Title

NAVIER V3.2 MODULE

APPLICATION MANUAL

	Name and Function	Date	Signature
Prepared by	**C. Theroude** **P. Chèoux-Damas**		
Verified by	**C. Theroude**		
Approved by	**P. Chèoux-Damas**		
Authorized by			
Application authorized by			

Document type	Nb WBS	Keywords

NbCars 27475
NbWords 5738
FileName Navier_AM_V5.doc

SUMMARY

This document is the application manual of the NAVIER 3.2 module.

Document controlled by

DOCUMENT CHANGE LOG

Issue/ Revision	Date	Modification Nb	Modified pages	Observations
00/00	27/11/02		All	First version
01/00	14/04/04		All	Update
02/00	20/03/06		All	Correction in the description of IWALL
03/00	20/12/13		All	Add §5.2.4
04/00	28/08/18		All	Definition of RATEX1, RATEX2 and TTAI

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	**01/00**										
All	**02/00**										
All	**03/00**										
All	**04/00**										

NAVIER V3.2 – Application Manual

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

Document issue :4.0

Software version : NAVIER Version 3.2

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 :

Airbus Defence and Space SAS

31 avenue des Cosmonautes

Z.I. du Palays

31402 TOULOUSE CEDEX 4

France

Page left intentionally blank

TABLE OF CONTENTS

1 REFERENCE DOCUMENTS

[RD1] "PLUME V3.2 – Interface files definition". P. Chèoux-Damas. Doc. MMS : S413/RT/41.97. 24/10/97.

[RD2] "PLUMFLOW V3.2 procedure – Application manual". C. Theroude. Doc. ASTRIUM : MOS.NT.CT.3682776.02 - Issue : 01, 14/04/2004.

[RD2] "TRAJET V3.2 Module – Application Manual". C. Theroude. Doc. ASTRIUM : M&S.NT.CT.4640.99 - Issue : 01. 14/04/04.

2 INTRODUCTION

The NAVIER computer program is used to compute the flow inside and in the vicinity of the nozzle using a Navier-Stokes solver.

3 NAVIER DESCRIPTION

3.1 FUNCTIONAL DESCRIPTION

The objective of NAVIER is :

according to :

- The condition in the chamber,
- The thermodynamic properties of gas,
- The thruster geometry,

compute :

- The gas flow in the subsonic and supersonic region,
- The gas expansion at the nozzle lip,
- The gas flow in the near space.

The flow computation consists in giving, for a set of points arranged in a meshing, the flow characteristics :

- Temperature,
- Density,
- Velocity vector.

3.2 METHOD OF ANALYSIS

The NAVIER code provides a numerical solution for the subsonic / supersonic, axisymmetrical, flow-field inside the nozzle and in its vicinity.

The computation is performed in two steps:

- The subsonic and supersonic flow inside the thruster is computed using a Navier-Stokes solver. The flow is assumed to be laminar in the nozzle. The computation domain is defined at the nozzle lip in order to obtain a supersonic flow at the wall.
- The computation of the flow outside the thruster in the vicinity of the nozzle (closer than 100 throat radii) is performed using an Euler solver. Viscous effects are assumed to be negligible. The flow field at the nozzle lip is imposed by a Prandtl-Meyer expansion at the nozzle wall.

4 NAVIER INPUT/OUTPUT

4.1 NAVIER ARCHITECTURE

The NAVIER input files are :

File	Function
PLUMFLOW.SYSINPUT	Run definition file
"thruster name".NSI	User input file
“thruster name”.THERMO	Gas table giving, versus the temperature :H, W_{mol}, γ, T, P, Cp, μ, Pr (only if OVER = .FALSE.)
“thruster name”.REP	File used to restart the simulation (only if REPLUM = .TRUE. or REPNOZ = .TRUE.)

The NAVIER output files :

File	Function
"thruster name".NSO	Output listing file
"thruster name".FLOW	Flow field description (interface file with MCLIP, PROLOG, TRAJET)
"thruster name".T45	Flow field description (interface file with MATFLOW)
"thruster name"_CNV.FLOW	Field description of the residuals
"thruster name".REP	File used to restart the simulation
"thruster name".THERMO	Gas properties (only if OVER = .FALSE.)

All these files are displayed on Figure 4-1.

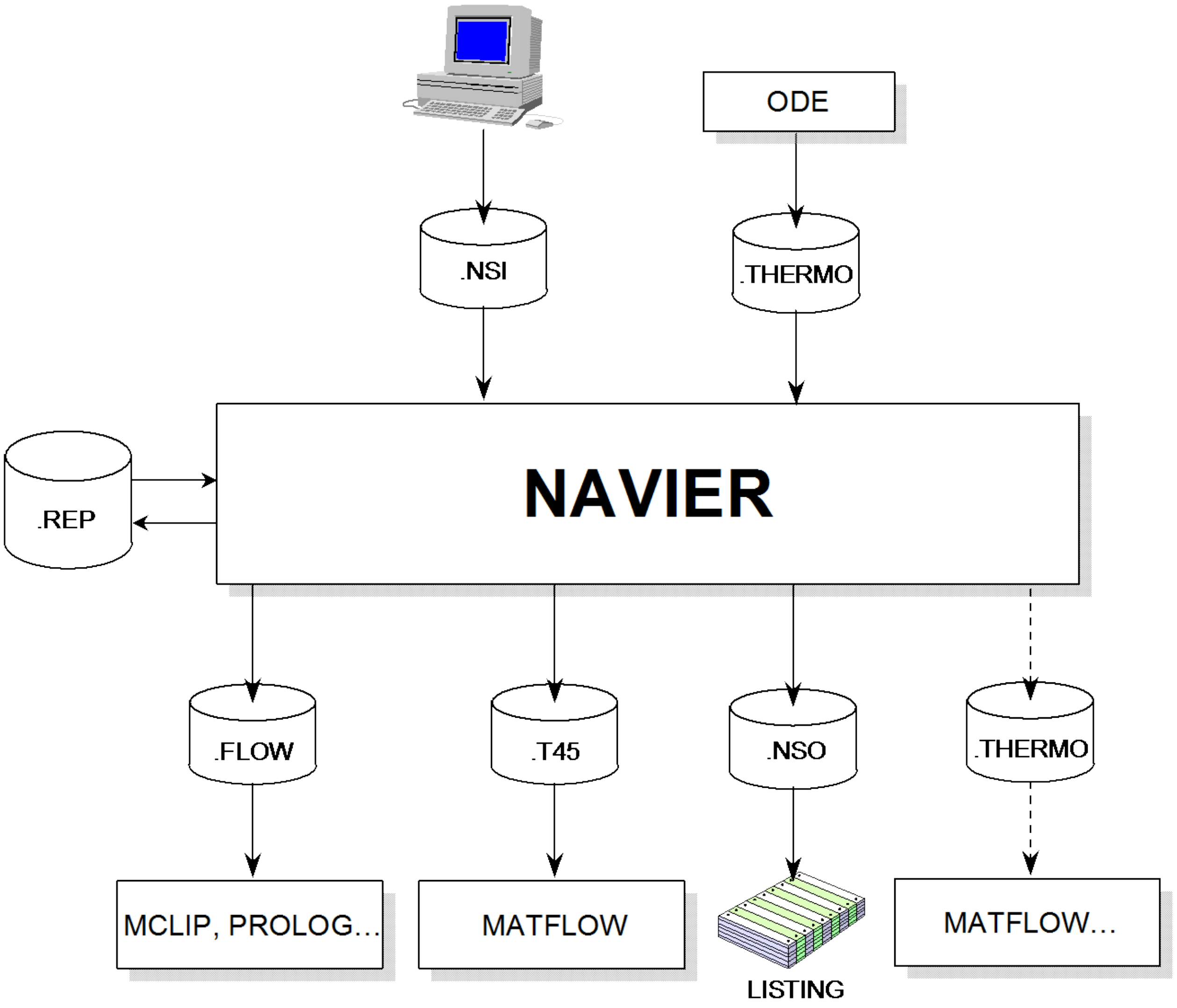


Figure 4-1 : architecture of NAVIER

4.2 NAVIER INPUT FILES DESCRIPTION

4.2.1 PLUMFLOW.SYSINPUT

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

4.2.2 .THERMO file

This is the interface file with ODE. It is the gas table giving, versus the temperature : H, W_{mol}, γ, T, P, C_p, μ, Pr.

4.2.3 .REP file

This is the NAVIER restart file. It allows to restart a calculation and is automatically created by the NAVIER module.

4.2.4 .NSI file

This is the user input file of NAVIER composed of 5 namelists, which are detailed below.

The specification of the namelist in the "thruster name".NSI file has to follow the ordering of the namelist description presented hereafter.

4.2.4.1 Namelist $CONTROL

Purpose

To specify the command parameters of the NAVIER run.

Format of the file :

```
$CONTROL
OVER=over           NOZZLE =nozzle        PLUME=plume
REPNOZ=repnoz       REPLUME=replume       RSTAR=rstar
IPRINT=iprint       KPRI=kpri1,kpri2      PLOTCNV=plotcnv
$END
```

Description :

OVER Specifies the definition of the gas table

Type : Logical

Range : F : NAVIER uses the gas table provided by ODE (.THERMO file)

: T : NAVIER do not use the gas table provided by ODE but the data defined in the $GASPROP namelist

Default : F

NOZZLE Specifies whether the flow is computed in the nozzle

Type : Logical

Range : T, F

Default : T

PLUME Specifies whether the flow is computed in the plume

Type : Logical

Range : T, F

Default : T

REPNOZ Specifies whether the calculation inside the nozzle starts from a previous calculation

Type : Logical

Range : T : NAVIER restarts the nozzle calculation from the .REP file

: F : NAVIER starts the nozzle calculation from the standard initialisation (.REP)

Default : F

REPLUME Specifies whether the calculation inside the plume starts from a previous calculation

Type : Logical

Range : T : NAVIER restarts the plume calculation from the .REP file

: F : NAVIER starts the plume calculation from the standard initialisation (.REP)

Default : F

RSTAR Throat radius of the thruster

Type : real

Unit : meter

Range : > 0

IPRINT Specifies the level of printing results in the listing file

Type : integer

Range : = 0 : minimum level

- Echo of the input data
- Progress of the run with the printing of the residuals at regular intervals
- Error messages

= 1 : same as previous level, plus, at the end of the run, a description of the flow field characteristics at the computation domain boundaries, depending on the KPRI parameter.

= 2 : same as previous level, plus, a description of the mesh.

Default : 0

KPRI Specifies the parameters to be printed

Type : integer vector

KPRI(1) = 1 : mesh point abscissa

KPRI(2) = 1 : mesh point ordinates

KPRI(3) = 1 : ratio of the total pressure to the chamber pressure

KPRI(4) = 1 : volume mass (ρ)

KPRI(5) = 1 : ρ u (u: axial velocity)

KPRI(6) = 1 : ρ v (v: radial velocity)

KPRI(7) = 1 : Mach number

KPRI(8) = 1 : time step per mesh

KPRI(9) = 1 : ratio between the static pressure to the chamber pressure

KPRI(10) = 1 : angle of the velocity with respect to the thruster axis

KPRI(I) = 0 : no impression

Default : (0, 0, 0, 0, 0, 0, 0, 0, 0, 0)

PLOTCNV Specifies whether a “_CNV.FLOW” file containing the field of the computation residuals is generated

Type : Logical

Range : T, F

Default : F

4.2.4.2 Namelist $GEOM

Purpose :

To specify the nozzle geometry.

Format of the file :

```
$NOZZ
NPIG=npig           NPIT=npit           NPII=npii           NPJG=npjg
IAA=iaa             IC=ic               IS=is
Qn=qn               NBF=nbf             D1=d1
RCURV1=rcurv1       RCURV2=rcurv2       TTA1=tta1           TTA2=tta2
EPS=eps             REXIT=rexit         ZEXIT=zexit
TTAEXIT=ttaexit     IWALL=iwall
RNOZ=rnoz           ZNOZ=znoz           RMAX=rmax           ZMAX=zmax
PMA=pma             RCURV=rcurv         TTA=tta
RATEX1=ratex1       RATEX2=ratex2
$END
```

Description :

NPIG	Total number of mesh points along the thruster symmetry axis Type : integer (must be of the form 8*N+1) Default : 121 Range : NPIT < NPII < NPIG
NPIT	Number of mesh points along the thruster axis for the definition of the thruster inside Type : integer (must be of the form 8*N+1) Default : 57 Range : NPIT < NPII < NPIG

NPII

Number of mesh points along the thruster axis for the definition of the thruster and intermediate zones

Type : integer (must be of the form 8*N+1)

Default : 65

Range : NPIT < NPII < NPIG

NPJG

Number of mesh points along the radial axis

Type : integer (must be of the form 8*N+1)

Default : 41

IAA

Index number of the point located at the thruster wall start (along the thruster axis)

Type : integer (must be of the form 8*N+1)

Default : 9

IC

Index number of the point located at the thruster throat

Type : integer (must be of the form 8*N+1)

Default : 9

IS

Type : integer (must be of the form 8*N+1)

Default : 0

QN

Radial geometric progression coefficient

Type : real

Default : .99

NBF

Number of mesh points defining the thruster lip (must be of the form 8*N+1)

Type : integer

Default : 8

D1 Length of the straight chamber wall

Type : real

Unit : adimensionned by the throat radius

Default : 6.

RCURV1 Upstream throat curvature radius

Type : real

Unit : adimensionned by the throat radius

Default : 2.

RCURV2 Downstream throat curvature radius

Type : real

Unit : adimensionned by the throat radius

Default : 2.

TTA1 Upstream throat convergence angle

Type : real

Unit : degrees

Default : 20

TTA2 Downstream throat convergence angle

Type : real

Unit : degrees

Default : 20

EPS Expansion ratio between the exit area and the throat area

Type : real

REXIT Exit radius of the nozzle,

Type : real

Unit : adimensionned by the throat radius

ZEXIT Nozzle length from throat to exit

Type : real

Unit : adimensionned by the throat radius

TTAEXIT Nozzle exit angle

Type : real

Unit : degrees

IWALL Specifies how the nozzle geometry is defined

Type : integer

Range : 0 : nozzle given by points
1 : nozzle is a cone
2 : nozzle is an arc of a parabola
3 : nozzle is an arc of a circle
4 : nozzle defined point by point then smoothed
5 : nozzle is a cone defined by its final point

RNOZ Array of nozzle points ordinates

Type : real

Unit : adimensionned by the throat radius

Note : up to 300 values can be specified, separated by commas

ZNOZ Array of nozzle points abscissa

Type : real

Unit : adimensionned by the throat radius

Note : up to 300 values can be specified, separated by commas

ZMAX Maximum abscissa

Type : real

Unit : adimensionned by the throat radius

Default : 100

RMAX Maximum size of the backflow boundary

Type : real

Unit : adimensionned by the throat radius

Default : 50

PMA Maximum angle of rotation of the last streamline

Type : real

Unit : degrees

Default : 110

RCURV Thruster lip curvature radius

Type : real

Unit : adimensionned by the throat radius

Default : 0.01

TTA Angle of the mesh line I =NIPG with respect to the radial direction

Type : real

Unit : degrees

Default : 180

TTAI Angle of the mesh line NPII wrt to the thruster axis

Type : real

Unit : degrees

Default : 45

RATEX1 Control of the size of the intermediate zone (between NPIT and NPII)

Type : real

Unit : adimensionnal

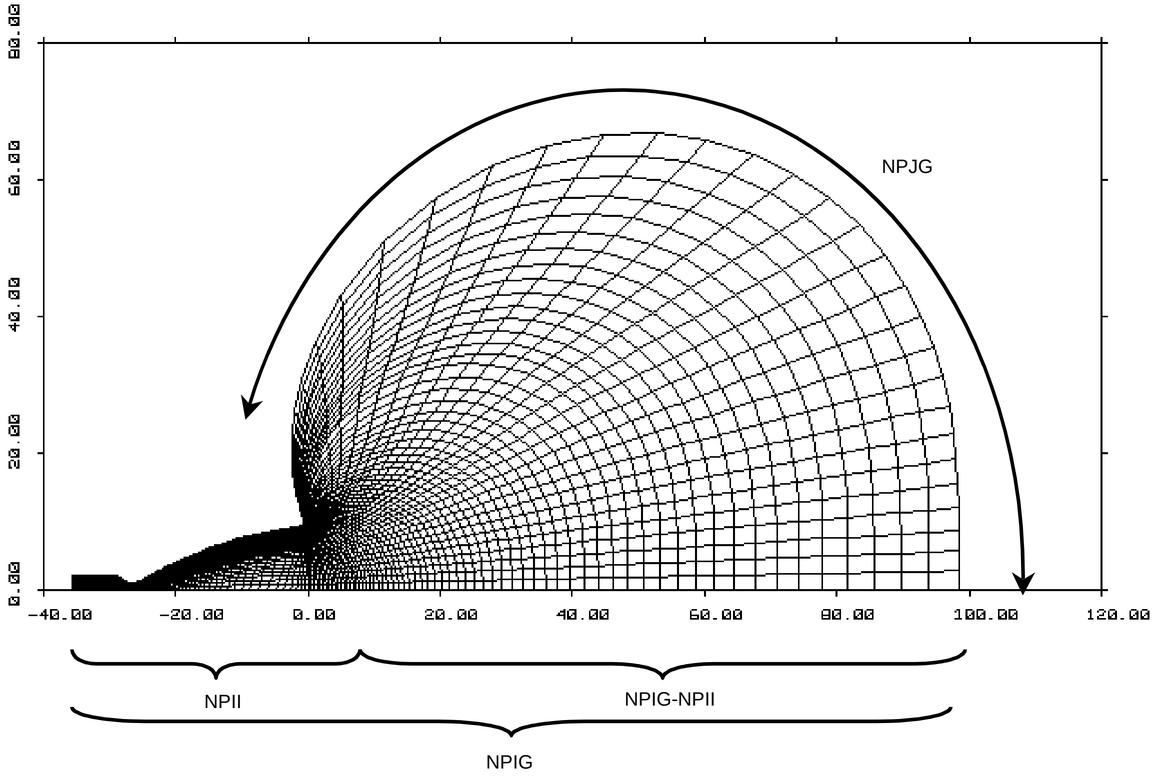
Default : 1

RATEX2 Control of the size of the external zone (between NPII and NPIG)

Type : real

Unit : adimensionnal

Default : 2



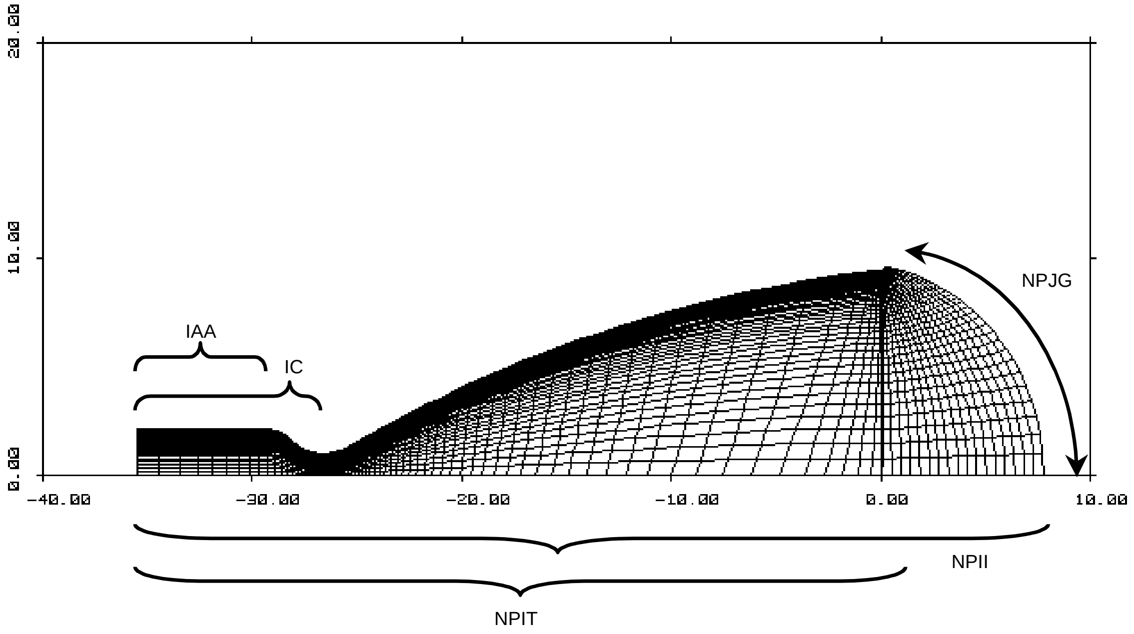


Figure 4-2 : mesh domain definition

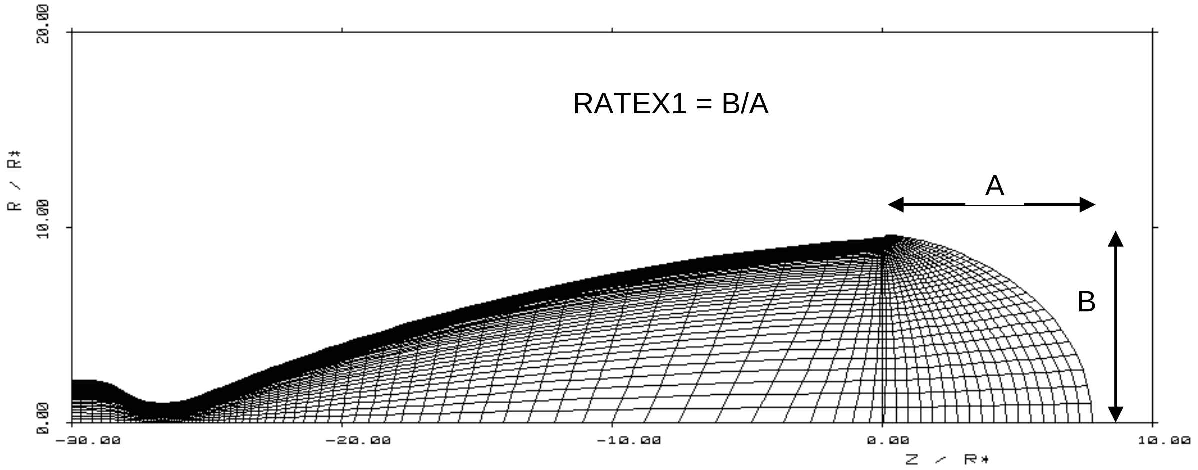


Figure 4-3 : RATEX1 definition

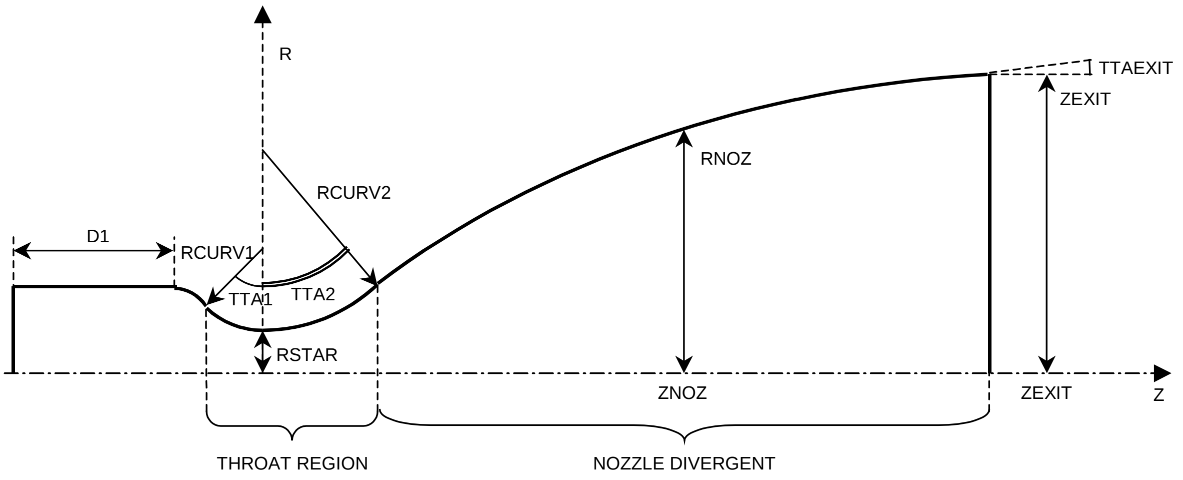


Figure 4-4 : nozzle geometry definition

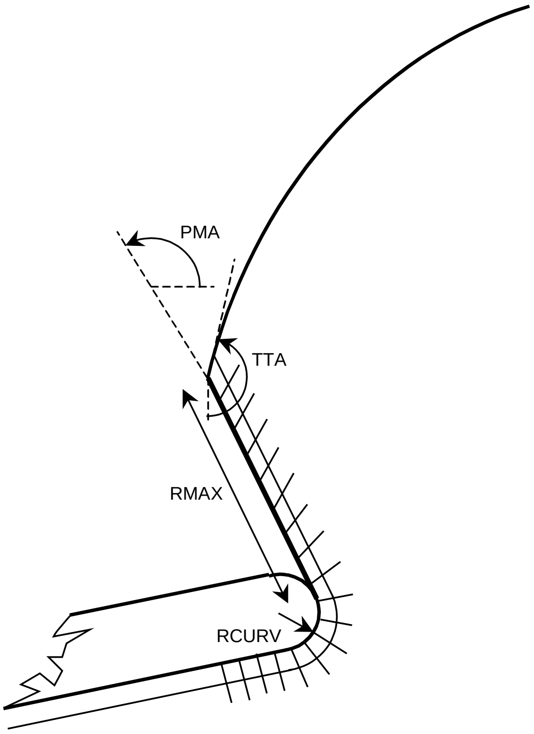


Figure 4-5 thruster lip definition

4.2.4.3 Namelist $TUYERE

Purpose :

To specify the plume computation parameters.

Format of the file :

```
$TUYERE
KEULER1=keuler     ETA1=eta1
NIT11=nit11        NIT21=nit21        NIT31=nit31
Q3I1=q3i1          Q4I1=q4i1          Q3J1=q3j1          Q4J1=q4j1
$END
```

Description :

KEULER1 Specifies the type of solver to be used

Type : 0 : Navier-Stokes

: 1 : Euler

Default : 0

NIT11 Number of iterations between two time steps recalculation

Type : integer

Default : 50

NIT21 Number of iterations between two calculations of the residuals (to be printed in the listing file)

Type : integer

Default : 50

NIT31 Total number of iterations

Type : integer

Default : 1

ETA1 Multiplicative factor of the time step to insure the stability of the numerical solver

Type : real

Range : [0 – 1] (CFL stability criterion)

Default : 0.7

Q3I1, Q4I1,

Q3J1, Q4J1 Multiplicative factors of the artificial viscosity along the thruster axis (I) or along the transverse axis (J)

Type : real

Default : 0.1, 0.01, 0.1, 0.01

4.2.4.4 Namelist $JET

Purpose :

To specify the plume computation parameters.

Format of the file :

```
$JET
KEULER2=keuler    NIT11=nit12    NIT21=nit22
NIT32=nit32       ETA2=eta2      Q3I2=q3i2
Q4I2=q4i2         Q3J2=q3j2      Q4J2=q4j2
$END
```

Description :

KEULER2 Specifies the type of solver to be used.

Type : 0 : Navier-Stokes

: 1 : Euler

Default : 0

NIT12 Number of iterations between two time steps recalculation

Type : integer

Default : 50

NIT22 Number of iterations between two calculations of the residuals (to be printed in the listing file)

Type : integer

Default : 50

NIT32 Total number of iterations

Type : integer

Default : 1

ETA2 Multiplicative factor of the time step to insure the stability of the numerical solver

Type : real

Range : [0 – 1] (CFL stability criterion)

Default : 0.7

Q3I2, Q4I2,

Q3J2, Q4J2 Multiplicative factors of the artificial viscosity along the thruster axis (I) or along the transverse axis (J)

Type : real

Default : 0.1, 0.01, 0.1, 0.01

4.2.4.5 Namelist $GASPROP

Purpose :

To specify the physical properties of the combustion gas, if ODE is not used to generate the .THERMO file.

Format of the file :

```
$GASPROP
TC=tc           PC=pc           CPG=cpg
GAM=gam         BMU0=amu0       OMEGAV=omega
PRTL=pr         WMOL=wmo        RGP=rgp
$END
```

Description :

TC

Specifies the temperature in the combustion chamber

Type : real

Unit : Kelvin

Default : 0

PC

Specifies the pressure in the combustion chamber

Type : real

Unit : bar

Default : 0

CPG

Specifies the heat capacity at constant pressure

Type : real

Unit : J/kg/K

Default : 0

GAM

Specifies the specific heat ratio

Type : real

Unit : adimensionned

Default : 0

BMU0

Specifies the dynamic viscosity at the chamber temperature

Type : real

Unit : Poiseuille (Pa.s)

Default : 0

OMEGAV

Specifies the exponent of the Sutherland law : $\mu=\mu_0\left(\frac{T}{T_0}\right)^{omega}$

Type : real

Unit : adimensionned

Default : 0

PRTL	Is the Prandtl number
	Type : real
	Unit : adimensionned
	Default : 0
WMOL	Is the gas molar mass
	Type : real
	Unit : g/mole
	Default : 0
RGP	Is the ratio of the perfect gas constant to the molar mass
	Type : real
	Unit : adimensionned
	Default : 0

Remarks :

This namelist is mandatory only when the results of the module ODE (.THERMO file) are not to used (i.e. when OVER=T is specified in the namelist $CONTROL).

The user has to provide:

- either WMOL or RGP,
- either CPG or GAM,

the other parameters are automatically calculated.

```
  Bi-propellant test thruster for PLUMFLOW demonstration
 $CONTROL
  OVER= T , NOZZLE=  T , PLUME= F ,
  REPNOZ= F, REPLUM= F ,
  IPRINT=1 , RSTAR=.00079375,KPRI=0,0,0,0,0,0,0,0,0,0
 $END
 $GEOM
  NPJG = 49 , NPIG = 145 , NPIT = 81 , NPII = 97 ,
  IAA = 9 , IC = 25 , QN = .93 ,
  D1 = 6. ,
  RCURV1 = 1.76 , RCURV2 = 0.81163,TTA1=42.5,TTA2=33.92163 ,
  IWALL = 3 ,
  REXIT=7.0991 ,ZEXIT=15.264,TTAEXIT=9.924694,
  ZMAX = 110 , RMAX = .2 , RCURV = .2 ,
  TTAI=40. ,TTA = 200. , PMA = 150. ,
  NBF = 15 ,
 $END
 $TUYERE
  KDTL1=1,KEULER1=0,ETA1=.3,Q3I1=.05,Q4I1=.005,Q3J1=.05,Q4J1=.005,
  NIT11=100,NIT21=100,NIT31=1000,ADH1=1,
 $END
 $JET
  KDTL2=1,KEULER2=1,ETA2=.6,Q3I2=.05,Q4I2=.005,Q3J2=0.05,Q4J2=.005,
  NIT12=1,NIT22=50,NIT32=2000,ADH2=2,
 $END
 $GASPROP
  TC=1120,PC=6.38,CPG=0.296E4,GAM=1.357,BMU0=0.386E-4,OMEGAV=0.691,
  PRTL=0.409,WMOL=10.7,
 $END
```

Figure 4-6 : test.NSI : example of NAVIER input file.

4.3 NAVIER OUTPUT FILES DESCRIPTION

4.3.1 .THERMO file

This is the interface file to MATFLOW and MATPLIMP. It is the gas table giving, versus the temperature : H, W_{mol}, γ, T, P, C_p, μ, Pr, and is created only if OVER = T.

4.3.2 .FLOW file

This is the interface file to MCLIP, PROLOG and TRAJET. It contains the results of the plume calculation (ρ, V, T, Mach number…) in the computation domain.

4.3.3 .T45 file

This is the interface file to MATFLOW. It contains the results of the plume calculation (ρ, V, T, Mach number…) in the computation domain.

4.3.4 _CNV.FLOW file

This is the interface file to TRAJET. It contains the residuals in all the computation domain allowing the user to check the convergence of the NAVIER calculation.

4.3.5 .REP file

This is the NAVIER restart file. It allows to restart a calculation and is automatically created by the NAVIER module.

4.3.6 .NSO file

This is the listing file of NAVIER, shared in several parts :

1. review of the input data (see Figure 4-7).
2. calculation progress : evolution of thrust and residuals with number of iterations for the nozzle and for the plume (see Figure 4-8)
3. results of calculation : flow parameters at different locations (see Figure 4-9) : Mach number, pressure, temperature, density ...

Hereafter, the listing is given in the case of the thruster *test*.

```
PROGRAM OPTIONS :

   KRAZ    =     0
   IPRINT  =     1      KPRI1   = 0,0,0,0,0,0,0,0,0,0

   NIT11   =   100      NIT12   =     1
   NIT21   =   100      NIT22   =    50
   NIT31   =  1000      NIT32   =  2000

 COMPUTATION OPTIONS :

   KEULER1 =     0      KEULER2 =     1
   NGR1    =     3      NGR2    =     3
   NITF1   = 10000      NITF2   = 10000
   NGRF1   =     1      NGRF2   =     1

 ARTIFICIAL VISCISITY :

   QIV1    =      1.000 QIV2    =      1.000
   QJV1    =      1.000 QJV2    =      1.000
   Q3I1    =      0.050 Q3I2    =      0.050
   Q4I1    =      0.005 Q4I2    =      0.005
   Q3J1    =      0.050 Q3J2    =      0.050
   Q4J1    =      0.005 Q4J2    =      0.005

 TIME STEPS :

   ETA1    =      0.300 ETA2    =      0.600
   KDTL1   =     1      KDTL2   =     1
   QDT     =      1.030

 MULTIGRID TRANSFER :

   ETP     =      1.000


 FLOW FIELD CHARACTERISTICS :

   CP/CV   =      1.357
   REN     =    10670.8
   PRTL    =      0.409
   BMU0    =  0.386E-04 KG/M/S
   OMEGAV  =      0.691
   TC      =  0.112E+04 K
   PC      =  0.638E+06 Pa
   WMOL    =  0.107E-01 KG/Mole
   RGP     =  0.777E+03 J/KG/K
   CPG     =  0.295E+04 J/KG/K
```

Figure 4-7 : test.NSO - review of the input data.

```
--    900 TH ITERATION :     NIT= 900     (NGRID = 3    ETP =  1.0000)

               RELATIVE INCREASE OF THE SOLUTION :

       DU0/DT  AVE :  9.820E-04   MAX : -1.632E-02    I=  16 , J=  49
       DU1/DT  AVE :  8.682E-04   MAX : -1.270E-02    I=  32 , J=  44
       DU2/DT  AVE :  5.323E-04   MAX :  8.943E-03    I=  12 , J=  44
       DU3/DT  AVE :  7.743E-04   MAX : -9.974E-03    I=  16 , J=  49
       DU0/U0  AVE :  1.173E-05   MAX : -5.075E-05    I=  55 , J=  48
       DU1/U   AVE :  1.324E-05   MAX :  6.826E-05    I=   6 , J=  49
       DU2/U   AVE :  5.923E-06   MAX : -5.319E-05    I=  82 , J=  49
       DU3/U3  AVE :  8.545E-06   MAX :  4.770E-05    I=  40 , J=   1


     THRUST   :   FX =  0.20405E+01 NEWTONS

     FLOW RATE   : DEBR = 0.88315E-03 KG/S
     (computed on the thrust line)


     FLOW RATE RATIO INLET/OULET DOMAIN :  RDEB =0.1001E+01



 --   1000 TH ITERATION :     NIT=1000     (NGRID = 3    ETP =  1.0000)

               RELATIVE INCREASE OF THE SOLUTION :

       DU0/DT  AVE :  7.085E-04   MAX : -1.347E-02    I=  15 , J=  49
       DU1/DT  AVE :  6.084E-04   MAX :  8.962E-03    I=   4 , J=  49
       DU2/DT  AVE :  3.444E-04   MAX :  6.379E-03    I=  12 , J=  44
       DU3/DT  AVE :  5.484E-04   MAX :  7.749E-03    I=   9 , J=  49
       DU0/U0  AVE :  9.092E-06   MAX : -4.210E-05    I=  56 , J=  48
       DU1/U   AVE :  9.753E-06   MAX :  5.122E-05    I=   6 , J=  49
       DU2/U   AVE :  3.922E-06   MAX : -3.991E-05    I=  82 , J=  49
       DU3/U3  AVE :  6.335E-06   MAX :  3.626E-05    I=  42 , J=   1


     THRUST   :   FX =  0.20401E+01 NEWTONS

     FLOW RATE   : DEBR = 0.88245E-03 KG/S
     (computed on the thrust line)


     FLOW RATE RATIO INLET/OULET DOMAIN :  RDEB =0.9937E+00
```

Figure 4-8 : test.NSO – progress summary

```
     **********  END OF NOZZLE COMPUTATION  **********
1RESULTS IN J =   1 :

    I    X       Y       MACH   PA/PA0 HA/HA0    ANGLE VELOCITY   RHO     TEMP

   1   -8.38    0.00     0.156  1.000  1.000      0.00  168.98 0.724E+00 1115.17
   2   -7.03    0.00     0.157  1.000  1.000      0.00  169.83 0.724E+00 1115.14
   3   -5.89    0.00     0.158  1.000  1.000      0.00  171.04 0.724E+00 1115.07
   4   -4.94    0.00     0.158  1.000  1.000      0.00  171.78 0.724E+00 1114.99
   5   -4.13    0.00     0.159  1.000  1.000      0.00  171.98 0.724E+00 1114.99
   6   -3.46    0.00     0.159  1.000  1.000      0.00  172.90 0.724E+00 1115.01
   7   -2.89    0.00     0.164  1.001  1.000      0.00  177.71 0.724E+00 1114.92
   8   -2.41    0.00     0.176  1.002  1.000      0.00  191.30 0.723E+00 1114.35
   9   -2.01    0.00     0.201  1.003  1.001      0.00  217.71 0.720E+00 1112.84
  10   -1.67    0.00     0.239  1.003  1.001      0.00  258.42 0.714E+00 1109.87
  11   -1.38    0.00     0.289  1.003  1.001      0.00  311.90 0.705E+00 1104.95
  12   -1.14    0.00     0.348  1.003  1.001      0.00  374.83 0.692E+00 1097.79
  13   -0.94    0.00     0.414  1.002  1.001      0.00  443.22 0.675E+00 1088.36
  14   -0.77    0.00     0.481  1.001  1.001      0.00  512.38 0.655E+00 1077.09
  15   -0.62    0.00     0.546  1.001  1.001      0.00  578.10 0.634E+00 1064.83
  16   -0.50    0.00     0.605  1.000  1.001      0.00  637.82 0.613E+00 1052.44
  17   -0.40    0.00     0.660  0.999  1.001      0.00  690.89 0.593E+00 1040.39
  18   -0.32    0.00     0.708  0.999  1.001      0.00  737.17 0.575E+00 1029.11
  19   -0.25    0.00     0.750  0.998  1.001      0.00  777.14 0.559E+00 1018.78
  20   -0.19    0.00     0.786  0.998  1.001      0.00  811.41 0.545E+00 1009.50
  21   -0.13    0.00     0.818  0.997  1.001      0.00  840.73 0.533E+00 1001.25
  22   -0.09    0.00     0.846  0.997  1.001      0.00  865.77 0.522E+00  993.97
  23   -0.06    0.00     0.869  0.997  1.001      0.00  887.19 0.512E+00  987.58
  24   -0.03    0.00     0.890  0.997  1.001      0.00  905.51 0.504E+00  981.98
  25    0.00    0.00     0.908  0.997  1.001      0.00  921.23 0.497E+00  977.09
  26    0.03    0.00     0.925  0.997  1.001      0.00  936.24 0.491E+00  972.38
  27    0.05    0.00     0.943  0.997  1.001      0.00  952.11 0.483E+00  967.32
  28    0.08    0.00     0.963  0.997  1.001      0.00  969.34 0.476E+00  961.73
  29    0.11    0.00     0.984  0.997  1.001      0.00  987.98 0.467E+00  955.57
  30    0.15    0.00     1.008  0.997  1.001      0.00 1008.12 0.458E+00  948.79
  31    0.19    0.00     1.034  0.997  1.001      0.00 1029.87 0.448E+00  941.31
  32    0.23    0.00     1.062  0.997  1.001      0.00 1053.36 0.437E+00  933.06
  33    0.27    0.00     1.093  0.997  1.001      0.00 1078.71 0.425E+00  923.95
  34    0.32    0.00     1.127  0.997  1.001      0.00 1106.02 0.412E+00  913.89
  35    0.37    0.00     1.164  0.997  1.001      0.00 1135.42 0.398E+00  902.79
  36    0.42    0.00     1.204  0.997  1.001      0.00 1167.02 0.383E+00  890.54
  37    0.48    0.00     1.249  0.998  1.001      0.00 1200.88 0.367E+00  877.04
  38    0.55    0.00     1.297  0.998  1.001      0.00 1237.04 0.350E+00  862.19
  39    0.62    0.00     1.351  0.998  1.001      0.00 1275.50 0.332E+00  845.91
  40    0.69    0.00     1.409  0.998  1.001      0.00 1316.25 0.313E+00  828.12
  41    0.77    0.00     1.472  0.998  1.001      0.00 1359.16 0.292E+00  808.77
  42    0.86    0.00     1.541  0.998  1.001      0.00 1404.12 0.272E+00  787.83
```

Figure 4-9 : test.NSO – Parameters along the stream lines

5 HOW TO USE NAVIER

The goal of this chapter is to present the use of the NAVIER module on a real case and to give advices to the user.

5.1 APPLICATION CASE

In this paragraph, a complete case of NAVIER application case is presented. Usually, the NAVIER calculation proceeds in three computation steps :

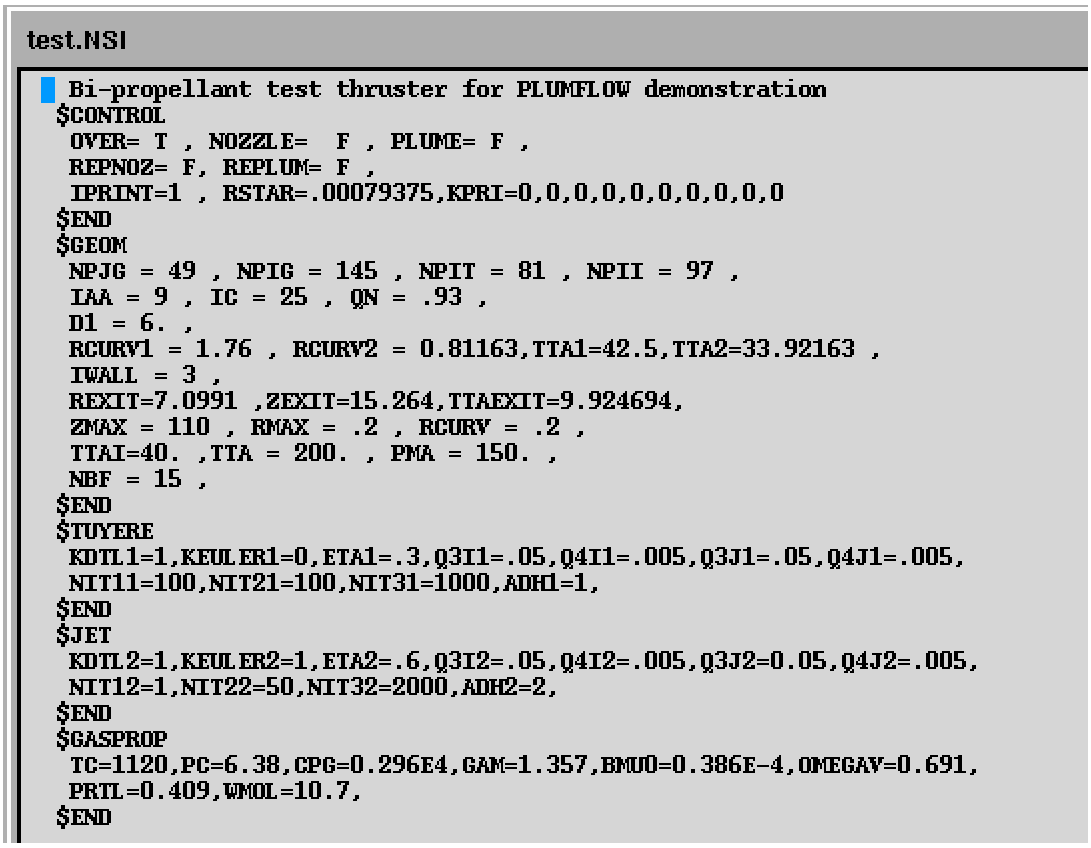
- Mesh generation,
- Nozzle flow field calculation,
- Plume flow field calculation.

5.1.1 External input file

As the thermodynamics characteristics are defined using the Namelist GASPROP, no external input file is required. Nevertheless, the same calculation can be performed after performing an ODE computation. In this case a .THERMO has been generated.

5.1.2 Verification of the mesh

First of all, the user shall generate the .NSI file containing the mesh definition and the run parameters. To do that that, the user can edit the .MLI file using the PLUMFLOW interface. In order to edit the NAVIER input file, the user has to click on ***Edit input file*** and then on ***NAVIER***. Using the editor, the user can enter the following file.



```
 Bi-propellant test thruster for PLUMFLOW demonstration
$CONTROL
 OVER= T , NOZZLE=  F , PLUME= F ,
 REPNOZ= F, REPLUM= F ,
 IPRINT=1 , RSTAR=.00079375,KPRI=0,0,0,0,0,0,0,0,0,0
$END
$GEOM
 NPJG = 49 , NPIG = 145 , NPIT = 81 , NPII = 97 ,
 IAA = 9 , IC = 25 , QN = .93 ,
 D1 = 6. ,
 RCURV1 = 1.76 , RCURV2 = 0.81163,TTA1=42.5,TTA2=33.92163 ,
 IWALL = 3 ,
 REXIT=7.0991 ,ZEXIT=15.264,TTAEXIT=9.924694,
 ZMAX = 110 , RMAX = .2 , RCURV = .2 ,
 TTAI=40. ,TTA = 200. , PMA = 150. ,
 NBF = 15 ,
$END
$TUYERE
 KDTL1=1,KEULER1=0,ETA1=.3,Q3I1=.05,Q4I1=.005,Q3J1=.05,Q4J1=.005,
 NIT11=100,NIT21=100,NIT31=1000,ADH1=1,
$END
$JET
 KDTL2=1,KEULER2=1,ETA2=.6,Q3I2=.05,Q4I2=.005,Q3J2=0.05,Q4J2=.005,
 NIT12=1,NIT22=50,NIT32=2000,ADH2=2,
$END
$GASPROP
 TC=1120,PC=6.38,CPG=0.296E4,GAM=1.357,BMU0=0.386E-4,OMEGAV=0.691,
 PRTL=0.409,WMOL=10.7,
$END
```

The mesh parameters are defined in the $GEOM section. Note that NOZZLE = F and PLUME = F. So, no NAVIER calculation will be performed and only the mesh will be generated to allow the user to check it.

To perform the calculation, the user has to select ***NAVIER*** and ***OK***. The NAVIER program is then executed and after few seconds the user can remark that a test.NSO and a test.FLOW files have been created.

After completion of the run the user can visualise the mesh using the TRAJET module. The generated mesh of the above case is presented at Figure 5-1.

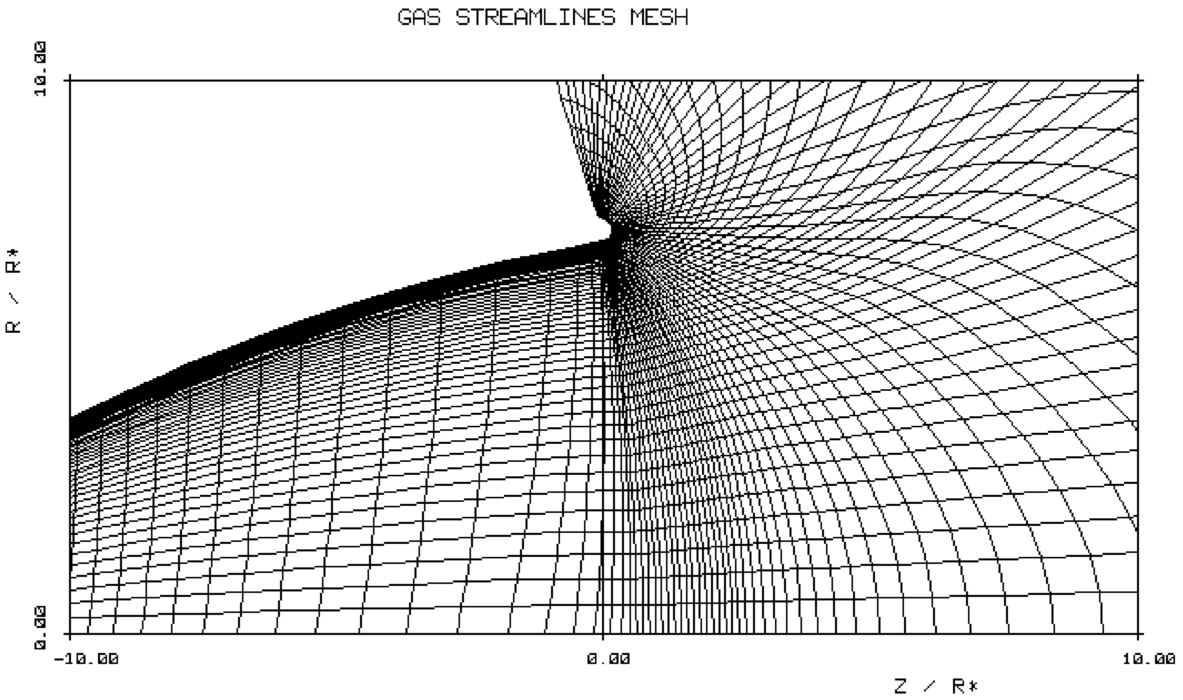
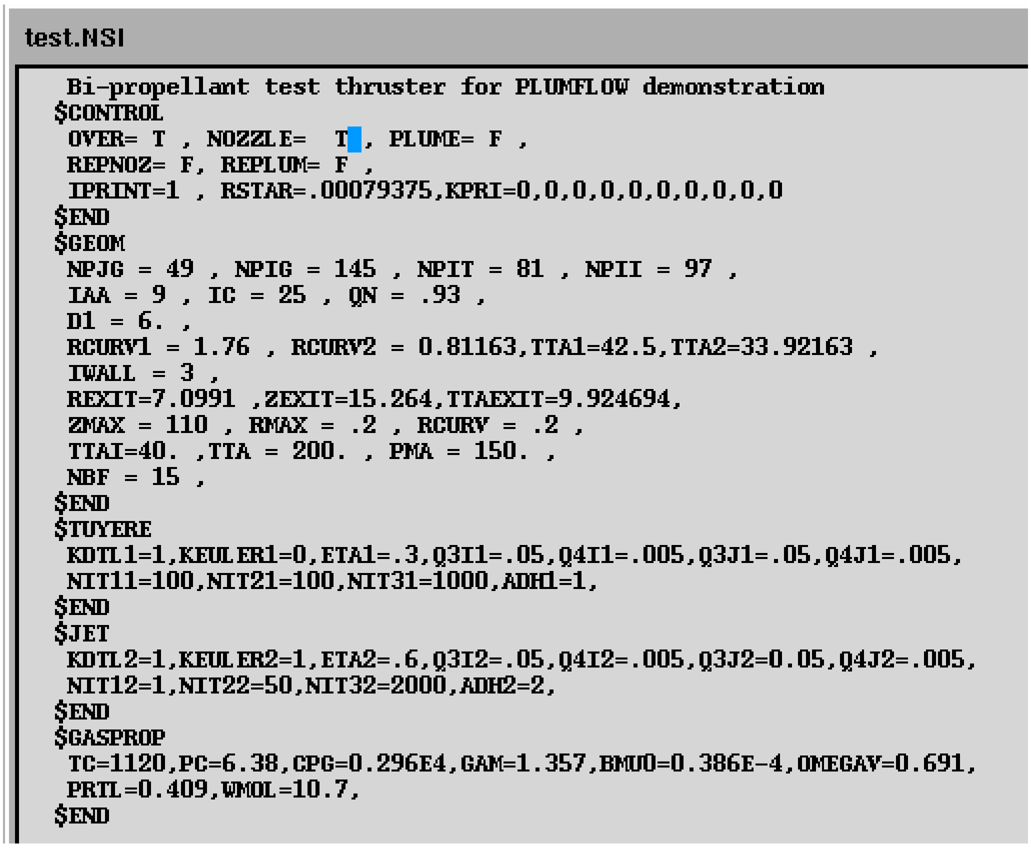


Figure 5-1 : Mesh generated by NAVIER

5.1.3 Calculation of the nozzle flow field

The first step of calculation consists in the calculation of the flow field properties inside the nozzle. To do that the user has to put the NOZZLE parameter at TRUE. The run parameters are defined in the $TUYERE section. The main parameters to be entered are the following :

- ETA1 = 0.3
- Q3I1 = 0.05, Q4I1 = 0.005, Q3J1 = 0.05, Q4J1 = 0.005
- NIT31 = 1000



```
 Bi-propellant test thruster for PLUMFLOW demonstration
$CONTROL
 OVER= T , NOZZLE=  T , PLUME= F ,
 REPNOZ= F, REPLUM= F ,
 IPRINT=1 , RSTAR=.00079375,KPRI=0,0,0,0,0,0,0,0,0,0
$END
$GEOM
 NPJG = 49 , NPIG = 145 , NPIT = 81 , NPII = 97 ,
 IAA = 9 , IC = 25 , QN = .93 ,
 D1 = 6. ,
 RCURV1 = 1.76 , RCURV2 = 0.81163,TTA1=42.5,TTA2=33.92163 ,
 IWALL = 3 ,
 REXIT=7.0991 ,ZEXIT=15.264,TTAEXIT=9.924694,
 ZMAX = 110 , RMAX = .2 , RCURV = .2 ,
 TTAI=40. ,TTA = 200. , PMA = 150. ,
 NBF = 15 ,
$END
$TUYERE
 KDTL1=1,KEULER1=0,ETA1=.3,Q3I1=.05,Q4I1=.005,Q3J1=.05,Q4J1=.005,
 NIT11=100,NIT21=100,NIT31=1000,ADH1=1,
$END
$JET
 KDTL2=1,KEULER2=1,ETA2=.6,Q3I2=.05,Q4I2=.005,Q3J2=0.05,Q4J2=.005,
 NIT12=1,NIT22=50,NIT32=2000,ADH2=2,
$END
$GASPROP
 TC=1120,PC=6.38,CPG=0.296E4,GAM=1.357,BMU0=0.386E-4,OMEGAV=0.691,
 PRTL=0.409,WMOL=10.7,
$END
```

After completion of the run, the user can edit the .NSO file using the PLUMFLOW integrated editor. The main results are summarized below :

- The mass flow rate conservation reaches less than 1 %,
- The maximum residuals is equal to 1.3 10^{-2},
- The Mach number at the lip edge (result in J = 49 and I = 97) reaches 1.14.

These results are satisfactory. In order to decrease the residuals, the user can restart the nozzle computation with 1000 more iterations.

To do that, the user has only to put the REPNOZ parameter at TRUE.

After completion of the run, it is possible to check the main characteristics of the run :

- The mass flow rate conservation reaches less than 0.1 %,
- The maximum residuals is equal 2.5 10^{-3},
- The Mach number at the lip edge (result in J = 49 and I = 97) reaches 1.10.

The user can also visualise the flow field characteristics inside the nozzle. As example the density is given at Figure 5-2.

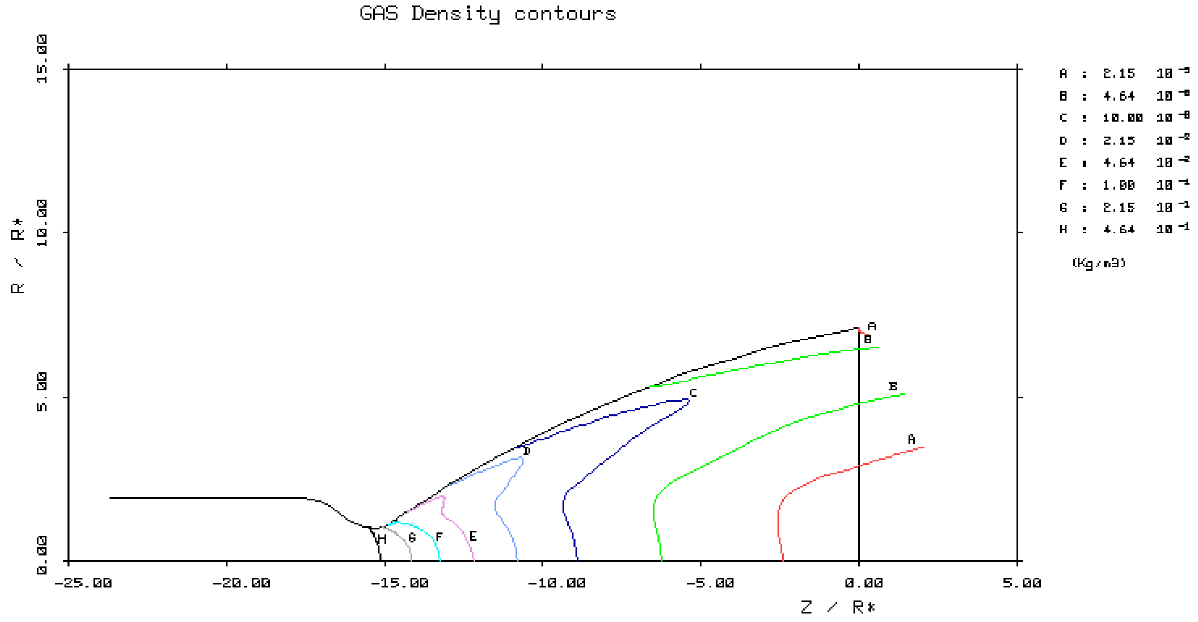


Figure 5-2 : Flow density inside the nozzle

The obtained results are sufficiently well converged to the be used to perform the external flow field calculation.

5.1.4 Calculation of the plume flow field

The second step of calculation consists in the calculation of the flow field properties inside the nozzle. To do that the user has to put the PLUME parameter at TRUE and REPNOZ at TRUE to keep the flow inside the thruster for visualisation. The run parameters are defined in the $PLUME section. The main parameters to be entered by the user are the following :

- ETA2 = 0.1
- Q3I2 = 0.1, Q4I2 = 0.01, Q3J2 = 0.1, Q4J2 = 0.01
- NIT32 = 1000

After NAVIER completion, the main results are summarized below :

- The mass flow rate conservation reaches 16 %,
- The maximum residuals is equal 2 10^{-2}.

In order to improve the mass flow conservation, it is necessary to restart the calculation (REPLUM = T) with an increased time step and a decreased artificial viscosity. The main parameters to be entered by the user are the following :

- ETA2 = 0.2
- Q3I2 = 0.05, Q4I2 = 0.005, Q3J2 = 0.05, Q4J2 = 0.005
- NIT32 = 1000

After NAVIER completion, the mass flow conservation is better achieved. The main results are summarized below :

- The mass flow rate conservation reaches 6 %,
- The maximum residuals is equal 8 10^{-5}.

The 6 % of mass flow conservation is not completely satisfactory and a final run has to be performed. The main parameters to be entered by the user are the following :

- ETA2 = 0.6
- Q3I2 = 0.05, Q4I2 = 0.005, Q3J2 = 0.05, Q4J2 = 0.005
- NIT32 = 1000

The main results are summarized below :

- The mass flow rate conservation reaches 2.8 %,
- The maximum residuals is equal 3 10^{-5}.

The results can be visualised using the flow field outside the nozzle using the TRAJET module. As example the density is given at Figure 5-3.

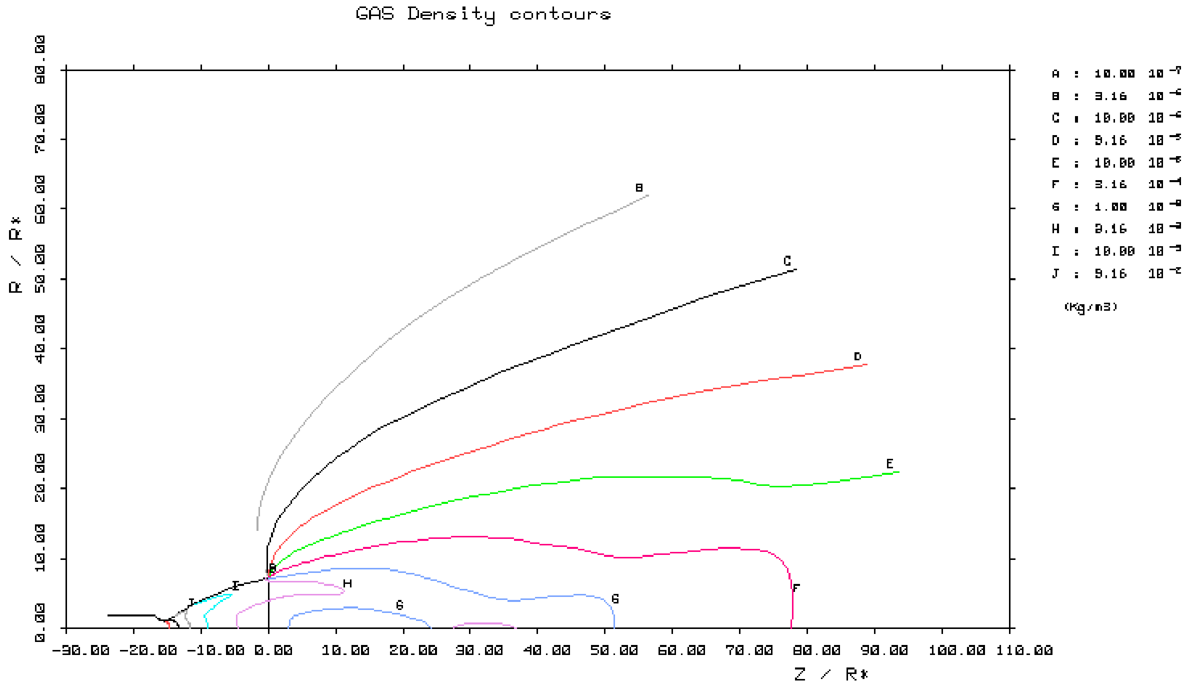


Figure 5-3 : Flow density inside the nozzle

5.2 ADVISES TO THE USER

5.2.1 On the methodology

The general methodology to perform a NAVIER calculation is the following :

- Computation of the nozzle flow,
- Then, computation of the plume flow.

Thus, it is recommended to proceed in this way :

- To generate only the mesh (the NOZZLE and PLUME parameters have to be defined as FALSE),
- To check the mesh using the TRAJET module,
- If the mesh is correct : no discontinuity, the mesh line crossing, no disturbed mesh, the nozzle and plume calculation may start.

5.2.2 On the convergence criterion

We can consider that the convergence of the calculation is achieved when :

- The residuals at the end of the calculation reach :
    - Nozzle part : $\left(\frac{du}{dt}\right)_{max} \cong 5.10^{-3}$
    - Plume part : $\left(\frac{du}{dt}\right)_{max} \cong 10^{-5}$
- The mass flow rate conservation is obtained with a few percents.

Moreover, for the nozzle part, the Mach number should be supersonic at the lip edge. This can be checked in the .MPO file, verifying the Mach number of the (NPII, NPJG) point. If it is not the case, the user shall restart the calculation with a lower artificial viscosity.

5.2.3 On the convergence improvement

The NAVIER convergence depends essentially on the time step, the artificial viscosity and the mesh. So, the user can play with these three aspects to improve the convergence. In particular, the lower is the time step and the higher is the artificial viscosity, the easier is the convergence.

If the NAVIER module diverges, the program stops. The parameters that can be modified by the user to improve the convergence are the following :

- For the nozzle part : to reduce the time step (ETA1), and eventually to increase the artificial viscosity.
- For the nozzle part : to reduce the time step (ETA2), and eventually to increase the artificial viscosity. It is also possible to reduce the backflow angle (PMA).

If the modification of these parameters is not sufficient, the solution is to modify the mesh. In particular, the following aspects may improve the computation convergence :

- Increasing of the number of mesh in the two directions,
- Modification of the mesh parameters in order to get more regular and less distorted cells.

5.3 ERRORS MANAGEMENT

5.3.1 NaN in the .NSI or in the .FLOW files

The presence of NaN in the output files generated by Navier is generally due to convergence problem. In order to solve this issue, it is recommended to restart the all the computation with:

- A lower time step
- An higher artificial viscosity

5.3.2 Anomalously slow progress of the computation

This is generally the case when there are convergence problem with generation of NaN in the Navier output file. To solve this issue refer to §5.3.1.

5.3.3 CONVERGENCE PROBLEM IN CINJET

This message is displayed when the plume computation cannot by initialised from the nozzle computation (intermediate zone). To solve this issue, it is recommended to reduce the size of the intermediate zone (between NPII and NPIT) by reducing the value of RATEX1 parameter.

Appendix A – THEORETICAL aspects

The NAVIER computation proceeds in two steps :

- Computation of the flow field inside the nozzle, taking into account the viscous effects (solving of the Navier-Stokes equations, computation of the boundary layer).
- Computation of the plume flow field without viscous effects, solving of the Euler equations.

Nevertheless, the choice between an Euler computation and a full Navier-Stokes calculation is up to the user.

The separation of the calculation in two stages has several advantages. Indeed, it allows :

- To work with different hypothesis in the nozzle and in the plume.
- To simulate the boundary layer in the nozzle using the full Navier-Stokes equations, and in the plume where the viscous effects are negligible, using a simple Euler solving. This allows to optimise the computation time.
- To check the proper convergence of the computation in the nozzle before starting the calculation of the plume.

The main advantages of the Navier-Stokes / Euler equations solving lies in the absence of a priori assumptions concerning the shape of the flow field at the nozzle lip. The computation converges itself towards the best solution. This is particularly important to properly compute the plume expansion from the boundary layer.

NAVIER module considers a gas with the following assumptions :

- The chemical composition is frozen,
- There is no condensed phase (particles or droplets),
- Cp and γ are constant,
- The viscosity depends on the temperature (Sutherland law).

Navier-stokes equations are solved using an explicit second-order finite-volumes scheme. The convergence is accelerated using a multi-grid method (four levels of grid are considered).

The numerical stability of the scheme relies on the CFL criterion limiting the integration time step. Because of the non-linearity of the flow (boundary layer, expansion at the nozzle lip, eventually compression waves), the stability is strengthened with an artificial viscosity allowing to smooth the numerical solution.

The artificial dissipation is written as :

$$\frac{\partial}{\partial x}\left\{\left(\frac{a\left|\dfrac{\partial^2 \varrho}{\partial x^2}\right|}{\varrho}+b\right)\frac{\partial U}{\partial x}\right\}$$

Where x is the variable of space and U is the variable representing the flow field ($U = (p, \rho u, \rho v, \rho E)$).

Resolving the above equation on the mesh directions, we obtain :

$$DU=\frac{\partial}{\partial i}\left\{\left[Q_{3i}\frac{\left|\dfrac{\partial^2\varrho}{\partial i^2}\right|}{\varrho}+Q_{4i}\right]\frac{\partial U}{\partial i}\right\}+\frac{\partial}{\partial j}\left\{\left[Q_{3j}\frac{\left|\dfrac{\partial^2\varrho}{\partial j^2}\right|}{\varrho}+Q_{4j}\right]\frac{\partial U}{\partial j}\right\}$$

This allows to retrieve the four parameters Q_{3i}, Q_{4i}, Q_{3j}, Q_{4j} defining the artificial viscosity.

DISTRIBUTION LIST

	Overall document		Summary
	Action	**Information**	