SYSTEMA PLUMFLOW

Interface file definition

October 2024

Document issue: 1

Page Left Intentionally Blank

Use of the software and of the present software tutorial 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:

Airbus Defence and Space

systema.business@airbus.com

Page Left Intentionally Blank

TABLE OF CONTENTS

1 Introduction

This document defines the format of interface files used within the PLUME software. These files can be generated either by PLUMFLOW itself or by other compatible software, and are subsequently required as input for specific modules within PLUME. Understanding the file format can therefore be useful for users who may need to generate or adapt these files. The files are in ASCII format for portability constraints.

2 The flow files

Several files are used by PLUME for the definition of the flow field generated by the thrusters. Depending on the PLUME module used, the files necessary for a run can be up to the three following ones:

- The « .FLOW » file for the flow-field description,
- The « .THERMO » file for the mean gaseous species thermodynamics description,
- The « .T07 » file for the gaseous species physical properties.

2.1 The flow-field file: « .FLOW »

2.1.1 Introduction

This file can be generated by the PLUMFLOW procedure or can be produced by any other flow field modelling software. It is the main interface between the flow field modelling software and the PLUME applications.
Its structure was generalised in order to transfer not only the gaseous mean flow field but also the particles flow fields and the flow fields for every specific chemical species convected by the plume.
The « .FLOW » file, formatted (ASCII) for compatibility between the various operating systems, is shared in four parts :

- The general data,
- The data concerning the mean gaseous flow field,
- The data concerning the particles flow fields,
- The data concerning the species flow fields.

The structure of the last three parts is the same and can be read by the same reading subroutine.

2.1.2 General data

Format of the file:

```
HEADER
LNC
information ...
...
... end of information
RSTAR          THRUST    FLOWR     REXIT     ZEXIT     RLIP
IDT            NGAS                ICALC
IPAR           ISPEC
NBNOZ
XNOZ(1)        RNOZ(1)   ...                 XNOZ(NBNOZ)
                         RNOZ(NBNOZ)
```

DESCRIPTION

Information part

The first lines of the 'general data' part of the « .FLOW » file are dedicated to the general information concerning the run, the references of the input data etc. Several lines are used to give to the user the information allowing to understand the contents of the file he will use for the plume impingement analysis. This information zone is essential if all the flow fields results (« .FLOW ») are stored in a thruster database.

HEADER: title of the run.
Type : character*80

LNC: number of information lines.
Type : integer

information: set of information lines (LNC lines) describing the hypothesis of the computation (date of the run, references of the input data, specific remarks etc).
Type : (character*80)*LNC lines

Thruster characteristics

RSTAR: throat radius of the thruster.
Type : real

THRUST: nominal thrust of the thruster.
Type : real

FLOWR: mass flow rate of the thruster.
Type : real

REXIT: exit radius of the thruster.
Type : real

ZEXIT: length of the nozzle.
Type : real

RLIP: radius of the lip.
Type : real

IDT: identifier characterising the thruster propellants.
Type : integer
Range :
- 0: not specified
- 1: cold gas thruster
- 2: solid propellant thruster
- 3: mono propellant thruster (N_2H_4)
- 4: bipropellant thruster (MMH/N_2O_4)

ICALC: identifier of the software used to generate the .FLOW file.
Type : integer
Range:
- 0: Unknown
- 1: TPPLUME
- 2: SESJET
- 3: NAVIER
- 4: MCLIP
- 5: MATFLOW
- 6: PROLOG

File characteristics

IPAR: number of particles groups described in the file (cf. the particles description).
Type : integer

ISPEC: number of species described in the file (cf. the species description).
Type : integer

Nozzle characteristics

NBNOZ: number of points used to visualise the nozzle.
Type : integer

XNOZ: abscissa of the thruster points.
Type : real

RNOZ: ordinate of the thruster points.
Type : real

2.1.3 Gaseous mean species data

Format of the file:

```
NPT             NSL                   NVT
NAME
IPAR            RADIUS    FACT
LAB(K)          UNIT(K)   TITLE(K)  k=1,nvt
(X(I,J)         j=1,nsl)                        i=1,npt
(R(I,J)         j=1,nsl)                        i=1,npt
(PSI(J)         j=1,nsl)
(VK(I,J)        j=1,nsl)                        i=1,npt
    k=1,nvt
```

DESCRIPTION

The first lines of the 'mean gas flow field' part of the « .FLOW » file are dedicated to the general information concerning the flow field mesh and the parameters stored in the file.
Then, the flow field meshing is described, followed by the description of the flow field parameters.

NPT: number of mesh point along the first direction of the mesh (the thruster axis direction).
Type : integer

NSL: number of mesh point along the second direction of the mesh (normal to the thruster axis direction).
Type : integer

NVT: number of flow field parameters stored in this mean gas species part.
Type : integer

NAME: unused there.
Type : character*80

IPAR, RADIUS, FACT: unused there.
Type : integer, real, real

The three following items are written on one line for a given flow field parameter. As the current part of the « .FLOW » file is containing NVT parameters, the description is repeated for each different parameter (NVT lines).

LAB(K): this label allows the PLUME application to identify the meaning of a given parameter. The variables, necessary for a further computation by the PLUME application, are corresponding to well defined labels that are:

'Density'	for the gas density
'Temperature'	for the gas temperature
'Velocity Module'	for the gas velocity module
'Velocity Angle'	for the gas velocity direction
'Bird Number'	for the Bird parameter value in the flow

Type : character*20

UNIT(K): unit of the current parameter. This unit must be compatible with the International System for the predetermined parameters (see LAB(K)). The parameter unit is printed by the 2D iso contours plot.
Type : character*12

TITLE(K): title of the current parameter. The title is only used for the 2D iso contours plot.
Type : character*40

The following records gather a column of value of the current item (I: normal to the thruster axis direction). This record is repeated for each meshing column (J: thruster axis direction).

X(I,J): matrix of the abscissa of the meshing points.
Type : real

R(I,J): matrix of the ordinate of the meshing points.

Type : real

PSI(I): streamline function determining the meshing lines (in the case where the mesh lines are stream lines).
Type : real

The following item is repeated for each of the NVT parameters of this part of the « .FLOW » file. For a given parameter (K), one record is containing the parameter values for the points of a given meshing column (I: normal to the thruster axis direction). This record is repeated for each meshing column (J: thruster axis direction).

VK(I,J): parameter number K describing the flow at the line J, column I of the flow meshing.
Type : real

Remarks:
For the PLUME application the following parameters are mandatory with the specified order. They are the first described parameters of the « .FLOW » file :

1. Gas density,
2. Gas temperature,
3. Gas velocity module,
4. Gas velocity direction.

The other parameters of the « .FLOW » file are used only for the 2D visualisation of the flow parameters.

2.1.4 Particle data

Format of the file:

```
NPT             NSL                    NVT
NAME
IPARRADIUS      FACT
LAB(K)          UNIT(K)    TITLE(K)    k=1,nvt
(X(I,J)         j=1,nsl)                          i=1,npt
(R(I,J)         j=1,nsl)                          i=1,npt
(PSI(J)         j=1,nsl)
(VK(I,J)        j=1,nsl)                          i=1,npt
    k=1,nvt
```

DESCRIPTION

The first lines of the 'particles flow field' part of the « .FLOW » file are dedicated to the general information concerning the flow field mesh and the parameters stored in the file.
Then, the particles flow filed meshing is described, followed by the description of the flow field parameters.

NPT: number of mesh point along the first direction of the mesh (the thruster axis direction).
Type : integer

NSL: number of mesh point along the second direction of the mesh (normal to the thruster axis direction).
Type : integer

NVT: number of flow field parameters stored in this particles part for the current group of particles.
Type : integer

NAME: unused there.
Type : character*80

IPAR: current particles group number.
Type : integer

RADIUS: mean particles radius of the current group.
Type : real
Unit : meter

FACT: mean mass of a particles of the current group.
Type : real
Unit : kilogram

The three following items are described on one line for a given particles flow field parameter. As the present part of the « .FLOW » file is containing NVT parameters, the description is repeated for each parameter (NVT lines).

LAB(K): this label allows the PLUME application to identify the meaning of a given parameter. The variables necessary for a further computation by the PLUME application are corresponding to well defined labels that are:

'Density '	for the particle density
'Temperature '	for the particle temperature
'Velocity Module '	for the particle velocity module

Type : character*20

UNIT(K): unit of the current parameter. This unit must be compatible with the International System for the predetermined parameters (see LAB(K)). The parameter unit is printed on the 2D iso contours plot.
Type : character*12

TITLE(K): title of the current parameter. The title is only used by the 2D iso contours plot.
Type : character*40

The following records gather a column of value of the current item (I: normal to the thruster axis direction). This record is repeated for each meshing column (J: thruster axis direction).

X(I,J): matrix of the abscissa of the meshing points.
Type : real

R(I,J): matrix of the ordinate of the meshing points.
Type : real

PSI(I): streamline function determining the meshing lines (the mesh lines are stream lines).
Type : real

The following item is repeated for each of the NVT parameters of this part of the « .FLOW » file. For a given parameter (K), one record is containing the parameter values for the points of a given meshing column (I: normal to the thruster axis direction). This record is repeated for each meshing column (J: thruster axis direction).

VK(I,J): parameter number K describing the particles flow field at the line J, column I of the flow meshing.
Type : real

2.1.5 Gaseous species data

Format of the file:

```
NPT             NSL                   NVT
NAME
IPAR            RADIUS    FACT
LAB(K)          UNIT(K)   TITLE(K)    k=1,nvt
(X(I,J)         j=1,nsl)                          i=1,npt
(R(I,J)         j=1,nsl)                          i=1,npt
(PSI(J)         j=1,nsl)
(VK(I,J)        j=1,nsl)                          i=1,npt
    k=1,nvt
```

DESCRIPTION

The first lines of the 'species flow field' part of the « .FLOW » file are dedicated to the general information concerning the flow field mesh and the parameters stored in the file.
Then, the flow field meshing is described, followed by the description of the flow field parameters.

NPT: number of mesh points along the first direction of the meshing (the thruster axis direction).

Type : integer

NSL: number of mesh points along the second direction of the mesh (normal to the thruster axis direction).
Type : integer

NVT: number of flow field parameters stored in this species part.
Type : integer

NAME: name of the current species.
Type : character*8

IPAR, RADIUS, FACT: unused there.
Type : integer, real, real

The three following items are described on one line for a given flow filed parameter. As the present part of the « .FLOW » file is containing NVT parameters, the description is repeated for each different parameter (NVT lines).

LAB(K): this label allows the PLUME application to identify the meaning of a given parameter. The variables necessary for a further computation by the PLUME application are corresponding to well defined labels that are:

'Density '	for the current species density
'Temperature '	for the current species temperature
'Velocity Module '	for the current species velocity module
'Velocity Angle '	for the current species velocity direction

Type : character*20

UNIT(K): unit of the current parameter. This unit must be compatible with the International System for the predetermined parameters (see LAB(K)). The parameter unit is printed on the 2D iso contours plots.
Type : character*12

TITLE(K): title of the current parameter. The title is only used for the 2D iso contours plot.
Type : character*40

The following records gather a column of value of the current item (I: normal to the thruster axis direction). This record is repeated for each meshing column (J: thruster axis direction).

X(I,J): matrix of the abscissa of the meshing points.
Type : real

R(I,J): matrix of the ordinate of the meshing points.
Type : real

PSI(I): streamline function determining the meshing lines (in the case where the mesh lines are stream lines).
Type : real

The following item is repeated for each of the NVT parameters of this part of the « .FLOW » file. For a given parameter (K), one record is containing the parameter values for the points of a given meshing column (I: normal to the thruster axis direction). This record is repeated for each meshing column (J: thruster axis direction).

VK(I,J) : parameter number K describing the flow of the current species at the line J, column I of the flow meshing.
Type : real

Remarks:
For the CONTAMINE application, the following parameters are mandatory with the specified order. They are the first described parameters of the « .FLOW » file:

1. Species density,
2. Species temperature,
3. Species velocity module,
4. Species velocity direction.

The other parameters of the « .FLOW » file are used only for the 2D visualisation of the flow parameters.

2.2 The thermodynamic file: « .THERMO »

2.2.1 Introduction

This file is also generated by the PLUMFLOW procedure or can be produced by any other flow field modelling software. It is associated with the « .FLOW » file and describes the thermodynamic properties of the mean gas (tabulation of the gas properties).
The « .THERMO » file is formatted (ASCII) for compatibility between the various operating systems.

2.2.2 Description of the structure

Format of the file:

```
HEADER
NLP
ENTH   WMOL   GAMMA TEMP   PRES   CPG    VISC   PRDTL     i=1,nlp
```

DESCRIPTION

The tabulation of the gas properties is related to the chamber pressure. The condition inside the chamber is on the second line of the tabulation.

HEADER: title of the computation.
Type : character*80

NLP: number of lines of tabulation of the gas properties.
Type : integer

ENTH: specific enthalpy of the gaseous mixture.
Type : real
Unit : J/kg

WMOL: molecular weight of the gaseous mixture (for number of Avogadro single molecules).
Type : real
Unit : g/mole

GAMMA: isentropic expansion ratio of the gaseous mixture.
Type : real
Unit : -

TEMP: temperature of the gaseous mixture.
Type : real
Unit : Kelvin

PRES: static pressure of the gaseous mixture.
Type : real
Unit : Pascal

CPG: specific heat at constant pressure of the gaseous mixture.
Type : real
Unit : J/kg/K

VISC: viscosity of the gaseous mixture.
Type : real
Unit : Poiseuille

PRDTL: Prandtl number of the gaseous mixture.
Type : real
Unit : -

2.3 The Species file: « .T07 »

2.3.1 Introduction

This file is generated by the PLUMFLOW procedure or can be produced by any other flow field modelling software.
The « .T07 » file is formatted (ASCII) for compatibility between the various operating systems.

2.3.2 Description of the structure

Format of the file:

```
TREF   TVISC
WTMOLM   VISCM   OMEGM   SDMM   OMEGAM   NDOFM

    The section bellow is repeated for each species group
     NAMGR
     NBSPEGR
     NAMSPEGR(i) (i = 1,NBSPEGR)
     WTMOLG   VISCG   OMEGG   SDMG   OMEGAG   NDOFG   FMOLG
```

DESCRIPTION

Mean species

TREF: Reference temperature for the computation of the viscosity: $\mu = \mu_{ref}\left(\frac{T}{T_{ref}}\right)^{\omega}$.

Type : real
Unit : Kelvin

TVISC: Temperature at which the viscosity (VISCM and VISCG) is provided.
Type : real
Unit : Kelvin

WTMOLM: Molecular weight of the gaseous mixture (for number of Avogadro single molecules).
Type : real
Unit : g/mole

VISCM: Viscosity of the gaseous mixture (at TVISC).
Type : real
Unit : Poiseuille

OMEGM: Exponent for the Sutherland law of the gaseous mixture.
Type : real
Unit : -

SDMM: Diameter for the VHS model of the gaseous mixture.
Type : real
Unit : m

OMEGAM: Exponent for the VHS model of the gaseous mixture.
Type : real
Unit : -

NDOFM: Number of degrees of freedom of the gaseous mixture.
Type : real
Unit : -

For each species group:

NAMGR: Name of the group.
Type : character*12

NBSPEGR: Number of species in the current group.
Type : integer

NAMSPEGR: Name of the species.
Type : character*12

WTMOLG: Molecular weight of the current group (for number of Avogadro single molecules).
Type : real
Unit : g/mole

VISCG: Viscosity of the current group (at TVISC).
Type : real
Unit : Poiseuille

OMEGG: Exponent for the Sutherland law of the current group.
Type : real
Unit : -

SDMG: Diameter for the VHS model of the current group.
Type : real
Unit : m

OMEGAG: Exponent for the VHS model of the current group.
Type : real
Unit : -

NDOFG: Number of degrees of freedom of the current group.
Type : real
Unit : -