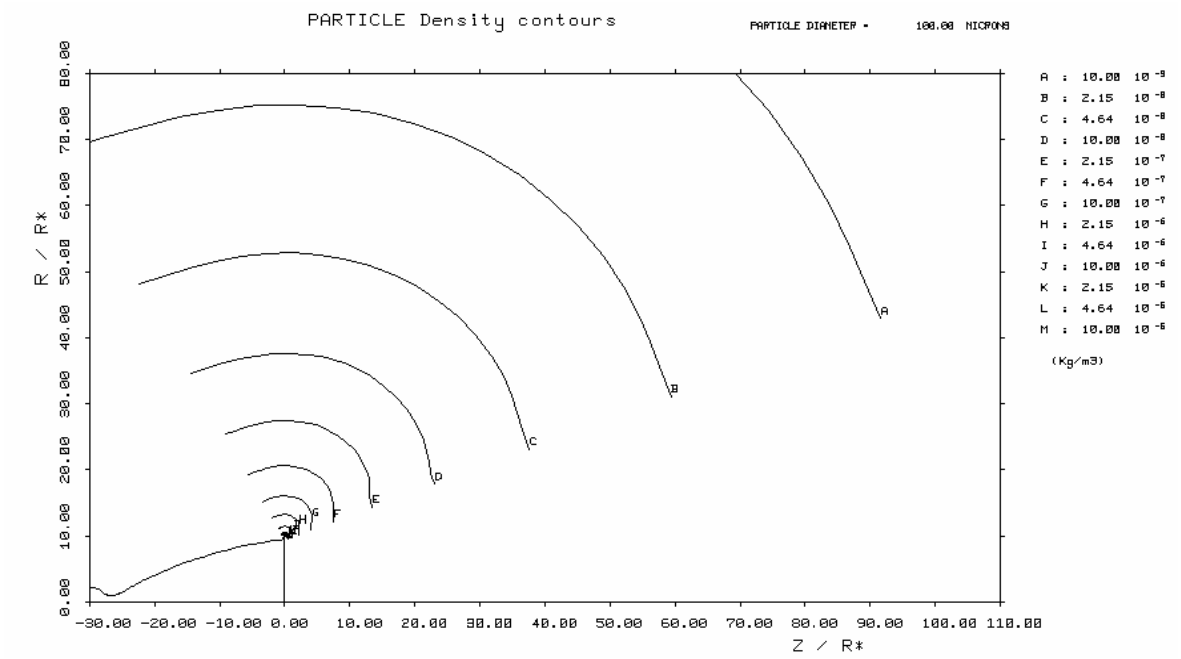

Figure 5.1-4 : Droplets density of the group 3

APPENDIX A - THEORETICAL ASPECTS

A-1 - CALCULATION OF THE DROPLETS EXPANSION

The DROPLET module allows to perform either a source-point expansion of the droplets outside the thruster or an integration of the droplets trajectory inside and outside the thruster.

Source-point expansion

The droplets / particles distribution is defined by the user. The source-point expansion is performed assuming that the velocity is constant and using the mass flow rate conservation law. The temperature of the droplets / particles is also assumed to be constant.

Integration of the droplets trajectory

In this case, the droplets / particles distribution is supposed to be uniform on a user defined streamline of the gaseous flow field (typically, the throat or the combustion chamber). The initial droplets / particles velocity and temperature are also defined by the user.

From the initial distribution, the droplets / particles trajectories are integrated taking into account the initial velocity and the drag force.

The evolution of the temperature is also modelled from the initial distribution taking into account the conducto-convective flux between the gas and the droplets /particles.

A- - CALCULATION OF THE DRAG FORCE

The particles are modelled by a sphere with a radius r_p immersed in a flow field with a uniform velocity $\vec{V} - \vec{V}_p$.

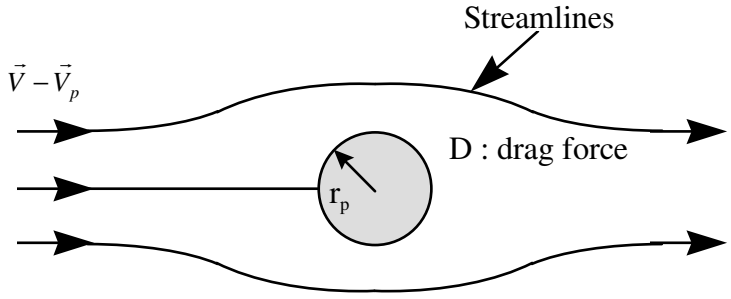

Figure A-1 : Modelling of the drag force

The parameters involved in the physical modelling are :

- The radius of the particle / droplet : r_p,
- The relative velocity of the flow : $\vec{V} - \vec{V}_p$,
- The density of the flow : ρ,
- The viscosity of the gas : μ.

A dimensional analysis allows to write :

$$\frac{D}{\rho 4 r_p^2 \left(\vec{V}-\vec{V}_p\right)^2} = f\left(\frac{\rho\left|\vec{V}-\vec{V}_p\right|}{\mu} 2r_p\right)$$

Where D is the drag force.

The expression : $\frac{\rho\left|\vec{V}-\vec{V}_p\right|}{\mu} 2r_p$ is the Reynolds number.

C_D is the drag coefficient defined by :

$$C_D = \frac{D}{\frac{1}{2}\rho\left(\vec{V}-\vec{V}_p\right)^2 A}$$

And A is defined by : $A = \pi . r_p^2$

So the drag force is equal to :

$$D = C_D(Re)\frac{1}{2}\rho\left(\vec{V}-\vec{V}_p\right)^2 A$$

Erreur ! Liaison incorrecte.

Figure A-2 : Drag coefficient of a sphere as function of the Reynolds number

So, a particle is submitted to an acceleration equal to :

$$\gamma_p = \frac{1}{2}\rho\left|\vec{V}-\vec{V}_p\right|^2 \pi . r_p^2 . C_D(Re)\frac{1}{\frac{4}{3}\pi . r_p^3 . \rho_m}$$

If $Re = \frac{\rho\left|\vec{V}-\vec{V}_p\right| 2r_p}{\mu}$ and if we put down $C_{d0} = \frac{24}{Re}$ we obtain :

$$\boxed{\vec{\gamma}_p = 4.\frac{C_D(Re)}{C_{D0}}.\frac{\rho_P}{r_p^2.\rho_m}.\mu.\left(\vec{V}-\vec{V}_p\right)}$$

Integration of the dynamic

During one time step the modification of the particle velocity is the following :

$$\boxed{\vec{V}_{p,new} = \vec{V}_p + \vec{\gamma}_p . \Delta t}$$

A-2 - CALCULATION OF CONDUCTO-CONVECTIVE FLUX

The calculation of the conducto-convective flux allows to handle the continuum regime, the free molecular regime and the transitional regime using a bridging method.

The flow regime is computed from the Knudsen Kn number :

$$Kn = \frac{\lambda}{r_p} = \mu\sqrt{\frac{\pi}{2RT}}\frac{1}{\rho}$$

If *Kn* is less than 0.1, the regime is supposed to be continuum, if *Kn* is greater than 10. the regime is supposed to be free molecular and if *Kn* is between 0.1 and 10. the regime is supposed to be transitional.

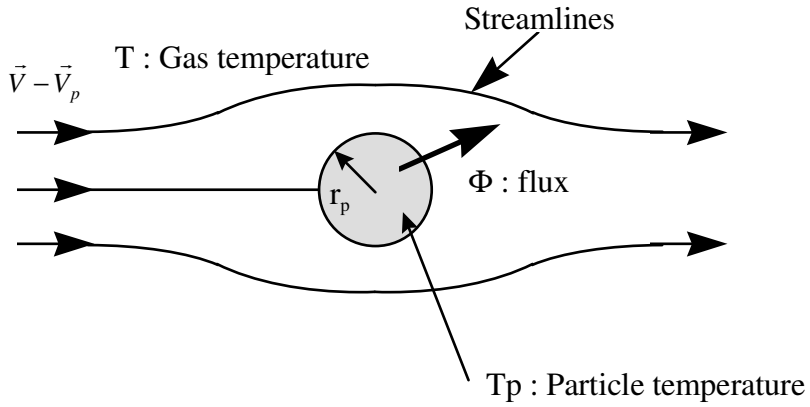

Figure A-3 : Modelling of the conducto-convective flux

In continuum regime

If we put down Φ the heat quantity exchanged between a particle and the gas.

$$\Phi = 4\pi.r_p^2.h.(T_p - T)$$

So, the heat quantity by mass is equal to :

$$Q_{p,c} = \frac{\Phi}{\frac{4}{3}\pi.r_p^3.\rho_m} = \frac{4\pi.r_p^2.(T_p - T)}{\frac{4}{3}\pi.r_p^3.\rho_m}$$

We have $Nu = \frac{2.h.r_p}{k}$ and $Pt = \frac{\mu.C_p}{k}$, which are respectively the Nusselt number and the Prandl number.

So, we obtain :

$$\boxed{Q_{p,c} = \frac{3}{2}Nu\frac{\mu.C_p}{Pt}\frac{1}{\rho_m.r_p^2}(T_p - T)}$$

In free molecular regime

In free molecular regime the formula is the one classically used (same as the PLUME/MATPLIMP module).

$$\Phi = P\sqrt{\frac{RT}{2}}\left\{A\left[s^2 + \frac{1}{RT}\int_{T_p}^{T} C_p.dt - \frac{1}{2}\left(1 - \frac{T_p}{T}\right)\right] + \frac{w(1 + erf(w))}{2}\right\}$$

Where :

$$s = \frac{\left|\vec{V} - \vec{V}_p\right|}{\sqrt{2RT}}$$

$w = s.\sin\theta$ with $\theta = \pi/2$

$$A = \frac{1}{\sqrt{\pi}} e^{-w^2} + w\left(1 + erf(w)\right)$$

And the heat quantity by mass is equal to :

$$\boxed{Q_{p,f} = \frac{\pi.r_p^2}{\frac{4}{3}\pi.r_p^3.\rho_p}.\Phi = \frac{3}{4.r_p.\rho_p}.\Phi}$$

In transitional regime

The heat quantity by mass is equal to :

$$Q_p = Q_{p,c} + \frac{\left(Q_{p,f} - Q_{p,c}\right)}{9.9}.(Kn - 0.1)$$

Integration of the temperature

During one time step the modification of the particle temperature is the following :

$$\boxed{T_{p,new} = T_p - \frac{Q_p}{C_p}.\Delta t}$$

DISTRIBUTION LIST

	Overall document		Summary
	Action	Information	