ESPSS

European Space Propulsion System Simulation

Software Verification and Validation Plan

Doc. 4000103800/11/NL/CP – TN-4120	
ESPSS Version: 3.0	Date: 30-11-2013

Client : ESA

Contract: ESPSS-3: 4000103800/11/NL/CP

Main Contractor Empresarios Agrupados

Contributors : Astrium / Cenaero / KopooS / VKI

ABSTRACT:

This document describes the tests designed, planned and run on Version 3.0 of the ESPSS Libraries under EcosimPro 5.2. The aims of this document are the following:

A/: To ensure that components, functions and models perform as expected.

B/: To check that the User's Manual and the code of all the elements of the libraries are in agreement.

C/: To get started with the use of the ESPSS libraries.

This version includes new features as the simulation of solid/hybrid combustors, ramjets, scramjets and the convection/mixing of combusted gases downstream a chamber. It also includes new libraries for steady/quasi-steady models: The STEADY Library designing rocket engine cycles, the SATELLITE Library for orbital and attitude motion and the ELECTRICAL_PROPULSION Library.

New useful tips and application examples have been added helping the user defining new models.

Revision table

ESPSS Version	Issue date	Modifications	Page
V1.0	29-01-2008	First version	
V1.4.1	05-06-2009	Version issued after the Industrial Validation Phase (ESPSS-II)	
V2.0	01-02-2010	This Version corresponds to the final delivery of ESPSS-II with Version 4.6 of EcosimPro	
V2.2	01-09-11	This version accounts for the upgrades and experience acquired since Version 2.0 was released. New tips and comments have been added. New test cases have been added to be compared with theoretical solutions.	
V2.4	30-04-12	Present version gains in robustness, includes new components for the mixture process of combusted gases, upgrades the TANKS library accounting for film boiling phenomena and better simulation of the generalized boiling process and includes new libraries for direct STEADY calculation designing cycles.	
V3.0.beta	29-09-13	This version includes new features as the simulation of solid/hybrid combustors, ramjets, scramjets and the convection/mixing of combusted gases downstream a chamber. It also includes new libraries for steady/quasi-steady models: The STEADY Library designing rocket engine cycles, the SATELLITE Library for orbital and attitude motion and the ELECTRICAL_PROPULSION Library.	
V3.0	30-11-13	*This version stabilizes ESPSS 3.0.beta and provides a first operational version for solid, hybrid and ramjets components, and for the simulation of absorption / desorption in Pipes and Tanks*	

INDEX

1. INTRODUCTION

1.1 PURPOSE OF THE DOCUMENT

This document describes the tests designed, planned and run on Version 3.0 of the ESPSS Libraries under EcosimPro 5.2. The aims of this document are the following:

- To ensure that components, functions and models perform as expected.
- To check that ESPSS User's Manual and the code of all the elements are in agreement.
- To get started with the use of the ESPSS libraries

There are only two applied verification methods: inspections of the documentation and acceptance tests. Any kind of check that requires running the software is considered to be an acceptance test case, and not an inspection. An inspection is always done at documentary level.

Testing shall be organised into particular models concerning the different areas covered by the libraries. Tests are of two kinds: unitary tests focused on the components containing the basic equations, and model tests focused on the system cases. If an error is found as a result of testing, the cause of the bug can be more easily isolated. There shall also be integrated test cases to demonstrate that the complete system behaves as expected.

Version 2.0 accounts for the upgrades and experience acquired under the Industrial Evaluation phase [RD-1] and for the new capabilities developed for the Phase II of the ESPSS.

Version 2.2 accounts for the upgrades and experience acquired since Version 2.0 was released mainly derived from the user feed-backs.

Version 2.4 gets a gain in robustness, accounts for the experience acquired since Version 2.0 was released, includes new components for the mixture process of combusted gases, upgrades the TANKS library accounting for film boiling phenomena and includes new libraries for direct STEADY calculation designing cycles.

This version 3.0 includes new features as the simulation of solid/hybrid combustors, ramjets, scramjets and the convection/mixing of combusted gases downstream a chamber. It also includes new libraries for steady/quasi-steady models: The STEADY Library designing rocket engine cycles, the SATELLITE Library for orbital and attitude motion and the ELECTRICAL_PROPULSION Library. New useful tips and application examples have been added to help the user define new models

New simple test cases have been added in this version, *so they can be compared with theoretical solutions*. These new cases cover interesting physical phenomena such as the flow through orifices, Fanno tubes and supersonic nozzles completing the existing test cases concerning the basic formulation of Volumes, Junctions and 1D pipes under gas, liquid or two-phase flow. Moreover, additional test cases have been included for the validation of hydraulic tanks and combustors equilibrium composition.

1.2 APPLICABLE DOCUMENTS

The following documents are applicable to this project:

AD-1 086-038-XY-L-X1820 "ESPSS. Management Proposal"

AD-2 086-038-XY-L-X1819 "ESPSS. Technical Proposal"

AD-3 4000103800/11/NL/CP – TN-4130 "ESPSS. Main libraries User Manual"

1.3 REFERENCE DOCUMENTS

RD-1 12205/07/NL/CP – TN-2110. ESPSS Phase II: Feasibility Demonstration of an European Space Propulsion System Simulation Tool. Industrial Evaluation

RD-2 C. Hirsch. Numerical Computation of Internal and External Flows. Volume I and II. Wiley Series in Numerical Methods in Engineering, USA, 1988

RD-3 E.F. Toro. Riemann Solvers and Numerical Methods for Fluid Dynamics. 2nd Edition. Springer, Germany, 1999.

RD-4 G.A. Sod. A Survey of Several Finite Difference Methods for Systems of Nonlinear Hyperbolic Conservation Laws. J. Comput. Phys., 27:1–31, 1978

RD-5 R.J. Leveque. Finite Volume Methods for Hyperbolic Problems. Cambridge Texts in Applied Mathematics, United Kingdom, 2002

RD-6 S. Karni and J.J. Quirk. On the Dynamics of a Shock Bubble Interaction. CR 194978, ICARE, NASA, NASA Langley Research Center, Hampton, VA 23681-0001, September 1994

RD-7 R. Abgrall. How to Prevent Pressure Oscillations in Multicomponent Flow Calculations: a Quasi Conservative Approach. J. Comput. Phys., 14:150–160, 1996

RD-8 R. Abgrall et al. Comparisons of Several Upwind Schemes for Multi-Component One-Dimensional Inviscid Flows. Rapport de Recherche 1253, INRIA, Le Chesnay, France, June 1990

RD-9 A.R. Simpson. Large water hammer pressures due to column separation in sloping pipes (transient, cavitation). PhD thesis, The University of Michigan, 1986

RD-10 J. Gale and I. Tiselj. Water hammer in elastic pipes. In roceedings of the International Conference Nuclear Energy for New Europe. Kranjska Gora, Slovenia, 2002

RD-11 A.S. Tijsseling and A. Anderson. The Joukowsky equation for fluids and solids. CASA Report, Technische Universiteit Eindhoven, Eindhoven, March 2006

RD-12 V.L. Streeter E.B. Wylie. Fluid Transients. McGraw-Hill, USA, 1978

RD-13 I. Tiselj et al. WAHA3 code manual. JSI Report, Jozef Stefan Institute, Ljubljana, Slovenija, March 2004

RD-14 S. Gordon & B.J. McBride, Computer Program for Calculation of Complex Chemical Equilibrium Compositions and Applications, (RP-1311), Technical report, NASA, 1994

RD-15 Fluent Inc. Fluent 6.3 User's guide. Lebanon, NH, 2006

RD-16 I. Gibek, Y. Maisonneuve. "Waterhammer Tests With Real Propellants". Joint Propulsion Conference and Exhibit. AIAA 2005-4081. Tucson, 10-13 July 2005.

RD-17 M. Hanif Chaudhry. "Applied Hydraulic Transients". Van Nostrand Reinhold Company, 1979.

RD-18 12205/07/NL/CP – TN-2110. ESPSS-2: Feasibility Demonstration of an European Space Propulsion System Simulation Tool. Industrial Evaluation.

RD-19 Renaud Lecourt & Johan Steelant. Experimental Investigation of Waterhammer in Simplified Feed Lines of Satellite Propulsion. AIAA 29269-804. JOURNAL OF PROPULSION AND POWER. Vol. 23, No. 6, November–December 2007

RD-20 ESPSS-TN5120. Absorption and desorption validation

1.4 ACCEPTANCE CRITERIA

Every customisation component/model shall be tested by:

- Inspecting the correspondence of the User's Manual with the code of the involved components.
- Running transient solutions and evaluating the model dynamic responses for reasonableness.
- Comparing the test results with those included in this document

The expected results for each one of the tests are included and commented in this document and can be reproduced by running the corresponding model included in specific application libraries provided with the code.

2. DESIGN OF TESTS

2.1 FLUID_FLOW_1D LIBRARY

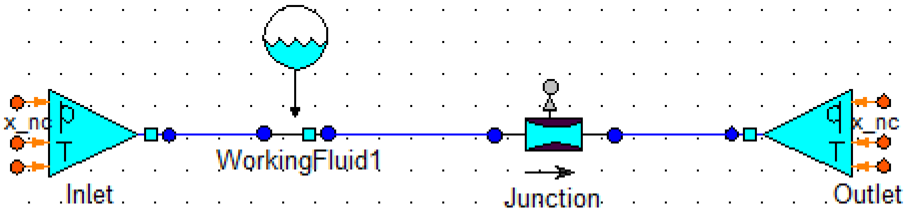
2.1.1 Orifice test case (T-FF-007)

Library:	FLUID_FLOW_1D_EXAMPLES
Model Name:	Test_ori
Partition Name:	default
Experiment Name:	exp_liq, exp_gas

2.1.1.1 Model description

This example validates the FLUID_FLOW_1D capabilities concerning the elementary Junction component formulation.

The model with ESPSS is representative of a concentrated pressure drop for liquid or gas conditions typically simulating an orifice or a Valve component. In both cases the choked flow limitation is tested (flushing flow in case of two phase flow).



2.1.1.2 Results

The test consists in evaluating the mass flow under a gradually greater pressure difference imposed through the Inlet/Outlet boundary conditions.

We summarize here below the theoretical expressions for the critical and laminar mass flows (see AD-3):

$$G_{lam} = \eta \, \mathrm{Re}_{lam} \, \mathrm{D}$$

$$G_{crit,gas} = \rho v \left(\frac{2 + (\gamma - 1) M^2}{\gamma + 1} \right)^{(\gamma+1)/2/(\gamma-1)}; \qquad M = v/c; \qquad \gamma = \frac{\rho c^2}{P}$$

$$G_{crit,subcooled} = \sqrt{2\rho (P - \alpha P_{sat})}; \qquad \alpha = 0.96 - 0.28 \sqrt{P_{sat} / P_{crit}}$$

Note that *Re_lam* and *D* (orifice diameter) are input data, so that laminar to turbulent transitions can be adjusted by the code. The expression for the theoretical mass flow should be:

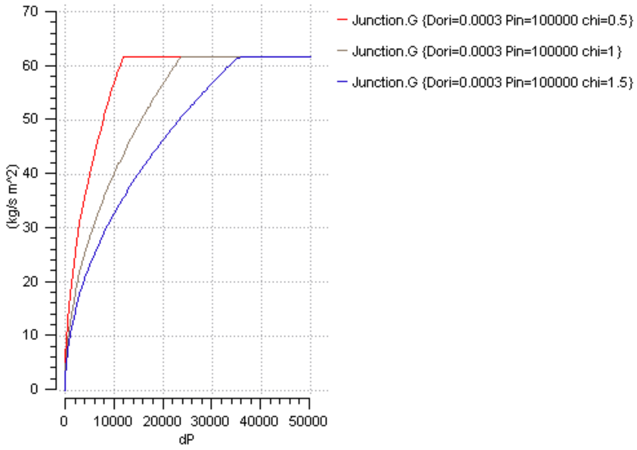
$$G_{ther} = \min(G_{crit}, \sqrt{2 \Delta P \rho_{in} / \xi})$$

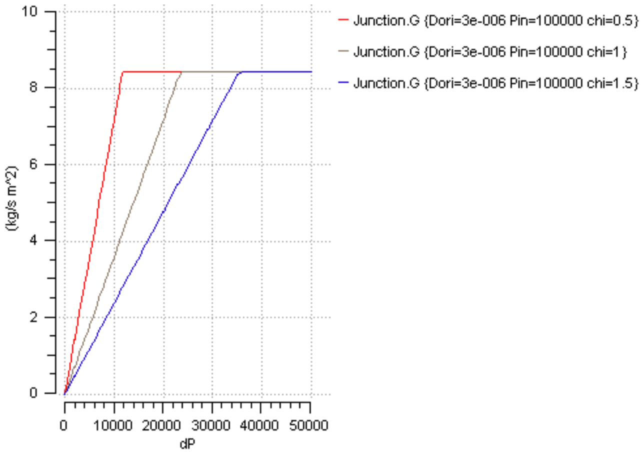
$$\xi : \text{pressure drop coef.}$$

Next figures show the mass flow per unit or area (G) for different pressure drop coefficients (chi):

Real H2 gas conditions

Pin = 1e5 Pa; Tin = 300 K; Gcrit= 61.65 kg/s/m2

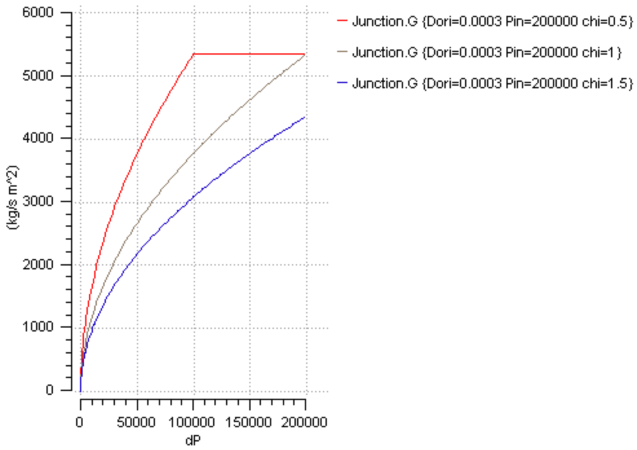


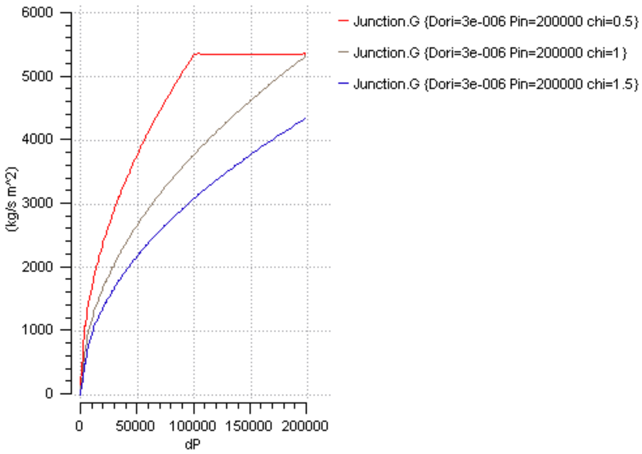


Re_lam = 200; Re = 2065.2 **Re_lam = 200; Re = 20.7**

Real H2 Liquid conditions.

Pin = 2e5 Pa; Tin = 20 K; Gcrit= 5353.6 kg/s/m2





Re_lam = 200; Re = 116415 **Re_lam = 200; Re = 1164**

Reynolds number is evaluated as: Re = Gcrit * D /visc. Main conclusions are:

- ESPSS model can simulate correctly subsonic- sonic transition.
- For lower Reynolds number (Re < Re_lam), it can be seen that the mass flow behaves linearly with dP, and that the critical flow cannot be reached if Gcrit < Glam (cases of very low pressure and orifices diameters).
- the results are very similar comparing real gas properties with perfect gas properties if the actual gas conditions are at low pressures and high temperatures
- For liquid conditions, ESPSS is able to calculate flushing conditions, see the limitation on the mass flow according to the Gcrit (sub cooling conditions) value

Theoretical (equilibrium) exact critical flow calculation (Junction.Gcr_exact =TRUE) would give nearly half values for the critical flow under liquid conditions, but the equilibrium conditions seems to be nonrealistic for flushing conditions.

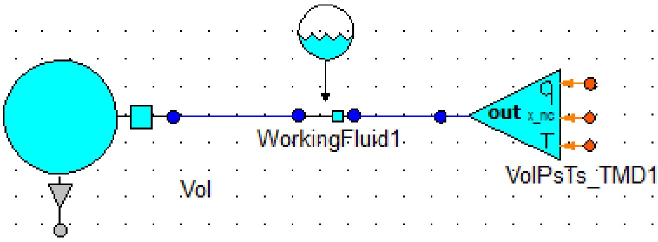
2.1.2 Volume test case (T-FF-008)

Library:	FLUID_FLOW_1D_EXAMPLES
Model Name:	Test_vol
Partition Name:	default
Experiment Name:	exp_liq, exp_gas

2.1.2.1 Model description

This example validates the FLUID_FLOW_1D capabilities concerning the elementary Volume component formulation.

The model with ESPSS is representative of an adiabatic lumped fluid volume for liquid, two-phase or gas conditions typically simulating an elementary tank component or a fluid capacity able to cumulate or dispense mass and energy through the connected ports.



Volume formulation (basically the mass and energy conservation equations) is stated in a very general way, without making use of any particular assumption concerning the fluid properties (for example making use of gamma value or assuming constant density for liquids, etc), so the formulation is valid for gas, liquid or two-phase flow.

Here below, we summarize these equations for a constant fluid geometrical volume in the particular case of a pure fluid (the case with non-condensable gases will be tested in the Tank library). For the general case formulation, see AD-3):

Mass conservation equation:

$$\frac{d\rho}{dt}V = \sum_{j \in Ports} m_j$$

Energy conservation equation:

$$\frac{d\rho}{dt}V\,E + \rho V \frac{dE}{dt} = \sum_{j \in Ports} (mH)_j; \qquad E = e + v^2/2$$

Previous conservation equations enable to calculate the derivatives of the mixture density and of the mixture energy. These variables can be integrated, so they are known at any time. Then, the complete thermodynamic state (pressure, temperature, *quality* …) *is calculated using the thermodynamic routines* entering with density and energy.

2.1.2.2 Results

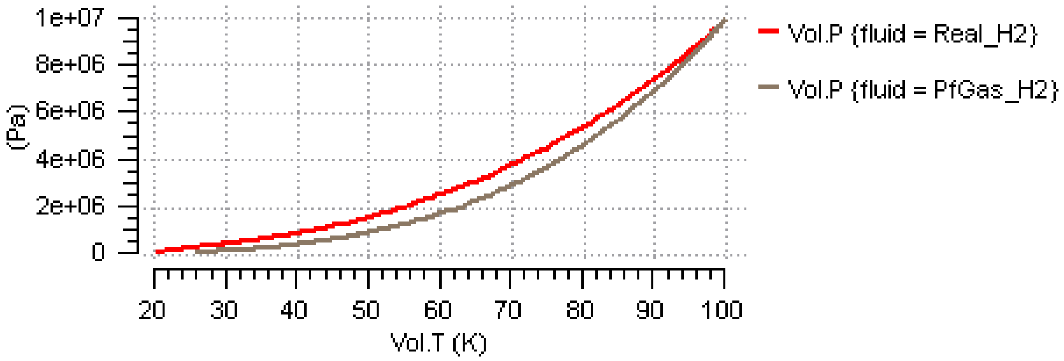
The test consists in calculating the pressure/temperature blow down evolution between an initial pressure of 100 bar and a fixed outlet pressure (VolPsTs_TMD1 component) of 1 bar. The exiting mass flow is calculated by this VolPsTs_TMD1 component as in the Junction component.

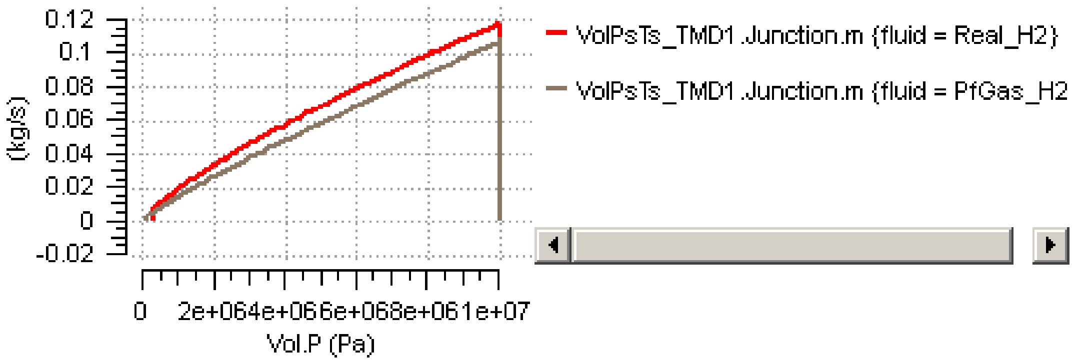
The volume exit area is 1e-5 m^2 and the geometrical volume is 1 m^3

Next figures show the pressure evolution as a function of the temperature in the volume. Two cases have been considered: two-phase flow and gas flow, in the last case real H2 properties vs. perfect gas properties have been compared:

Gas conditions

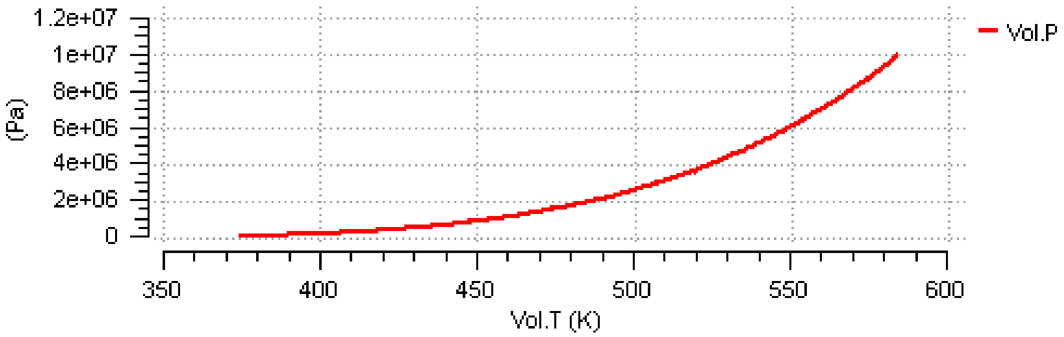
Fluid: H2; Pini = 100e5 Pa; Tini = 100 K

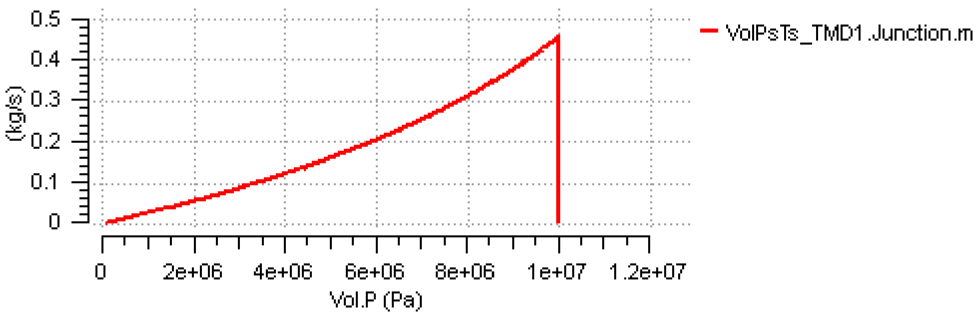




Two phase conditions

Fluid: H2O; Pini = 100e5 Pa; x_ini (quality)= 0.1





Main conclusions are:

- ESPSS model can simulate correctly the conditions of a fluid capacity under real or ideal fluid properties accordingly with the inlet/outlet mass exchanged.
- Using perfect gases properties, the pressure/temperature evolution corresponds exactly with the theoretical expression of $P/Po = (T/To)^{\gamma/(\gamma-1)}$, see figures above.
- Using real gases properties, the pressure/temperature evolution is correctly delayed with respect to an ideal expansion, see figures above.
- For two-phase conditions, the pressure/temperature evolution correctly corresponds to the saturation line.

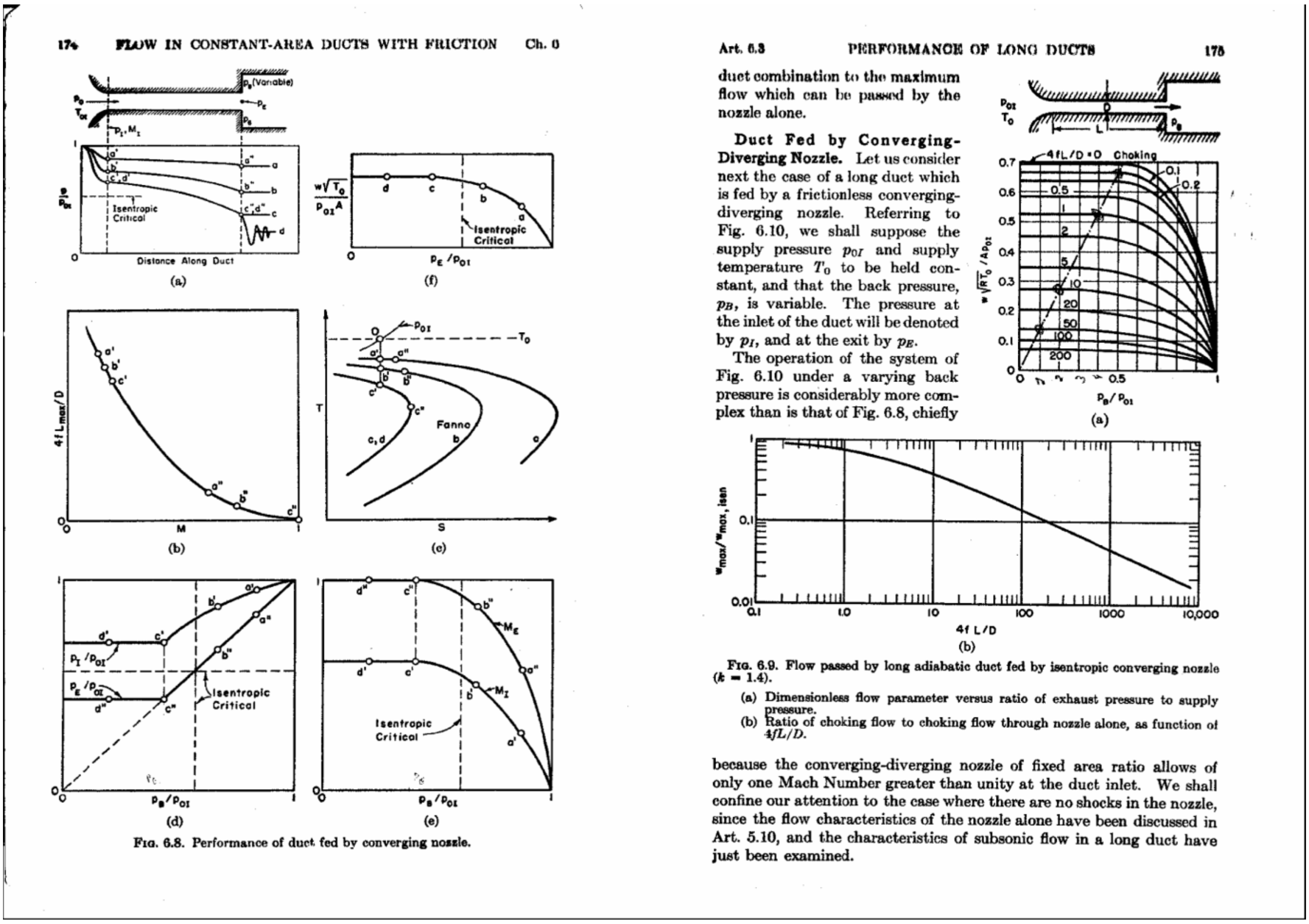
2.1.3 Pipe test: Subsonic Fanno flow (T-FF-009)

Library:	FLUID_FLOW_1D_EXAMPLES
Model Name:	aDemoShapiroFannoAdia
Partition Name:	default
Experiment Name:	+shapiro+p175+comparison

2.1.3.1 Model description

This example validates the FLUID_FLOW_1D capabilities concerning high speed compressible flow analysis in a pipe using real properties fluids. The pipe is simulated by the Tube component, which takes into account inertia forces, pressure losses and density changes.

We take as reference the Shapiro example contained in the book *"The Dynamics and Thermodynamics Of Compressible Fluid Flow", Ascher H. Shapiro, Volume 1, 1953, pages 174, 175:*



174 FLOW IN CONSTANT-AREA DUCTS WITH FRICTION Ch. 6

FIG. 6.8. Performance of duct fed by converging nozzle.

Art. 6.3 PERFORMANCE OF LONG DUCTS 175

duct combination to the maximum flow which can be passed by the nozzle alone.

Duct Fed by Converging-Diverging Nozzle. Let us consider next the case of a long duct which is fed by a frictionless converging-diverging nozzle. Referring to Fig. 6.10, we shall suppose the supply pressure p_{0I} and supply temperature T_0 to be held constant, and that the back pressure, p_B, is variable. The pressure at the inlet of the duct will be denoted by p_I, and at the exit by p_E.

The operation of the system of Fig. 6.10 under a varying back pressure is considerably more complex than is that of Fig. 6.8, chiefly

FIG. 6.9. Flow passed by long adiabatic duct fed by isentropic converging nozzle ($k = 1.4$).

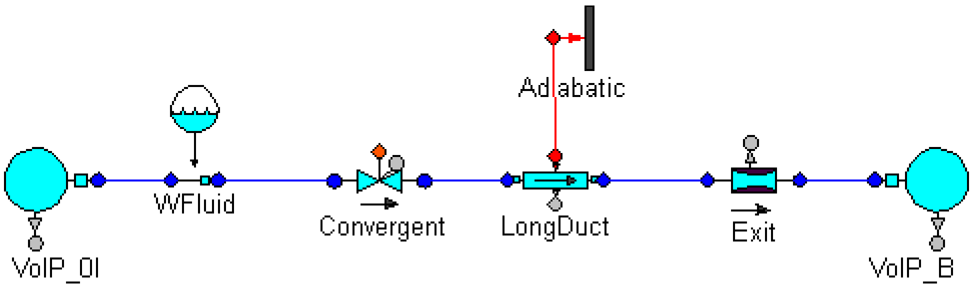
(a) Dimensionless flow parameter versus ratio of exhaust pressure to supply pressure.

(b) Ratio of choking flow to choking flow through nozzle alone, as function of $4fL/D$.

because the converging-diverging nozzle of fixed area ratio allows of only one Mach Number greater than unity at the duct inlet. We shall confine our attention to the case where there are no shocks in the nozzle, since the flow characteristics of the nozzle alone have been discussed in Art. 5.10, and the characteristics of subsonic flow in a long duct have just been examined.

- The reference Shapiro uses a long duct that is fed by a frictionless converging nozzle. In order to get such behavior, the component Valve "Convergent" is set with a very small pressure losses coefficient zeta (1e-9) i.e; a very small dynamic pressure loss.
- The reference uses an outlet of the long duct which is directly connected to a large volume. The component Junction "Exit" shall be set with a pressure losses coefficient zeta to 1 dynamic pressure loss (and other value of zeta produces a significant shift with respect to the reference).
- The reference uses an adiabatic duct. The model rely on a component Tube "LongDuct" without any thermal capacity into the walls, the temperatures of the thermal port of the tube are connected to the Insulation component called Adiabatic (without any thermal transfer to the LongDuct).
- The reference uses a k = 1.4 = Cp/Cv. The gas used is the perfect N2.

The model with ESPSS is representative of the Fanno tube:



A dedicated component called "ReferenceDataFile" for managing the digitalized data table has been used for the purpose of a direct comparison within EcosimPro of non-dimensional results.

In order to get similar models for each setting of λ L/D, the geometry is fixed (the number of nodes is fixed). For each branch the pressure losses coefficient (zeta) of the long duct is fixed. The branches are labeled with respect to the values of λ L/D (=4f.L/D): those are 0, 0.1, 0.2, 0.5, 1, 2, 5, 10, 20, 50, 100, 200.

2.1.3.2 Results

The plots shown below are directly output from the EcosimPro monitor. :

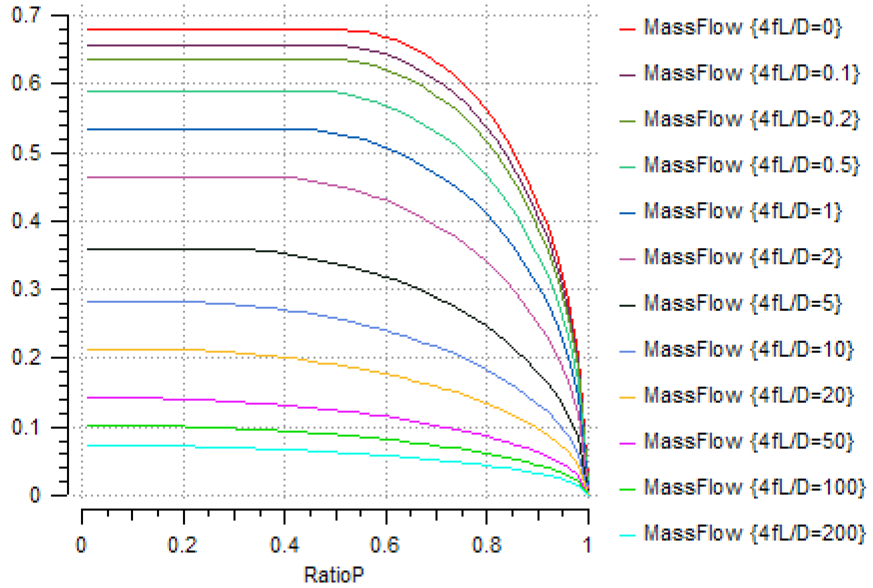
- Plot of the non-dimensional mass flow rate with respect to the pressure ratio of the background/upstream (MassFlowAddim versus RatioPB_P0I, plot similar to fig. 6.9-a of Shapiro-P175). The "non_dimensional mass flow rate" is defined by non_dimensional_mfr = mass flow *sqrt(8.314/Mmol*Tup)/(Pup* tubeSection))

The mass flow rate given by ESPSS is always slightly lower or equal than the theoretical Shapiro values. For very low values of λL/D -0, 0.1, 0.2-, under sonic (chocked) conditions, ESPSS gives lower values of mass flow rate, with a maximum error less than 5%.

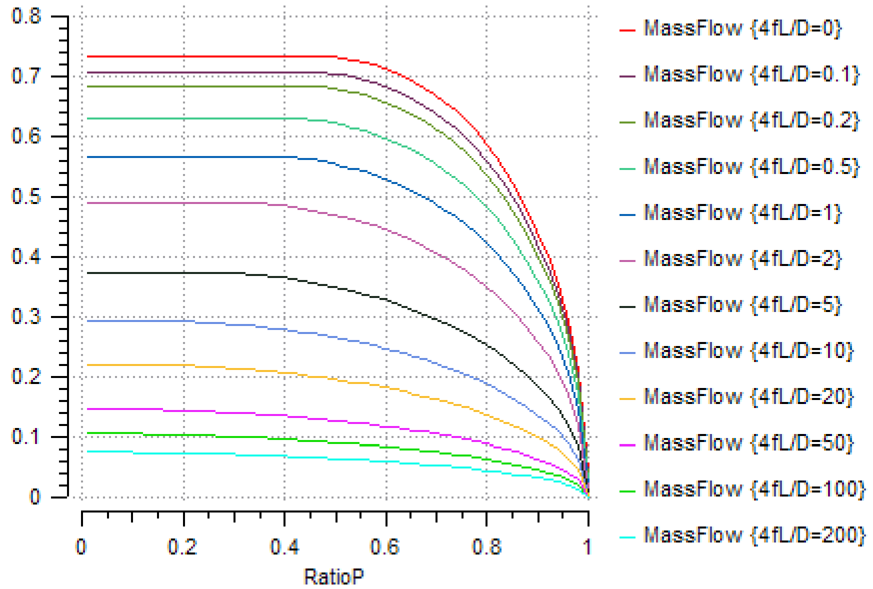
This error is probably a consequence of the numerical dissipation intrinsic to the Centred scheme (not due to the artificial dissipation that is always zero under steady flow). This small error should be acceptable because a null friction is not a realistic case.

The results are repeated for **N2** and **Xe** gases. The influence of taken real properties is not negligible for the Xenon as it can be shown comparing the non-dimensional performances of mass flow vs pressure.

N2: Real properties



Xe: Real properties



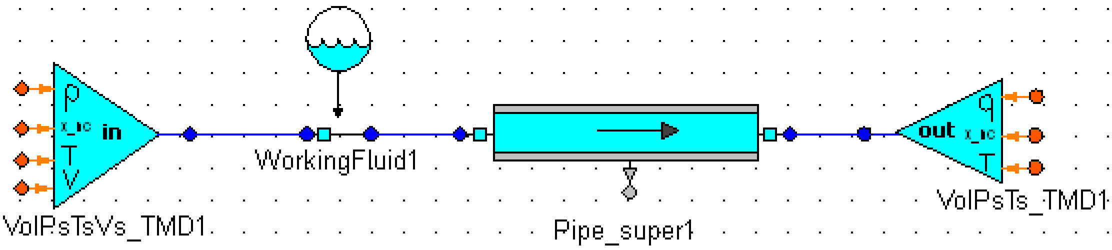
2.1.4 Pipe test: Supersonic intake case (T-FF-010)

Library:	FLUID_FLOW_1D_EXAMPLES
Model Name:	Test_supersonic_intake
Partition Name:	default
Experiment Name:	exp1

2.1.4.1 Model description

This example validates the FLUID_FLOW_1D capabilities concerning a supersonic intake followed by a nozzle decelerating the fluid using real/perfect gases properties. The nozzle is simulated by a Pipe component, which takes into account acceleration forces, pressure losses and density changes. In the present case, constant area is considered in the Pipe.

The model with ESPSS is representative of a typical intake where the inlet (flight) conditions are input data: Inlet Mach number = 4; Static pressure (altitude) = 0.135 bar; Temperature = 286 K



2.1.4.2 Results

The calculated friction looses as a function of Mach number can be compared with an analytical solution for a full supersonic pipe flow (cf. "Gas Dynamics" by M.J.Zucrow):

$$zeta\_anal_i = zeta\_anal_{i-1} + \frac{M_{i-1}^2 - M_i^2}{\gamma M_{i-1}^2 M_i^2} + \frac{\gamma+1}{2\gamma}\log\frac{M_{i-1}^2(1+(\gamma-1)/2M_i^2}{M_i^2(1+(\gamma-1)/2M_{i-1}^2}$$

$$zeta\_cal = f\Delta L / D; \quad f: wall\ \ friction\ \ factor = f(\mathrm{Re}, Rug)$$

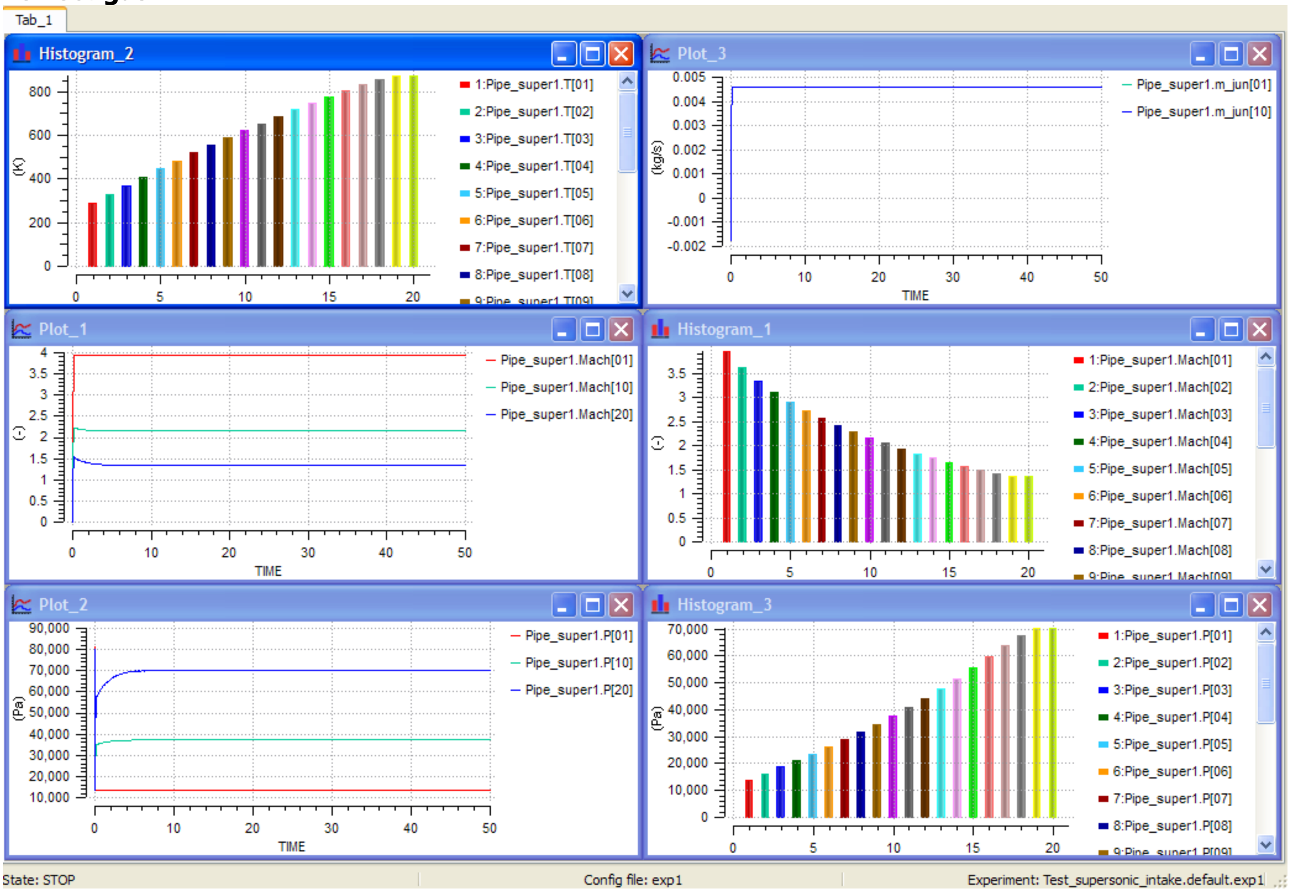
The comparison between the ESPSS calculated friction coefficients (depending on the Mach number) and the theoretical ones using the Mach numbers ESPSS solution at node 10 and 20 (exit) is:

Node	10	20	
Mach no.	2.10407	1.31008	
gamma	1.39549	1.38586	(real H2 properties)
zeta_cal	0.293652	0.609484	
zeta_anal	0.299676	0.570841	

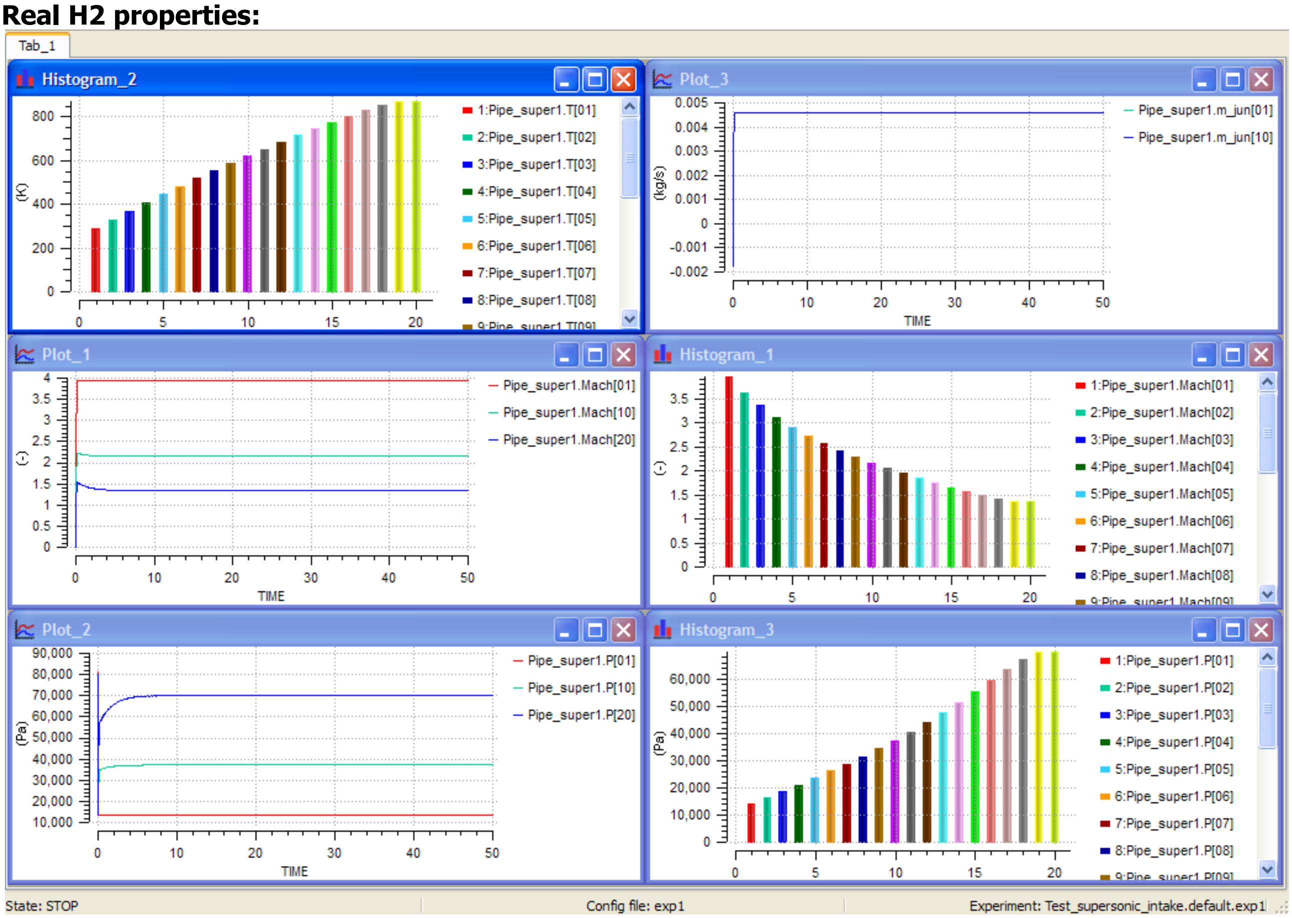
The plots shown below are directly output from the EcosimPro monitor:

- ESPSS model can simulate correctly the supersonic deceleration in the intake due to friction. Comparison with the expected theoretical results show good agreement
- While the intake remains supersonic, results are independent on the downstream conditions
- Using real properties of H2, the extrapolated conditions at T > Tmax are coherent with the expected results using perfect gases properties

Perfect gas H2:



Real H2 properties:

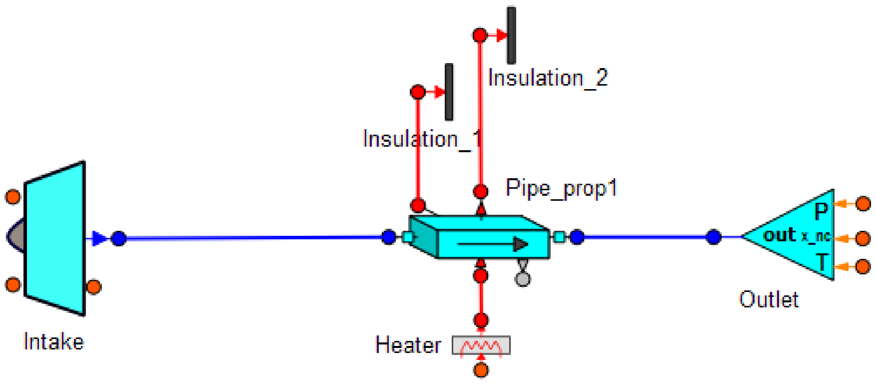


2.1.5 Pipe test: High speed Fanno tube (T-FF-012)

Library:	FLUID_FLOW_1D_EXAMPLES
Model Name:	test_fanno_Roe
Partition Name:	default
Experiment Name:	exp1

2.1.5.1 Model description

This example validates the FLUID_FLOW_1D capabilities for a *subsonic/supersonic fanno* Pipes using air as a perfect gas. The pipe is simulated by a Tube_Rect component to be compared with the Ramjet component results (§0). The ESPSS model is provided with an intake where the inlet (flight) conditions are input data.



Signals "s_mach" and "s_alt" give variable flight conditions to be imposed at the inlet of the intake. The static pressure, temperature and velocity will be calculated and applied as initial conditions and as external conditions at outlet (Outlet component). Below the additional code included in the model:

```
INIT
    atm_cond(s_alt.signal[1], s_mach.signal[1], P_o, T_o, v_o)
    m_o = P_o/(RGAS/28.9644*T_o)*v_o*Pipe_prop1.a*Pipe_prop1.b

CONTINUOUS
    Outlet.s_pres.signal[1] = intake.meas_out.signal[2]
    Outlet.s_temp.signal[1] = intake.meas_out.signal[3]
    Outlet.s_xNonCond.signal[1] = 0
```

where P_o, T_o and v_o are the calculated ambient conditions and velocity according to the standard atmosphere and the inlet Mach number.

It is pointed out that the static pressure (port variable) at the inlet of the Pipe component will become implicit with the Intake equations. *The inlet pressure of the Pipe will be coupled to the intake equations and to the fixed discharge pressure, P_o, the ambient pressure*.

Main dimensions are:

- Combustor length: L = 1.298 m;
- Characteristic cross section width and height= 0.04 m
- Constant width and height profiles
- Total Pressure Recovery coefficient equal to 1 for any flight Mach number

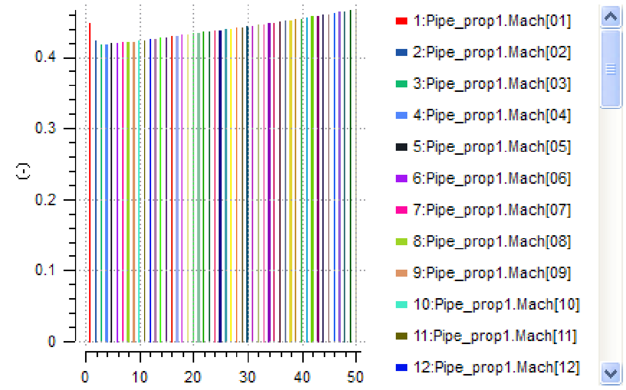
Two different schemes are used and compared through the execution of this model: the centered scheme and the *Roe scheme with the option Isent_Correl = TRUE*.

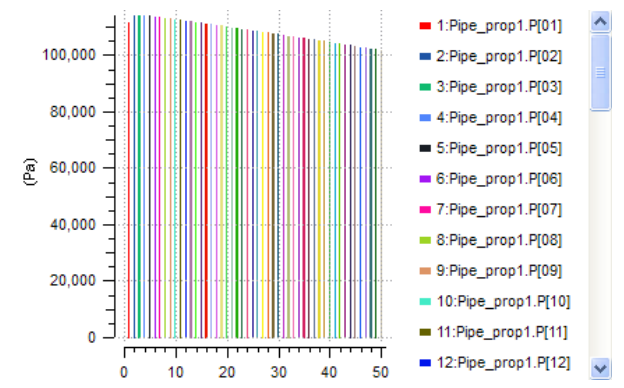
2.1.5.2 ESPSS Results

The plots shown below are directly output from the EcosimPro monitor. First are presented the results obtained with the pipe discretized to 50 nodes, comparing the centred and Roe schemes for inlet Mach numbers of 0.6 and 3. Next page shows the same results but for 200 nodes.

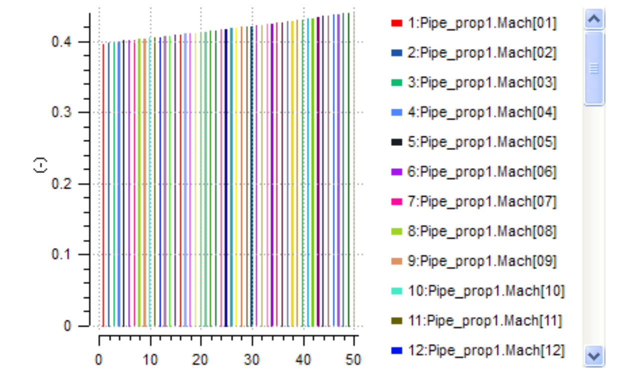
50 nodes, M=0.6

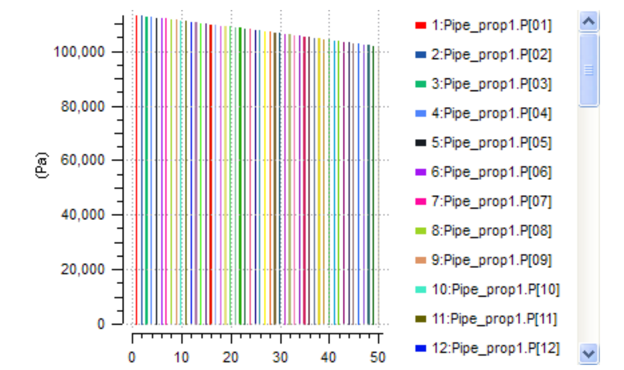
Centred scheme





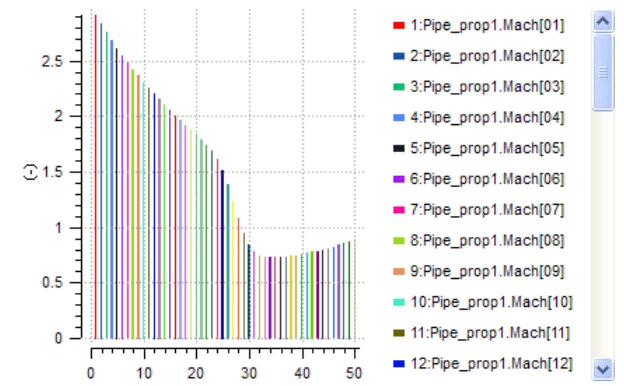
Roe scheme

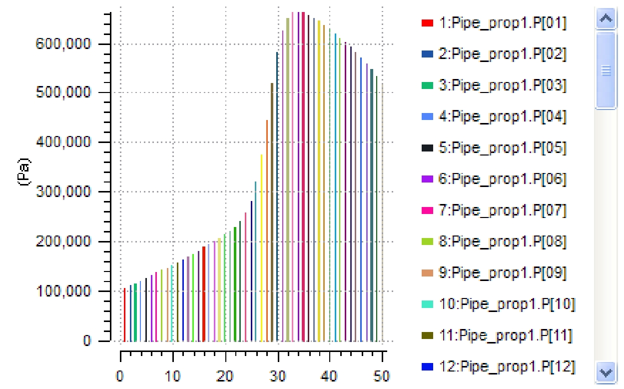




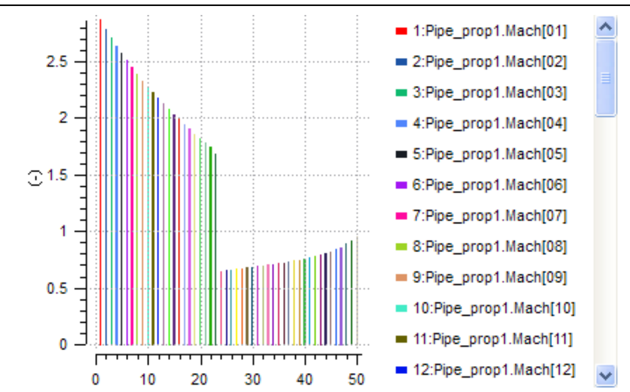
50 nodes, M=3

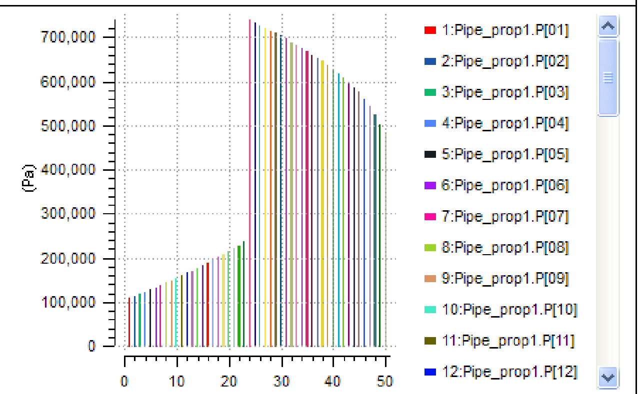
Centred scheme





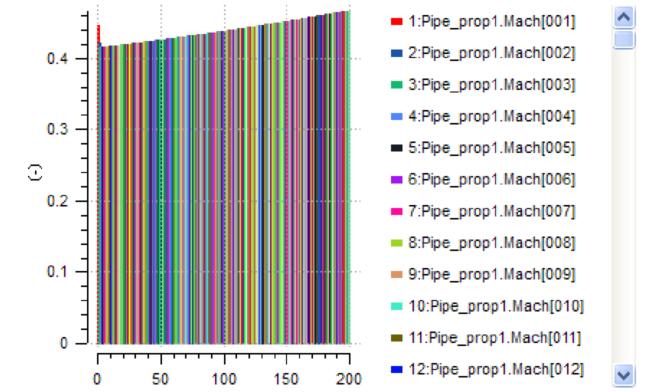
Roe scheme

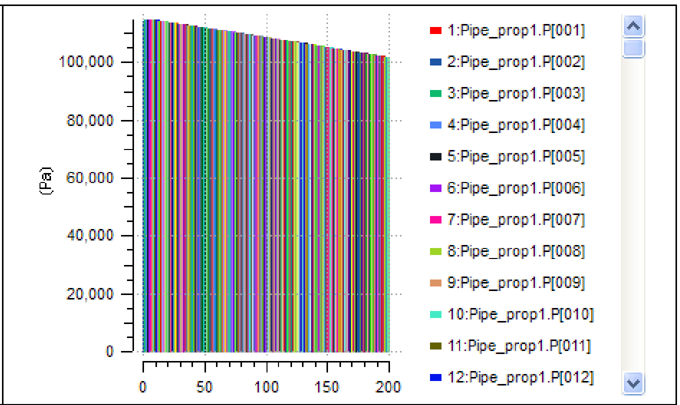




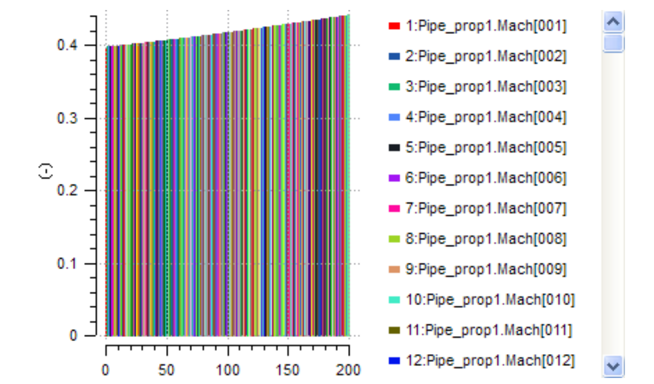
200 nodes, M=0.6

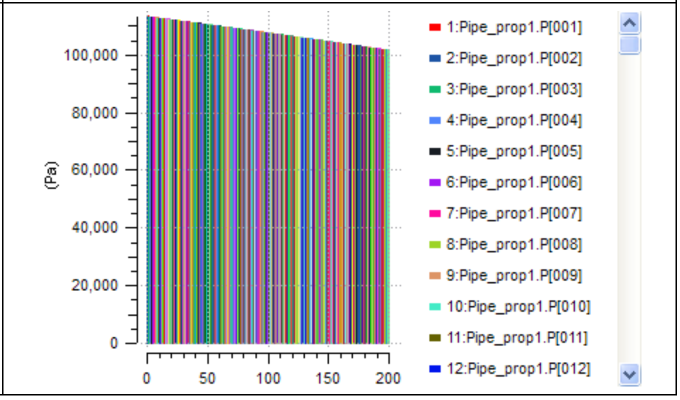
Centred scheme





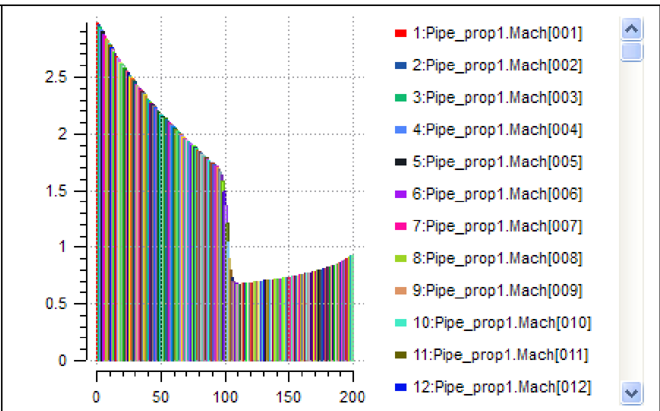
Roe scheme

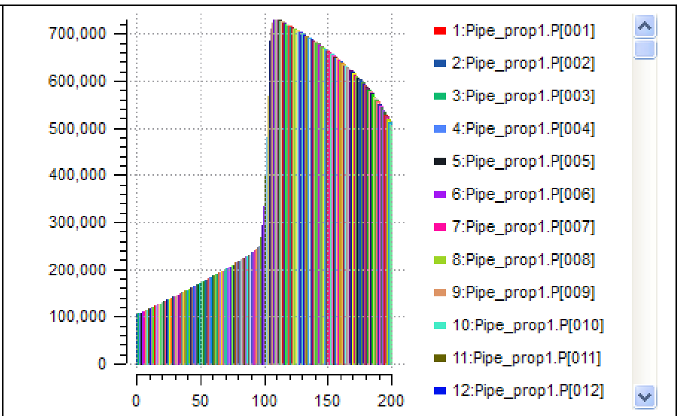




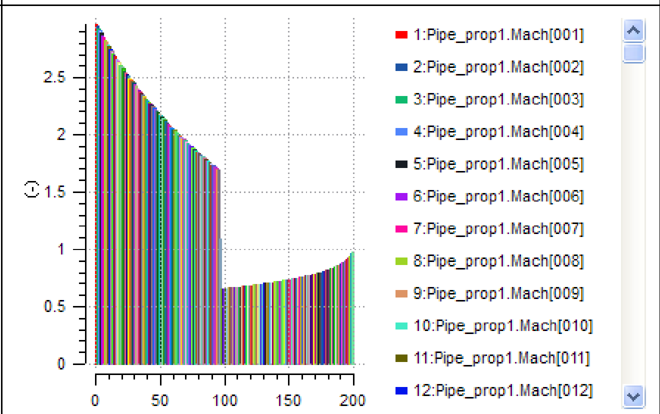
200 nodes, M=3

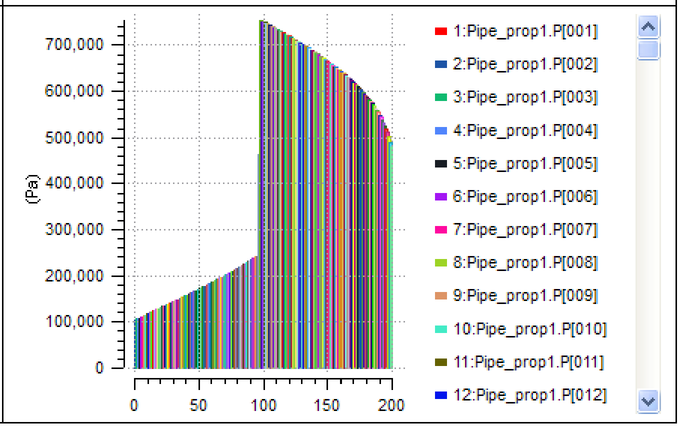
Centred scheme





Roe scheme





The following physical phenomena and conclusions can be shown:

- For a Mach number of 0.6, the pipe pressure increases with respect to the ambient (exit pressure) due to the friction losses, while for M = 3 the supersonic inlet conditions are unperturbed but a shock appears at around 65-70% of the total length and the exit becomes blocked at M = 1.
- *The Roe scheme represents the shock with a high precision, whereas the centred scheme detects the discontinuity as well but smoothing the shock. For subsonic cases, the Centred scheme introduces a small non-physical perturbation at the intake imposing the flight conditions.*
- The bigger the number of nodes, the better the centred scheme represents the discontinuity. The Roe scheme is less affected by the discretization of the pipe.
- In case the mass flow becomes zero or close, the Roe scheme can fail. This is due to the fact that the upwind enthalpy is supposed to be continuous and a function of the upwind enthalpy flow, but actually it is discontinuous. This issue has been avoided preventing the flow being under a positive small value, but the problem still remains.

2.1.5.3 Analytical solution comparison

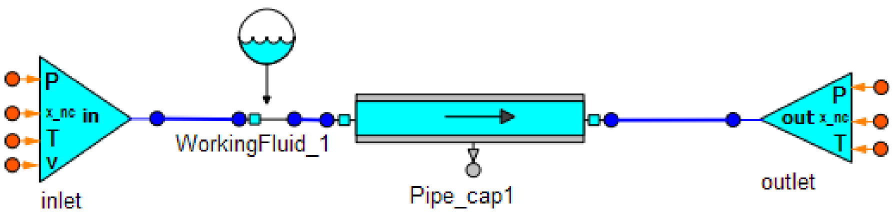
Appendix A1 is an extract of document ACP7-GA-2008-21 1485 LAPCAT II, Reference Number: D4.3.2 - Part B. "Development of One-dimensional Propulsion Model", where an analytical solution is presented for supersonic/subsonic Fanno Tubes.

This analytical solution (for constant Cp and viscosity) is coded under EcosimPro in the specific component "**analyticalFanno**" in the FLUID_FLOW_1D_EXAMPLES library.

Constant input data are:

```
R = 287.114          (Gas constant, Air)
gamma = 1.376        (Isentropic coefficient)
mu = 3.03153e-5      (Viscosity)
D = 0.004            (Pipe diameter)
L_p = 1.3            (Pipe length)
```

The following ESPSS model has been used to test the **capacitive** Pipe under different boundaries and schemes against the analytical solution:

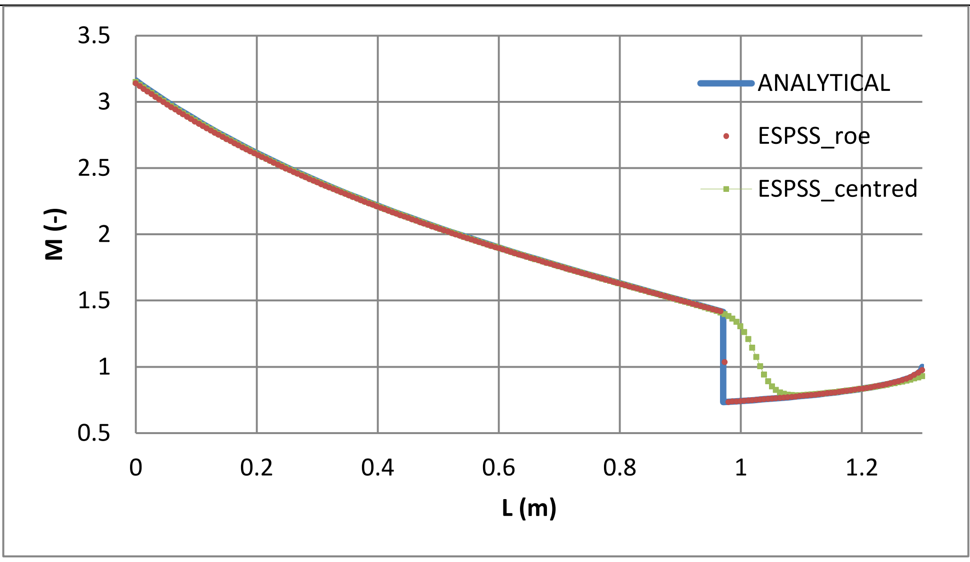
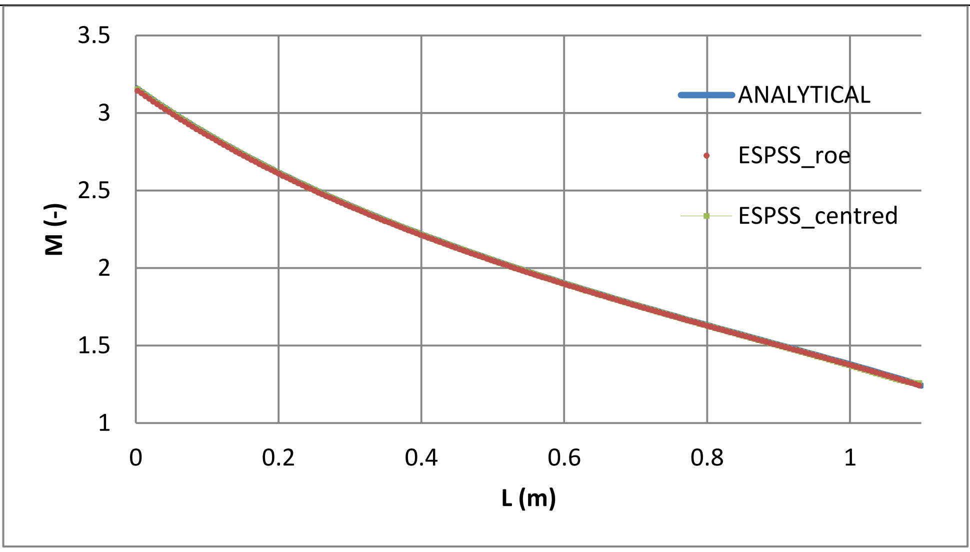
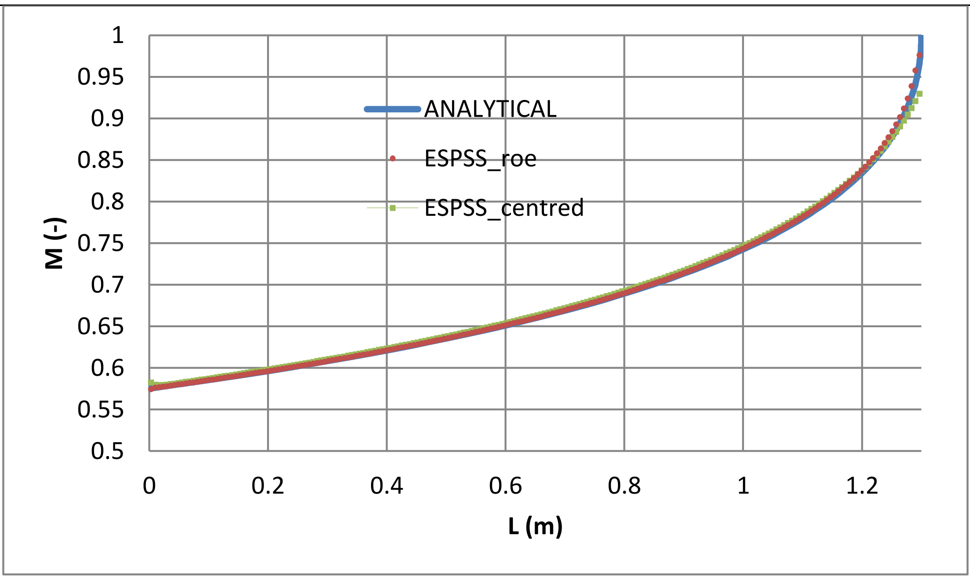


Model Name: FPL1 (default part. Experiment exp1)

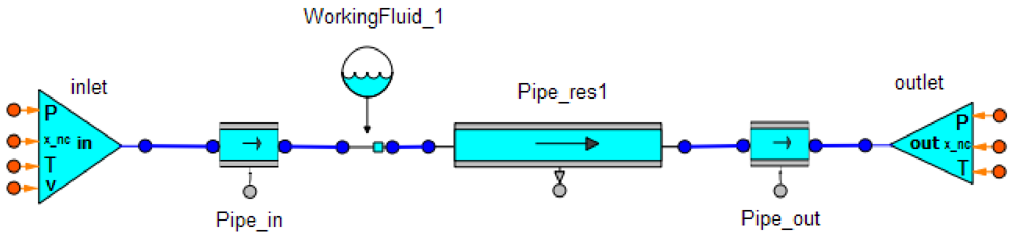
The inlet/outlet components can reproduce the same conditions (static pressure, temperature and speed, see tables below) as in the analytical solution (Calorically perfect gas and circular Pipe). The properties file used is the "PfGas_Usr2" representing Air at the same constant Cp and viscosity as for the analytical model:

```
PfGas_Usr2   28.958538
 Temp   Cp            visc                   cond
 200    1051.1525     3.03152893e-005     0.0455684556
 1000   1051.1525     3.03152893e-005     0.0455684556
```

- The plots in next page had shown a very good matching with the theoretical subsonic or supersonic solutions including shock capture. Plots have been automatically obtained using EXCEL files (attached to the experiment folder) by inserting the reports generated with EcosimPro.
- *The "Centred" scheme behaves more robustly and is less time consuming than the Roe scheme but is less precise capturing shock discontinuities.*

Case	Capacitive Pipe Comparison (200 nodes)
Supersonic with shock and subsonic exit Po = 50812 To = 417 Uo = 1283.0 Pipe_cap1.L = 1.3 P_out = 2e5	
SUPERSONIC fully Po = 50812 To = 417 Uo = 1283.0 Pipe_cap1.L = 1.1 P_out = 2e5	
SUBSONIC chocked Po = 451622 To = 657.7 Uo = 298.1 Pipe_cap1.L = 1.3 P_out = 2e5	

The following ESPSS model has been used to test the **resistive** Pipe under the same conditions as before:



Model Name: FPL1_res (default part. Experiment exp1)

In this case, because the pipe is resistive and to be able to use the same boundaries as in the previous case, we are obliged to add two very small capacitive pipes (0.1 mm length) of same diameter as the main Pipe.

The plots below show the comparison results:

- The Centred scheme is able to obtain similar results as in the capacitive Pipe, Nevertheless, the added capacitive Pipes in between the boundaries seems to produce relatively important perturbations (losses) in the global results
- *The Roe scheme has numerical troubles and produces numerical exceptions*

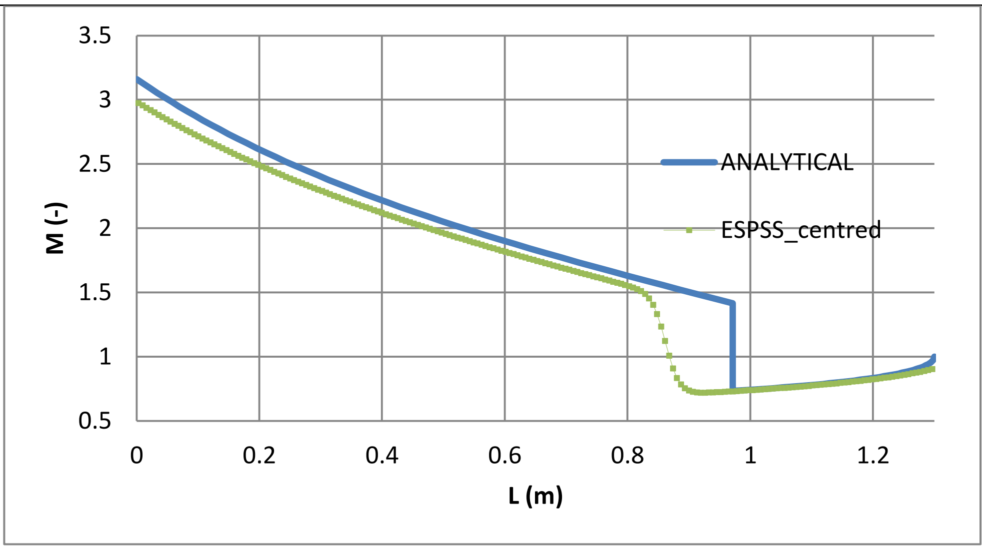
<table>
<tr><th>Case</th><th>Resistive Pipe Comparison (200 nodes)</th></tr>
<tr><td>

Supersonic with shock and subsonic exit

Po = 50812
To = 417
Uo = 1283.0
Pipe_cap1.L = 1.3

P_out = 2e5

</td><td>



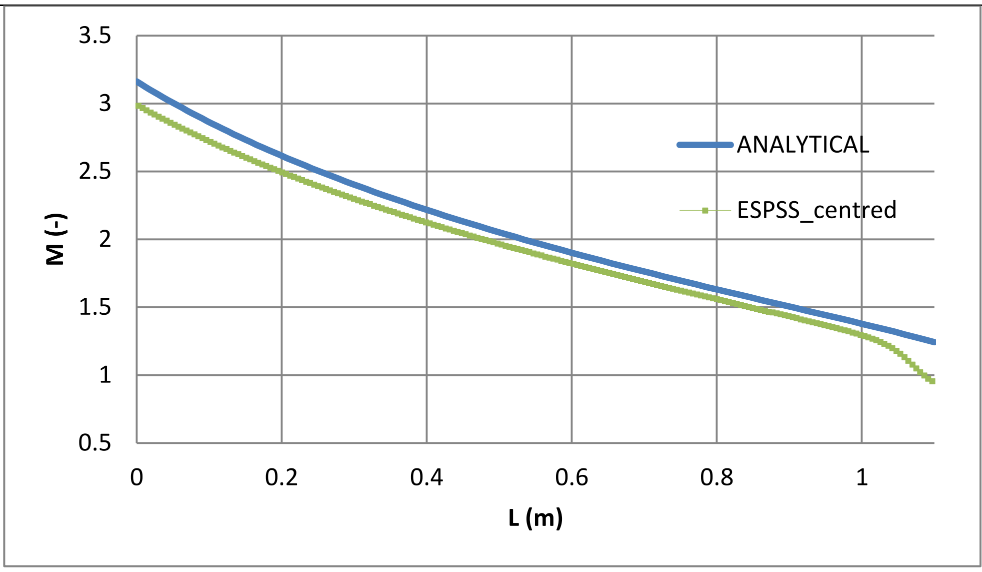
</td></tr>
<tr><td>

SUPERSONIC fully

Po = 50812
To = 417
Uo = 1283.0
Pipe_cap1.L = 1.1

P_out = 2e5

</td><td>



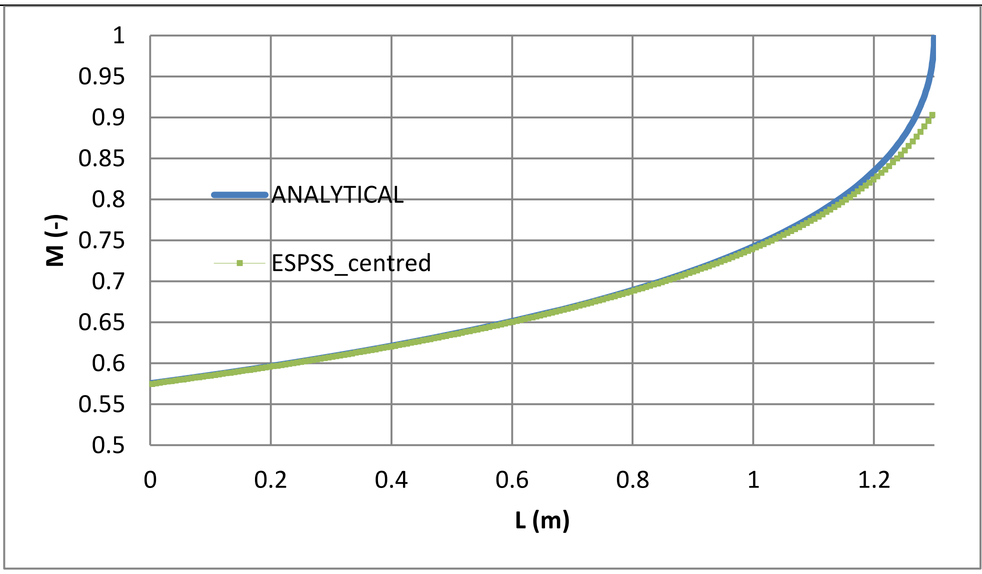
</td></tr>
<tr><td>

SUBSONIC chocked

Po = 451622
To = 657.7
Uo = 298.1
Pipe_cap1.L = 1.3

P_out = 2e5

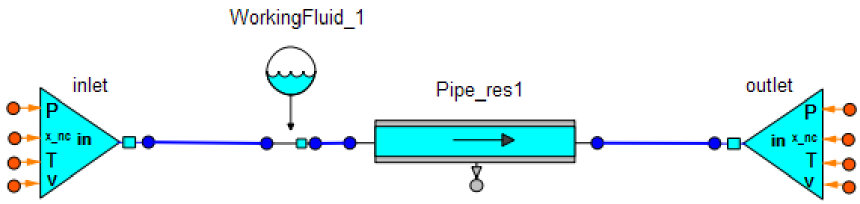
</td><td>



</td></tr>
</table>

A new ESPSS model has been built to test the **resistive** Pipe alone.

Library:	FLUID_FLOW_1D_EXAMPLES
Model Name:	FPL2_res
Partition Name:	default
Experiment Name:	exp1



New capacitive "inlet" and "outlet" components have been taken for this case, making them capacitive so they can be directly connected to the resistive pipe. This way, the perturbation produced by the capacitive pipes can be avoided

The plots below had shown a good matching with the theoretical subsonic or supersonic solutions including shock capture:

- The Centred scheme is able to obtain good results in subsonic and supersonic cases, *but introduce additional losses at the inlet for supersonic cases*, which is anyhow conservative
- The Roe scheme is able to reproduce exactly the analytical solution.

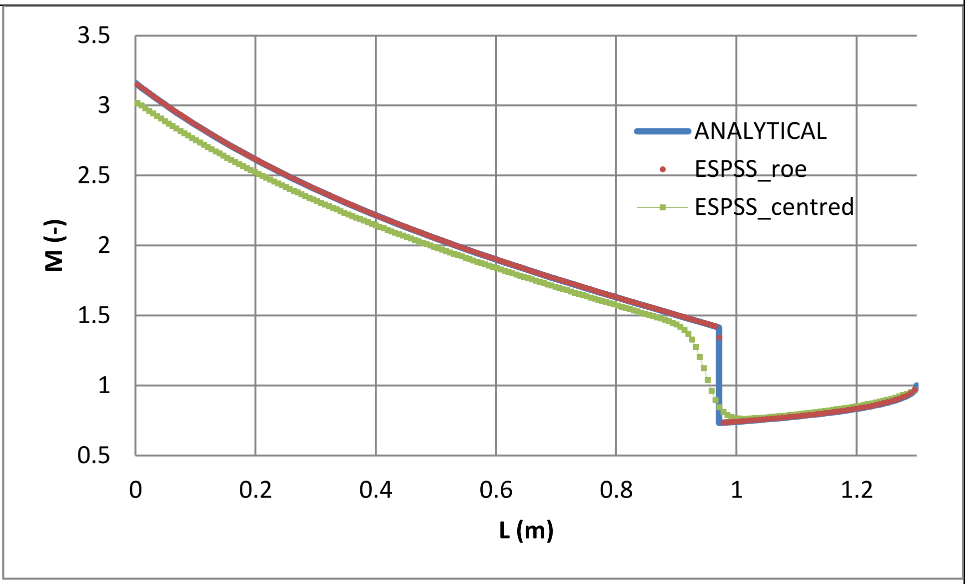
<table>
<tr><th>Case</th><th>Resistive Pipe Comparison (200 nodes)</th></tr>
<tr><td>

Supersonic with shock and subsonic exit

Po = 50812
To = 417
Uo = 1283.0
Pipe_cap1.L = 1.3

P_out = 2.5e5

</td><td>



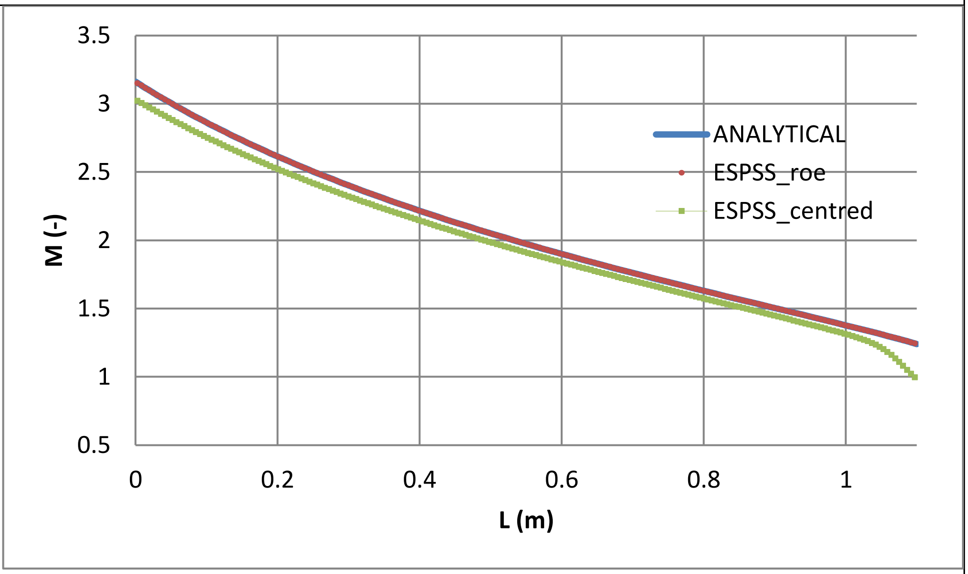
</td></tr>
<tr><td>

SUPERSONIC fully

Po = 50812
To = 417
Uo = 1283.0
Pipe_cap1.L = 1.1

P_out = 2.5e5

</td><td>



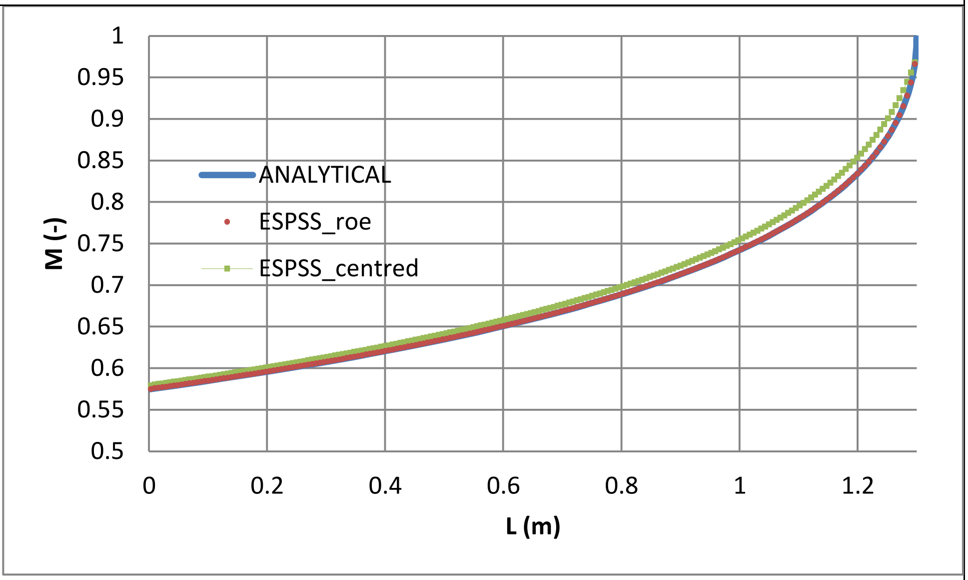
</td></tr>
<tr><td>

SUBSONIC chocked

Po = 451622
To = 657.7
Uo = 298.1
Pipe_cap1.L = 1.3

P_out = 2.5e5

</td><td>



</td></tr>
</table>

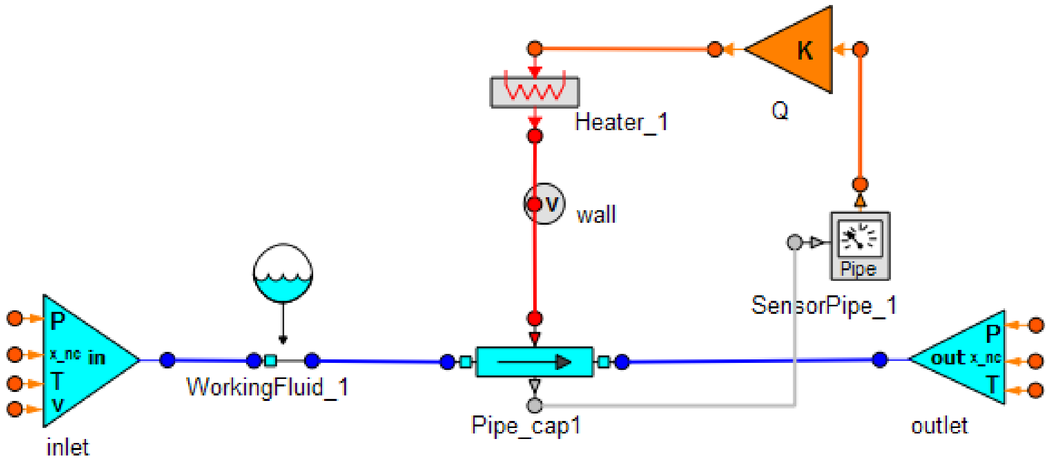
2.1.6 Pipe test: High speed Rayleigh tube (T-FF-013)

Library:	FLUID_FLOW_1D_EXAMPLES
Model Name:	RB1 (capacitive Pipe) RB2 (resistive Pipe)
Partition Name:	default
Experiment Name:	exp1

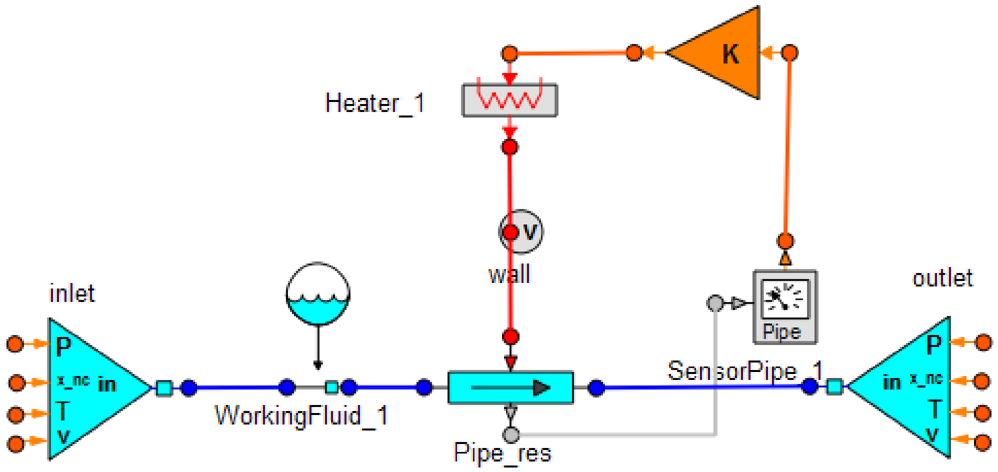
2.1.6.1 Model description

This example validates the FLUID_FLOW_1D capabilities for a *subsonic/supersonic* Rayleigh tube using air as a perfect gas. Two ESPSS models are provided depending on the Pipe type (re4sisitive or capacitive):

Capacitive Pipe with resistive boundary conditions (RB1):



Resistive Pipe with capacitive boundary conditions (RB2):



The inlet/outlet components can reproduce the same conditions (static pressure, temperature and speed, see tables below) as in the analytical solution (Calorically perfect gas and circular Pipe).

Heat addition and heat rejection is controlled by a gain in such a way the power applied to the tube (Watts) is the results of a fixed quantity (the delta energy value in Joules) multiplied by the mass flow (Pipe sensor)

Two different schemes are used and compared through the execution of this model: the centered scheme and the *Roe scheme with the option Isent_Correl = TRUE***.**

2.1.6.2 Analytical solution comparison

Document ACP7-GA-2008-21 1485 LAPCAT II. Reference Number: D4.3.2 - Part B. "Development of One-dimensional Propulsion Model" presents an analytical solution for supersonic/subsonic Rayleigh Tubes.

This analytical solution (for constant Cp and viscosity) is coded under EcosimPro in the specific component "**analyticalRayleigh**" in the FLUID_FLOW_1D_EXAMPLES library.

Constant input data are:

```
R = 287.114          (Gas constant, Air)
gamma = 1.376        (Isentropic coefficient)
D = 0.043701937      (Pipe diameter)
L_p = 1.2            (Pipe length)
```

Plots in next pages have been automatically obtained using EXCEL files (attached to the experiment folder) by inserting the reports generated with EcosimPro: There is nearly no differences between 50 and 200 nodes.

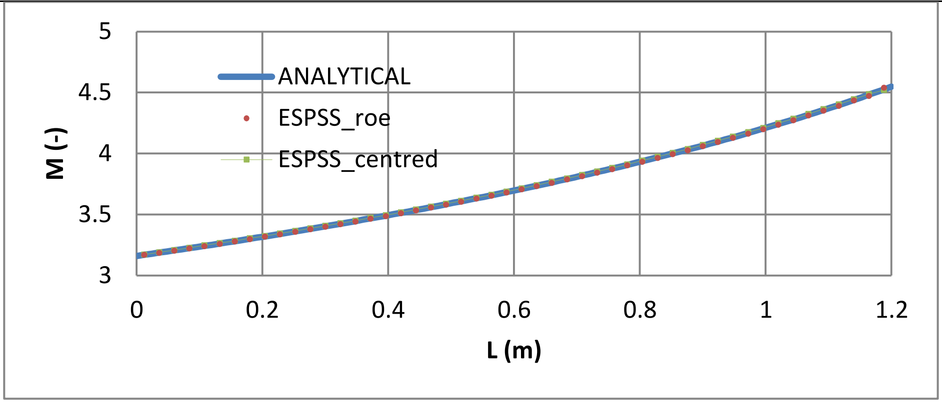
Main conclusions for the **capacitive** Pipes are:

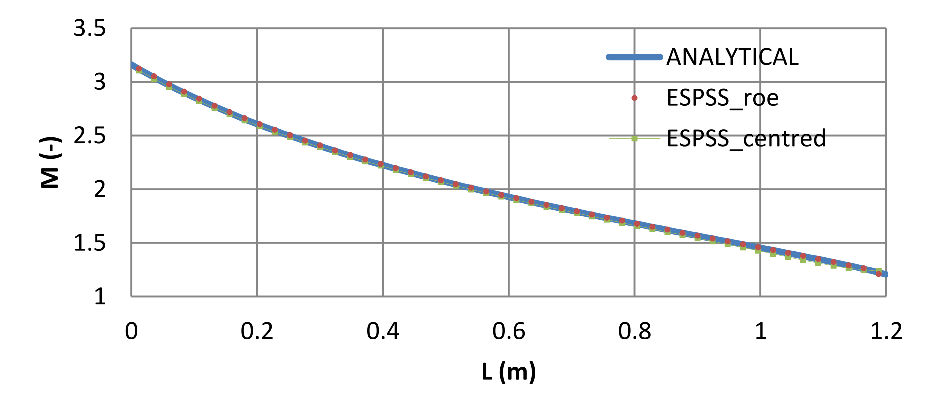
- The Centred scheme is able to obtain good results in subsonic and supersonic cases, even though the small perturbation at the inlet in subsonic cases
- The Roe scheme is able to reproduce exactly the analytical solution *but it does not converge in the subsonic heat rejection case, and has numerical troubles in other cases*.

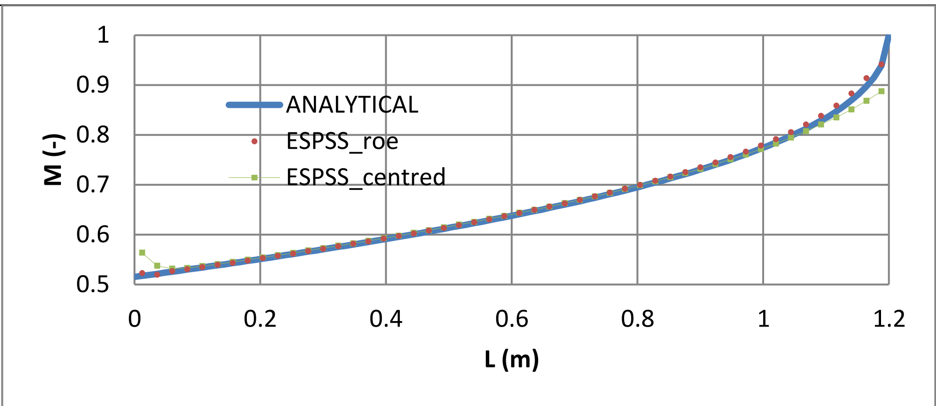
Main conclusions for the **resistive** Pipes are:

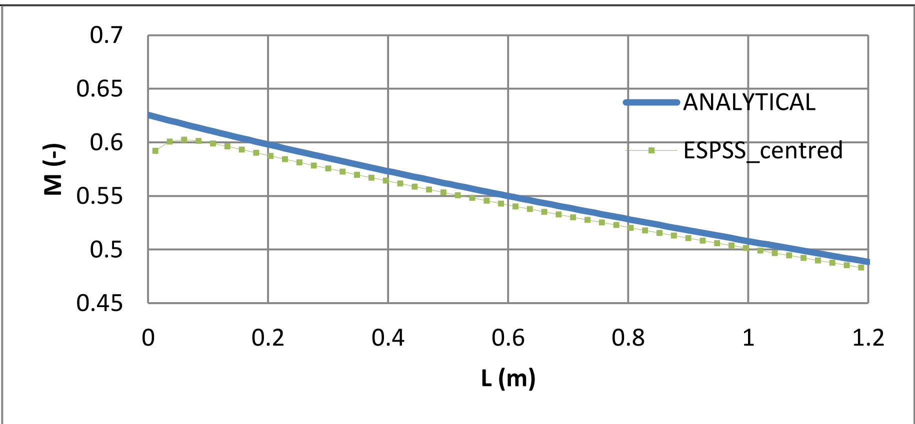
- The Centred scheme presents appreciable differences with the analytical solution. *On the contrary to the model using capacitive Pipes, the resistive pipe needs a capacitive outlet boundary condition where the outlet pressure is always applied, even for supersonic cases*. Results have been then adjusted tuning the outlet pressure. This was not the case using a resistive outlet condition (capacitive Pipes) because in this case the boundary includes a Junction calculating the mass flow. This Junction has intelligence to "know" when a supersonic flow is established.
- The Roe scheme is able to reproduce exactly the analytical solution and it seems not affected by the type of the boundary conditions, *except for the subsonic heat rejection case*. The problem in this case is *not* that the ESPSS pipe formulation gives a wrong solution, but that the inlet boundary component is changing the actual inlet conditions in the pipe. If we impose to the analytical solution the pipe inlet conditions obtained with ESPSS, then the results are exactly the same. Be aware that the inlet conditions in the analytical model are directly applied to the Pipe inlet, while in the ESPSS model these conditions need to be applied to the inlet boundary component.
- The modifications of the static conditions through the inlet boundary component (a VolPsTsVsCap_TMD type) are more important using the Roe scheme than the Centred scheme, as it can be seen in the document.

<table>
<tr><th>Case</th><th>Capacitive Pipe Comparison (50 nodes)</th></tr>
<tr><td>Supersonic heat rejection

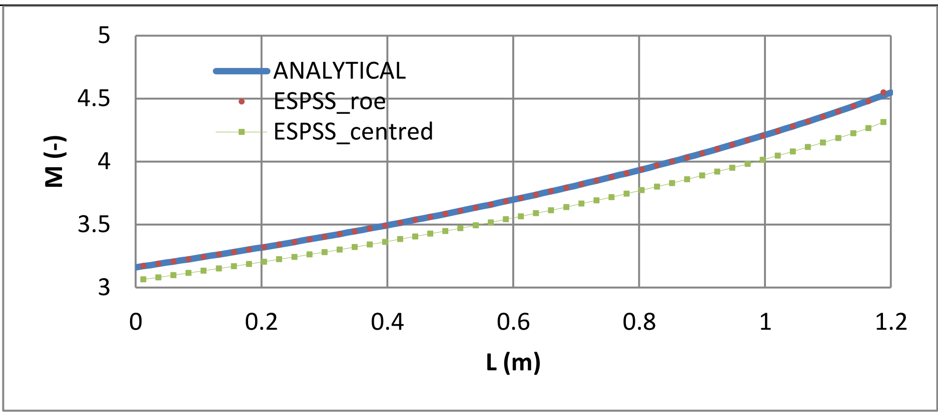
Po = 50812
To = 417
Uo = 1283
P_ex = 200000
Q.k[1] = -150000</td><td>
</td></tr>
<tr><td>SUPERSONIC heat addition

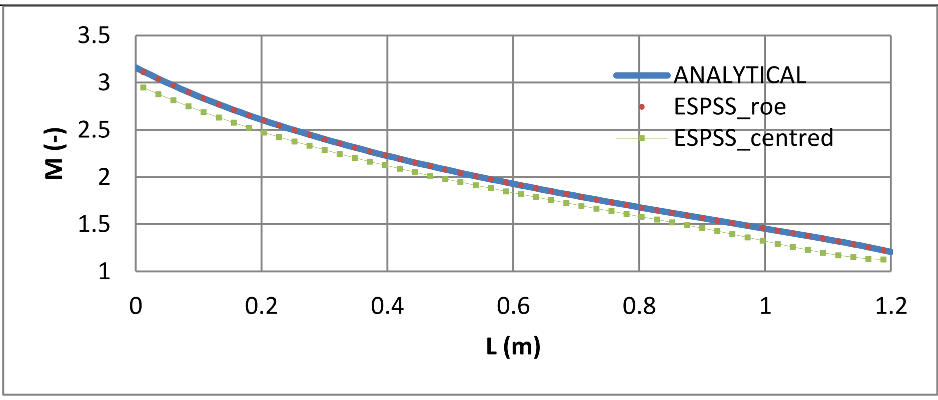
Po = 50812
To = 417
Uo = 1283
P_ex = 200000
Q.k[1] = 700000</td><td>
</td></tr>
<tr><td>SUBSONIC heat addition chocked

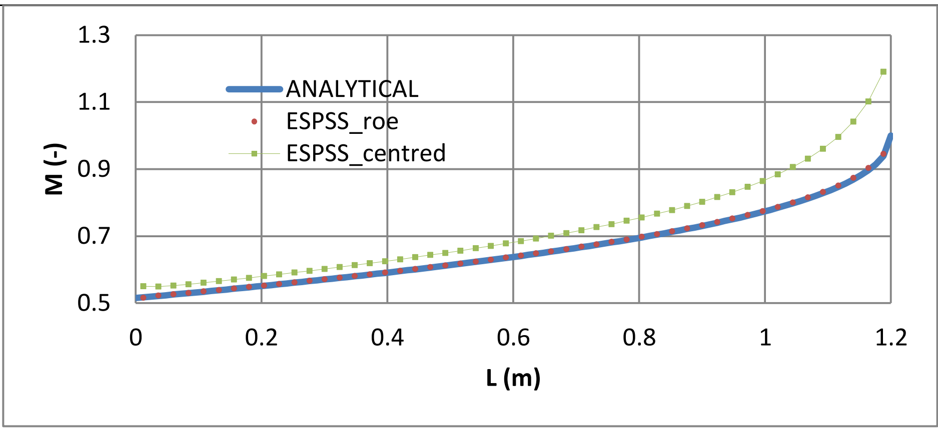
Po = 451622
To = 657.7
Uo = 298.1
P_ex = 200000
Q.k[1] = 300000</td><td>
</td></tr>
<tr><td>SUBSONIC heat rejection

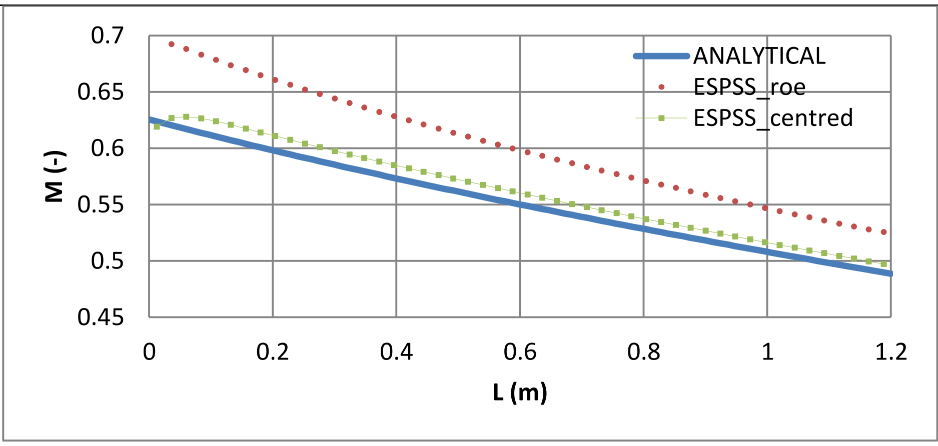
Po = 451622
To = 657.7
Uo = 298.1
P_ex = 500000
Q.k[1] = -150000</td><td>
</td></tr>
</table>

<table>
<tr><th>Case</th><th>Resistive Pipe Comparison (50 nodes)</th></tr>
<tr><td>Supersonic heat rejection

Po = 50812
To = 417
Uo = 1283
P_ex = 20000
Q.k[1] = -150000</td><td>
</td></tr>
<tr><td>SUPERSONIC heat addition

Po = 50812
To = 417
Uo = 1283
P_ex = 260000
Q.k[1] = 700000</td><td>
</td></tr>
<tr><td>SUBSONIC heat addition chocked

Po = 451622
To = 657.7
Uo = 298.1
P_ex = 200000
Q.k[1] = 300000</td><td>
</td></tr>
<tr><td>SUBSONIC heat rejection

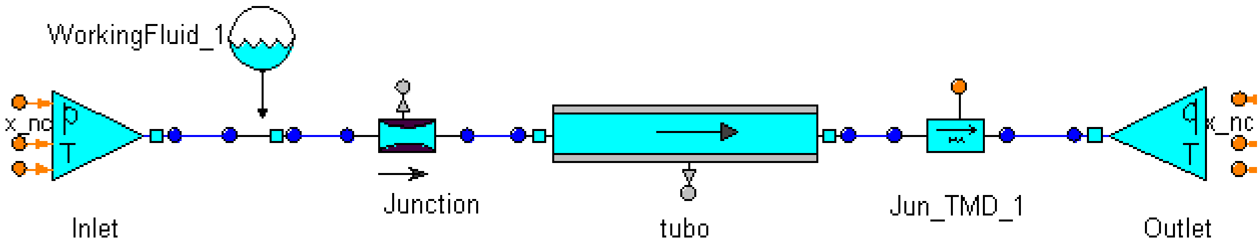
Po = 451622
To = 657.7
Uo = 298.1
P_ex = 500000
Q.k[1] = -150000</td><td>
</td></tr>
</table>

2.1.7 Pipe test: Water hammer in a Liquid line (T-FF-002)

Library:	FLUID_FLOW_1D_EXAMPLES
Model Name:	Test_Pipe
Partition Name:	default
Experiment Name:	exp1

2.1.7.1 Model description

This example validates the FLUID_FLOW_1D capabilities concerning water-hammer analysis in a pipe using a real properties fluid (two phase flow). The pipe is simulated by the Pipe component, which takes into account inertia forces, pressure losses and phase changes.



The model represents a pipe between two boundaries: The left one imposing P-T (5 bar, 300 K). The right one forces the mass flow circulating through the pipe. The pipe data are:

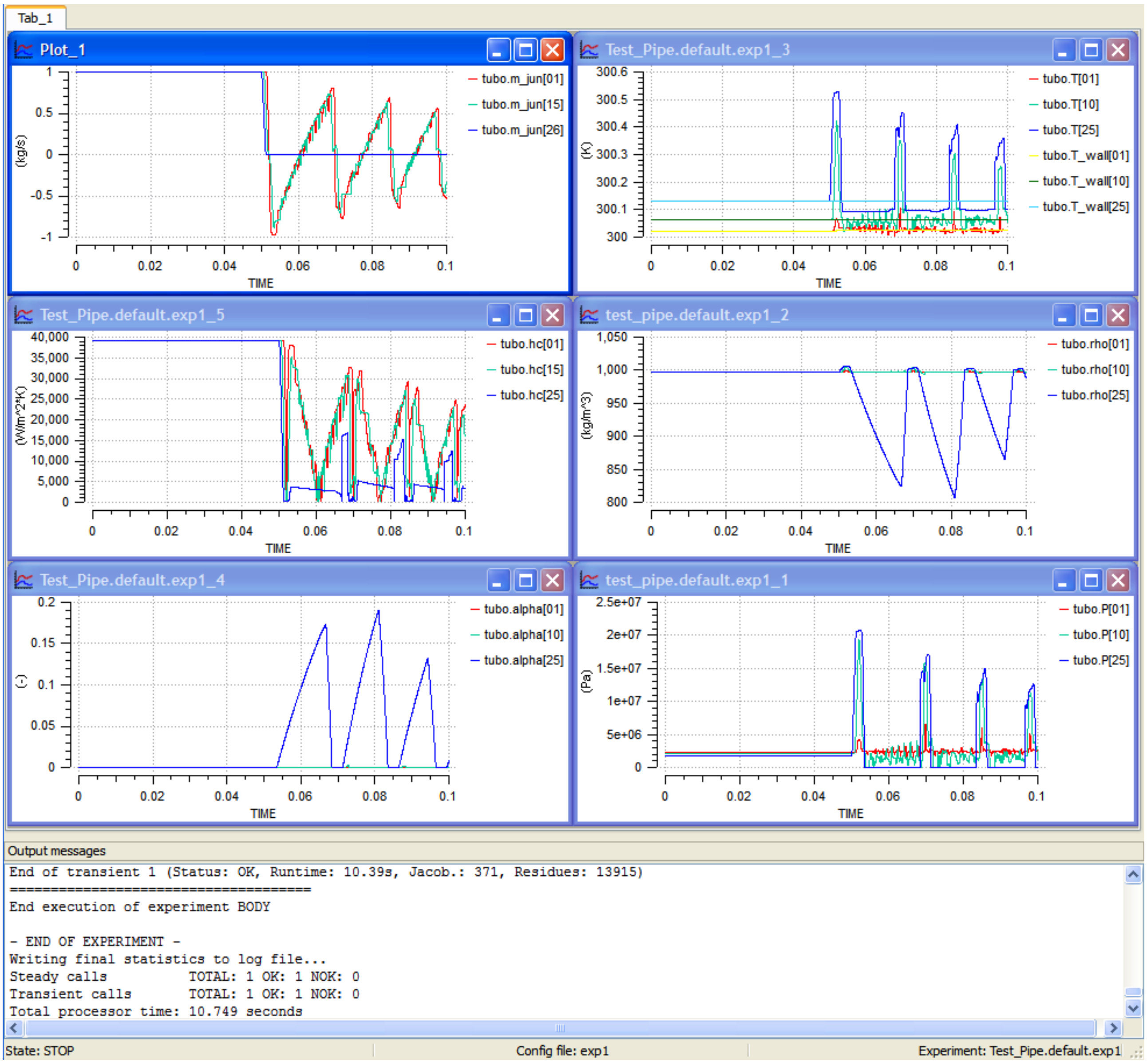
- Internal diameter: 10 mm. Roughness: 50 μm
- Nodes: 25
- Length: 2 m

A previous steady state is calculated to determine the pipe conditions at a constant mass flow of 1 kg/s. The transient will be started by a sudden flow reduction at the right side, that is, the time dependant mass flow is set to zero in 1 millisecond at TIME = 0.05.

2.1.7.2 Results

The following phenomena can be shown:

- Pressure rises (*that matches with the theoretical value* ρcv) are due to the wave trips caused by a sudden stop of the fluid. This wave is reflected in the open end as a negative flow.
- When this backflow is stopped again at the closed end, the "negative" pressures that should be created are limited to the vapor pressure.
- Then, the corresponding vapor bubble formation takes place, a water column separation is caused, and the column enters the tank.
- The bubble collapse begins when this liquid column is stopped by the tank pressure and begins to enter the pipe, collapsing the vapor.
- A new cycle starts when the liquid column is stopped at the closed end when the vapor is eliminated.

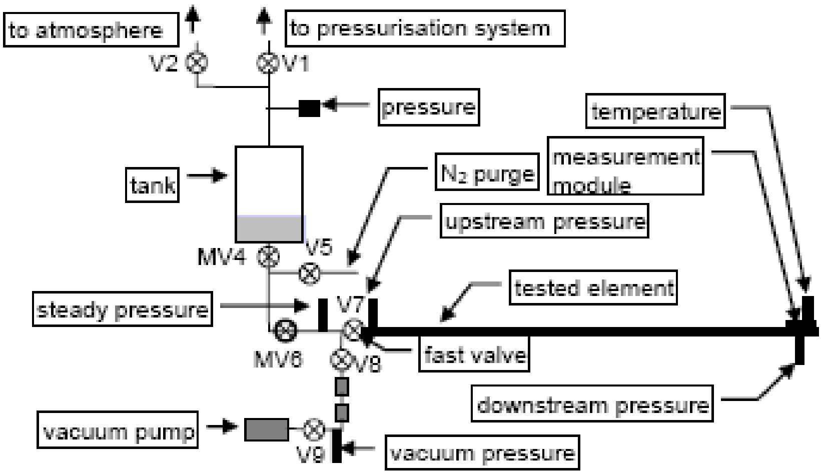

Tab_1
Plot_1
tubo.m_jun[01]
tubo.m_jun[15]
tubo.m_jun[26]
(kg/s)
TIME
Test_Pipe.default.exp1_3
tubo.T[01]
tubo.T[10]
tubo.T[25]
tubo.T_wall[01]
tubo.T_wall[10]
tubo.T_wall[25]
(K)
Test_Pipe.default.exp1_5
tubo.hc[01]
tubo.hc[15]
tubo.hc[25]
(W/m^2*K)
test_pipe.default.exp1_2
tubo.rho[01]
tubo.rho[10]
tubo.rho[25]
(kg/m^3)
Test_Pipe.default.exp1_4
tubo.alpha[01]
tubo.alpha[10]
tubo.alpha[25]
(-)
test_pipe.default.exp1_1
tubo.P[01]
tubo.P[10]
tubo.P[25]
(Pa)
Output messages
End of transient 1 (Status: OK, Runtime: 10.39s, Jacob.: 371, Residues: 13915)
====================================
End execution of experiment BODY
- END OF EXPERIMENT -
Writing final statistics to log file...
Steady calls TOTAL: 1 OK: 1 NOK: 0
Transient calls TOTAL: 1 OK: 1 NOK: 0
Total processor time: 10.749 seconds
State: STOP
Config file: exp1
Experiment: Test_Pipe.default.exp1

2.1.8 Pipe test: Priming Case (T-FF-003)

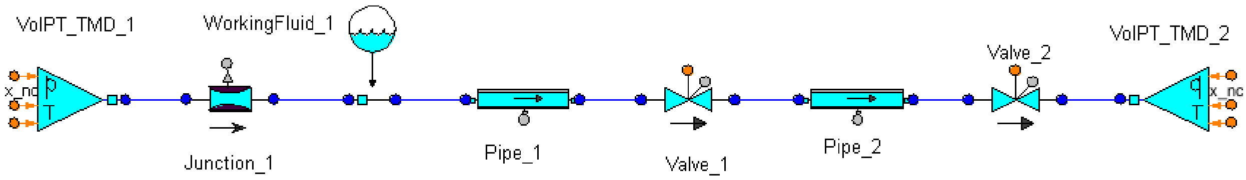
Library: FLUID_FLOW_1D_EXAMPLES
Model Name: Priming / Priming_Roe
Partition Name: default
Experiment Name: exp_Ppipe_001MPa / exp_MMH

2.1.8.1 Model description

The model used for priming verification is representative of the hardware set-up described in [RD-16]:



The EcosimPro schematic of the model appears in figure below.



Pipe_1 is initialised in liquid conditions. Pipe_2 initial conditions can be either saturated vapor (Ppipe < Pvap(Ttank)), either non-condensable gas (**GN2**) with 100% of humidity (Ppipe > Pvap(Ttank)). Two liquids were tested, **water** and **MMH**, this one compared with real rest results. See RD-19.

Input data with Water

Water case	Pipe_1 (Centred scheme)	Pipe_2 (Centred scheme)
Number of nodes	20	50
Pipe length (m)	0.670	2.000
Pipe inner diameter (m)	0.010	0.00553
Thickness (m)	0.001	0.00041
Material	Titanium	Titanium
Initial Pressure	Ptank	Ppipe
Initial Temperature	Ttank	Ttank
Heat transfer	Two-phase correlations	**Two-phase correlations**

	Junction_1	**Valve_1**	**Valve_2**
Area (m2)	7.8540E-05	2.4018e-005	0
Forward loss coefficient	5	5	1
Backward loss coefficient	5	5	1
Valve Position	Open	Linear opening in 25 ms. Starts at time 1 ms	Closed

Setup input data with MMH.

MMH case	**Pipe_1 (Centred scheme)**		**Pipe_2 (Roe-3 scheme)**
Number of nodes	20		50
Pipe length (m)	1.9		2.000
Pipe inner diameter (m)	0.010		0.00553
Thickness (m)	0.001		0.00041
Material	Titanium		Titanium
Initial Pressure	Ptank		Ppipe
Initial Temperature	Ttank		Ttank
Wall/fluid Heat transfer	Two-phase correlations		**Imposed zero**
	Junction_1	**Valve_1**	**Valve_2**
Area (m2)	7.8540E-05	2.4018e-005	0
Forward loss coefficient	3	3	1
Backward loss coefficient	3	3	1
Valve Position	Open	Linear opening in 25 ms. Starts at time 1 ms	Closed

2.1.8.2 Results with water (centred scheme)

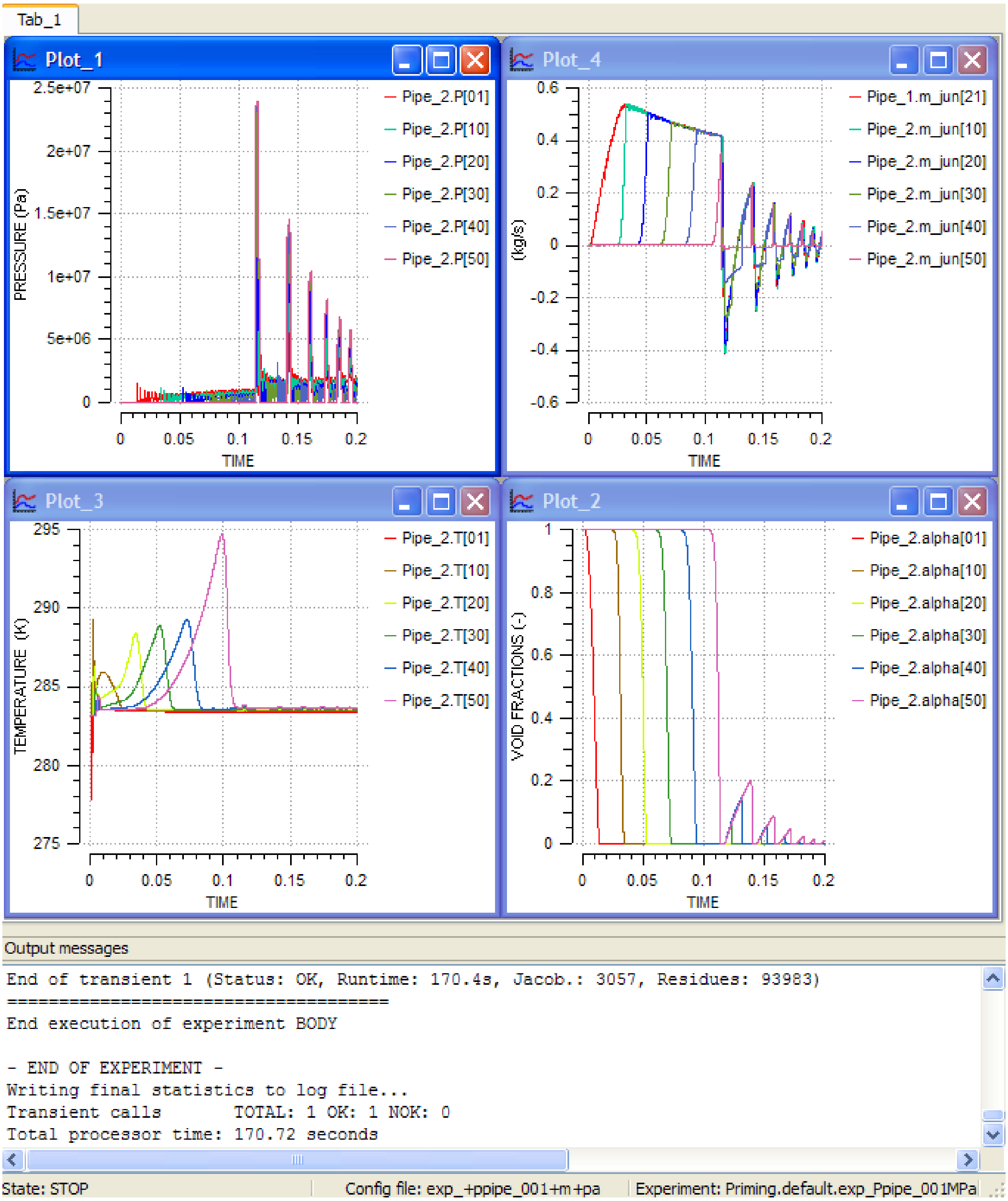
After linearly opening of Valve1 in 25 milliseconds, the transient will simulate the filling of Pipe2. This is one of the most difficult processes to be simulated. The following phenomena can be shown:

- Liquid front collapsing the vapor or mixing with a non-condensable gas. See the differences of having a very low initial downstream pressure (vapor) or having a GN2 initially filling Pipe2.
- Pressure peaks due to the abrupt stop of the liquid front at the end of the downstream line. Similar water-hammer phenomena than in previous test case can be observed
- Repetitive pressure cycles due to the very low natural damping in a pipe in case of no non-condensable gases

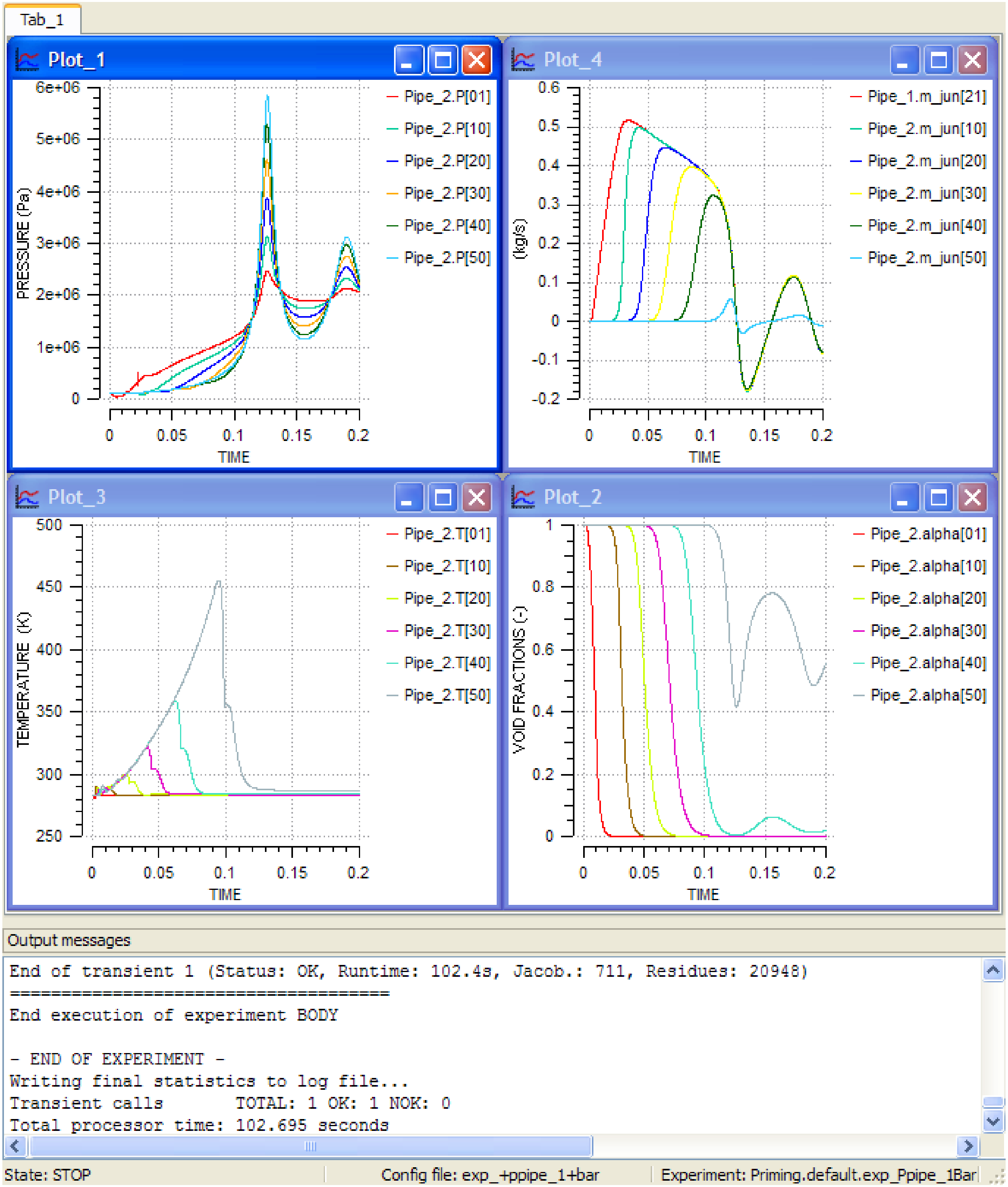
With non-condensable gases, the fluid temperature increase is about 140 K because the heat exchange with walls. Without non-condensable gases, the temperature variation is low (10 K) because the vapor is condensing when it is pressurized. Assuming adiabatic compression (input data "ht_option=HT_constant" and "hc_dat=0"), the temperature rise would be greater.

It should be noted the influence of the "Damp" global parameter (see the experiment file) on the CPU timing. Attached figures correspond to Damp=1. Using Damp = 0.4, we obtain basically the same results but the CPU time is greater.

Initial Pipe2 Pressure: 0.00123 MPa (sat. vapour)



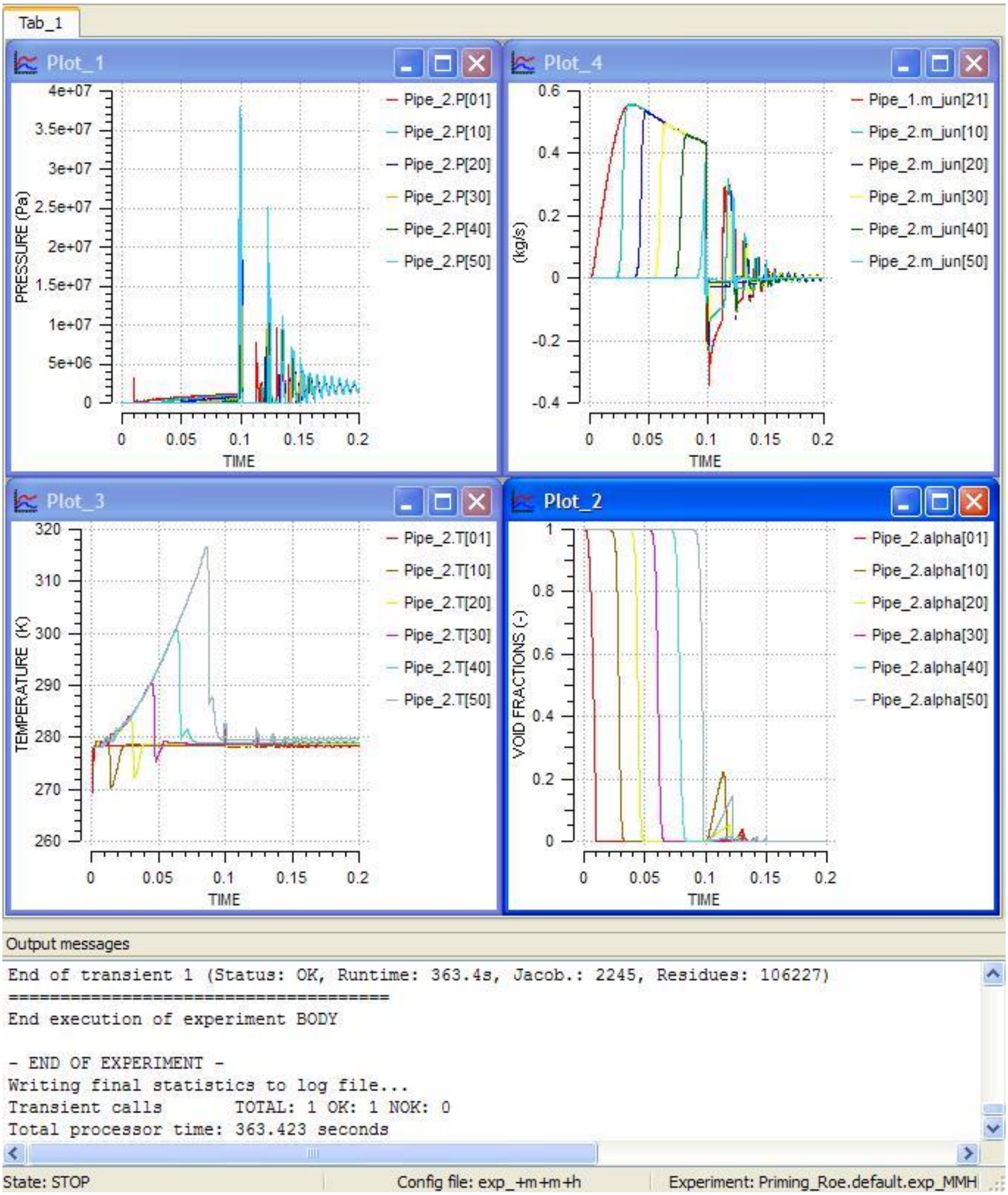
Initial Pipe2 Pressure: 0.1 MPa (GN2)



2.1.8.3 Results with MMH

Using real MMH as working fluid, similar results than with real water can be found. In this case, the simulation has been run with the Roe-3rd scheme in the filling pipe, obtaining more precise results for the first water hammer peak as explained in RD-18, section 2.1.3.

Initial Pipe2 Pressure: 0.001 MPa (vapour)

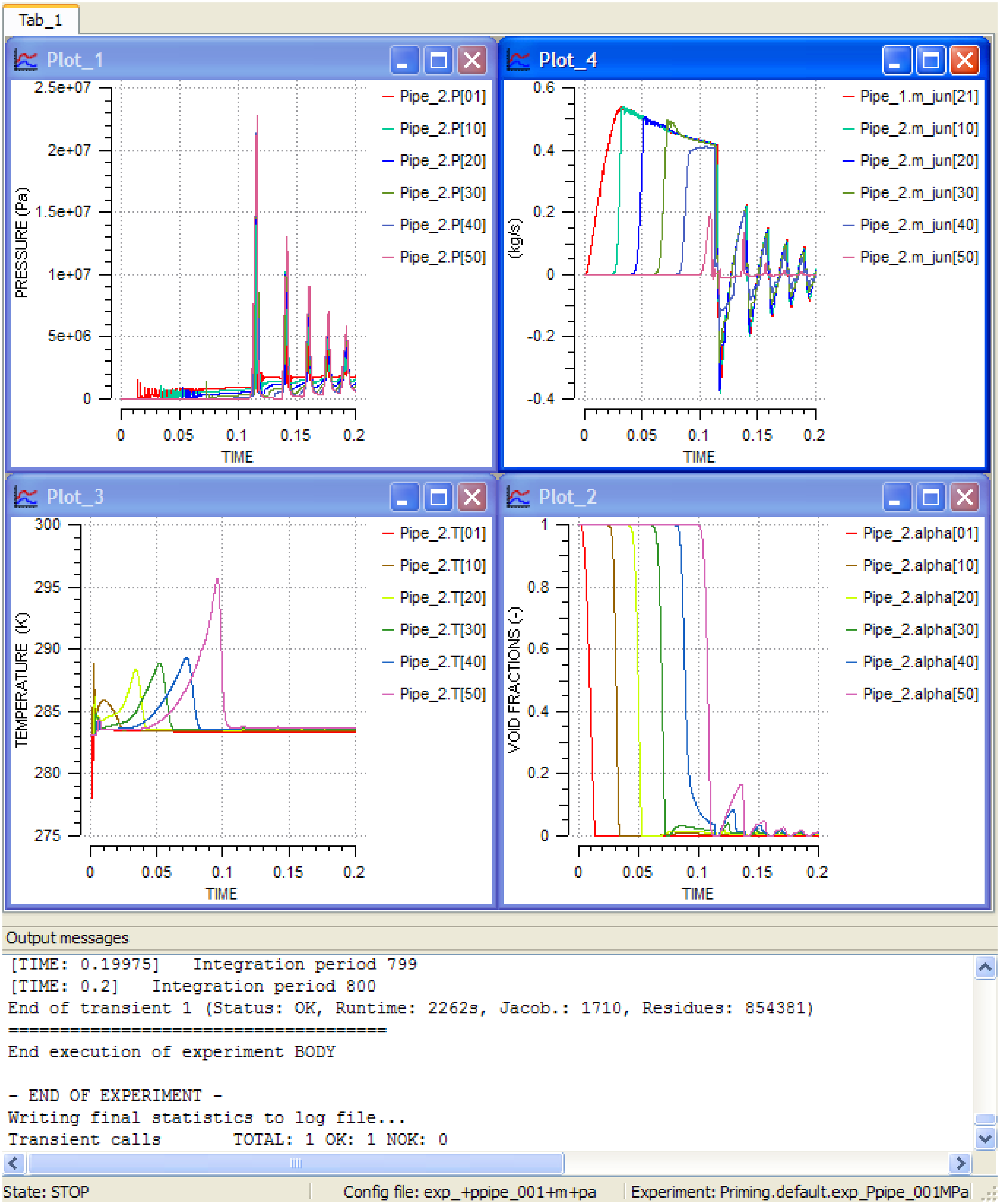


2.1.8.4 Results with water (gas desorption activated)

This case uses real water as working fluid but with gas absorption/desorption activated. The liquid upstream de valve has initially a mass concentration of diluted gas (N2) of xd_nco = 1e-4.

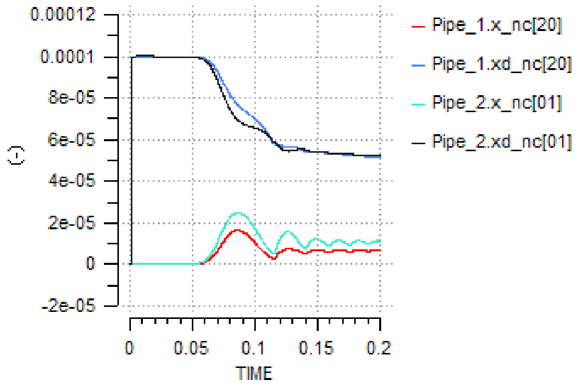
With respect to the case with absorption/desorption deactivated, results are slightly smoothed because of the non-condensable gas (N2) released when the liquid is filling the pipe 2 downstream the valve.

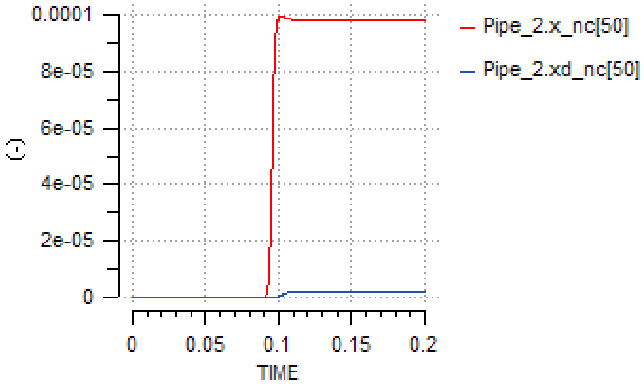
Initial Pipe2 Pressure: 0.00123 MPa (sat. vapour)



Results should change depending on the absorption/desorption coefficients (A_coef_sol, A_coef_sol, Ca, Cd input data).

The plots below show how the diluted gas initially present upstream the valve (Pipe_1) has been released during the filling process of Pipe_2 smoothing the water hammer.





The CPU time is too much high with the absorption/desorption option activated. This is because, with this option, it has to be calculated the small amount of non-condensable gases released affecting in great measure the liquid properties and the reflection of the acoustic waves.

2.1.9 Pipe test: Cold thruster (T-FF-011)

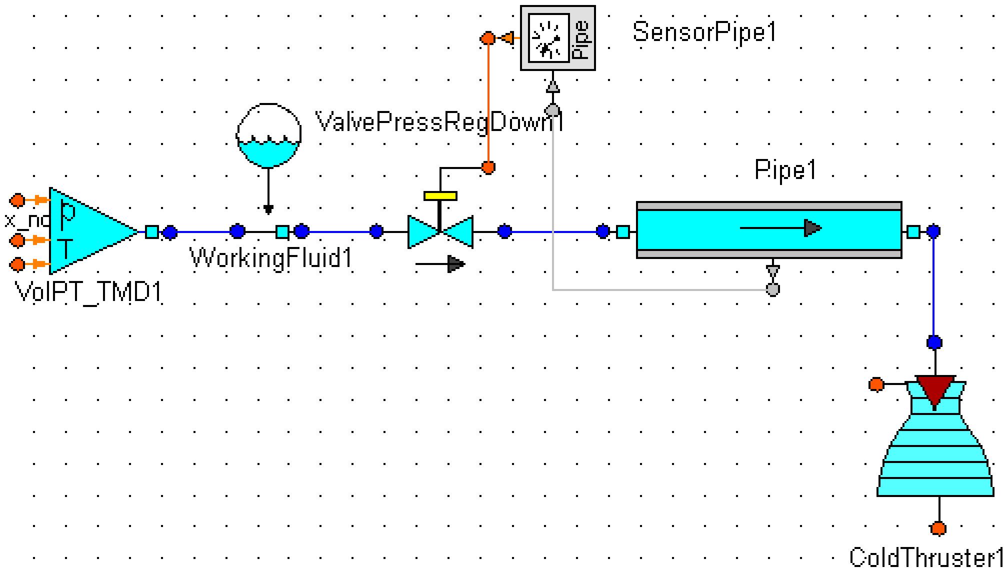
Library:	FLUID_FLOW_1D_EXAMPLES
Model Name:	test_coldThruster
Partition Name:	default
Experiment Name:	exp1

2.1.9.1 Model description

This example shows the FLUID_FLOW_1D capabilities concerning subsonic/supersonic transitions in a Nozzle using real properties fluids. The nozzle is simulated by a particular Pipe component, which takes into account variable area, inertia forces, pressure losses and density changes.

Shock capture inside the supersonic part is modelled depending on the outlet pressure

The model with ESPSS is representative of a typical cold thruster system including a pressure regulation valve from HP to medium pressure, a pipe for driving gases at medium pressure to the thruster and finally the thruster itself provided with an on/off valve:

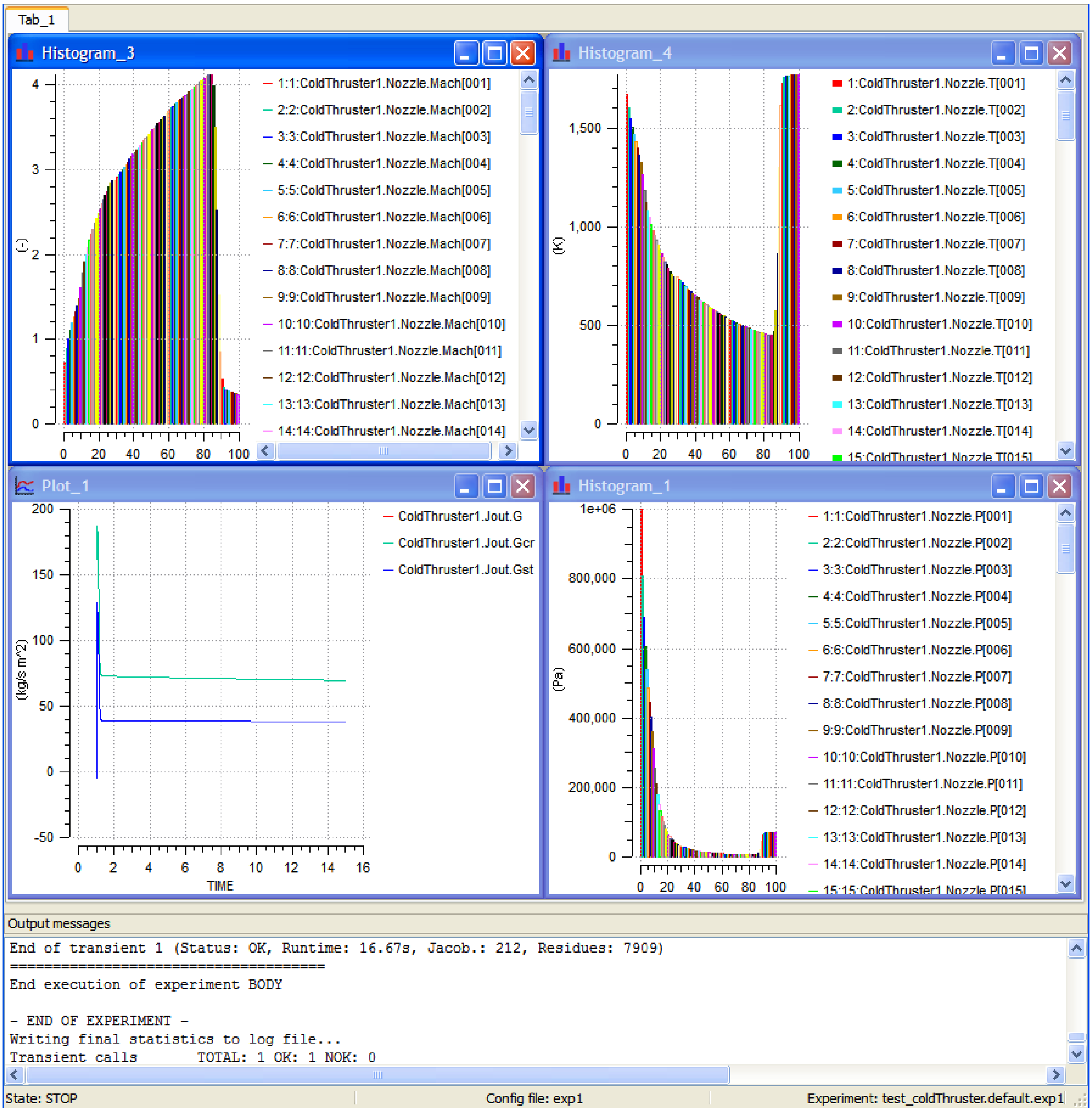


2.1.9.2 Results

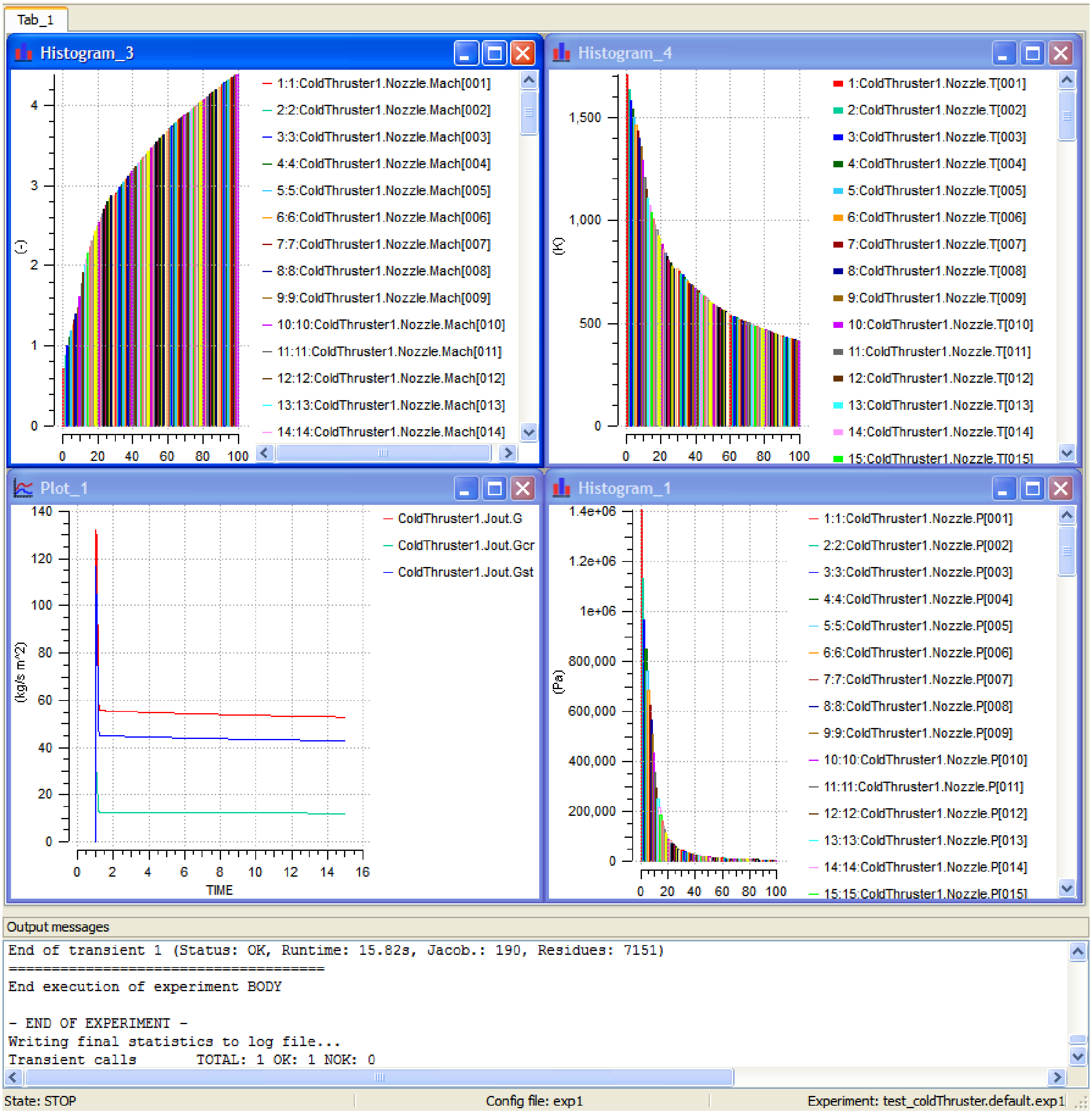
The plots shown below are directly output from the EcosimPro monitor:

- ESPSS model can simulate correctly the three stationary cases: Shock inside the nozzle, outside the nozzle and attached to the exit
- ESPSS model can simulate correctly transient cases passing from adapted conditions to non-adapted conditions and vice versa
- Real fluid conditions can be calculated, even if the fluid is condensing inside the nozzle due to the cold temperatures reached in supersonic flow

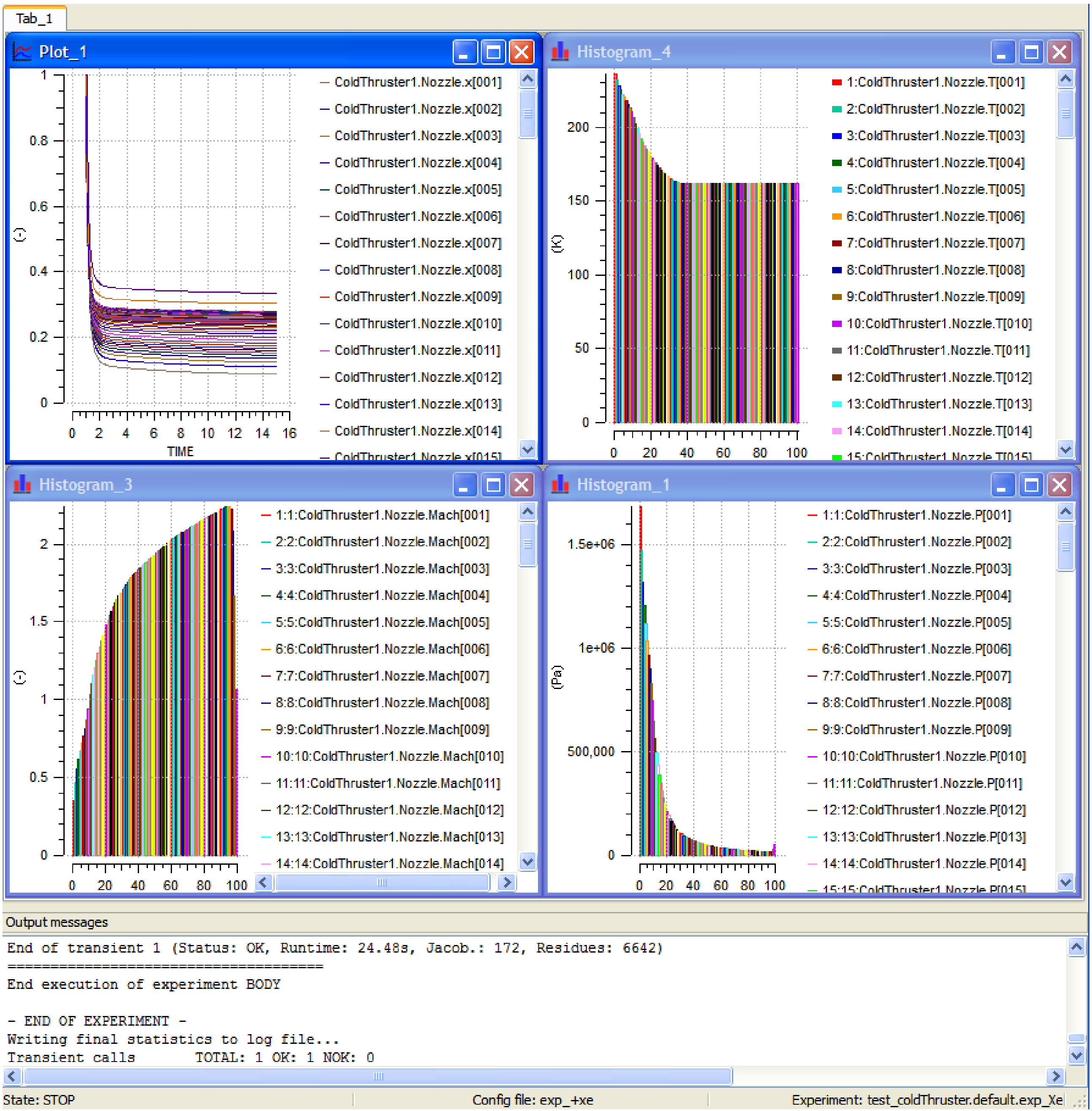
Perfect gas N2 (Tin= 2000 K; Pin = 20 bar; Pamb = 0.7 bar): Shock inside the nozzle:



Transition to Pamb = 0.2 bar: Shock attached to the nozzle exit



Real Xe properties (Tin= 250 K; Pin = 20 bar; Pamb = 0.7 bar): Liquefaction in supersonic region



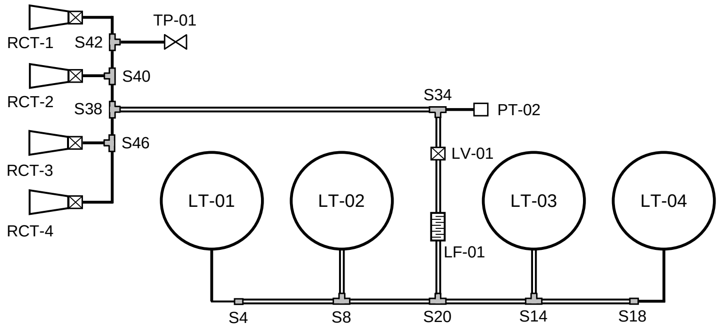
2.1.10 RCS Case. Thrusters feeding system (T-FF-004)

Library: FLUID_FLOW_1D_EXAMPLES

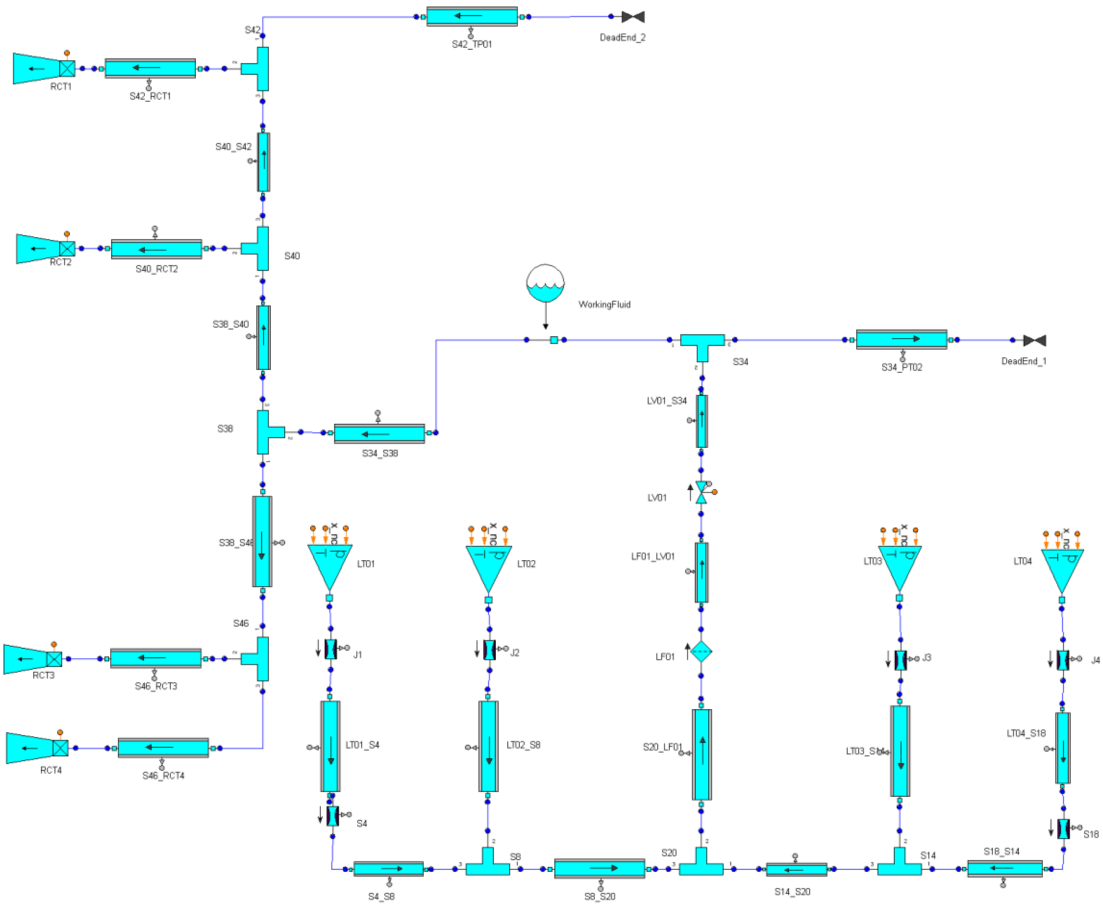
Model Name:	Test_RCS
Partition Name:	default
Experiment Name:	exp2

2.1.10.1 Model description

This model is representative of a satellite propulsion system with **hydrazine**



The EcosimPro schematic of the model appears in figure below, where the main feeding lines are included:



Main data are described below:

PIPE	LENGTH [mm]	DIAMETER [in]	PIPE	LENGTH [mm]	DIAMETER [in]
LT01-S4	290	0.25	S34-S38	113	0.375
LT02-S8	275	0.25	S38-S40	189	0.25
LT03-S14	273	0.25	S40-S42	1980	0.25
LT03-S18	287	0.25	S42-RCT1	1750	0.25
S4-S8	2355	0.375	S40-RCT2	1684	0.25
S8-S20	212	0.375	S38-S46	1498	0.25
S20-S14	1647	0.375	S46-RCT3	1579	0.25
S14-S18	2117	0.375	S46-RCT4	3771	0.25
S20-LF01	1127	0.375	TP01-S42	520	0.25
LF01-LV01	263	0.375			
LV01-S34	113	0.375			

RCT (Reaction Control Thruster) Performance Characteristics:

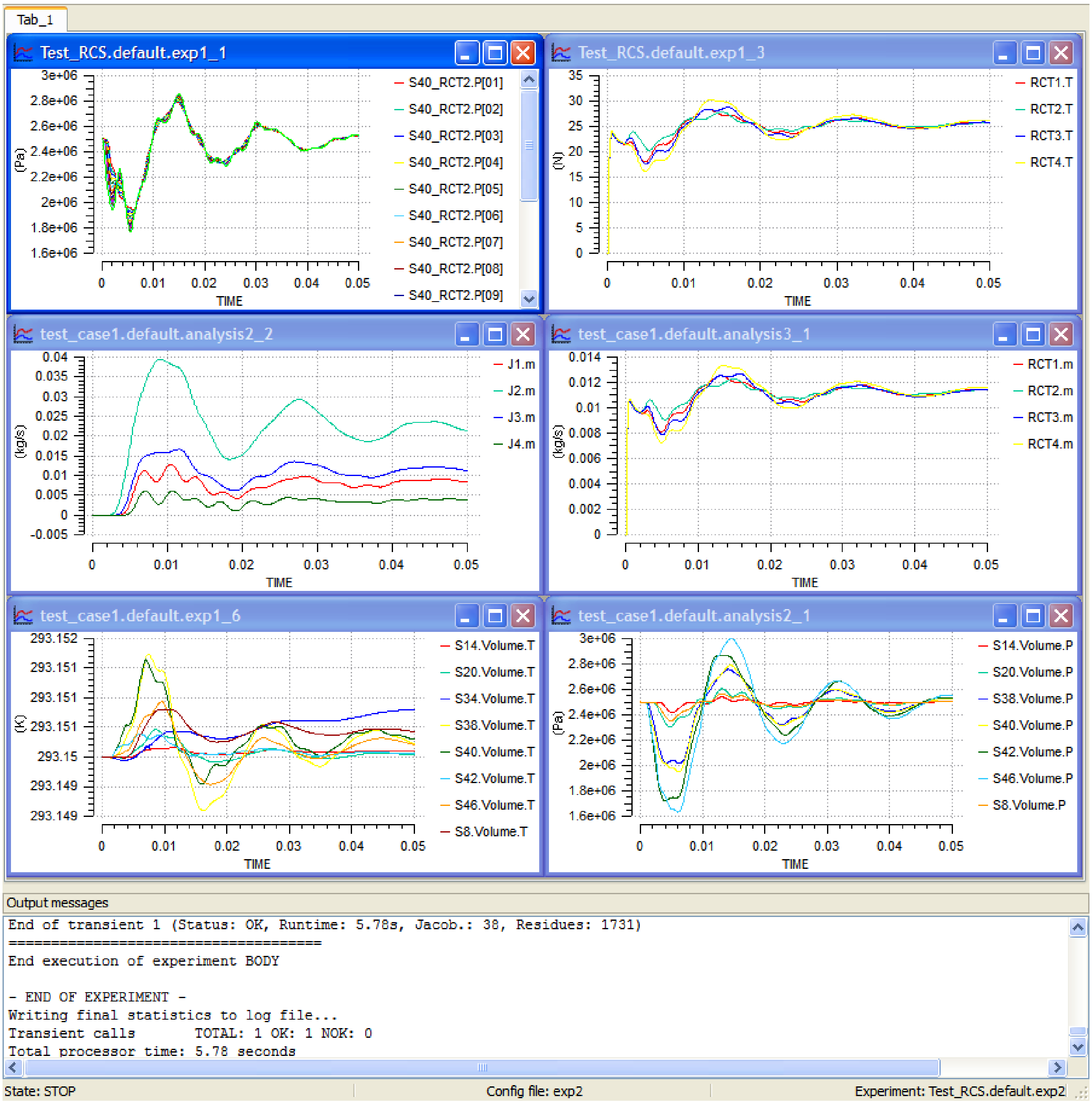
$$thrust: \quad T[N] = A \cdot \sqrt{P_{in}[bar]} + B$$
$$specific\ impulse: \quad I_{sp}[Ns/Kg] = 2.04 \cdot P_{in}[bar] + 2203.5$$
$$mass\ \ flow\ rate: \quad \dot{m}[Kg/s] = T/I_{sp}$$

RCT	A	B
1	6.628	-7.669
2	6.588	-7.619
3	6.641	-7.713
4	6.602	-7.388

2.1.10.2 Results

The following phenomena can be shown:

- Simultaneous opening of the 4 RCT valves (tanks initial pressure of 25 bar, tank liquid temperature kept constant at 20°C) produce important shifts in the line pressures.
- Steady conditions can be easily calculated in a few CPU seconds by running the model.


Tab_1
Test_RCS.default.exp1_1
(Pa)
TIME
S40_RCT2.P[01]
S40_RCT2.P[02]
S40_RCT2.P[03]
S40_RCT2.P[04]
S40_RCT2.P[05]
S40_RCT2.P[06]
S40_RCT2.P[07]
S40_RCT2.P[08]
S40_RCT2.P[09]
Test_RCS.default.exp1_3
(N)
TIME
RCT1.T
RCT2.T
RCT3.T
RCT4.T
test_case1.default.analysis2_2
(kg/s)
TIME
J1.m
J2.m
J3.m
J4.m
test_case1.default.analysis3_1
(kg/s)
TIME
RCT1.m
RCT2.m
RCT3.m
RCT4.m
test_case1.default.exp1_6
(K)
TIME
S14.Volume.T
S20.Volume.T
S34.Volume.T
S38.Volume.T
S40.Volume.T
S42.Volume.T
S46.Volume.T
S8.Volume.T
test_case1.default.analysis2_1
(Pa)
TIME
S14.Volume.P
S20.Volume.P
S38.Volume.P
S40.Volume.P
S42.Volume.P
S46.Volume.P
S8.Volume.P
Output messages
End of transient 1 (Status: OK, Runtime: 5.78s, Jacob.: 38, Residues: 1731)
====================================
End execution of experiment BODY
- END OF EXPERIMENT -
Writing final statistics to log file...
Transient calls TOTAL: 1 OK: 1 NOK: 0
Total processor time: 5.78 seconds
State: STOP
Config file: exp2
Experiment: Test_RCS.default.exp2

2.1.11 Vacuum network (T-FF-001)

Library:	FLUID_FLOW_1D_EXAMPLES
Model Name:	Test_vacuum_network
Partition Name:	default
Experiment Name:	exp1

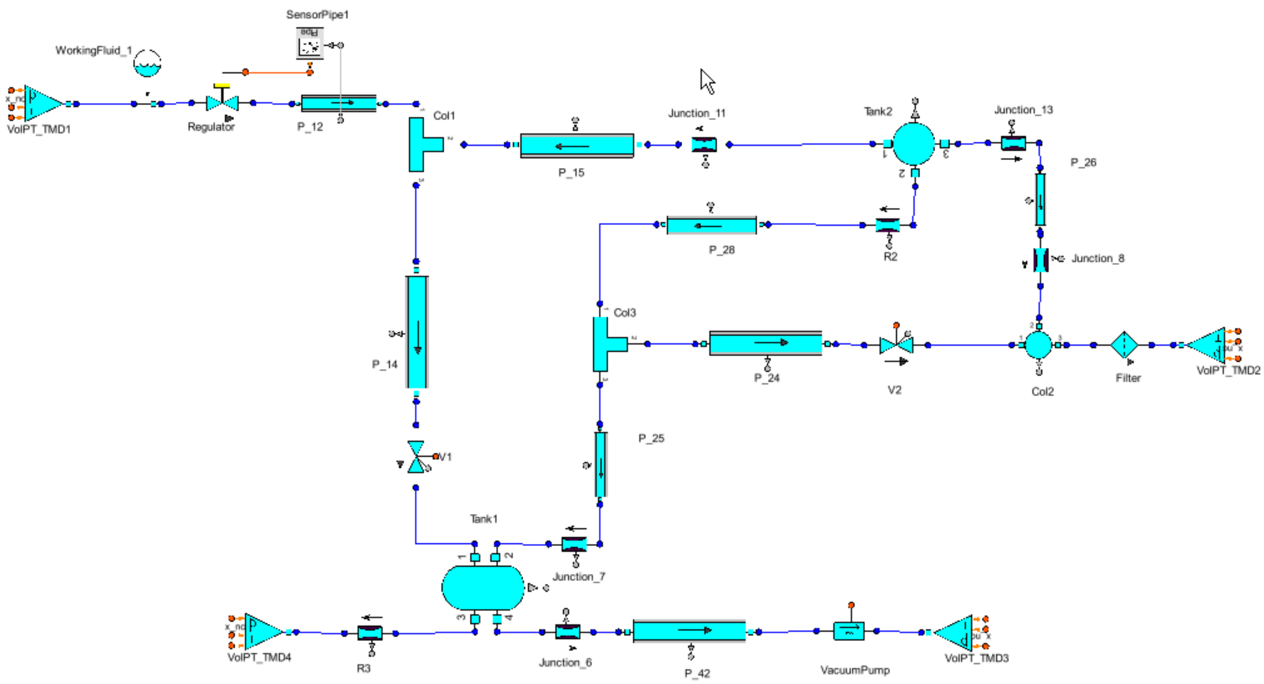
2.1.11.1 Model description

The following example shows some of the FLUID_FLOW_1D capabilities modeling a vacuum network. It contains two single tanks, a pressure regulator, a vacuum pump, some pneumatic valves, Tees and calibrated orifices. The aim of the simulation is to control the pressure level in the tanks Tank1 and Tank2 by switching the valves V1 and V2.

The model represents a system that deals with the extraction of polluted air from several places where high cleaning conditions are required. Tank1 and Tank2 are 0.01m3 each. The working fluid is air and it is set by the component "WorkingFluid1".

The pressure-regulator component keeps the vacuum-level at a certain value (0.2 bar in this case). It is a valve with variable throat section driven by the upstream / downstream pressure difference (the allowable pressure difference is set to 0.02 bar). The maximum throat section is 1E-5m2.

The circuit contains a porous filter. The pressure loss in this component is calculated as a linear relation between flow and pressure-drop for given reference conditions. For this model, those conditions are a pressure drop of 0.8 bar for 0.0002 kg/s mass flow.



Vacuum is achieved by means of a water-ring pump that is simulated by a Jun_TMD (time dependant mass flow) component.

The following table is implemented in the model (CONTINUOUS BLOCK) to be used in the interpolation of the imposed mass flow vs. speed and vacuum pressure:

	Vacuum pressure (Pa)									
Speed (rpm)	0	10159	20318	30477	40637	50796	60955	71114	81273	91432
1450	**0.039176**	**0.035872**	**0.03304**	**0.028792**	**0.025016**	**0.020768**	**0.016048**	**0.010384**	**0.005664**	**0.0005664**
1750	**0.053808**	**0.04956**	**0.042952**	**0.03776**	**0.032096**	**0.025016**	**0.019824**	**0.01416**	**0.008496**	**0.0006608**

Two on-off valves (V1 and V2) control the incoming atmospheric air, either from the pressure regulator branch or the filter branch. Additionally, there are two calibrated holes R2 and R3 (quadratic restrictors).

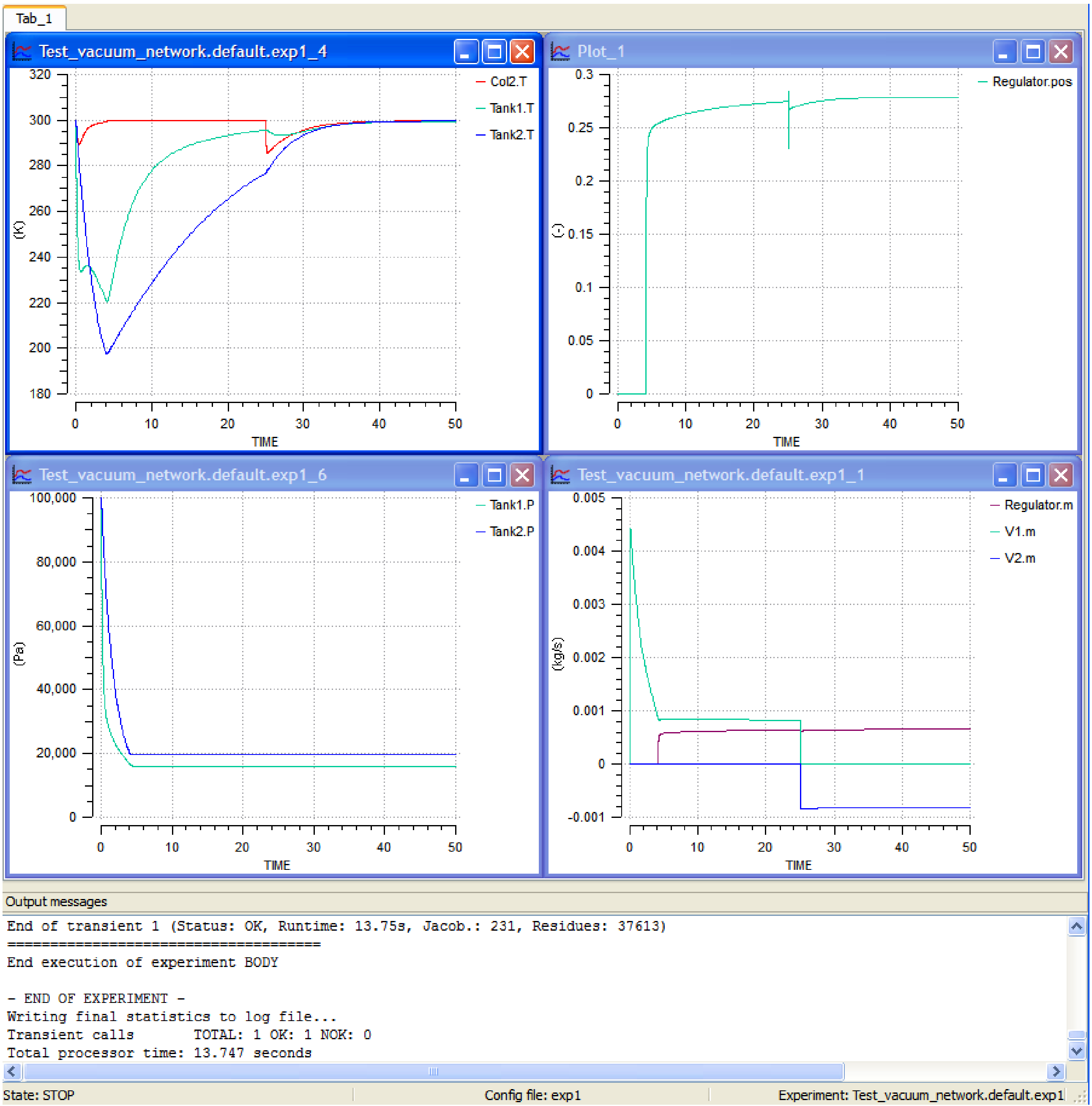
Some other elements are used to connect all the parts of the circuit, such as pipes, junctions and tee joints. The diameter considered in all these items is 0.030 m.

Finally, the environmental conditions such as ambient pressure and temperature are simulated using constant boundary conditions.

2.1.11.2 Results

In the beginning of the simulation, the valve V1 is opened and the V2 is closed, permitting a direct air flow from the external atmosphere (HP_tank) to the Tank1. In this first stage, the air in the low pressure area is pumped out reaching the desired pressure conditions in Tank1 in less than 5 s. The pressure drops until the pressure difference in the regulator is enough to open it. After this, the pump keeps the pressure level in the tanks pumping out the incoming air flow from the regulator and filter branches and the tank leaks.

The valves are kept in their initial status up to 25 s, when V1 is closed and V2 is opened. Then, the air coming from the pressure regulator reaches Tank1 after passing through Tank2. The change of valve status can be clearly seen in the mass flow plots. The pressure level of the tanks is kept in the same way, as in the previous stage. Finally, the simulation is stopped at 50 s.

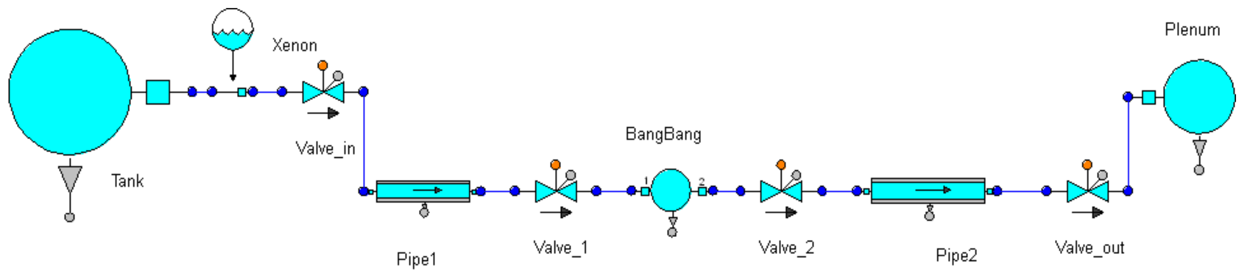

Tab_1
Test_vacuum_network.default.exp1_4
Col2.T
Tank1.T
Tank2.T
(K)
TIME
Plot_1
Regulator.pos
(-)
TIME
Test_vacuum_network.default.exp1_6
Tank1.P
Tank2.P
(Pa)
TIME
Test_vacuum_network.default.exp1_1
Regulator.m
V1.m
V2.m
(kg/s)
TIME
Output messages
End of transient 1 (Status: OK, Runtime: 13.75s, Jacob.: 231, Residues: 37613)
====================================
End execution of experiment BODY
- END OF EXPERIMENT -
Writing final statistics to log file...
Transient calls TOTAL: 1 OK: 1 NOK: 0
Total processor time: 13.747 seconds
State: STOP
Config file: exp1
Experiment: Test_vacuum_network.default.exp1

2.1.12 Bang-Bang case (T-FF-005)

Library: FLUID_FLOW_1D_EXAMPLES
Model Name: Test_BangBang
Partition Name: default
Experiment Name: exp1

2.1.12.1 Model description

This model is representative of the bang-bang regulation system for electrical propulsion. The EcosimPro schematic of the model appears in figure below:



Main data are described below:

	BangBang	Plenum	Pipes	Tank	Valves
Volumes (m3)	$5\ 10^{-7}$	0.005		1.0	
Pipe length (m)			0.21+0.39		
Pipe inner diameter (m)			0.00177		
Area (m2)					1.365e-8
Initial Pressure (Pa)	2e5	2e5	2e5	80e5	
Initial Temperature (K)	280	280	280	280	

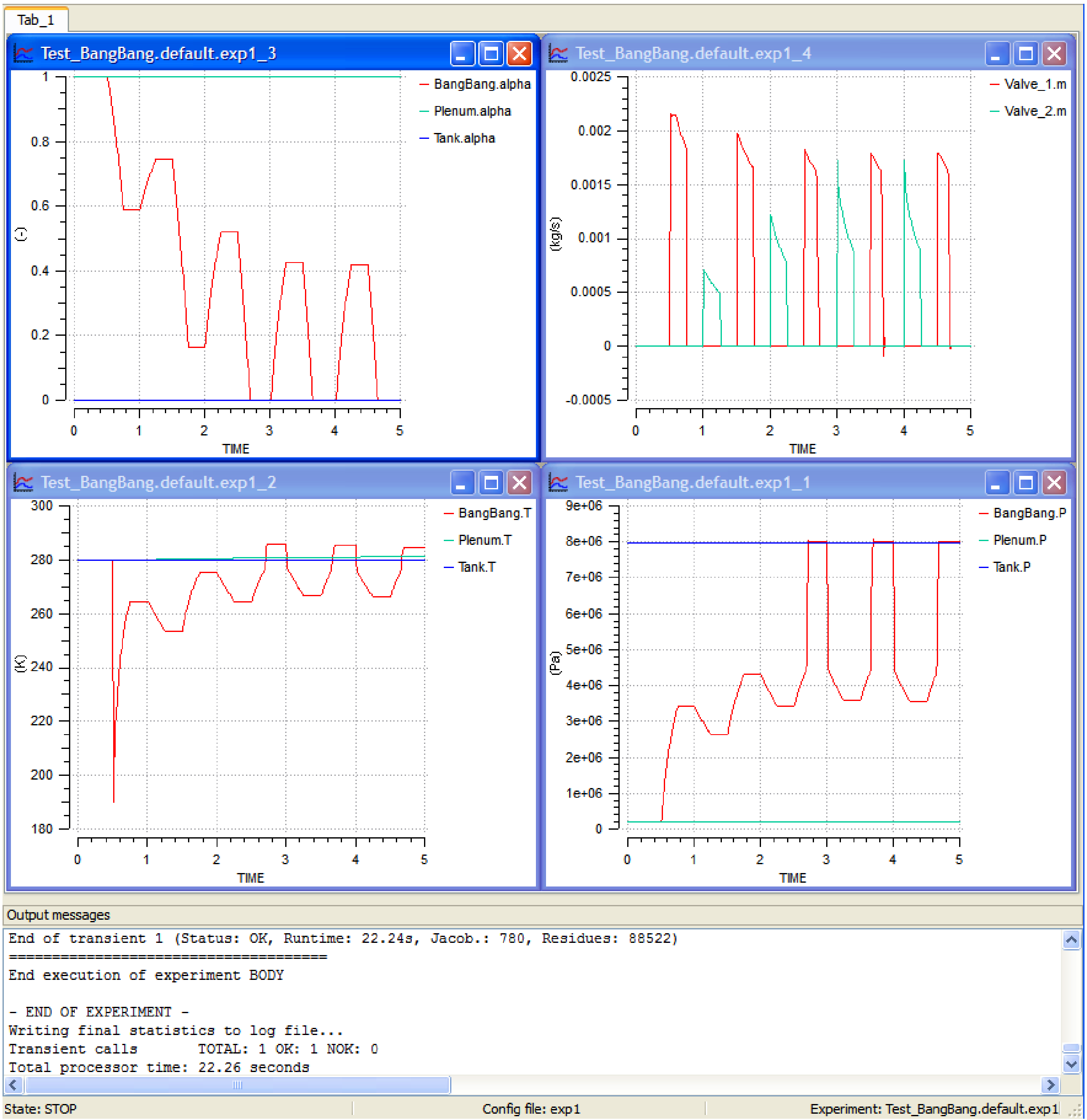
Valve activation is controlled by the following activation laws:

```
Valve_1.s_pos.signal[1] = pulse(TIME-0.5, 1, 0.25)
Valve_2.s_pos.signal[1] = pulse(TIME-1,   1, 0.25)
```

2.1.12.2 Results

The following phenomena can be shown:

- Very low mass flow goes through the pipes due to the extremely thin valve areas.
- Xenon flow crosses the saturation line entering into the cavity.
- The BangBang cavity evolves between liquid and two-phase conditions depending on the pressure level (valve actuation) and on the temperature.

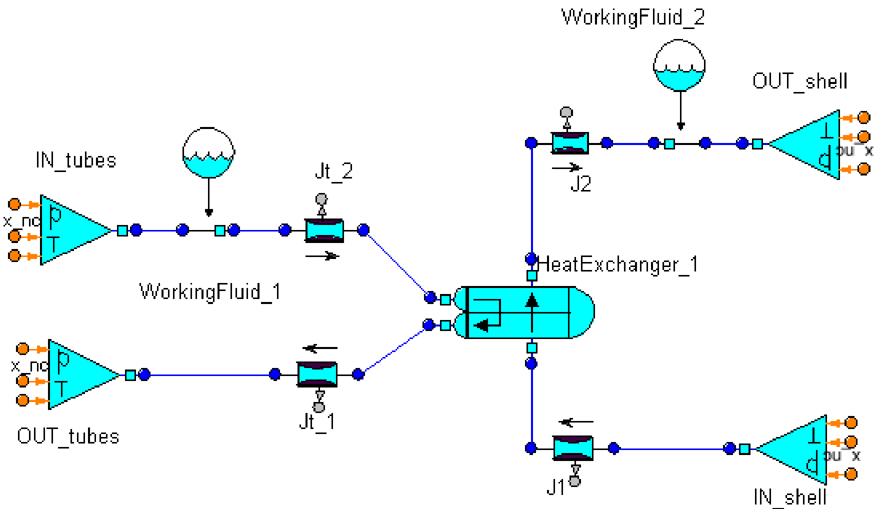

Tab_1
Test_BangBang.default.exp1_3
BangBang.alpha
Plenum.alpha
Tank.alpha
(-)
TIME
Test_BangBang.default.exp1_4
Valve_1.m
Valve_2.m
(kg/s)
TIME
Test_BangBang.default.exp1_2
BangBang.T
Plenum.T
Tank.T
(K)
TIME
Test_BangBang.default.exp1_1
BangBang.P
Plenum.P
Tank.P
(Pa)
TIME
Output messages
End of transient 1 (Status: OK, Runtime: 22.24s, Jacob.: 780, Residues: 88522)
======================================
End execution of experiment BODY
- END OF EXPERIMENT -
Writing final statistics to log file...
Transient calls TOTAL: 1 OK: 1 NOK: 0
Total processor time: 22.26 seconds
State: STOP
Config file: exp1
Experiment: Test_BangBang.default.exp1

2.1.13 Boiler device. Heat Exchanger (T-FF-006)

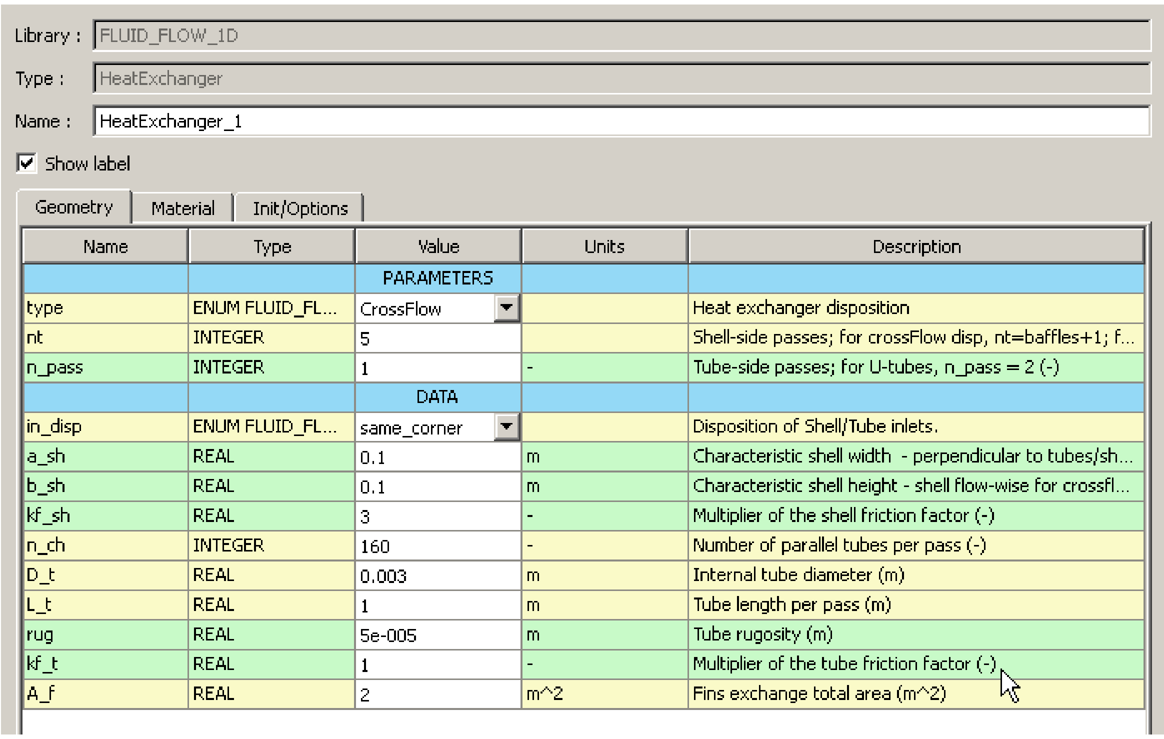
Library:	FLUID_FLOW_1D_EXAMPLES
Model Name:	Test_Hex
Partition Name:	default
Experiment Name:	exp1

2.1.13.1 Model description

This example shows capabilities concerning the simulation of a boiler using a shell-and-tube heat exchanger component. The model represents boiler including the hot gases and water circuits.



The heat exchanger data are intentionally set to simulate the case where the liquid water will reach the boiling conditions:



Library : FLUID_FLOW_1D
Type : HeatExchanger
Name : HeatExchanger_1
Show label
Geometry | Material | Init/Options

Name	Type	Value	Units	Description
		PARAMETERS		
type	ENUM FLUID_FL...	CrossFlow		Heat exchanger disposition
nt	INTEGER	5		Shell-side passes; for crossFlow disp, nt=baffles+1; f...
n_pass	INTEGER	1	-	Tube-side passes; for U-tubes, n_pass = 2 (-)
		DATA		
in_disp	ENUM FLUID_FL...	same_corner		Disposition of Shell/Tube inlets.
a_sh	REAL	0.1	m	Characteristic shell width - perpendicular to tubes/sh...
b_sh	REAL	0.1	m	Characteristic shell height - shell flow-wise for crossfl...
kf_sh	REAL	3	-	Multiplier of the shell friction factor (-)
n_ch	INTEGER	160	-	Number of parallel tubes per pass (-)
D_t	REAL	0.003	m	Internal tube diameter (m)
L_t	REAL	1	m	Tube length per pass (m)
rug	REAL	5e-005	m	Tube rugosity (m)
kf_t	REAL	1	-	Multiplier of the tube friction factor (-)
A_f	REAL	2	m^2	Fins exchange total area (m^2)

Geometry | Material | Init/Options

Name	Type	Value	Units	Description
		DATA		
mat_c	ENUM THERMAL....	Carbon_Steel		Case wall material
cp_c	REAL	500	J/kg*K	Case wall specific heat if Case_mat=None (J/kg*K)
rho_c	REAL	1000	kg/m^3	Case wall density if Case_mat=None (kg/m^3)
th_c	REAL	0.002	m	Case wall thickness (m)
mat_t	ENUM THERMAL....	Copper	-	Tube-Fins material (-)
cp_t	REAL	500	J/kg*K	Tube wall specific heat if Tube_mat=None (J/kg*K)
rho_t	REAL	1000	kg/m^3	Tube wall density if Tube_mat=None (kg/m^3)
th_f	REAL	0.002	m	Fins thickness (m)
th_t	REAL	0.002	m	Tube wall thickness (m)

Geometry | Material | Init/Options

Name	Type	Value	Units	Description
		DATA		
P_o	REAL	180000	Pa	Initial Tube-pressure (Pa)
P_sh_o	REAL	100000	Pa	Initial Shell-pressure (Pa)
T_o	REAL	293.15	K	Initial temperature (K)
T_outside	REAL	293.15	K	Environment temperature (K)
x_nco	REAL	0	-	Initial non-condensable mass fraction at tubes side (-)
x_sh_nco	REAL	0	-	Initial non-condensable mass fraction at shell side (-)
h_outside	REAL	0	W/m^2*K	Heat transfer coefficient with the environment (W/m...
ht_option	ENUM FLUID_PR...	HT_tube_1ph		Tubes-internal fluid heat transfer option
hc_dat	REAL	1	W/m^2*K	Heat transfer coefficient - only if ht_option = Ht_con...
fr_option	ENUM FLUID_PR...	FR_tube_1ph		Tubes-internal fluid friction correlation option

Inlet/outlet heat exchanger boundary conditions are supposed to be fixed:

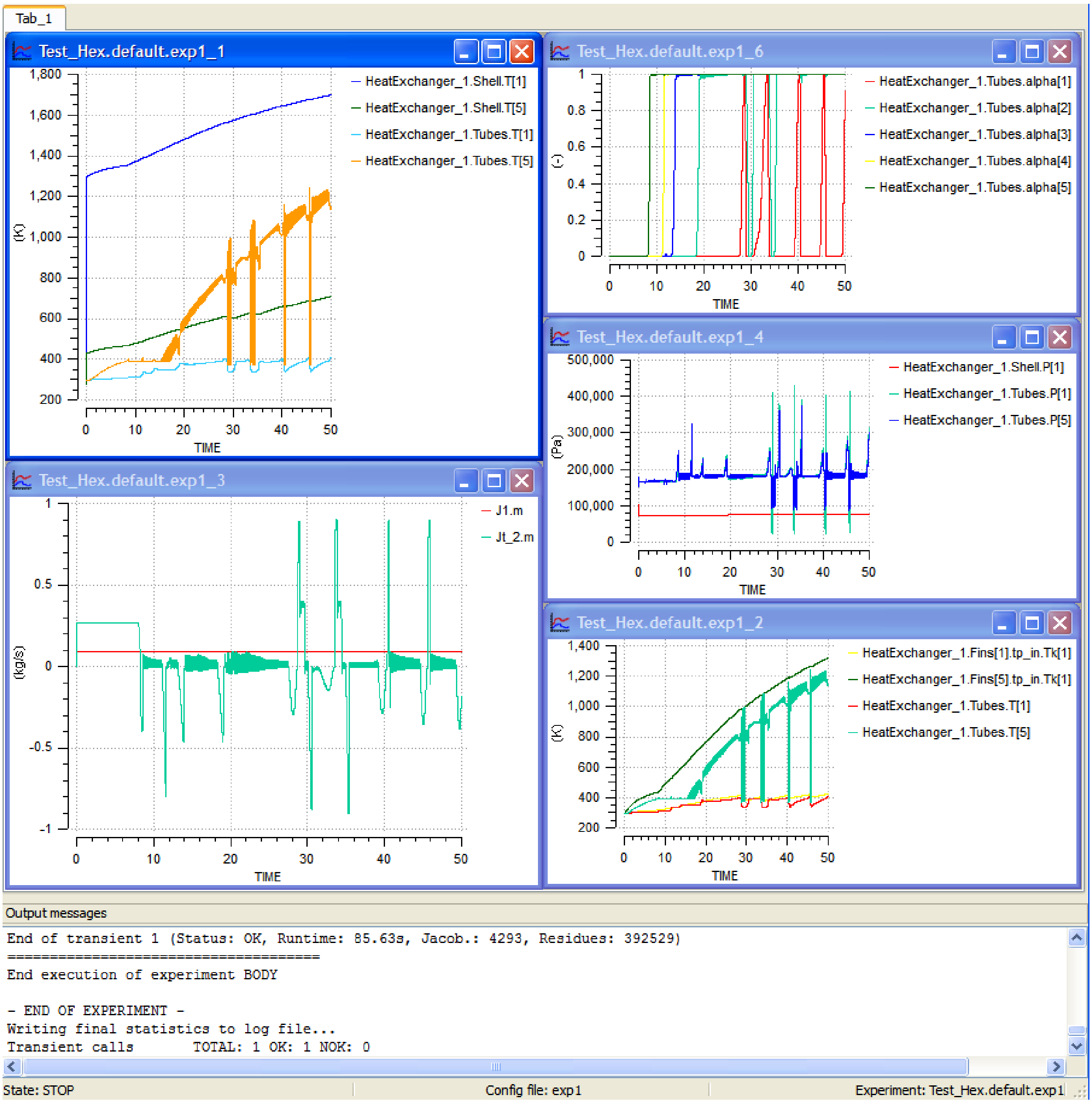
```
IN_shell.s_pres.signal[1] = 1e5
IN_shell.s_temp.signal[1] = 2000
OUT_shell.s_pres.signal[1] = 0.6e5
OUT_shell.s_temp.signal[1] = 2000

IN_tubes.s_pres.signal[1] = 1.8e5
IN_tubes.s_temp.signal[1] = 300
OUT_tubes.s_pres.signal[1] = 1.5e5
OUT_tubes.s_temp.signal[1] = 300
```

2.1.13.2 Results

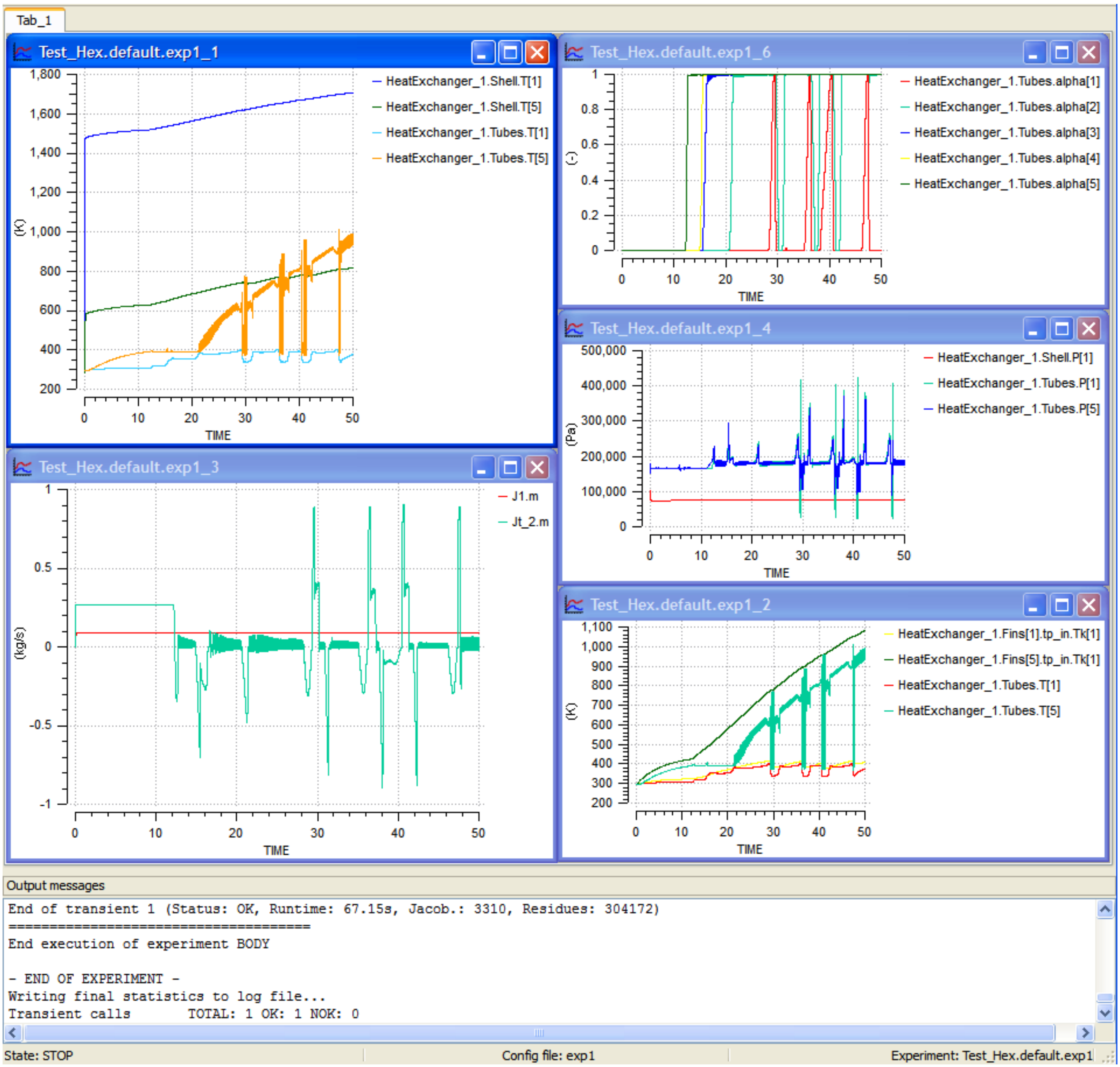
The following phenomena can be shown:

- The water flow suffers sudden reductions (even momentary reverse flow) with the corresponding water-hammer peaks when the boiling point is reached along the tubes. In these points the quality (alpha variable) goes from zero to one.
- After some time, the outgoing vapor becomes reheated, but the tube inlet conditions are oscillatory due to the coupling of the heat exchange coefficient (that is strongly dependent on the mass flow and the phase conditions) with the temperature.
- A lesser number of tubes or a lesser hot gases temperature would take the water outlet temperatures under the saturation point making the simulation too much simpler.

Counter flow disposition:



Cross flow disposition with 5 gas passes (same shell geometry as before):



2.2 TANKS LIBRARY

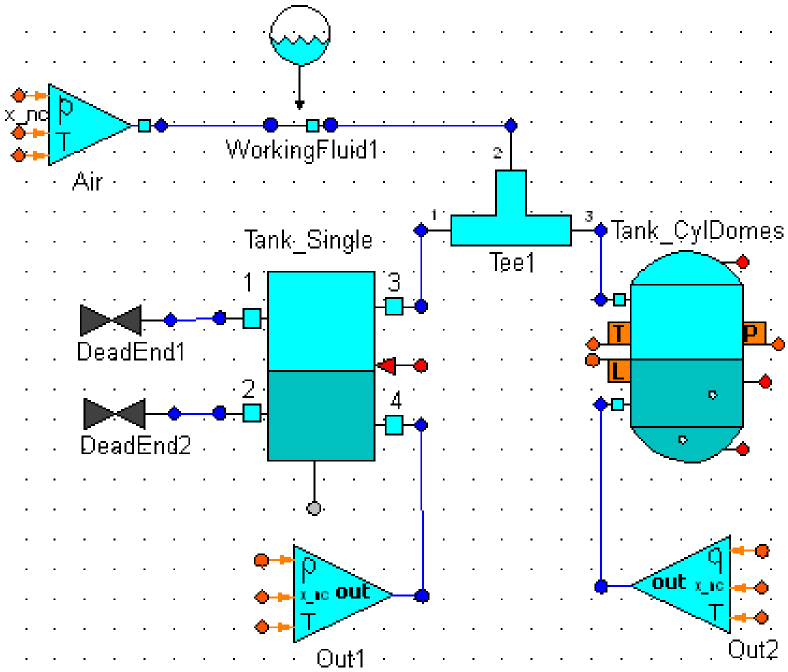
2.2.1 Tank test case (T-TNK-003)

Library:	ROCKET_EXAMPLES
Model Name:	Test_tank_bis
Partition Name:	default
Experiment Name:	exp1

2.2.1.1 Model description

This example validates the TANK capabilities concerning the Tank basic formulation.

The model with ESPSS is representative of two kind of Tanks (Tank_Single vs. 1D Tank) partially filled with liquid in contact with a gas volume in presence of gravity. The tanks have two fluid ports, one connected to the atmosphere, and another in the bottom producing a free loss of liquid.



This simple test case provides an easy way to test the tank formulation because the liquid level evolution can be compared with an analytical solution:

- In the case of the Tank_Single component, this test also validates the volume component (see §2.1.2) because the Tank_Single component inheriting the Volume formulation concerning the conservation equations, the Tank adding the equations corresponding to the level calculation assuming that all the liquid is in the lower part of the tank
- In the case of the 1D Tank (Tank_CylDomes) the liquid level evolution is the result of the complex formulation taking in to account two moving spatial discretizations, one in the gas and another in the liquid side of the tank, and the equilibrium pressure equation at the gas/liquid interface

Tank radius is 1m and the tank height is 10m. Initial liquid level is 5m. The holes in the bottom are 0.01 m2, while the venting diameter is 0.1 m. The liquid fluid is water and the non-condensable gas is Air.

2.2.1.2 Results

The test consists of in the calculation of the bottom pressure/level evolution due to a free loss of liquid by a "hole" in the tank bottom. The gas side filling process will be calculated accordingly with the loss of liquid at nearly constant gas pressure because of the big area communicating the tank with the atmosphere.

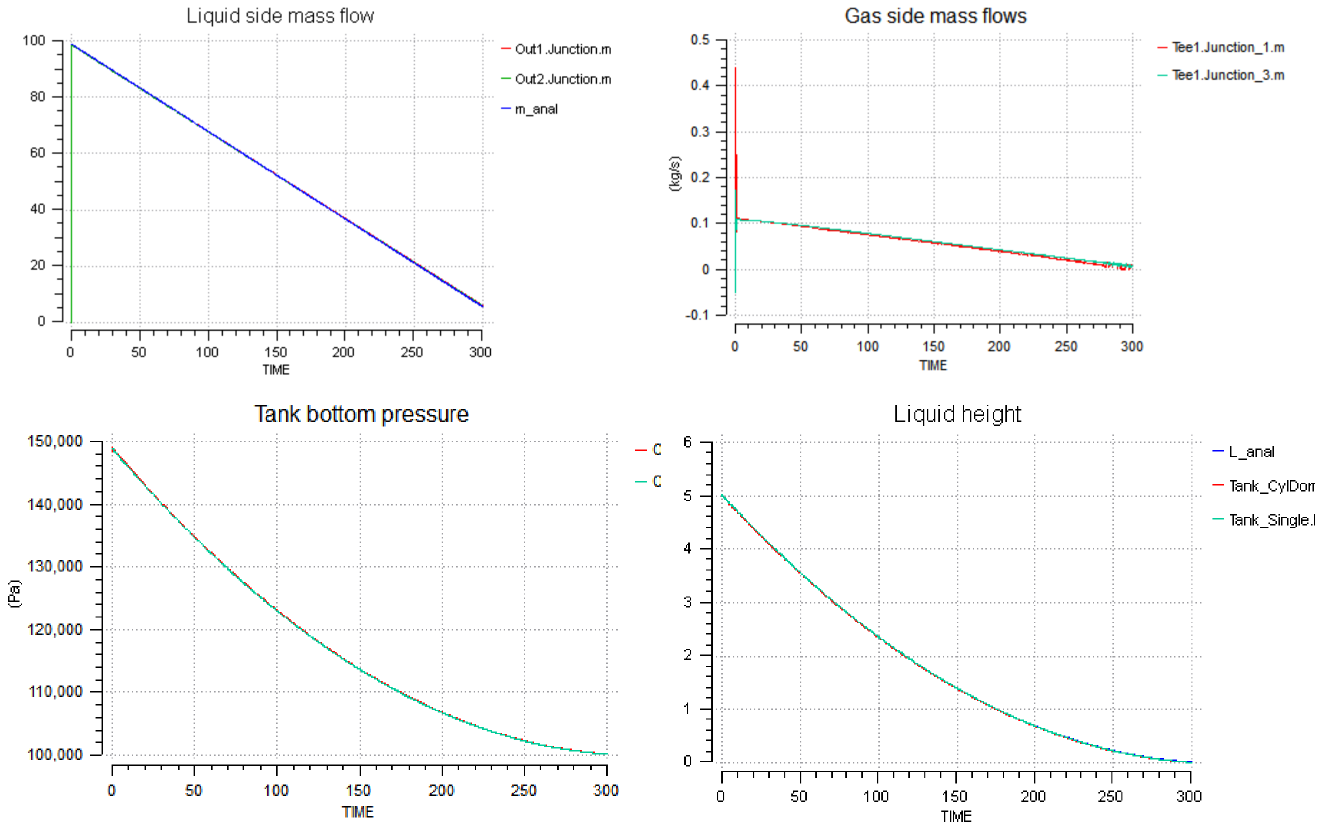
Then, the analytical solution would be as follows:

$$m_{anal} = A_{hole}\rho_{liq}\sqrt{2gH}$$

$$H' = m_{anal}A_{\tan k} / \rho_{liq}$$

Where "m_{anal}" is the outlet mass flow and H is tank height

Next figures show the pressure, liquid height and mass flows evolution compared with the analytical values:



As it can be seen, analytical and calculation results are the same

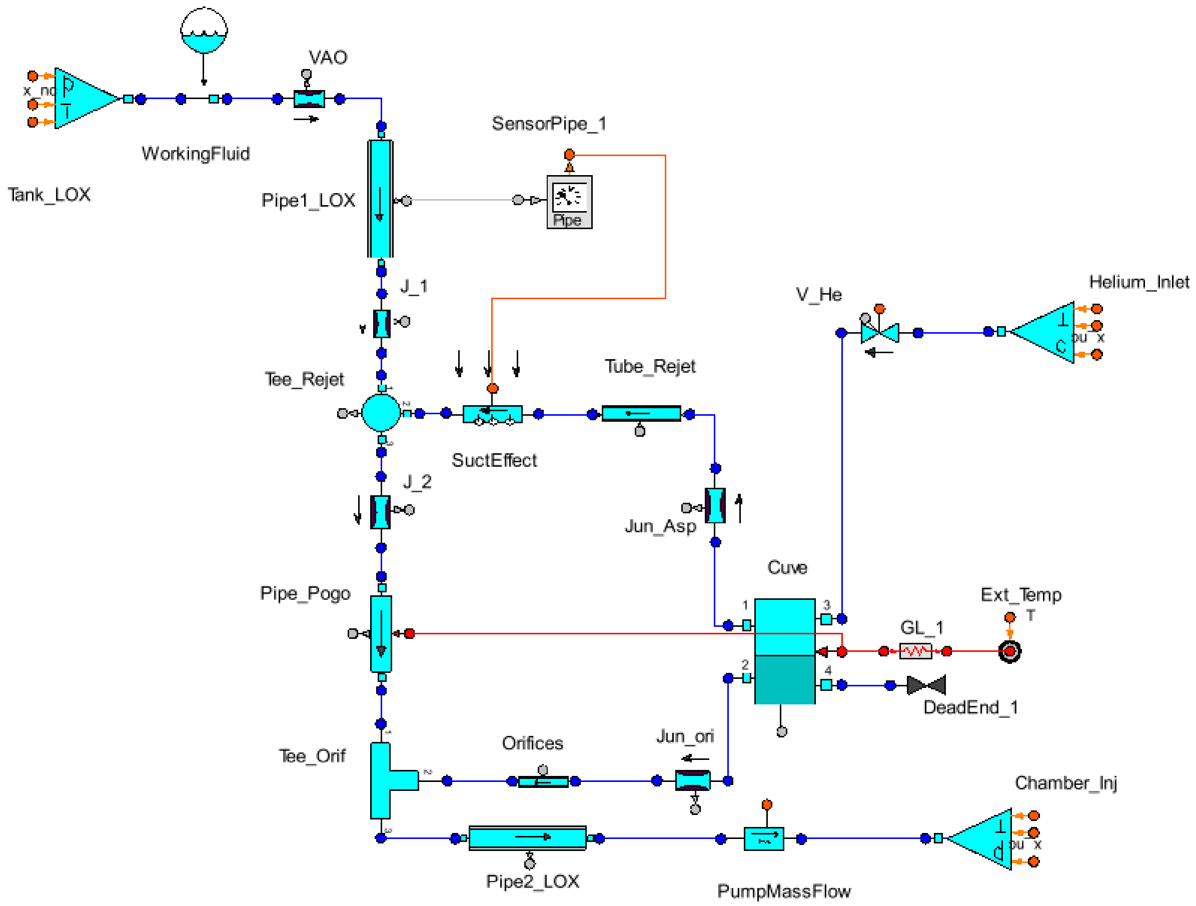
2.2.2 Pogo Model (T-TNK-001)

Library:	ROCKET_EXAMPLES
Model Name:	PogoSystem
Partition Name:	default
Experiment Name:	exp1

2.2.2.1 Model description

A model showing a device to smooth the POGO oscillations in a cryogenic feeding line is presented in this example.

The following physical phenomena will be retained: overflowing a tank outlet (reject tube), damping of the pressure oscillations with non-condensable bubbles in a liquid line, pressure rise due the hydraulic height, suction and plugging effects in a tube, etc.



The system consists of an accumulator tank (Cuve) working under two-phase conditions (liquid and vapor Oxygen) with Helium. The homogeneous equilibrium model (HEM) is applied

The tank is pressurized with Helium to maintain a gas cavity in the accumulator. The excess of helium is rejected to the main liquid line through the "Tube_Rejet" pipe. Another communication between the liquid in the lower part of the tank and the main line is made through holes (Orifices component with inertia) giving a new characteristic response time to the system

All the orifices connecting the main line with the accumulator have been modeled by only one pipe with the same orifice length and an equivalent hydraulic diameter

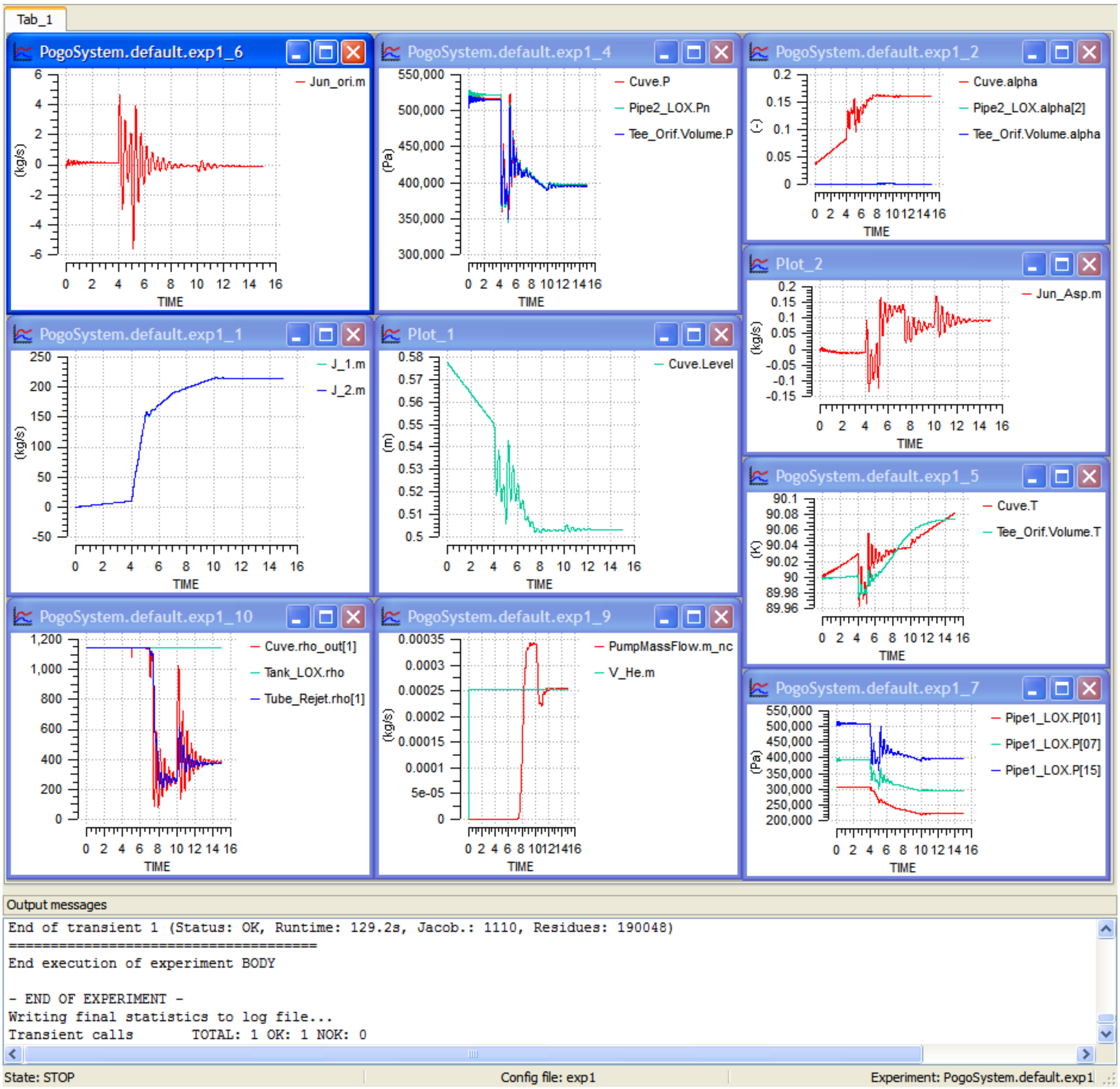
The main line mass flow is imposed by a Jun_TMD component ("PumpMassFlow") simulating the Pump.

The boundaries, valve activation orders and some input geometrical data are given in the experiment file.

2.2.2.2 Results

Without an accumulator system, the pressure oscillations in the main line forced by the start up of the pump would be very sharp. Following plots show how big these oscillations are reduced. Similarly, the frequency of the oscillation is also reduced. Here below the main plots obtained:

- Depending on the ΔP calculated by the "SuctEffect" and "Orifices" components and on the Helium mass flow (V_He valve), different stationary liquid levels can be foreseen in the accumulator.
- For low rates of Helium, the equilibrium is in a semi-plugged situation, so that the tube flow regime is two-phase with Helium.
- Trade-offs performed over some geometrical dimensions (orifices, tube and accumulator) shows a very non-linear behavior of the system.
- Line pressures plot shows the Δz of the line (20 meters).

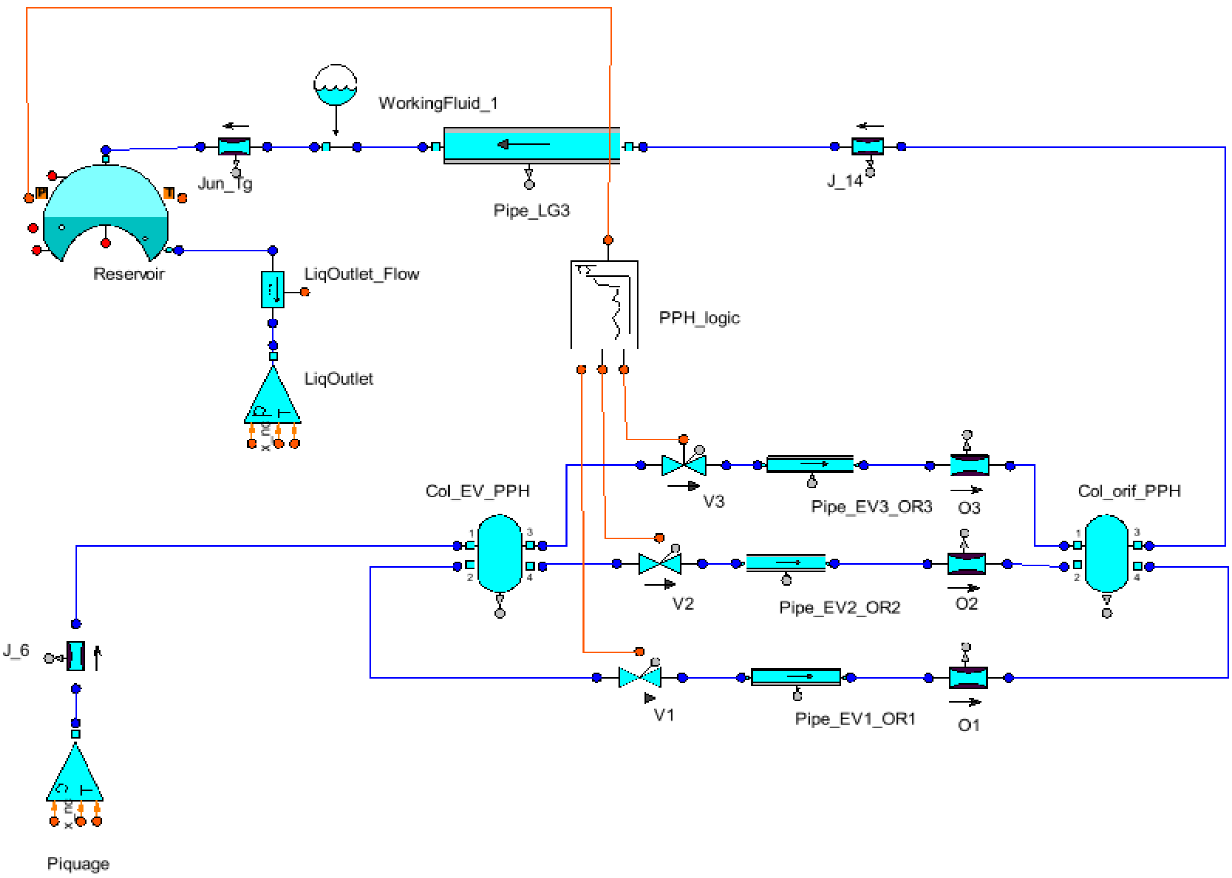


2.2.3 Pressurization System (T-TNK-002)

Library:	ROCKET_EXAMPLES
Model Name:	PressureCircuit_LH2
Partition Name:	default
Experiment Name:	exp1, exp1_boil

2.2.3.1 Model description

A model showing a pressurization system by means of electro-valves is presented below. The Tank is simulated with the more complex component capturing vaporization, fluid temperature distribution, etc.



The pressurization system consists of a set of three lath valves. One of them is normally open. The second will open when the Tank pressure goes down a predefined (but variable in time, see PPH_logic component) threshold, and will close when the Tank pressure goes up another predefined threshold. The third one is normally closed.

The tank is pressurized with Hydrogen. The initial condition in the Tank is: 95% full of LH2 with the ullage volume full of Helium at 80% in mass. By simplicity, in this example all the thermal ports of the tank are supposed adiabatic (no heat flux).

The boundaries, valve activation orders and some input geometrical data are given in the experiment file.

2.2.3.2 Results

Here below the main plots obtained. We remark the following:

- The tank pressure evolves (in accordance with the latch valve activations) between the predefined thresholds.

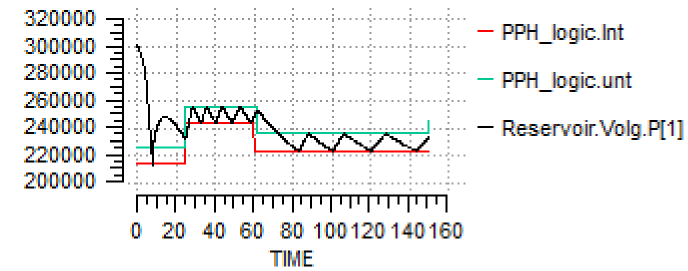
- The vaporization level is very important in order to know the number of valve activations.
- The liquid volume discretization can have non-negligible influence on the model results: Indeed, the local wall temperature seen by a liquid volume will depend on its size, the wall temperature being constant inside each wall slab. The consequence is that the evaporation of the liquid volume can be affected by the discretization.

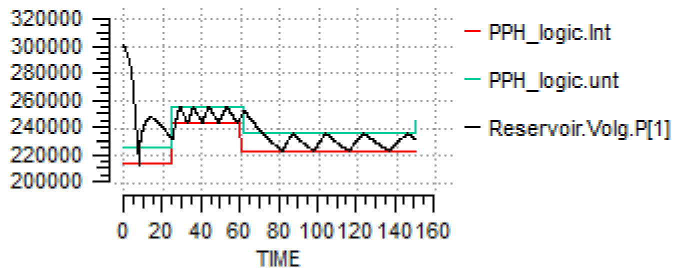
  While pressurizing a tank with an *inflow jet*, vaporization results can be better with an only gas volume in which a complete non-condensable gas / vapor mixture is calculated. The evaporation rate is highly dependent on the vapor partial pressure that can be overestimated with a 1D model at the gas volume near to the interface.
- The "heatExch" option = Diffusion seems to be not adapted to tank conditions where the vapour mass fraction in the ullage part of the Tank is important, giving soon a negative value for the evaporation rate (injected GH2 condenses) once the non-condensable mass fraction of helium (x_nc) goes down of 0.6. Nevertheless, with "heatExch" option = EvapPool, the evaporation rate is positive for the same conditions, what seems more logic (negative values would only happen for lower values of x_nc or temperature).

A new simulation experiment is presented below with the same model testing the film boiling formulation. To do that, an important heating of the tank vertical walls is procured (10KW per wall slab, in total 25o KW). The bubble production and the corresponding pressurisation are showed in next plots:

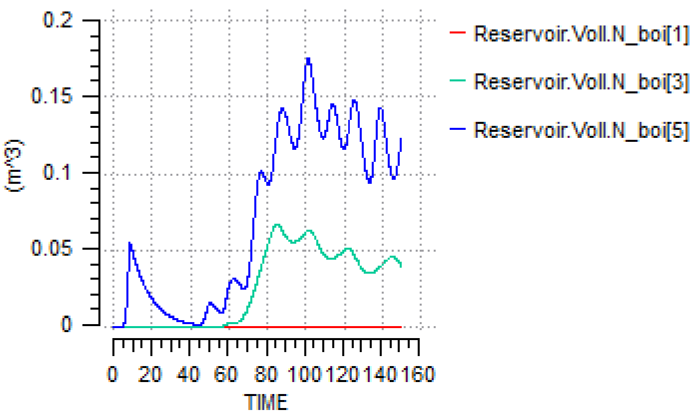
Film boiling Option activated **Film boiling Option deactivated**

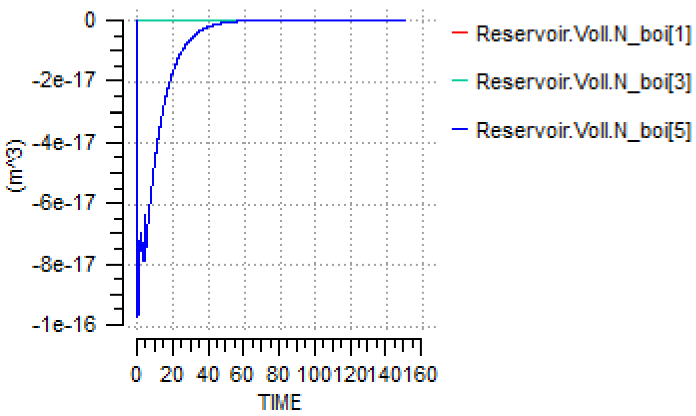
Tank pressurization



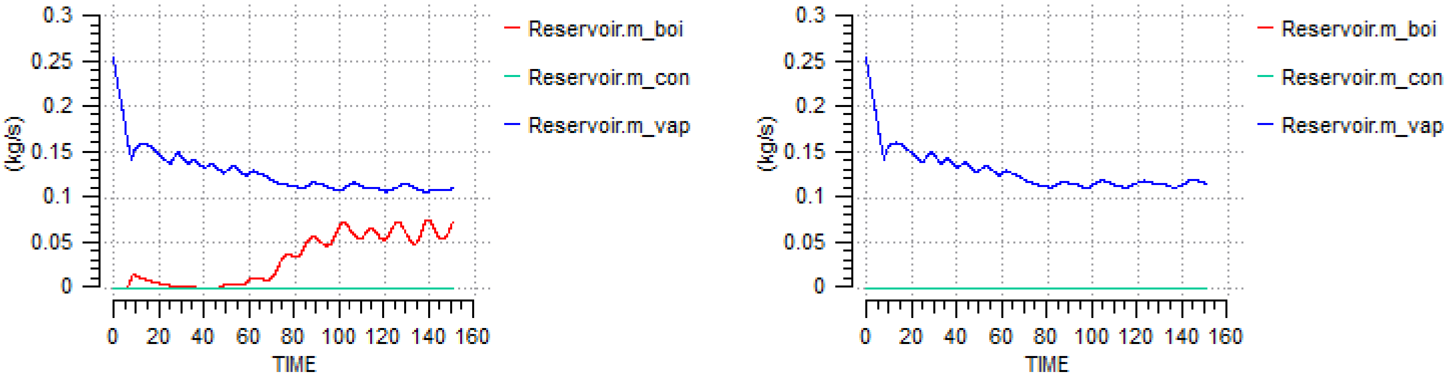


Bubbles volume

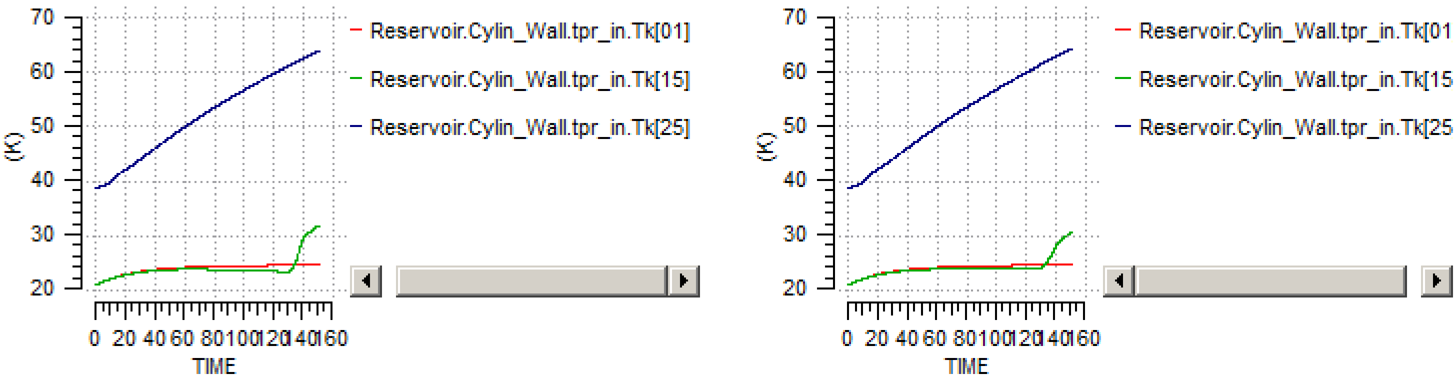




Boiling mass flow



Wall temperatures



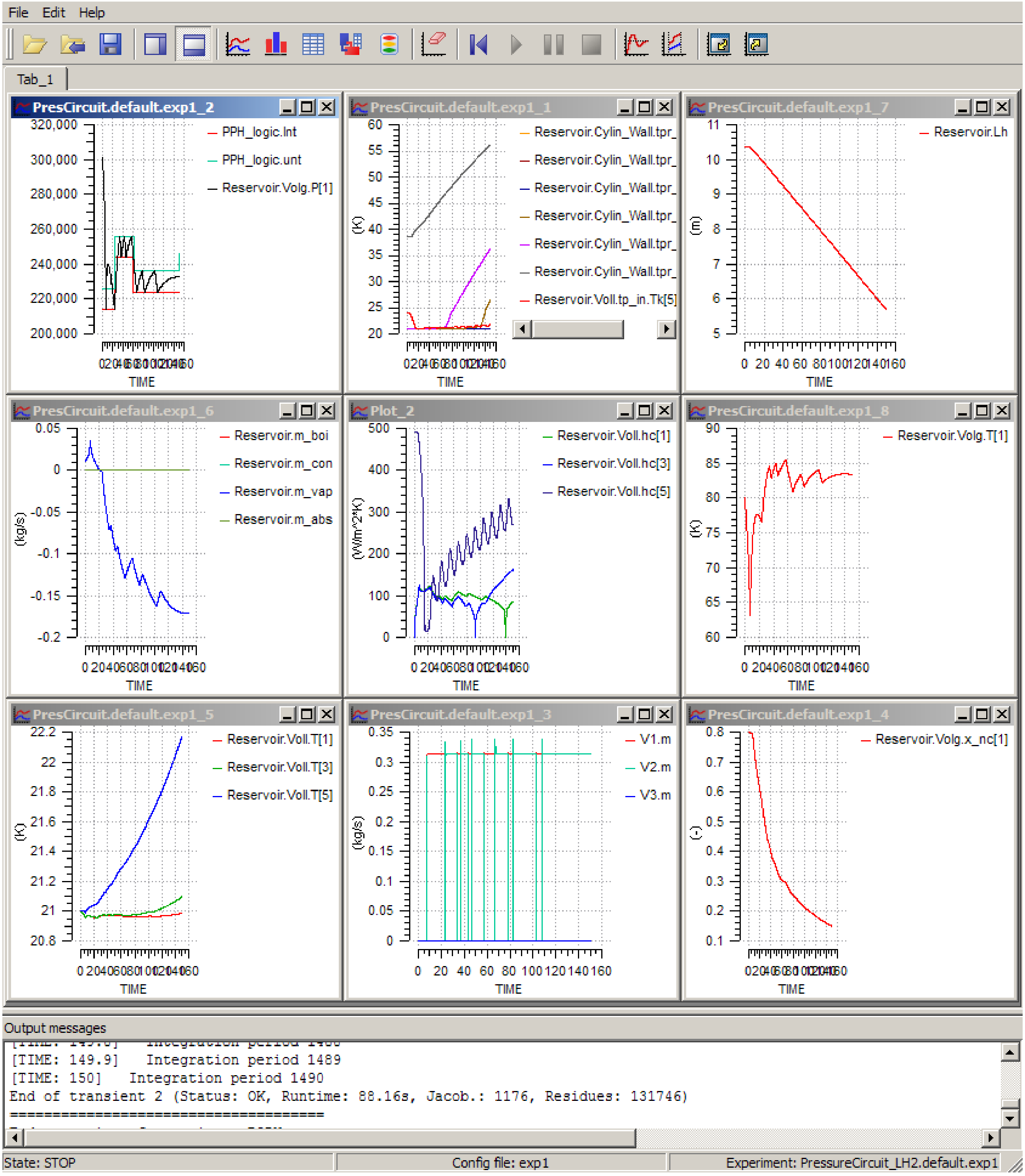
Main conclusion is:

- The boiling production in this case is only due to film boiling at tank walls, the liquid being in sub cooling conditions. Its value is about 60 gr/s (depending on the tank pressure), and increasing because of the continuous wall heating.
- The film boiling effect on the number of valve activations to maintain pressure in the tank is not negligible: from 80 to 150 seconds we have 3 valve activations instead of 4 with no film boiling effects
- The bubbles volume is about 0.1 m3, depending on the level, being small compared with the total liquid volume (more than 150 m3).
- As regard of the wall temperature evolution, it can be seen that the wall temperature increases faster at levels over the liquid side. Under the liquid side, wall temperatures are close to the saturation temperature, see figures.
- The boiling production (m_boi*latent heat) can be compared with the total heating:

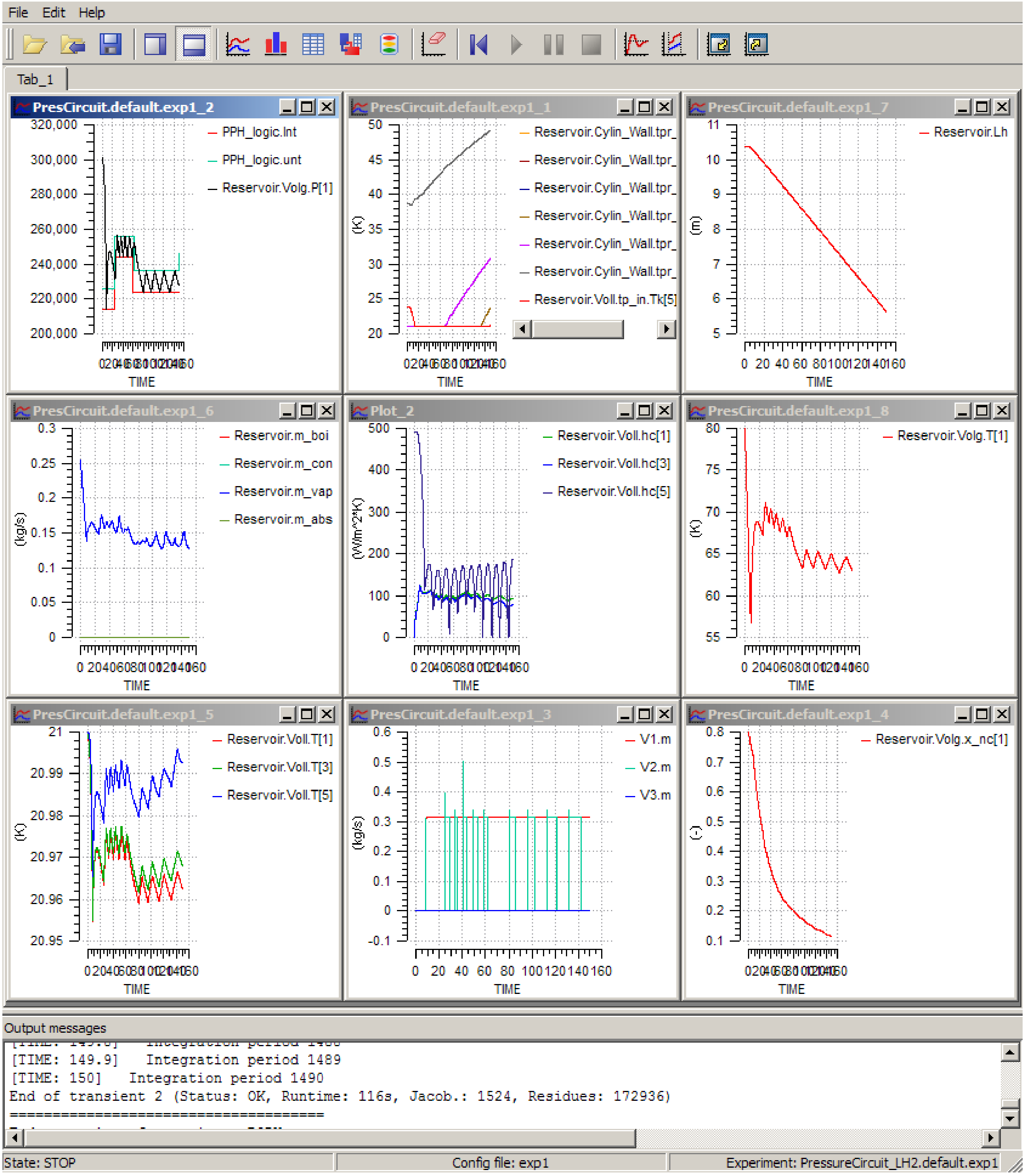
  m_boi*latent heat = 0,06*(849709- 31973) = 50 KW

  this quantity remains of course below the total heating (250 KW) because of the power employed in heating the wall and the liquid itself, and because no steady conditions have been reached yet

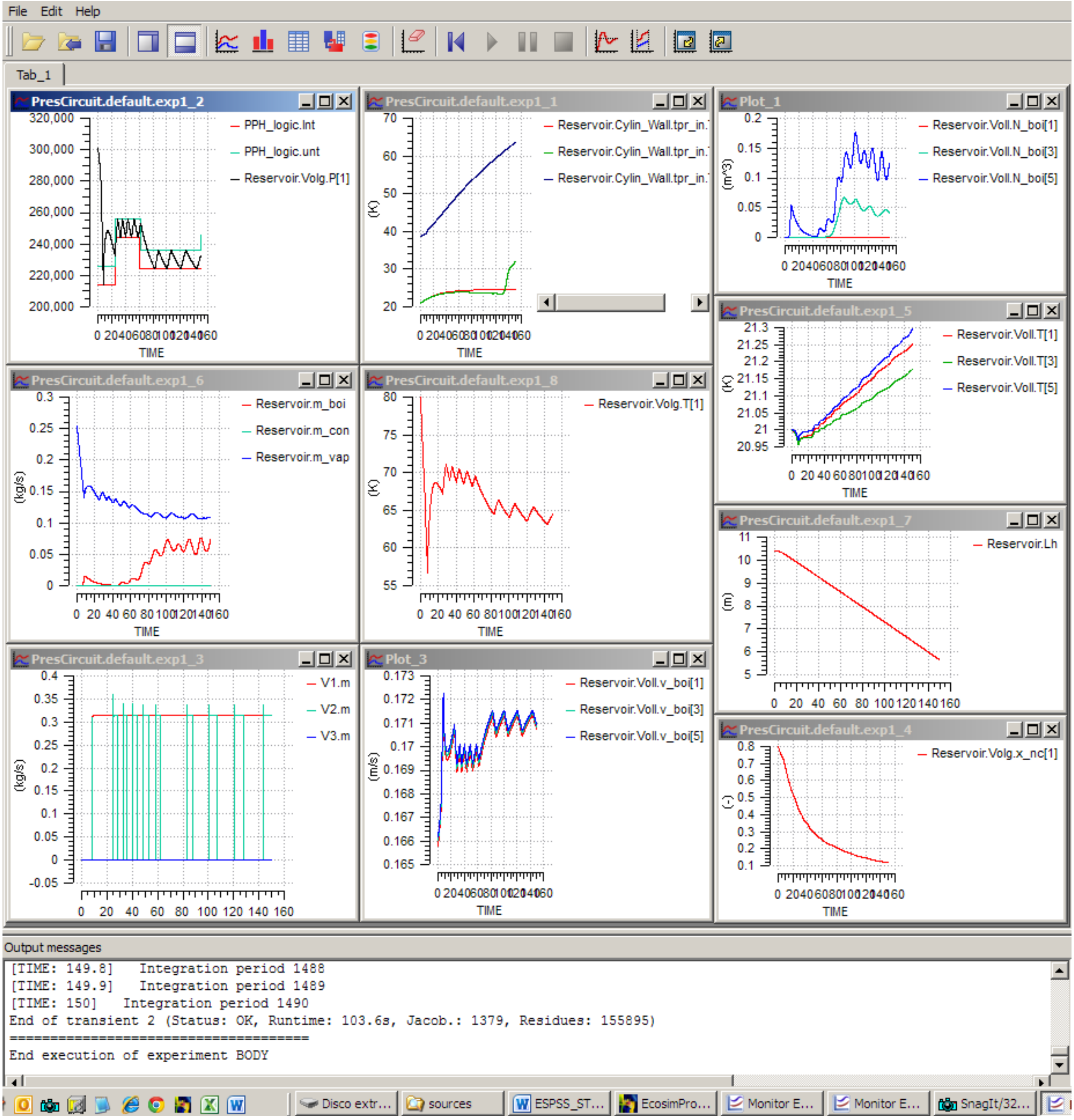
Modified orifice diameters (multiplied by 1.28). "heatExch" option = Diffusion:



Modified orifice diameters (multiplied by 1.28). "heatExch" option = EvapPool:



Modified orifice diameters (multiplied by 1.28). "heatExch" option = EvapPool. Film boiling Option + 250 KW applied:



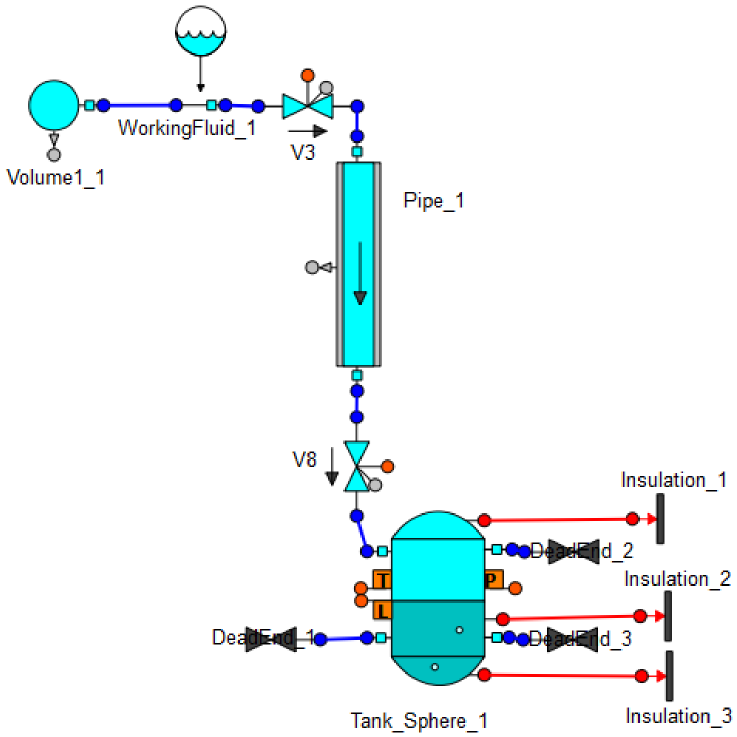
2.2.4 Absorption/Desorption validation (T-TNK-004)

Library:	ROCKET_EXAMPLES
Model Name:	TestcinetiquedissolutionCYL
Partition Name:	default
Experiment Name:	exp1

2.2.4.1 Model description

The test case simulates an almost-cylindrical tank where the two domes have a radius of $1m$ each with a cylindrical pipe of $0.1479m$ of radius, and a pipe $0.2959m$ long. The liquid in the tank is real NTO while the pressurant gas is Helium considered as a perfect gas (it will never condensate during the simulation). The pipe is straight and simulated with 5 nodes with no adsorption. The tank is thermally insulated simplifying the temperature behavior inside the component. In fact, applying a temperature variation will cause the gas to reach different equilibrium states resulting in a different mass of absorbed gas and a different pressure.

The schematic of such a case is reported hereafter.



2.2.4.2 Gas absorbed mass profile Results

The simulation is done for 5 nodes only. The absolute and relative errors are set to $1\text{E}-6$. The objective is to use different diffusion coefficients in the code to compare the reaction of ESPSS with the simple FVM (finite volume method) diffusion equation.

Setting the diffusivity to $2.498E-5\ m^2/s$, we want to compare the non condensable gas absorbed mass profile at the time $3220\ s$. The maximum solubility data changes slightly with time, due to the pressure change, so a value $5.85E-5$ for the dissolved mass fraction is taken from the ESPSS and used inside the FVM as a steady boundary condition. The actual value decays from $6.4072E-5$ to $5.835E-5$.

In the next picture it is possible to see that the error is equivalent to the tolerance imposed to the ESPSS solver. By taking into account all the simplifications this comparison could be considered satisfactory. The results of these two simulations are displayed in the following figure. It is worth note the $y - axis$ has the origin at the bottom of the wall therefore, the right bound of the graph actually represents the gas/liquid interface.

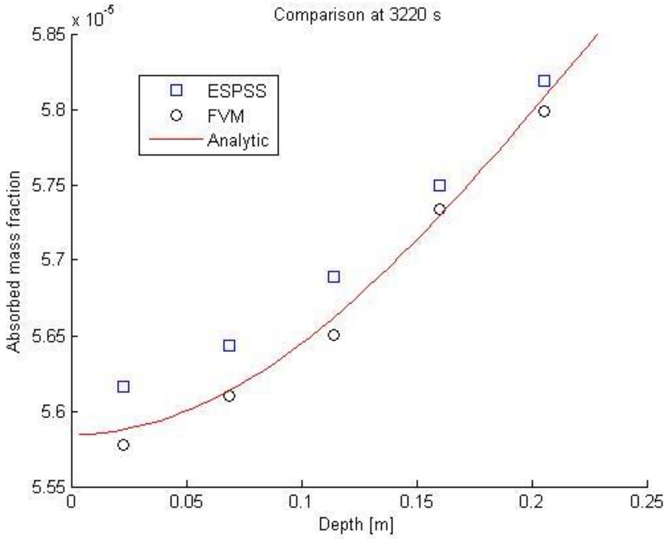


Figure 1: comparison FVM, analytic ESPSS for diffusivity $2.498\mathrm{E}-5\ \mathrm{m}^2/s$

A second case was computed lowering the diffusivity to $2.495E - 06m^2/s$. The maximum mass fraction of non condensable gas taken from the ESPSS simulation in this case is $6.25E - 5$.

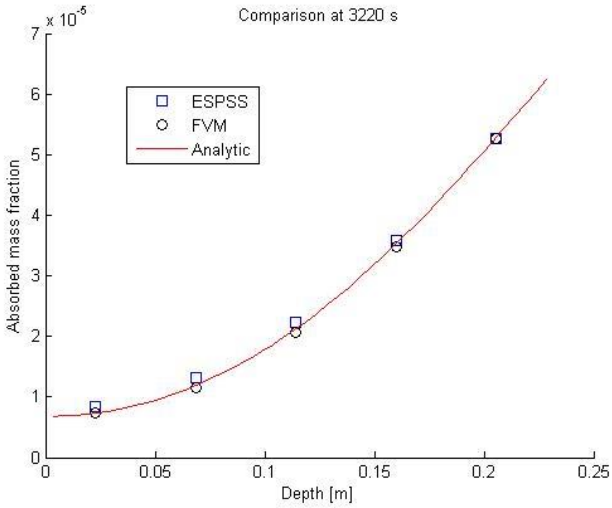


Figure 2: comparison FVM, analytic ESPSS for diffusivity 2.495E-6 m^2/s

Also in this case a difference of the order of $1E - 6$ is retreived. A third simulation is performed lowering the diffusivity to $2.72E - 08m^2/s$ and with the maximum absorbed mass fraction equal to $6.712E - 5$. The comparison is still performed at $3220\ s$.

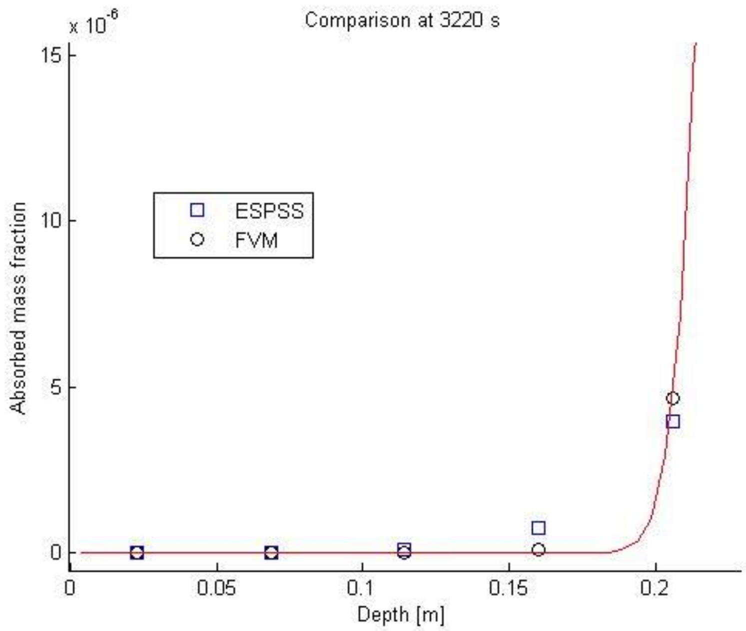


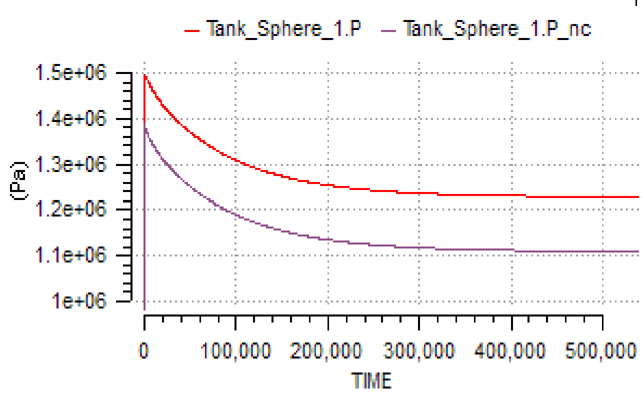
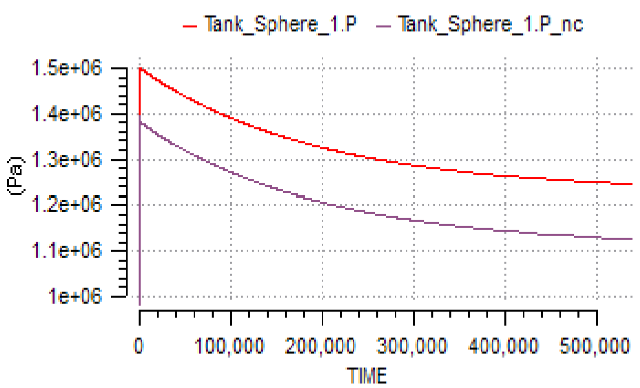
Figure 3: comparison FVM, analytic ESPSS for diffusivity 2.72E-8 m^2/s

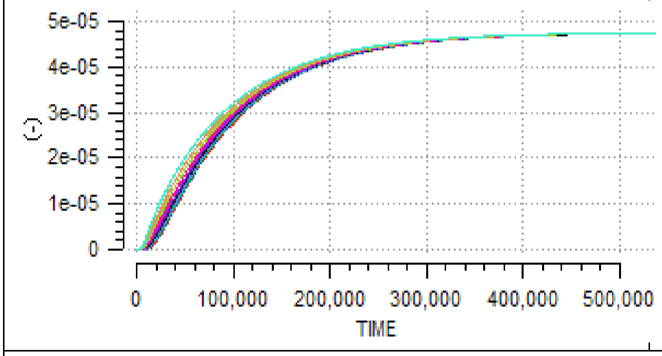
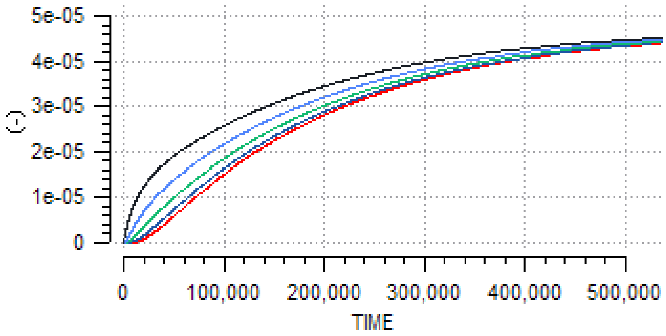
The two solutions for thise case are still comparably close, yelding a maximum error of the order of $1E-6$. Apparently the difference increase slightly with decreasing diffusivity but it is still bounded. It is worth noticing that in the last simulation the two profile intersect around $0.18\,m$, giving quite a different representation of the final gradient.

2.2.4.3 *Influence of the number of nodes of the Tank*

The first thing to report is that, due to the use of many coupled equations, the evaluation of real properties and many other details that make ESPSS a valid support for system engineering, it is rather unpractical to discretize the tank with more than 20 nodes. For this reason, the Tank model results should have no influence of the number of nodes.

To be able to conclude about this, the previous simulation was repeated for different number of nodes in the liquid side of the tank. Next figures compare the time histories of the Tank pressure and the diluted gas mass fraction for 5 and 20 nodes:

20 nodes	**5 nodes**
	
Tank pressure (Pa)	

<table>
<tr><th>20 nodes</th><th>5 nodes</th></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td colspan="2">Diluted gas mass fraction (-)</td></tr>
</table>

The following conclusions can be dropped out:

- The steady value of the tank conditions is few influenced by the number of nodes
- *There is an important influence of the number of nodes in the time needed to stabilize the tank conditions, and according to reference RD-20, this influence is proportional to the log of the number of nodes.*

2.2.4.4 Conclusions

The ESPSS solver has been verified to reproduce correctly the typical behavior of adsorption phenomena when compared to a similar FV solver for a diffusion equation.

The FVM, ESPSS and the analytical results show a reasonable agreement, though the ESPSS solver (virtually the same as the FVM implementation) has a more diffusive behavior.

On the other hand, concerning the time response, there is an important influence of the number of nodes in the time needed to stabilize the tank conditions (pressure and amount of dissolved gas). More studies should be done to asses about this problem.

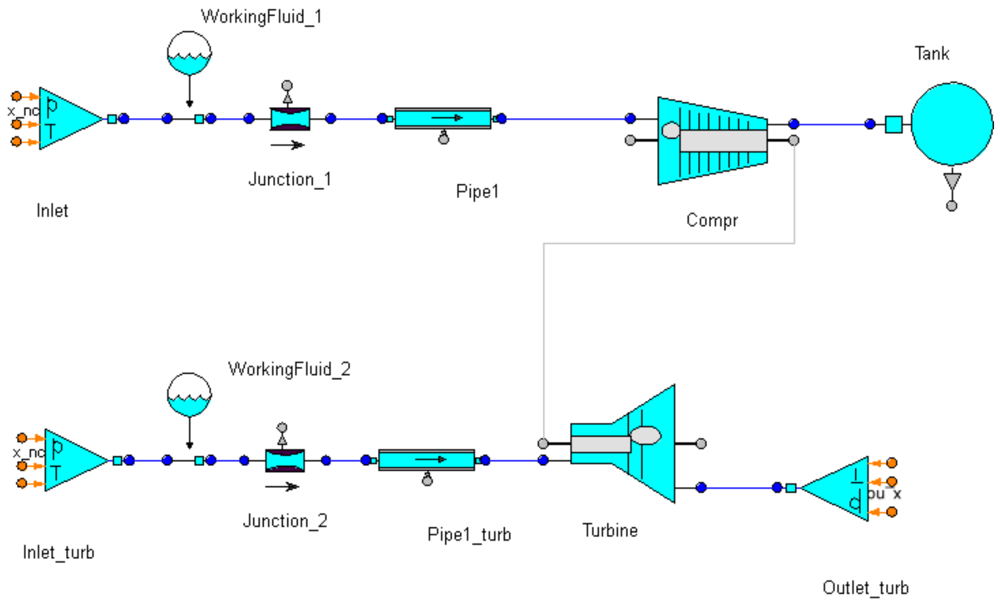
2.3 TURBO_MACHINERY LIBRARY

2.3.1 Turbo-fed System (T-TM-001)

Library:	ROCKET_EXAMPLES
Model Name:	Test_compressor_turbine
Partition Name:	default
Experiment Name:	exp1

2.3.1.1 Model description

This example shows the library capabilities concerning a turbine/compressor coupling. In the model below, the turbine is fed by variable boundary conditions. The compressor feeds a Tank represented by a simple fixed fluid Volume



Turbine / compressor non dimensional performances (characteristic of the maps) are the default ones, i.e., they are arbitrary (do not correspond to any real case) but in principle they are physic.

The turbine inlet pressure increase from 1 bar to a maximum of 3 bar. Then the pressure ratio is maintained constant

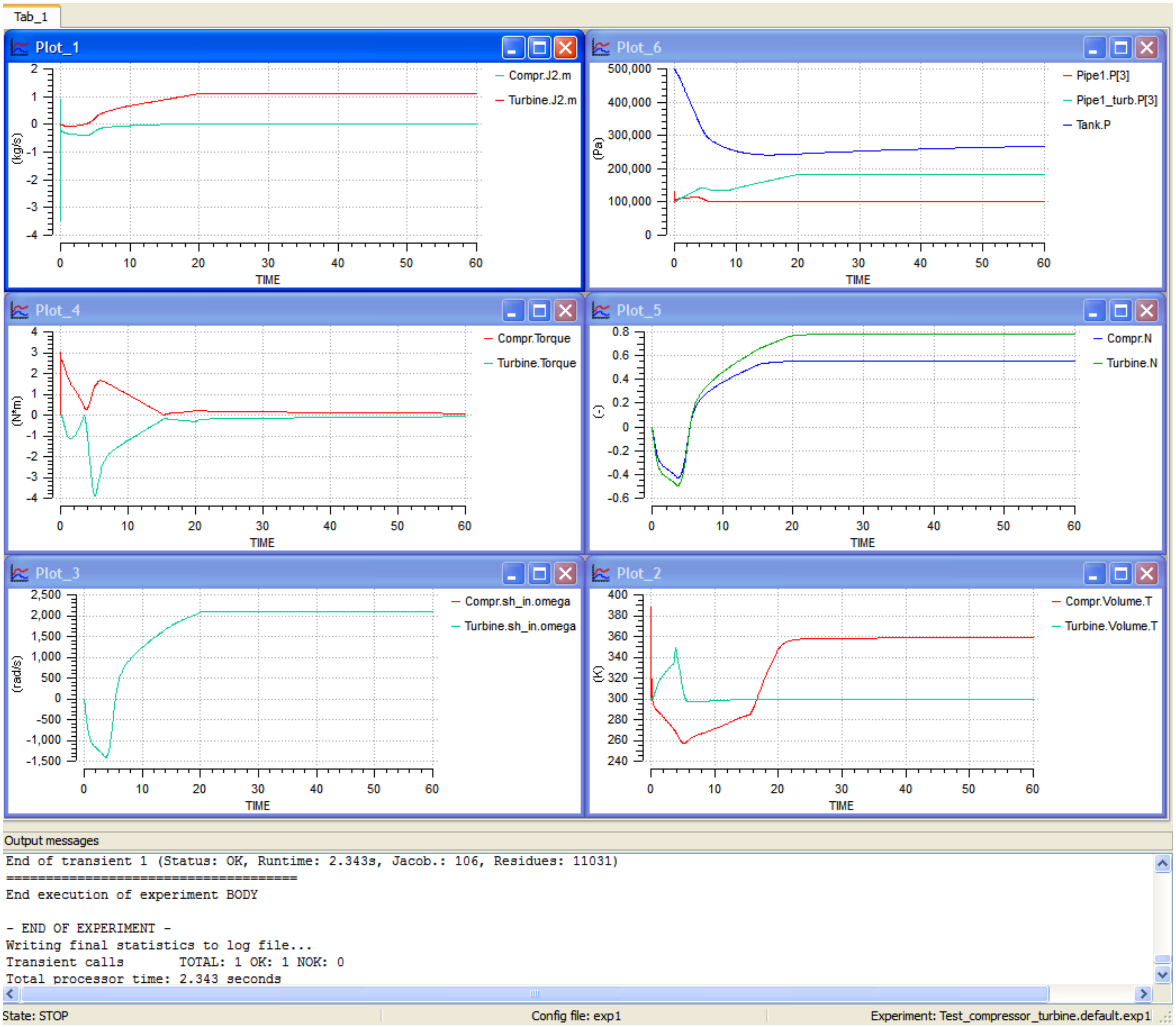
2.3.1.2 Results

Initially, the Tank pressure is intentionally high, and the turbo system is starting from zero axial speed.

Then, the axial speed will be negative till the turbine power and the Tank depressurization are enough to reverse the torque balance.

At the end, an equilibrium position is reached where the Tank inlet flow is null for a determined tank pressure.

Some result plots are given below:


Tab_1
Plot_1
Compr.J2.m
Turbine.J2.m
(kg/s)
TIME
Plot_6
Pipe1.P[3]
Pipe1_turb.P[3]
Tank.P
(Pa)
TIME
Plot_4
Compr.Torque
Turbine.Torque
(N*m)
TIME
Plot_5
Compr.N
Turbine.N
(-)
TIME
Plot_3
Compr.sh_in.omega
Turbine.sh_in.omega
(rad/s)
TIME
Plot_2
Compr.Volume.T
Turbine.Volume.T
(K)
TIME
Output messages
End of transient 1 (Status: OK, Runtime: 2.343s, Jacob.: 106, Residues: 11031)
====================================
End execution of experiment BODY
- END OF EXPERIMENT -
Writing final statistics to log file...
Transient calls TOTAL: 1 OK: 1 NOK: 0
Total processor time: 2.343 seconds
State: STOP
Config file: exp1
Experiment: Test_compressor_turbine.default.exp1

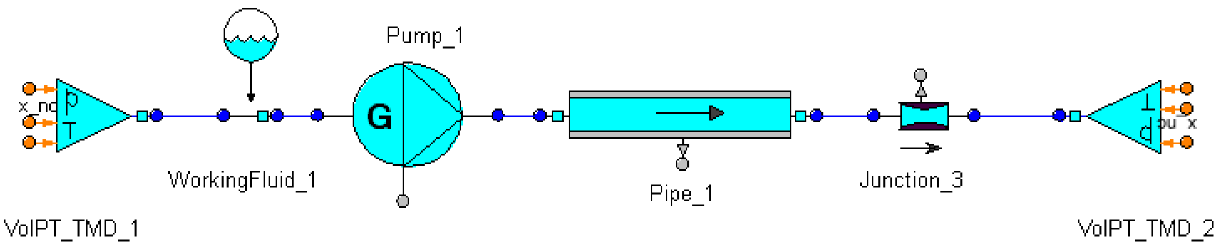
2.3.2 Pump Power failure (T-TM-002)

Library:	ROCKET_EXAMPLES
Model Name:	Test_Pump
Partition Name:	default
Experiment Name:	exp2

2.3.2.1 Model description

This example validates the library capabilities concerning a liquid pump power failure.

The model represents a Pump which ports are connected to two constant volumes through pipes and junctions. The pipe conducts water from SL to a height of 60 meters.



The pipe data are

- Internal diameter = 750 mm. Length = 1000 m. Δz = 60m
- Roughness = 0.05 mm.

The pump data are:

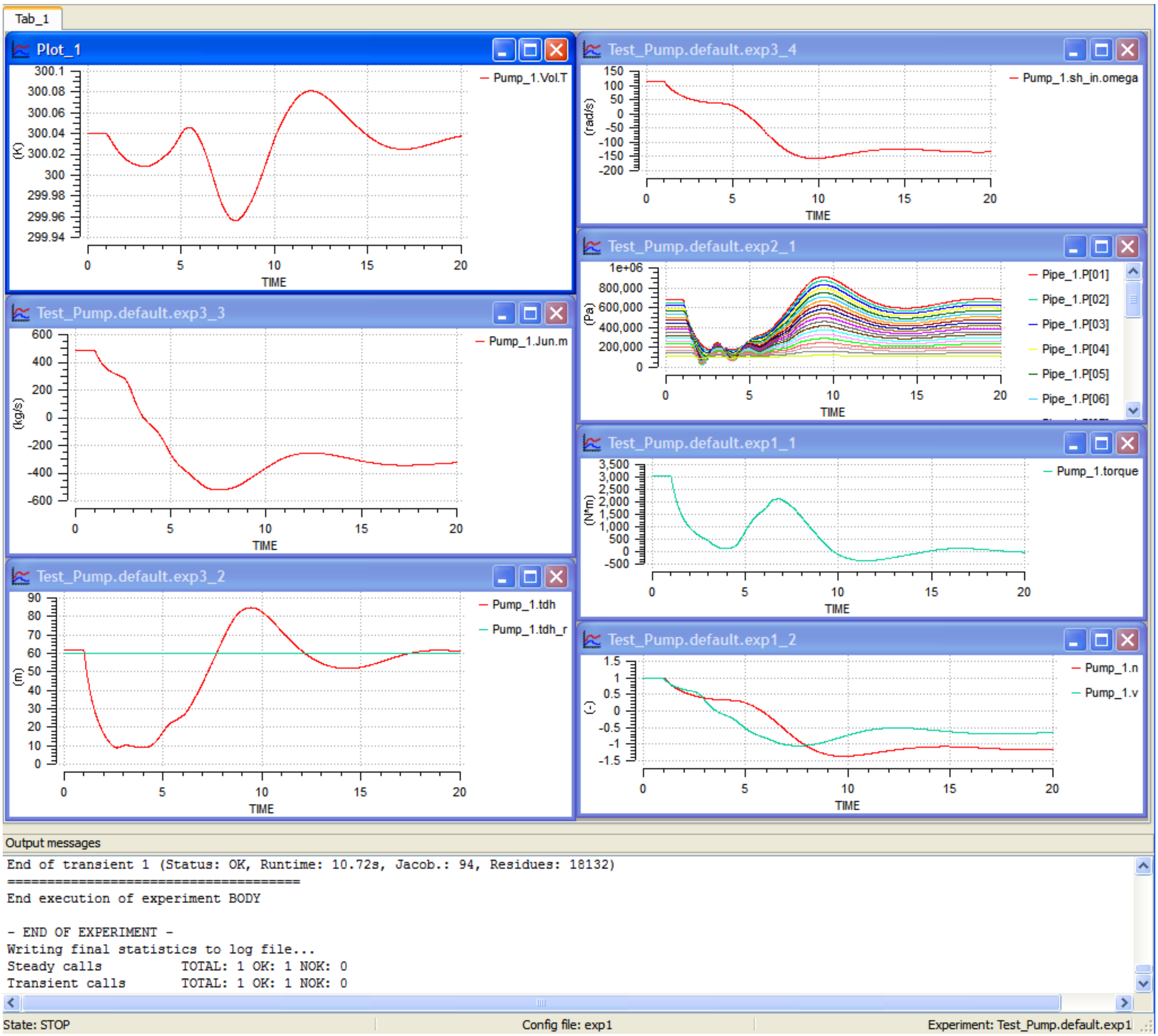
- Pump inertia = 33.7 Kg·m2.
- Total dynamic head at rated conditions = 60 m.
- Volume flow at rated conditions = 0.5 m3/s.
- Pump speed at rated conditions = 1100 rpm.
- Efficiency at rated conditions = 0.84.
- Specific speed = 25.

2.3.2.2 Results

First, a steady state will be calculated via the STEADY() command at the pump nominal flow under the imposed ΔZ, bound pressures and nominal pump speed. The external torque is obtained imposing the nominal pump speed.

Then, the transient to be calculated consist of setting the external torque to zero at TIME = 1 s. The model will simulate how the reverse flow and the pump speed behave.

The results show a very similar behavior as in the Chaudhry reference [RD-17]. Some result plots are given below:


Tab_1
Plot_1
Pump_1.Vol.T
(K)
TIME
Test_Pump.default.exp3_4
Pump_1.sh_in.omega
(rad/s)
TIME
Test_Pump.default.exp2_1
Pipe_1.P[01]
Pipe_1.P[02]
Pipe_1.P[03]
Pipe_1.P[04]
Pipe_1.P[05]
Pipe_1.P[06]
(Pa)
TIME
Test_Pump.default.exp3_3
Pump_1.Jun.m
(kg/s)
TIME
Test_Pump.default.exp1_1
Pump_1.torque
(N*m)
TIME
Test_Pump.default.exp3_2
Pump_1.tdh
Pump_1.tdh_r
(m)
TIME
Test_Pump.default.exp1_2
Pump_1.n
Pump_1.v
(-)
TIME
Output messages
End of transient 1 (Status: OK, Runtime: 10.72s, Jacob.: 94, Residues: 18132)
=====================================
End execution of experiment BODY
- END OF EXPERIMENT -
Writing final statistics to log file...
Steady calls TOTAL: 1 OK: 1 NOK: 0
Transient calls TOTAL: 1 OK: 1 NOK: 0
State: STOP
Config file: exp1
Experiment: Test_Pump.default.exp1

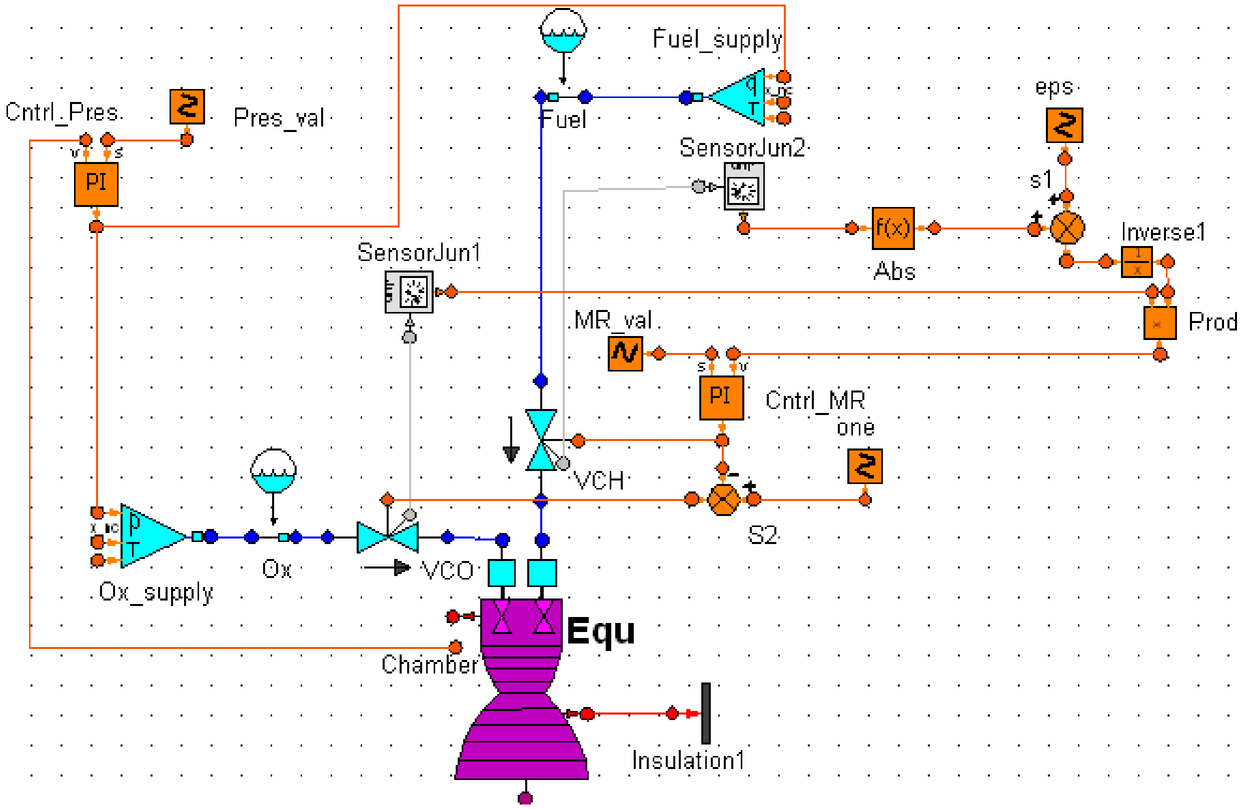
2.4 COMB_CHAMBERS LIBRARY

2.4.1 Combustor test-bench example (T-CCN-004)

Library:	ROCKET_EXAMPLES
Model Name:	Test_ChamberNozzle_eq
Partition Name:	default
Experiment Name:	exp_CH4_LOX, exp_LH2_LOX, exp_MMH_NTO

2.4.1.1 Model description

We present below a model simulating a combustor regulated in pressure and MR. It could represent a simplified real test bench with PIDs controlling pressure and valves position, so that the value of the chamber pressure and MR can be imposed as set points. The model could serve to tune and prepare a real test of a main thruster from the startup to the stabilized conditions:



It can be seen that there is one Controller for the chamber pressure that is fed back by the current chamber pressure, and another Controller for the MR taking measure of the valves mass flows. The output of the first controller will regulate the feeding pressure of both injectors, while the second one will regulate the valve positions (one the complement to one of the other).

The value of the Chamber pressure set point is constant, equal to the "p_cha" parameter. The value of the MR set point is a table of time, so that we dispose of two seconds to stabilize the chamber to MR=0.2 from the beginning of the simulation. Then we study the chamber behavior evaluating the MR from 0.2 to 10 evolving in quasi-steady state:

Time	0	2	12	100
MR	0.2	0.2	10	10

Most of the input data are default values, with the exception of some geometrical/control values and other modified in the experiment file. The experiment file is presented below. We remark that the experiment redefine the chamber initial conditions so as to have initially an ignited chamber; no limitation on the mixture ratio. Other important data of the model is defined in the experiment file to control different model cases and initialization options:

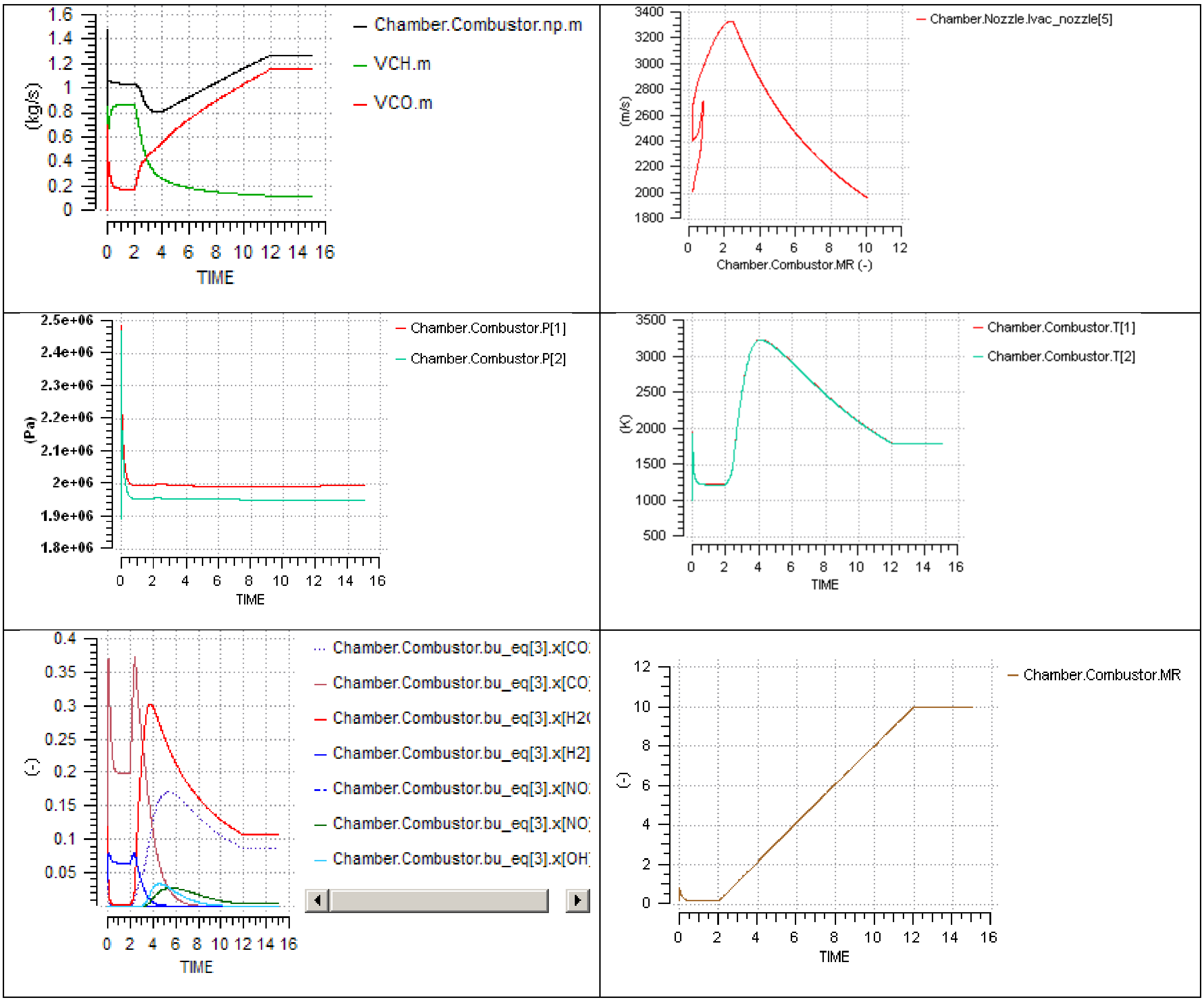
```
EXPERIMENT exp_CH4_LOX ON Test_ChamberNozzle_eq.default
DECLS
INIT
-- initial values for state variables
Cntrl_Pres.vi[1] = 60e5 -- initial value of the feeding pressure
Chamber.Nozzle.np_in.m = 0
Chamber.Nozzle.np_in.rho = 0
Cntrl_MR.vi[1] = 0
BOUNDS
-- Set equations for boundaries: boundVar = f(TIME;...)
Chamber.Combustor.IgnitFlag = 1-- step(TIME,0.5)
Chamber.Combustor.starter_T = 1000
Chamber.Combustor.starter_m = 0
Chamber.np_out.P = p_ext
Chamber.tp_inj.q[1] = 0
FLUID_FLOW_1D.Damp = 1
FLUID_FLOW_1D.GRAV = 9.806
FLUID_FLOW_1D.Re_lam = 2000
FLUID_PROPERTIES.MinMolarFr = 1e-008
Fuel_supply.s_temp.signal[1] = th0
Fuel_supply.s_xNonCond.signal[1] = 0
Ox_supply.s_temp.signal[1] = to0
Ox_supply.s_xNonCond.signal[1] = 0

BODY
Cntrl_Pres.k[1] = 2  -- Controllers gains
Cntrl_MR.k[1] = -1
Chamber.Dt = 0.03
p_cha = 50e5 -- imposed chamber pressure
p_ext = 100
t_ext = 300
th0 = 110
to0 = 90

Chamber.T_ch = 1000 -- ingnited condit. initialisation
Chamber.P_ch = p_cha
Chamber.Tcav_red = th0
Chamber.Tcav_oxy = to0
Chamber.AR_sup = 50
Chamber.MR_ini = 0.2
Chamber.MR_min = -1
Chamber.MR_max = 1e20

Fuel.fluid = Real_CH4
Ox.fluid = Real_O2
REL_ERROR = 1e-4
ABS_ERROR = 1e-4
REPORT_MODE = IS_STEP
TIME = 0
TSTOP = 15
CINT = 0.1
INTEG()
END EXPERIMENT
```

For illustrative purposes, the "Test_PreBurner_setup" model of the ROCKET_EXAMPLES library presents a similar case but using a pre-burner component connected to fixed boundary conditions at exit. In this case, a starter of solid propellant gases is used between 0 to 0.6 seconds (see DM_law table in the corresponding experiment); ignition is produced at time 0.5 s and no limitation on the mixture ratio, so we can study a real startup from the ambient pressure and temperature. Same procedure could also be used in the present case ("Test_ChamberNozzle_eq").

2.4.1.2 Results

Here below the main plots obtained:

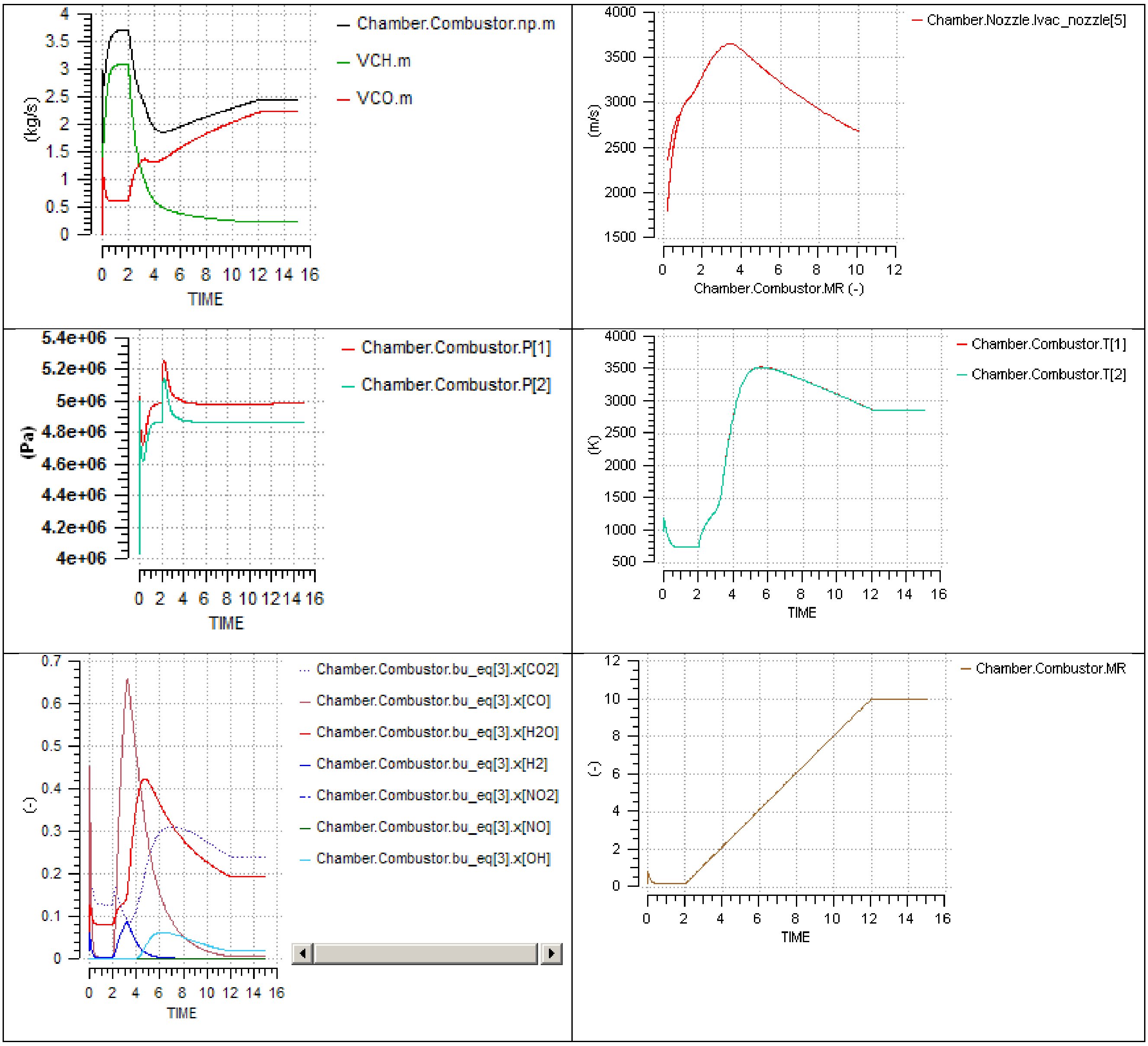
MMH/NTO (REAL FLUIDS)



It can be remarked the following:

- There is an initial transient to accommodate the ignited initial conditions in the chamber to the valve/feeding conditions. This transient produce big pressure variations
- The valve position is calculated by the PI controllers to assure the specified MR at the specified pressure. The specified chamber pressure can be maintained even for the lower MR
- The overshot in chamber pressure due to the change in the MR law is very low.
- The maximum Ivac is for MR of about 2.3

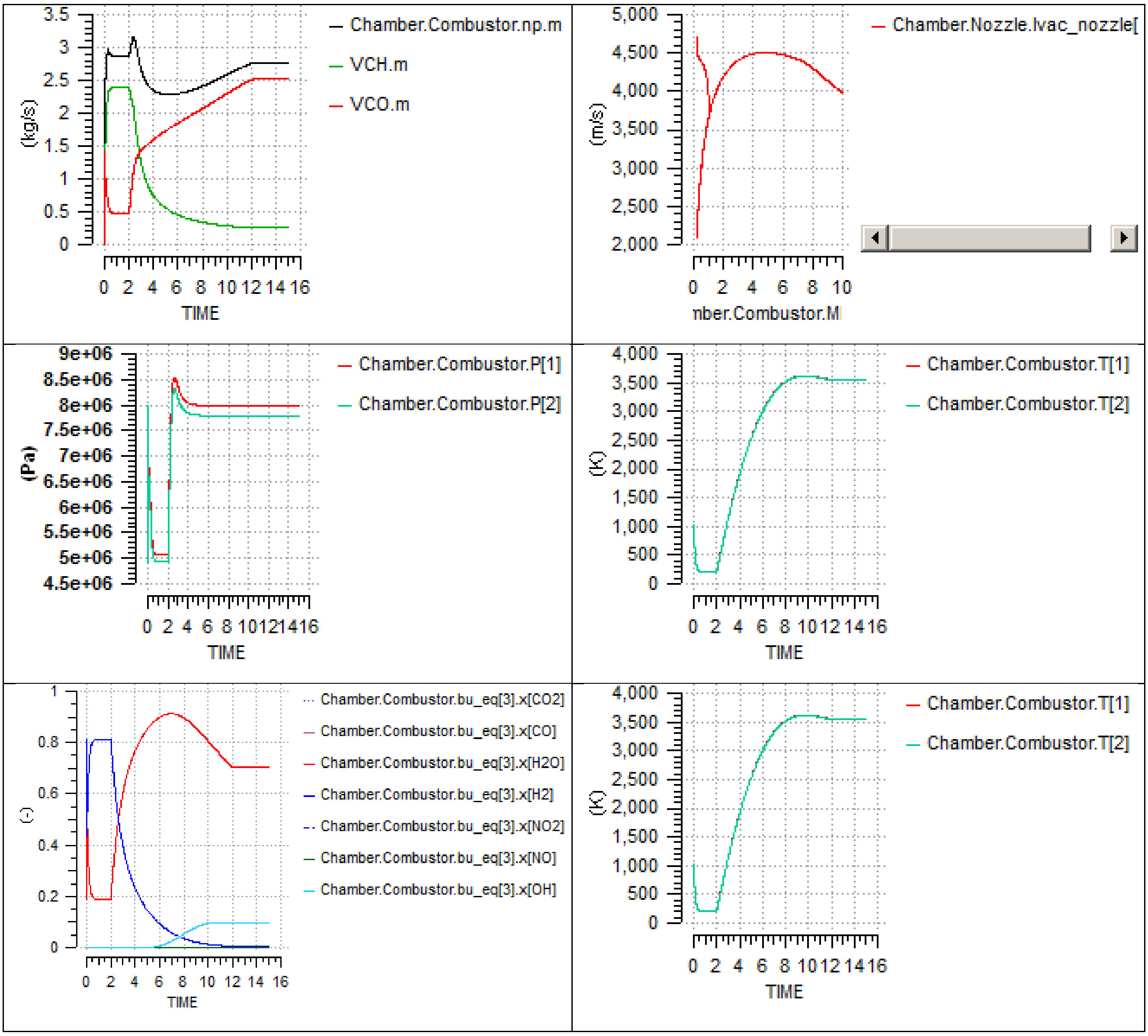
CH4/LOX



It can be remarked the following:

- As before, there is an initial transient to accommodate the ignited initial conditions in the chamber to the valve/feeding conditions. This transient produce big pressure variations
- The valve position is calculated by the PI controllers to assure the specified MR at the specified pressure. The specified chamber pressure can be maintained even for the lower MR
- The overshot in chamber pressure due to the change in the MR law is about 2 bar.
- The maximum Ivac is for MR of about 3

LH2/LOX



It can be remarked the following:

- As before, there is an initial transient to accommodate the ignited initial conditions in the chamber to the valve/feeding conditions. This transient produce big pressure variations
- The valve position is calculated by the PI controllers to assure the specified MR at the specified pressure. In this case (Pch=80 bar), the specified chamber pressure cannot be maintained for the lower MR
- The overshot in chamber pressure due to the change in the MR law is about 5 bar.
- The maximum Ivac is for MR of about 5

Tables below validate the main chamber properties and composition (molar fractions) compared with CEA code at a particular MR (one particular point in previous figures):

Chamber cond.(MMH/NTO): MR=2.1454.
Oxidizer Injector Cond.: P=2.01449e+006 T=294.039. Reduzer Injector Cond.: P=2.00089e+006 T=295.385

	P (Pa)	T (K)	MW (gr/mol)	Mach (-)	Thrust (N)	Ivac (m/s)	CF (-)
Subsonic combustor conditions (Asub/Ath=4)							
ESPSS(*)	1.95E+06	3334.26	22.2343	0.14638			
ESPSS	1.95E+06	3219.75	22.6554	0.146717			
CEA	1.95 E+06	3252.48	22.518	0.151			
Throat conditions							
ESPSS(*)	1.15E+06	3171.71	22.5208	1			
ESPSS	1.15E+06	3056.54	22.9192	1			
CEA	1.141 E+06	3089.78	22.791	1			
Supersonic nozzle conditions (Asub/Ath=50)							
ESPSS(*)	3366.51	1411.02	23.8486	4.23196	2750.76	3449.88	1.89348
ESPSS	3156.06	1241.44	23.8484	4.3437	2720.86	3320	1.87624
CEA	3206	1294.82	23.855	4.305		3361.3	1.8663

	y(CO) (-)	y(CO2) (-)	y(H2O) (-)	y(H2) (-))	y(NO2) (-)	y(NO) (-)	y(OH) (-)
Subsonic combustor conditions (Asub/Ath=4)							
ESPSS(*)	0.0976423	0.055835	0.348264	0.075943	7.04E-06	0.0120714	0.0475461
ESPSS	0.0918822	0.0645049	0.373704	0.0680463	5.09E-06	0.00942375	0.0372816
CEA	0.09381	0.06160	0.36452	0.07044	0.00001	0.01007	0.04303
Throat conditions							
ESPSS(*)	0.0931382	0.0623415	0.36573	0.0706334	4.33E-06	0.00951618	0.0396969
ESPSS	0.0868677	0.0713653	0.389767	0.0631264	2.87E-06	0.00703237	0.0296873
CEA	0.08890	0.06839	0.38117	0.06536	0.00000	0.00768	0.03502
Supersonic nozzle conditions (Asub/Ath=50)							
ESPSS(*)	0.0448432	0.11978	0.422591	0.0713519	9.44E-16	1.91E-08	1.01E-06
ESPSS	0.0371262	0.127408	0.415045	0.0789069	7.89E-16	3.30E-10	3.43E-08
CEA	0.03971	0.12493	0.41774	0.07617	0.00000	0.00000	0.00000

(*): this case corresponds to simplified liquid properties of MMH/NTO (no latent heat considered)

There are small differences between CEA code and ESPSS (< 1%). The reasons can be the followings:

- Different criteria evaluating the real properties of liquids at injection. It can be see that ESPSS solution for perfect liquid (no latent heat considered) is more optimist that CEA, but the solution with real properties at injection gives more conservative results that CEA
- The CF coefficient given by ESPSS seems to have an error of 1% because the evaluation of the total pressure in the corresponding nozzle section does not consider the change in composition. Nevertheless, this error only concerns the calculation of the CF coefficient itself, not the calculation of nozzle variables (thrust, speed, pressure, etc)

Chamber cond.(CH4/O2): MR=3.00064
Oxidizer Injector Cond.: P=5.15573e+006 T=93.3296. Reduzer Injector Cond.: P=5.03539e+006 T=113.366

	P (Pa)	T (K)	MW (gr/mol)	Mach (-)	Thrust (N)	Ivac (m/s)	CF (-)
Subsonic combustor conditions (Asub/Ath=4)							
ESPSS	4.89E+06	3430.23	20.3946	0.14793			
CEA	4.874 E+06	3425.85	20.383	0.151			
Throat conditions							
ESPSS	2.87E+06	3250.56	20.623	1			
CEA	2.848 E+06	3246.58	20.613	1.000			
Supersonic nozzle conditions (Asub/Ath=50)							
ESPSS	8299.38	1382.44	21.386	4.28495	6822.62	3615.14	1.87209
CEA	8262	1385.50	21.393	4.279		3615.9	1.8606

	y(CO) (-)	y(CO2) (-)	y(H2O) (-)	y(H2) (-))	y(NO2) (-)	y(NO) (-)	y(OH) (-)
Subsonic combustor conditions (Asub/Ath=4)							
ESPSS	0.217509	0.100312	0.475452	0.126133	0	0	0.0425586
CEA	0.21760	0.09996	0.47366	0.12627			0.04504
Throat conditions							
ESPSS	0.214182	0.107243	0.491164	0.124529	0	0	0.0331063
CEA	0.21422	0.10693	0.48954	0.12461			0.03526
Supersonic nozzle conditions (Asub/Ath=50)							
ESPSS	0.147429	0.185662	0.483692	0.18297	0	0	2.77E-07
CEA	0.14781	0.18552	0.48405	0.18262			0.00000

There are small differences between CEA code and ESPSS (about 0.1%). The reasons can be the followings:

- Different criteria evaluating the real properties of liquids at injection. ESPSS consider real properties according to REFPROP.
- The CF coefficient given by ESPSS seems to have an error of 0.5% because the evaluation of the total pressure in the corresponding nozzle section does not consider the change in composition. Nevertheless, this error only concerns the calculation of the CF coefficient itself, not the calculation of nozzle variables (thrust, speed, pressure, etc)

Chamber cond.(LH2/LOX): MR=6.07453
Oxidizer Injector Cond.: P=8.34571e+006 T=92.8307. Reduzer Injector Cond.: P=8.13208e+006 T=26.9083

	P	T	MW	Mach	Thrust	Ivac	CF
	(Pa)	(K)	(gr/mol)	(-)	(N)	(m/s)	(-)
Subsonic combustor conditions (Asub/Ath=4)							
ESPSS	7.82E+06	3512.76	13.6005	0.14854			
CEA	7.797 E+06	3504.20	13.597	0.151			
Throat conditions							
ESPSS	4.58E+06	3323.59	13.7457	1			
CEA	4.553E+06	3315.30	13.742	1			
Supersonic nozzle conditions (Asub/Ath=50)							
ESPSS	12994.9	1383.85	14.2591	4.28203	10912.6	4474.47	1.87107
CEA	12924	1382.73	14.261	4.280		4470.6	1.8609

	y(CO)	y(CO2)	y(H2O)	y(H2)	y(NO2)	y(NO)	y(OH)
	(-)	(-)	(-)	(-))	(-)	(-)	(-)
Subsonic combustor conditions (Asub/Ath=4)							
ESPSS	0	0	0.674683	0.240266	0	0	0.0438232
CEA	0	0	0.67303	0.24084	0	0	0.04582
Throat conditions							
ESPSS	0	0	0.69592	0.237107	0	0	0.0339608
CEA	0	0	0.69440	0.23761	0	0	0.03566
Supersonic nozzle conditions (Asub/Ath=50)							
ESPSS	0	0	0.765232	0.234762	0	0	3.17E-07
CEA	0	0	0.76537	0.23462	0	0	0.00000

Same conclusions as before.

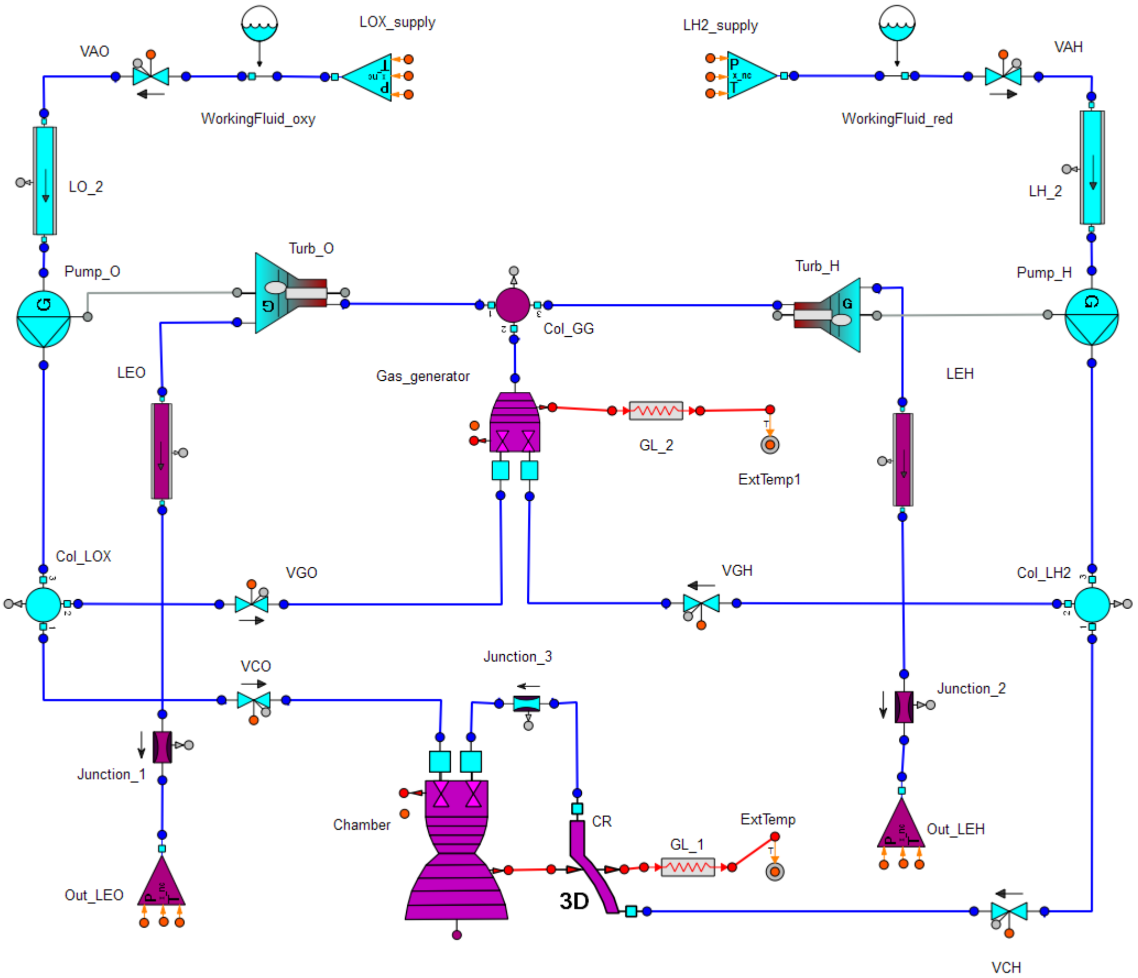
2.4.2 GG engine cycle (T-CCN-001)

Library: ROCKET_EXAMPLES
Model Name: GGEngine
Partition Name: default
Experiment Name: exp1

2.4.2.1 Model description

This model represents a Gas Generator engine cycle type. Input data are fictitious values, the aim this example being just to show the ESPSS Libraries capabilities regarding this type of engine.

In this model the generalized turbine model have been used. This is because no adapted maps are available for this type of turbines



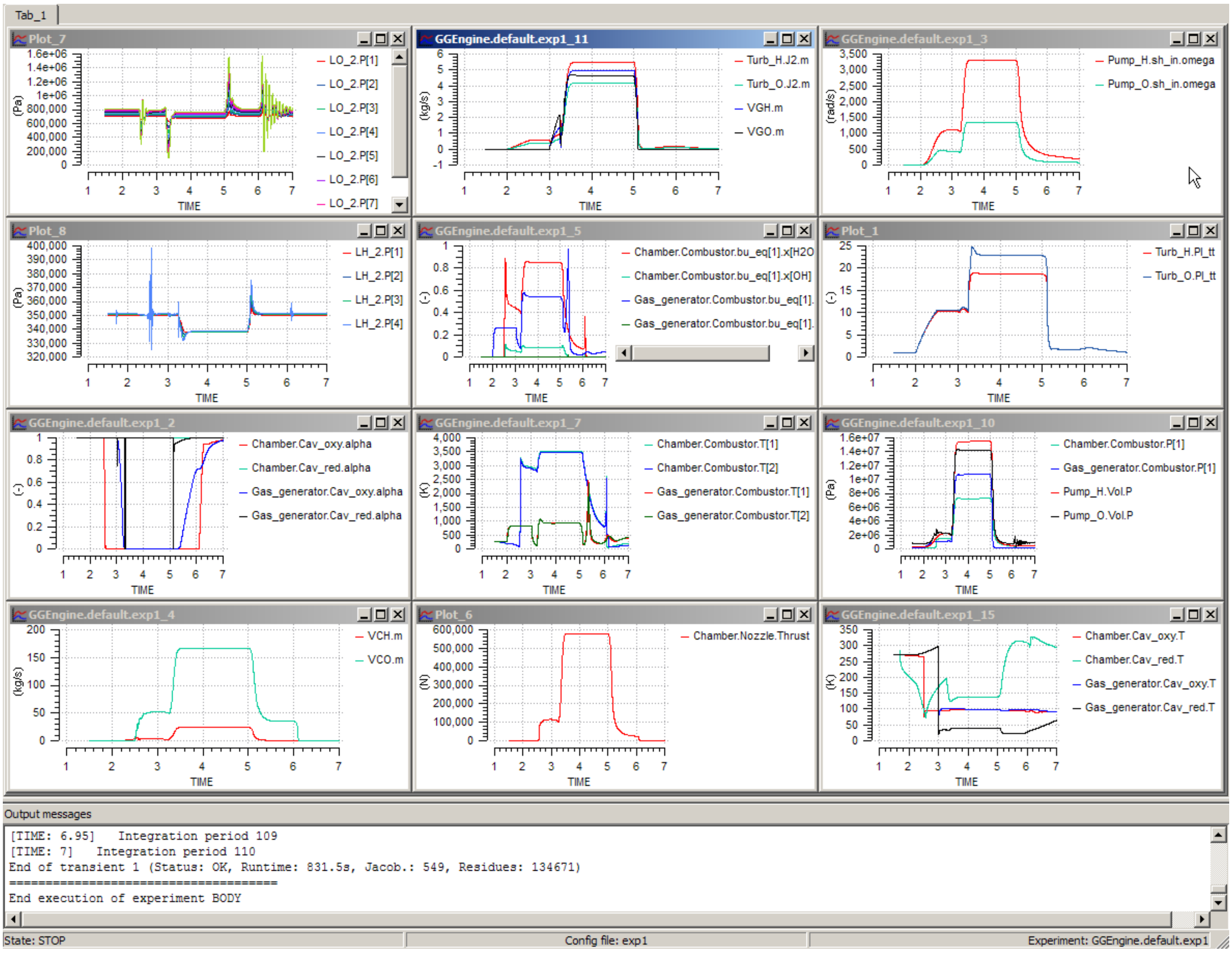
The boundaries, valve activation orders and some input geometrical data are given in the experiment file.

2.4.2.2 Results

Here below the main plots obtained. We remark the strong transients that the start-up of the engine carries out:

- Due to a no optimized input data (Turbo-Pump characteristics, injector and Valve flow areas, etc), the chamber pressure can go very close to the pump pressures during some instants before the engine stabilization. This can result in important chamber/pump pressure oscillations.
- Cavity bubble collapse produces important pressure oscillations.

- The starter gases (see table "`DM_law`" in the experiment file) must be defined correctly to allow a proper stat-up. The mixture process between the injected vapors and the solid propellant gases of the starter (which chemical composition is now an input data) allows a correct ignition process inside the 1D combustors (gas generator and main combustor)
- Temperatures can be very low in the chamber before ignition because the feeding tanks are cryogenic. The simplified model for the liquid vaporization inside the combustor prevents the chamber ignition if the injection's conditions do not have an appropriate mixture ratio, letting the liquid jets leave the chamber without vaporization



ESPSS 3.0 allows the simulation of the engine shutdown:

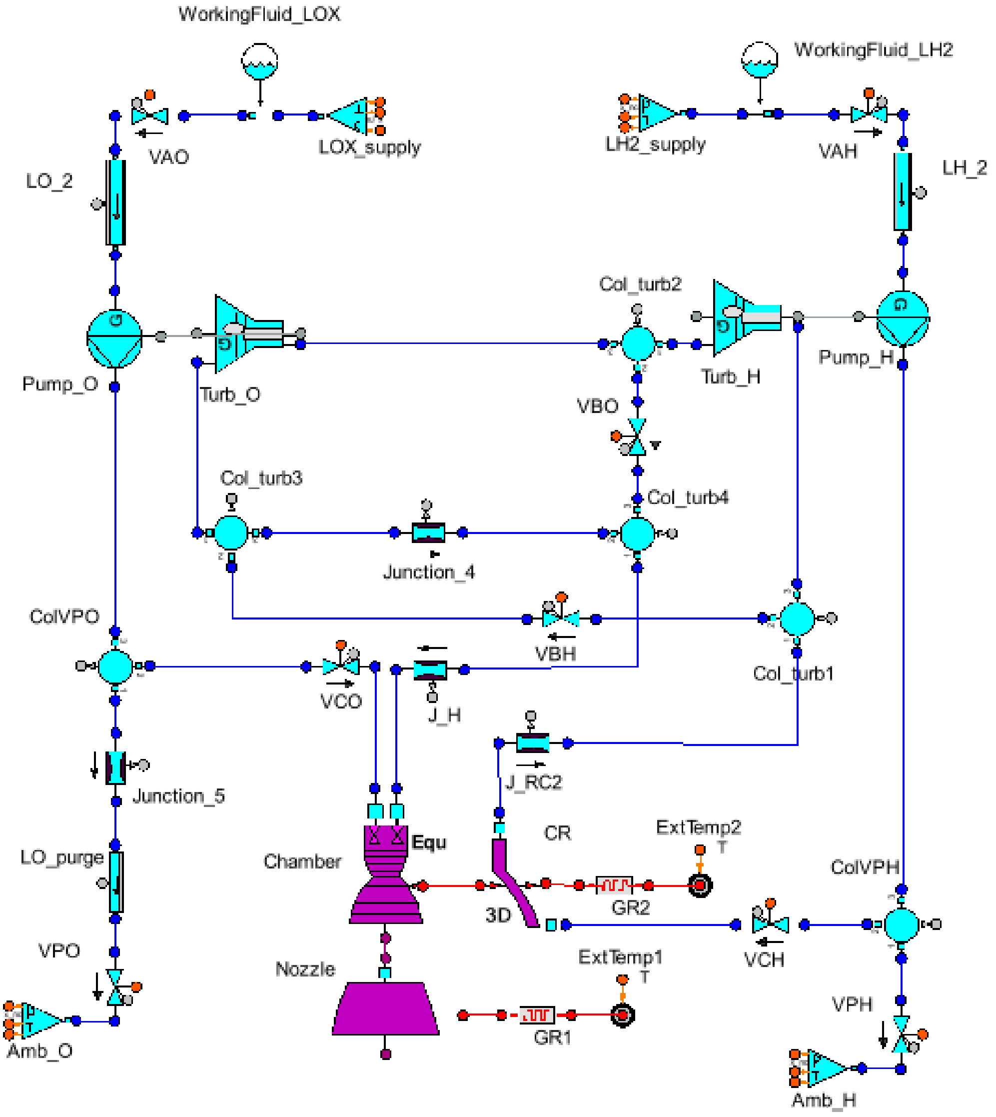
- Not all the valve closing sequences are possible, some of them causing very adverse conditions in the combustor and in the turbines and the pumps could work in cavitation. Selected closing sequence is: first, the GG valves and one second later the main chamber valves.
- It can be appreciated the water hammer produced in the feeding lines due to the valves closing.
- Burning is allowed during the shutdown even though the MR conditions and mass flow are far away from the nominal ones, and the turbines quickly going to nearly null pressure ratios.
- At the end of the engine shutdown, the combustor cavities are under two phase flow, all the residual liquids being vaporized.

2.4.3 Expander engine cycle (T- CCN -002)

Library:	ROCKET_EXAMPLES
Model Name:	ExpanderEngine_eq
Partition Name:	default
Experiment Name:	exp1

2.4.3.1 Model description

This model represents an Expander engine cycle type. Input data are fictitious values, the aim this example being just to show the ESPSS Libraries capabilities regarding this type of engine.



The boundaries, valve activation orders and some geometrical input data are given in the experiment file.

2.4.3.2 Results

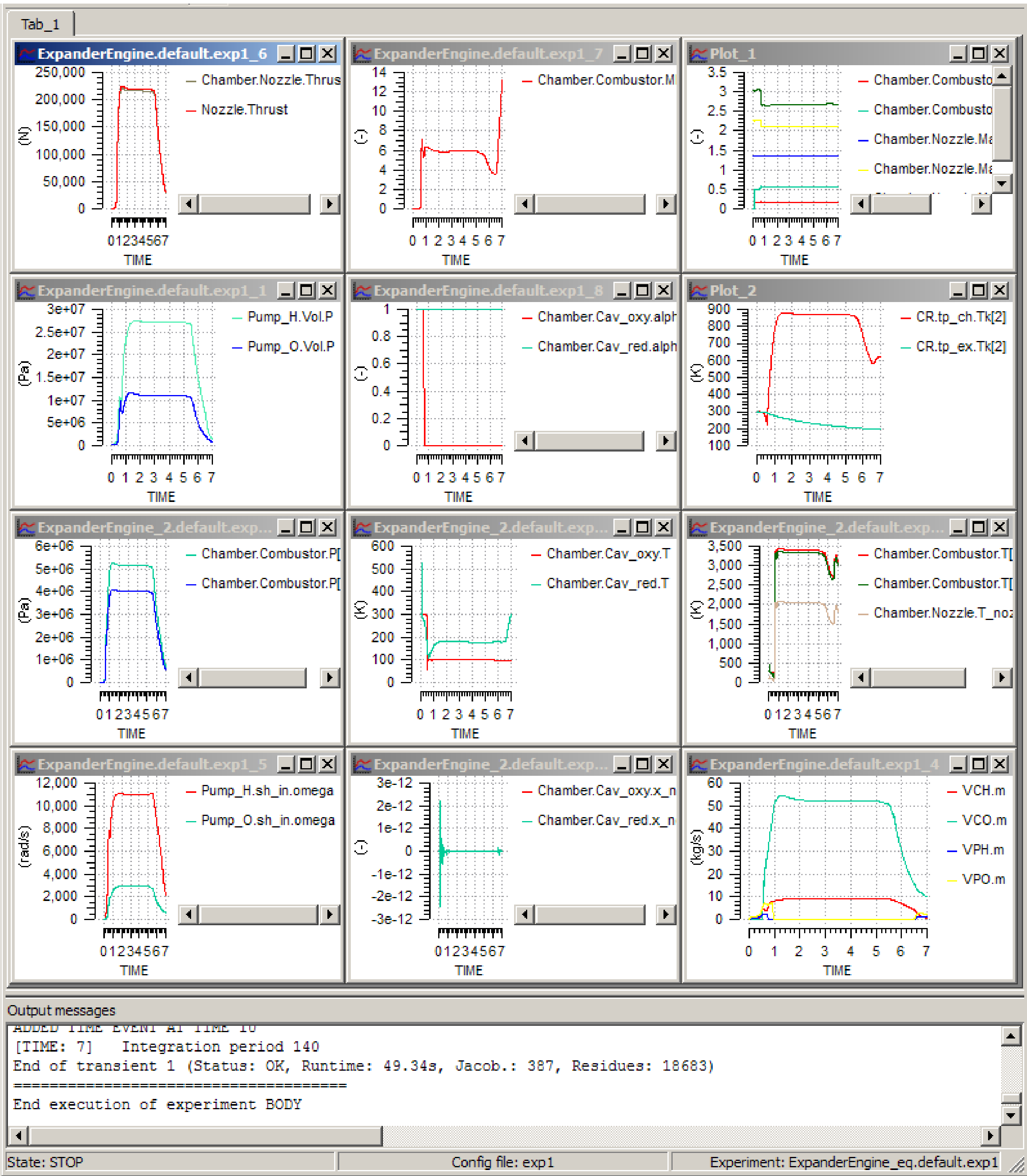
Here below the main plots obtained. We remark the following:

- In this model the generalized turbine model have been used.
- The Regenerative circuit is enough to allow the start-up of the engine without any auxiliary starter. The coupling of this element with the turbines produces some pressure and axial speed oscillations.

The model is able to simulate the start-up and the shut-down of the engine. As for the previous model (GG engine), main simulated processes during the startup are:

- Priming process in the cavities
- Vapor pressurization inside the combustor before ignition
- Simplified model for the LOX vaporization inside the combustor
- Pressure/temperature rise in the chamber during ignition

At the end of the shutdown it can be found generalized two-phase flow in the feeding lines.



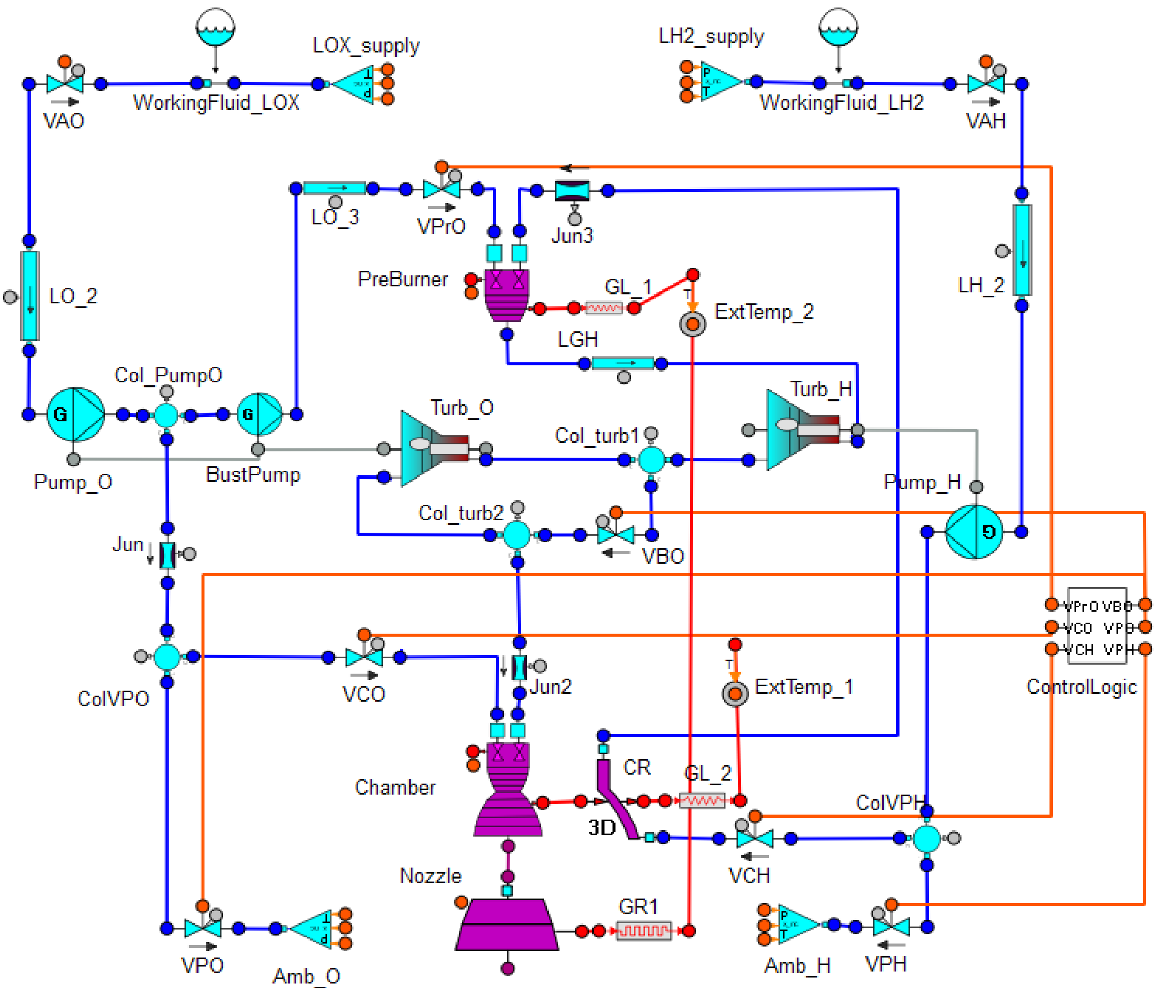
2.4.4 Staged engine cycle (T- CCN -003)

Library:	ROCKET_EXAMPLES
Model Name:	StagedEngine_rate
Partition Name:	default
Experiment Name:	exp1

2.4.4.1 Model description

This model represents a Staged engine cycle type. Input data are fictitious values, the aim this example being just to show the ESPSS Libraries capabilities regarding this type of engine.

In this model, the default performances maps of the turbines (subsonic) have been used.



The boundaries, valve activation orders and some input geometrical data are given in the experiment file.

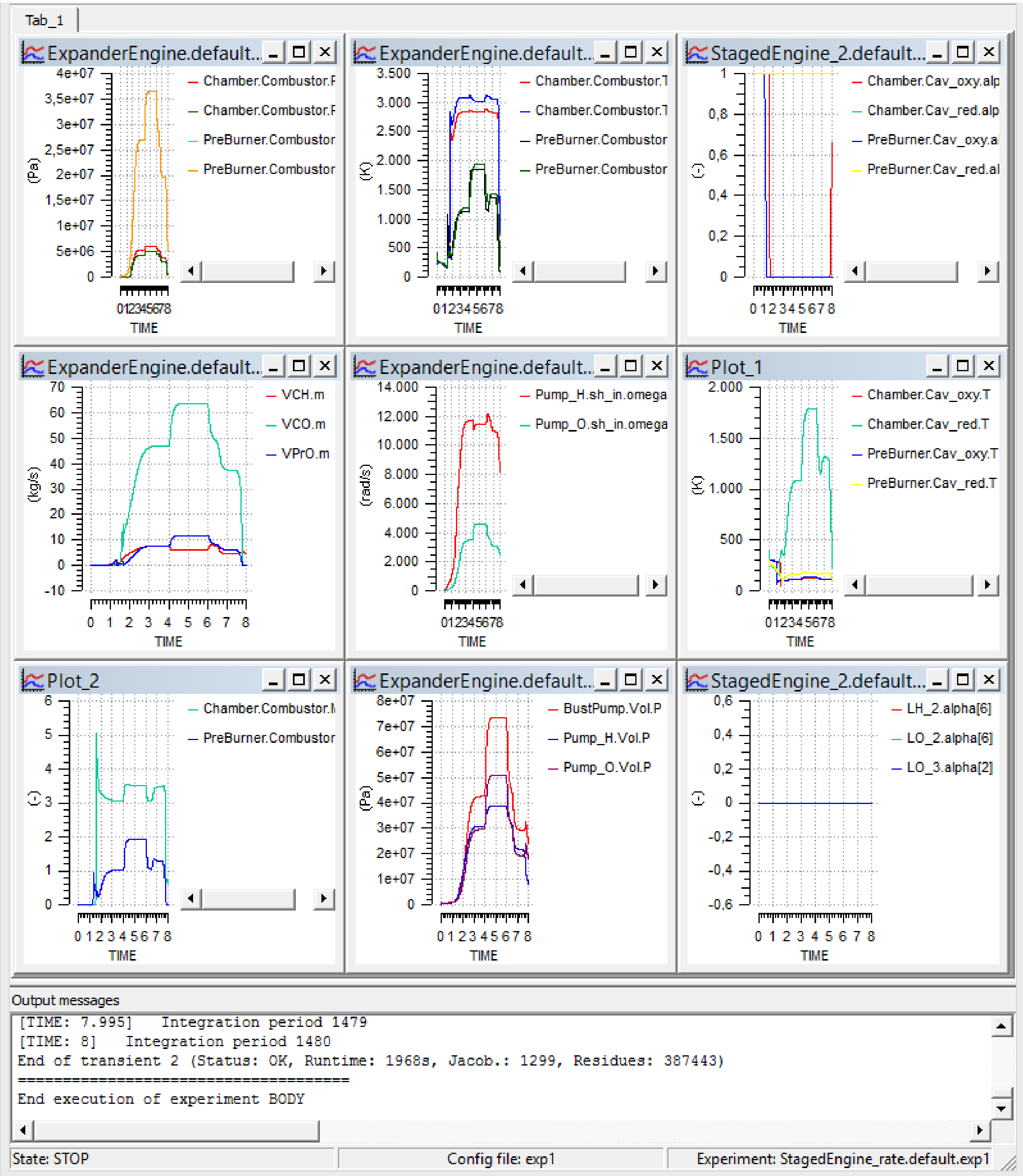
2.4.4.2 Results

Here below the main plots obtained. We remark the following:

- In this model, the "combustor_rate" component is used. With the default burning and evaporation times (see UM, AD-4, Section 8), the chamber temperatures are lesser than the equilibrium ones.
- The model is able to simulate the start-up and the shut-down of the engine, including the relevant phenomena: cavities priming, liquid vaporization, pressure/temperature rise in the chamber during ignition, cavitations in the feeding lines, etc.

- Two steady working points have been simulated by controlling the VPrO valve. See the input data tables on the "ControlLogic" component.

As for the others engine models, the molar fraction of the products entering and exiting the combustion chambers can be found by exploring the results on the monitor tool.

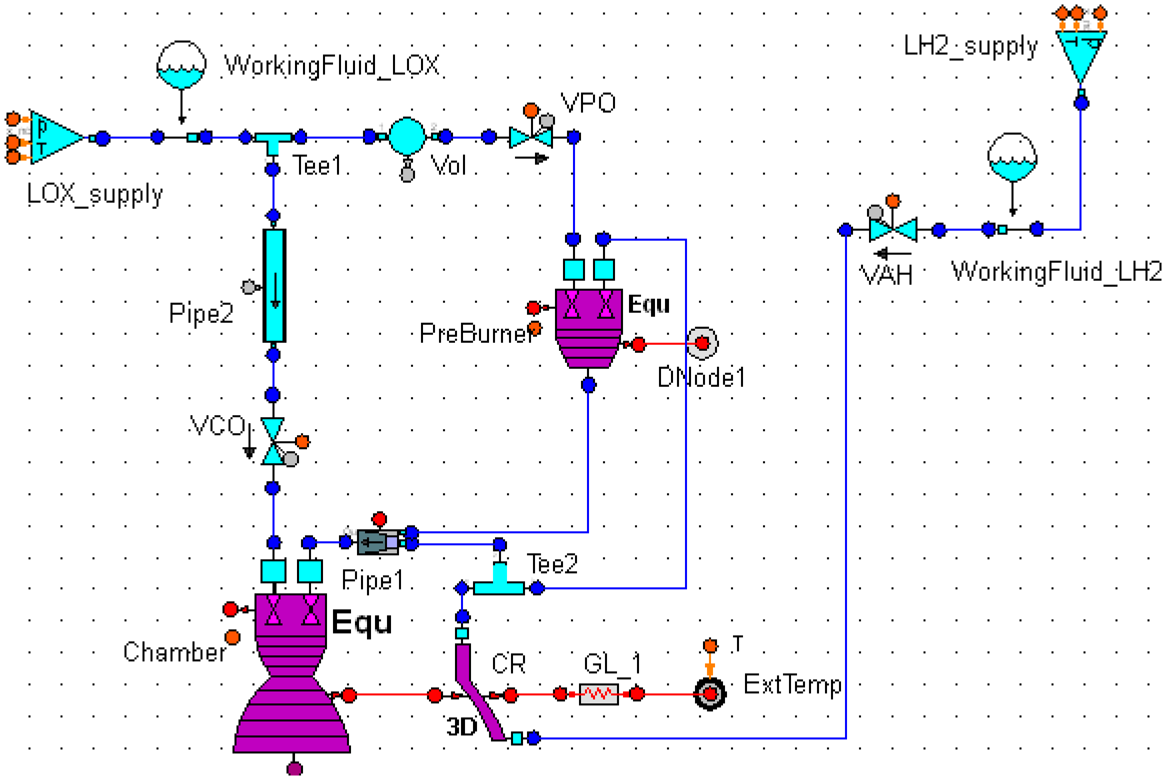


2.4.5 Pre-burner pure fluid mixer cycle (T- CCN -005)

Library:	ROCKET_EXAMPLES
Model Name:	Test_PreBurner
Partition Name:	default
Experiment Name:	exp1

2.4.5.1 Model description

This model represents an example of a Pre-burner cycle with a main chamber feed with combusted gases and two pure fluids as propellants. Input data are fictitious values, being the aim of this example just to show the ESPSS Libraries capabilities regarding this type of engine.



It is pointed out the use of the component "ProdMixer_tee" (aliased Pipe1 in this case) for the simulation of more than 2 injectors in a chamber. This component mixes the combustion gases leaving the Preburner with the pure fluid coming from the reducer tank.

- Before firing, liquid methane is injected both in pre-burner and in the mixer downstream the combustor
- After firing, combusted gases will be mixed with pure methane

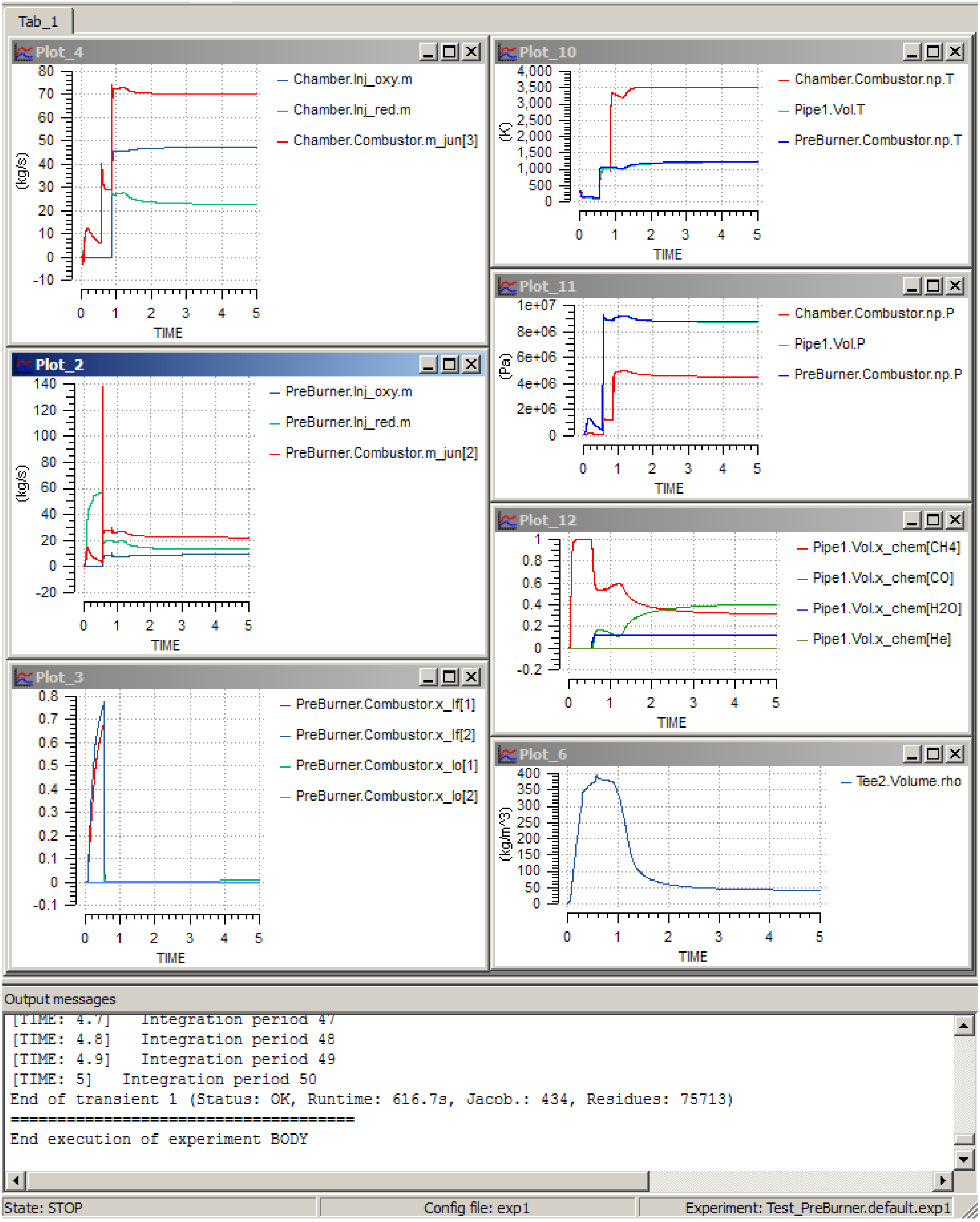
The boundaries, valve activation and some input geometrical data are given in the experiment file. *The cooling jacket bleeding is made intentionally high*.

2.4.5.2 Results

Here below the main plots obtained with **liquidExitAllowed = FALSE**. We remark the following:

- Before firing, an important amount of liquid is present in the chambers. Because in this case, liquid exit is not allowed, an important shift in combustor mass flows can be appreciated
- After firing, all the injected liquid is vaporized in the chambers producing a big increase in pressure and temperature
- In contrast with previous ESPSS versions, chemicals and density values are correctly calculated

- Simulation results can be instable if the mixer inlet flows are not positive,



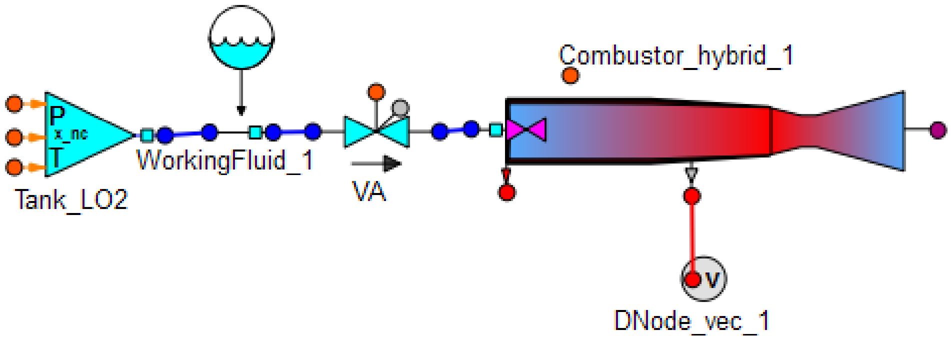
With **liquidExitAllowed = TRUE**, the **simulation fails under the same conditions** as before because of the difficulty to deals with liquids treated as "combusted gases". Note that in components downstream the combustor, any fluid property is calculated according to CEA coefficients. *Simulation would be possible if we advance simultaneously the ignition and the valve opening of LOX and CH4, thus avoiding the formation of important amount of liquid inside the combustor.*

2.4.6 Hybrid engine (T- CCN -006)

Library:	ROCKET_EXAMPLES
Model Name:	Test_hybrid
Partition Name:	default
Experiment Name:	exp1 (Test_hybrid), exp2 (test_solid)

2.4.6.1 Model description

This model represents an example of a Hybrid/Solid rocket engine with a main chamber containing a solid propellant, optionally fed with a liquid or gaseous oxidizer and a nozzle plus a liquid oxidizer injector system (closed while simulating a solid rocket engine). Input data are fictitious values, being the aim of this example to show the ESPSS Libraries capabilities regarding this type of engines.



Main dimensions are:

Cylinder: L = 1 m; D = 0.2 m *(Uniformly in 90% of the length; linear decreasing till throat)*
Grain propellant thickness = 0.03 m *(Uniformly in 90% of the length; linear to 0 at throat))*
Nozzle D_out/D_th ratio = 2.4

Grain factors are set to 5 (effective grain surface area with respect to a cylindrical one).

2.4.6.2 Results for a solid propellant case (exp2)

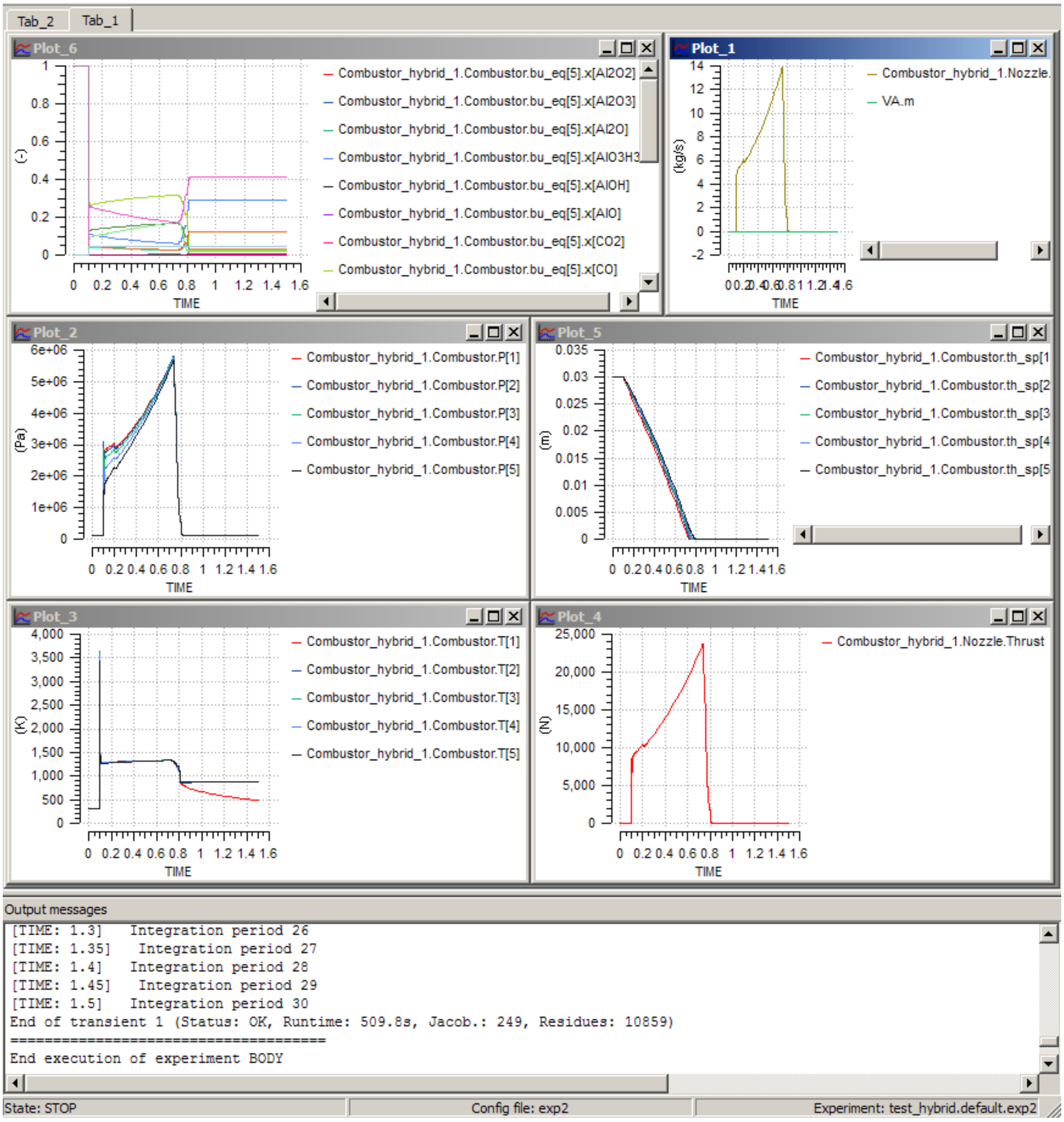
The boundaries (*closed LOX valve*), the time-dependant law to control the ignition, and some input data related to the combustion chamber and the solid propellant are given in the experiment file. *Note in this case we need the starter active (during 0.2 s) to help the release of rubber*.

```
EXPERIMENT exp2 ON test_hybrid.default
DECLS
...
BOUNDS
        Combustor_hybrid_1.Combustor.IgnitFlag = step(TIME,0.1)
        Combustor_hybrid_1.Combustor.starter_m = 0.2*(step(TIME,0.1) - step(TIME,0.2)) – starter
        ...
        -- Grain/gas exchange area factors :
        Combustor_hybrid_1.Combustor.f_s[01] = 5 ...
        Combustor_hybrid_1.Combustor.f_s[10] = 5

        VA.s_pos.signal[1] = 0. -- closed LOX valve
        Combustor_hybrid_1.np_out.P = 100000
        Combustor_hybrid_1.tp_inj.q[1] = 0
BODY
        Combustor_hybrid_1.Dt = 0.06        -- Small throat because NO LOX injection

        WorkingFluid_1.fluid_nc = PfGas_Air
        Combustor_hybrid_1.x_nco = 0
```

--Solid propellant characteristics
Combustor_hybrid_1.GasSolOption = **stdSolid**
Combustor_hybrid_1.a_sp = 1e-005 -- 3e-3 -- **constant regression rate (3 mm/s !!)**
Combustor_hybrid_1.b_sp = 0.45 -- 0 --
Combustor_hybrid_1.tau_c = 1e-3
Combustor_hybrid_1.tau_b = 1e-4
Combustor_hybrid_1.Tsat_sp = 400

-- Mass fractions of solid propellant constituents
Combustor_hybrid_1.rubComp[**HTPB**] = 0.5
Combustor_hybrid_1.rubComp[IPDI] = 0.
Combustor_hybrid_1.rubComp[RubUsr] = 0.
Combustor_hybrid_1.rubComp[KNO3_a] = 0
Combustor_hybrid_1.rubComp[**Al_cr**] = 0.1
Combustor_hybrid_1.rubComp[S_a] = 0
Combustor_hybrid_1.rubComp[NH4NO3_IV] = 0
Combustor_hybrid_1.rubComp[**NH4CLO4**_I] = 0.4
…

Below some plots obtained. The model calculates the chemical composition of the products at the equilibrium temperature and pressure in correspondence with the grain composition, the consumption of solid propellant and the corresponding outlet mass flow for the given geometry.



Following conclusions can be achieved:

- The model calculates the P/T 1D axial distributions during the startup, quasi-steady conditions and shutdown.
- A small pressure surge at the start up is detected, then, the thrust is increasing because the grain/gas effective area is increasing with the grain consumption.
- The model is able to evaluate the influence of different grain compositions coupled with the shape of the grain and the combustor geometry.
- It is also appreciated that the extinction of the solid propellant is produced nearly at the same time for all the nodes along the chamber length.

2.4.6.3 Results for a hybrid propellant case (exp1)

The boundaries, *the time-dependant law to control the LOX valve opening* and ignition, and some input data related to the combustion chamber and the solid propellant are given in the experiment file:

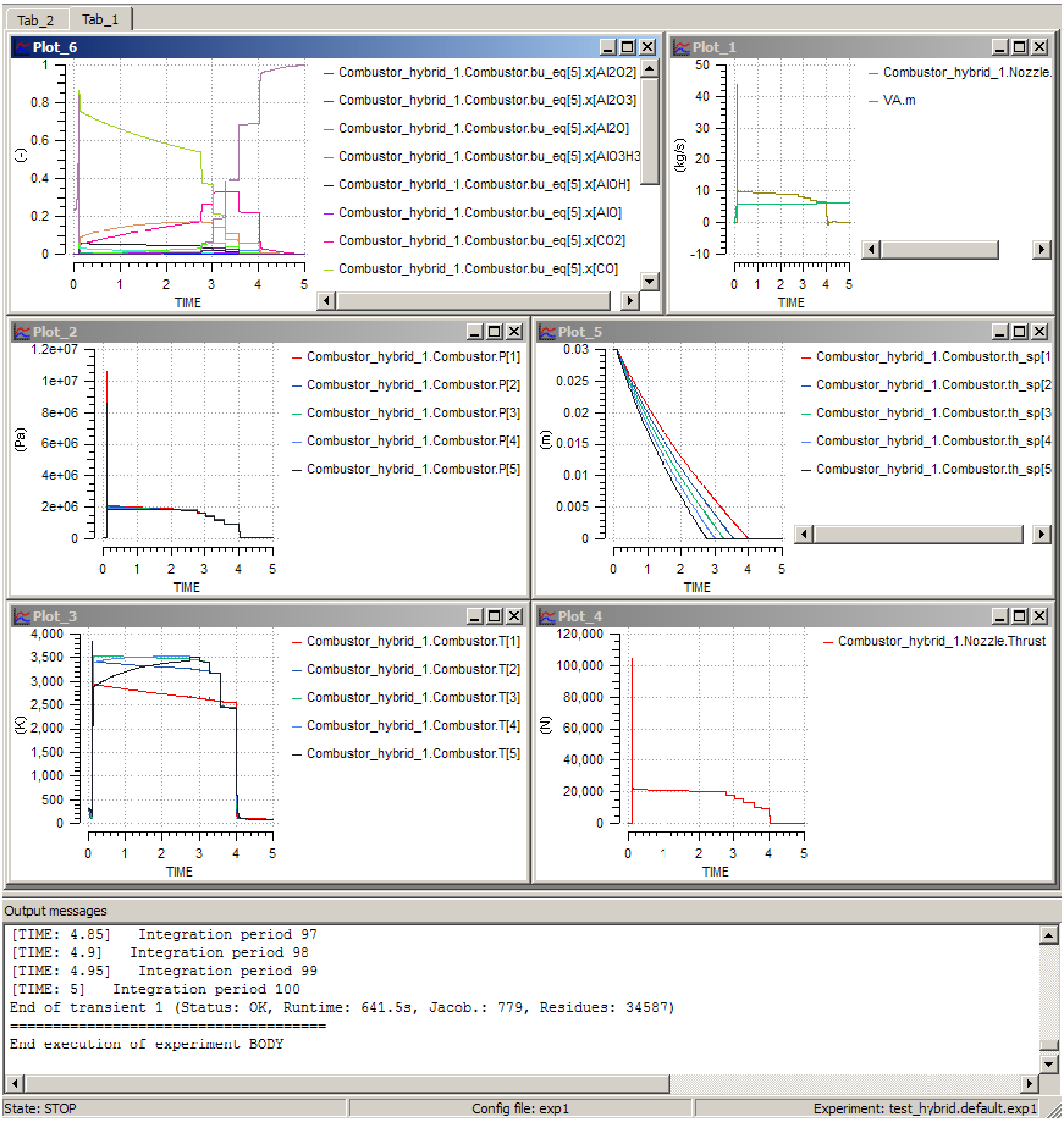
```
EXPERIMENT exp1 ON test_hybrid.default
DECLS
        TABLE_1D law_VAH = {{0., 0.1,0.2, 9.5, 10., 100} , {0, 1, 1, 1 , 0. , 0. } }
…
BOUNDS
        Combustor_hybrid_1.Combustor.IgnitFlag = step(TIME,0.1)
        …
        -- Liquid oxygen injection conditions
        Tank_LO2.s_pres.signal[1] = 100e5
        Tank_LO2.s_temp.signal[1] = 90
        Tank_LO2.s_xNonCond.signal[1] = 0
        VA.s_pos.signal[1] = 0.5*timeTableInterp(TIME,law_VAH)   -- open LOX valve

        -- Grain/gas exchange area factors :
        Combustor_hybrid_1.Combustor.f_s[01] = 5 …
BODY
        Combustor_hybrid_1.Dt = 0.10        -- Greater throat because LOX injection
        VA.Ao = 1e-4                        -- LOX injection area

        --Solid propellant characteristics
        Combustor_hybrid_1.GasSolOption = stdHybrid
        Combustor_hybrid_1.a_sp = 1e-5 ---- constant regression rate (3 mm/s !!)
        Combustor_hybrid_1.b_sp = 0.9
        …
        -- Molar fractions of solid propellant constituents
        Combustor_hybrid_1.rubComp[HTPB] = 0.9
        Combustor_hybrid_1.rubComp[IPDI] = 0.
        Combustor_hybrid_1.rubComp[RubUsr] = 0.
        Combustor_hybrid_1.rubComp[Al_cr] = 0.1
        …
```

It is pointed out:

- The high pressure peak (nearly an explosion) during the startup because of the fact we suppose valid the rubber release regression law (proportional to the mass flow) during this first stage.
- Then, the temperature evolution is different for each node because of the local mixture ratio depends on the LOX consumption and rubber "vapor" released calculated by the code along the chamber.
- The model calculates the chemical composition of the products and the equilibrium temperature and pressure in correspondence with the grain composition and oxidizer injection conditions, the consumption of solid propellant and the corresponding outlet mass flow for the given geometry. Ç
- The simulation results in this case are strongly dependent on the oxidizer injection conditions (VA area and injection pressure) because the combusted gases compositions depends on the mixture of the evaporated LOX, also simulated in the ESPSS Combustor_hybrid component.

- It is also appreciated the extinction of the solid propellant at different stages, the latest nodes being finishing before because they stayed at higher mass flow.

Below some plots obtained:

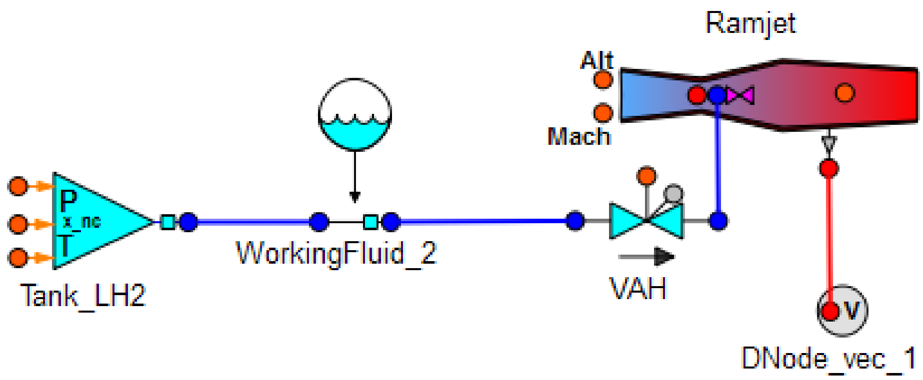


2.4.7 Ramjet engine (T- CCN -007)

Library:	ROCKET_EXAMPLES
Model Name:	Test_ramjet
Partition Name:	default
Experiment Name:	exp1

2.4.7.1 Model description

This model represents an example of a Ramjet engine with a main chamber taking the air from the atmosphere and fed with the LH2 fuel. Input data are fictitious values, being the aim of this example just to show the ESPSS Libraries capabilities regarding this type of engine.

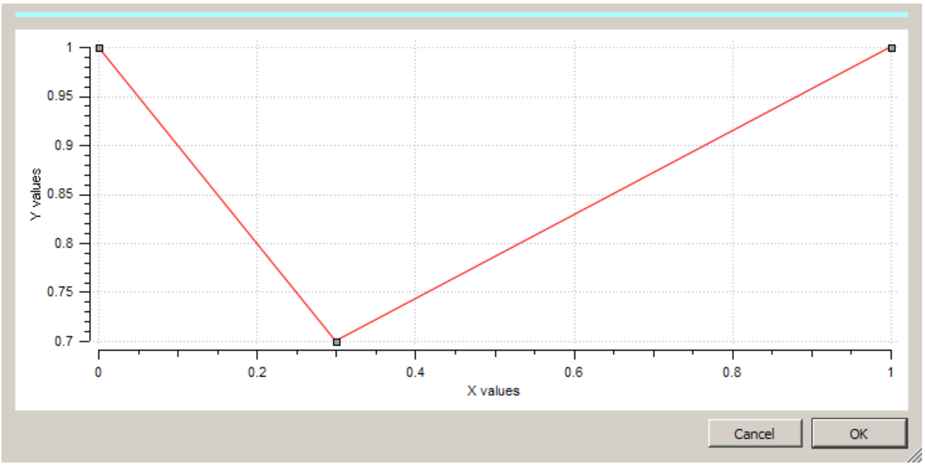


The system consists of a main thruster (aliased "Ramjet" in this case) internally provided with a non-ideal intake, an area varying supersonic combustion chamber, plus a liquid fuel injector system. Main dimensions are:

- Combustor length: L = 1 m;
- Inlet cross section width and height= 0.5 m
- Number of nodes: **30**

Combustor height Profile →

Constant width



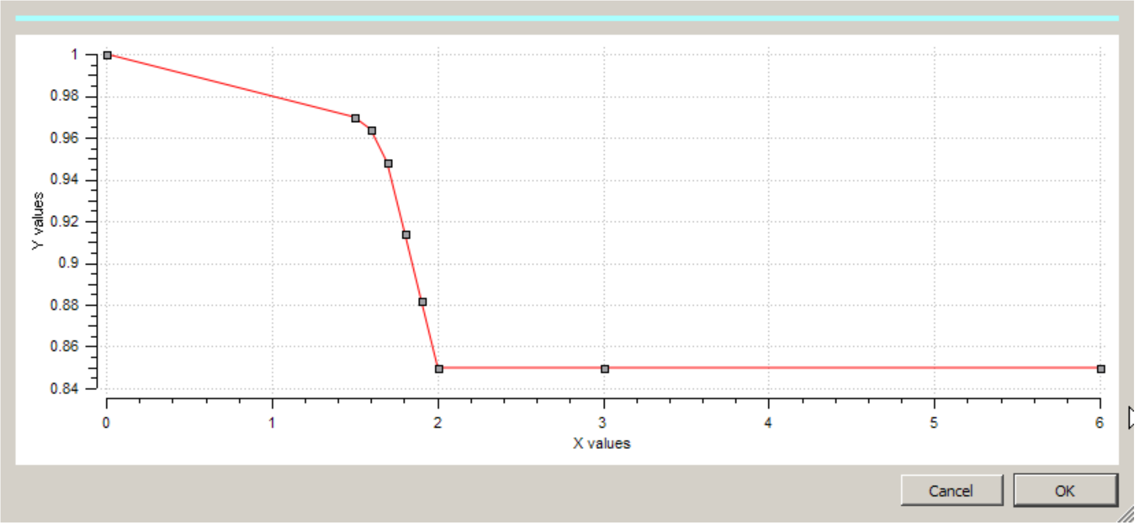
Liquid LH2 injection conditions are:

Pressure = 20 bars; Temperature: 21 K

A_inj_red = 1E-4 m2

VAH.Ao = 1E-3 m2

The Total Pressure Recovery vs. flight Mach number at the intake is an "arbitrary" input data table as showed below.



The inlet pressure of the combustor is calculated in the Ramjet component accordingly with the flow regime in the combustor and to intake pressure losses at the current flight Mach number and altitude. The discharge pressure is fixed to the static ambient pressure.

The boundaries, the time-dependent law to control the valve opening and the Flight conditions, are given in the experiment file:

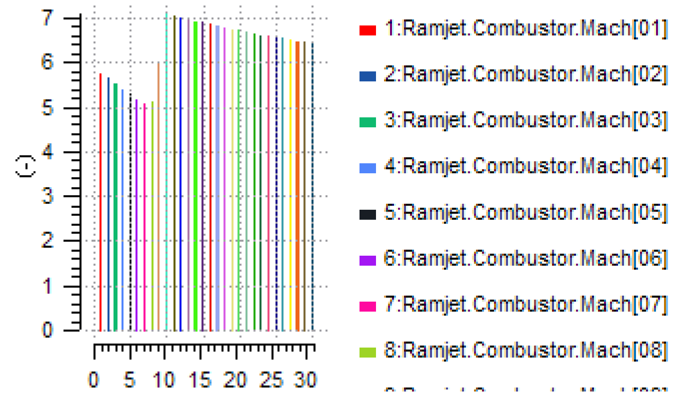
```
…
BOUNDS
        FLUID_FLOW_1D.Damp = 1
        FLUID_FLOW_1D.GRAV = 9.806
        FLUID_PROPERTIES.VDW_option = 0

        Ramjet.Combustor.IgnitFlag = step(TIME,15.01)        -- ignition time
        Ramjet.Combustor.starter_T = 1000
        Ramjet.Combustor.starter_m = 0
        Ramjet.tp_inj.q[1] = 0

        Tank_LH2.s_pres.signal[1] = 20e5
        Tank_LH2.s_temp.signal[1] = 21
        Tank_LH2.s_xNonCond.signal[1] = 0
        VAH.s_pos.signal[1] = 0.1*step(TIME,10.01)           -- LH2 injection time

        Ramjet.s_alt.signal[1] = 20000                       -- Altitude
        Ramjet.s_mach.signal[1] = 6--*min(1,TIME*10)         -- Flight Mach number
        …
        Ramjet.Combustor.A_rel_red[15] = 0.2
        Ramjet.Combustor.A_rel_red[16] = 0.2
        Ramjet.Combustor.A_rel_red[17] = 0.2
…
```

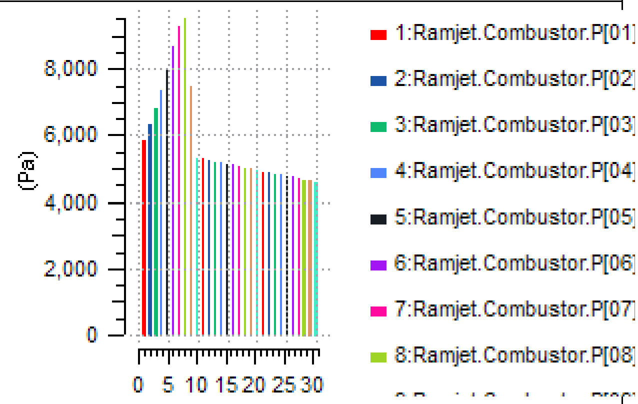
LH2 injection nodes are from node 13 to 17 (30 nodes in total uniformly distributed):

2.4.7.2 Results

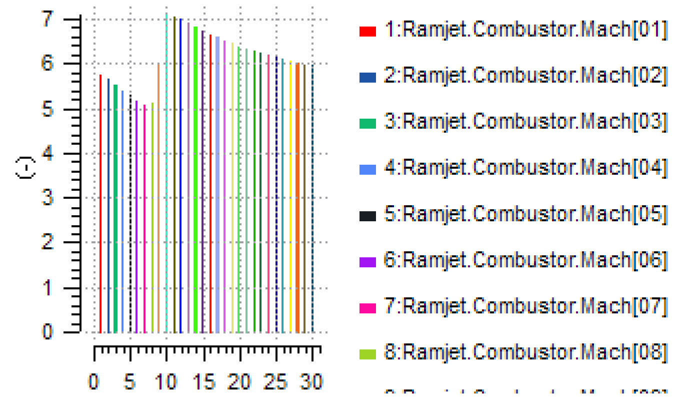
Below the main plots obtained before and after LH2 injection, and after stabilized combustion at a flight Mach number of 6:

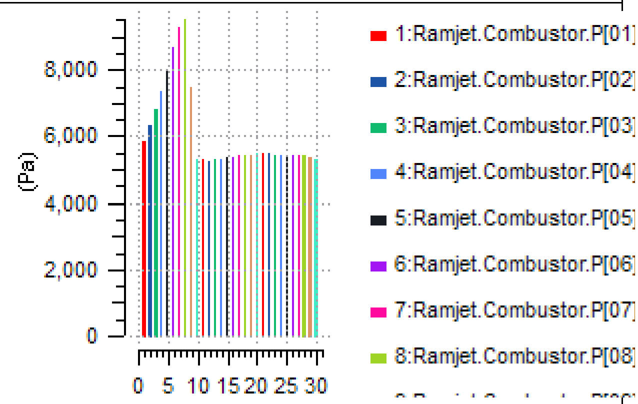
Flight Conditions: M =6, Alt = 20000 m (before LH2 injection)



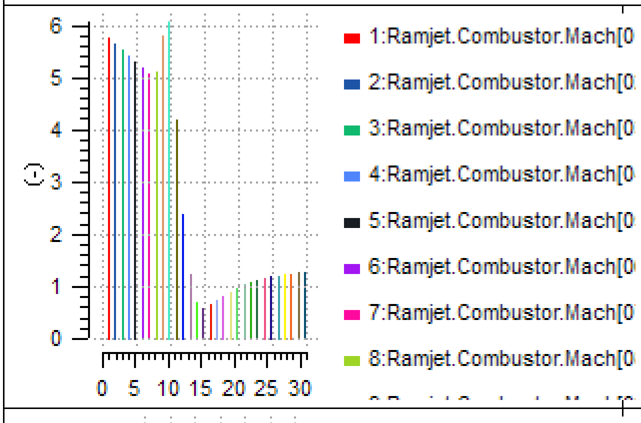


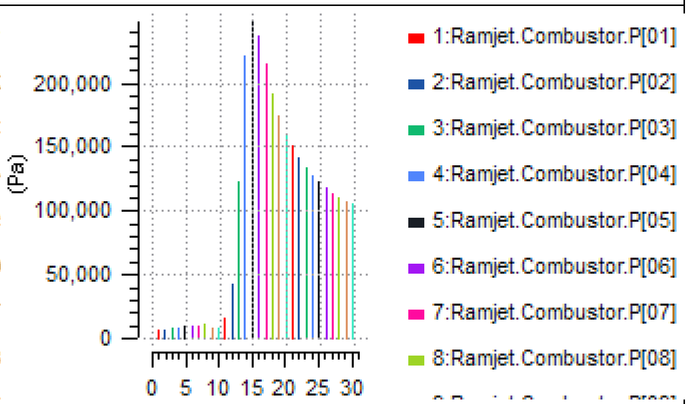
(after LH2 injection, time = 10 s)

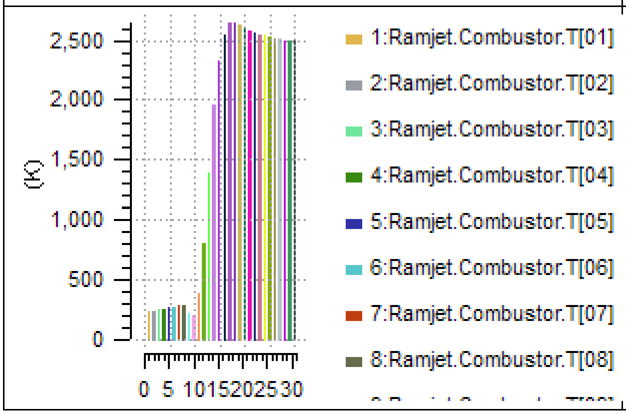


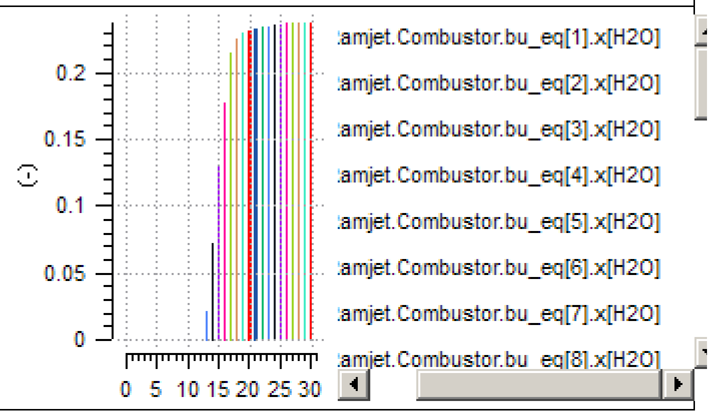


(after combustion, time = 15 s)



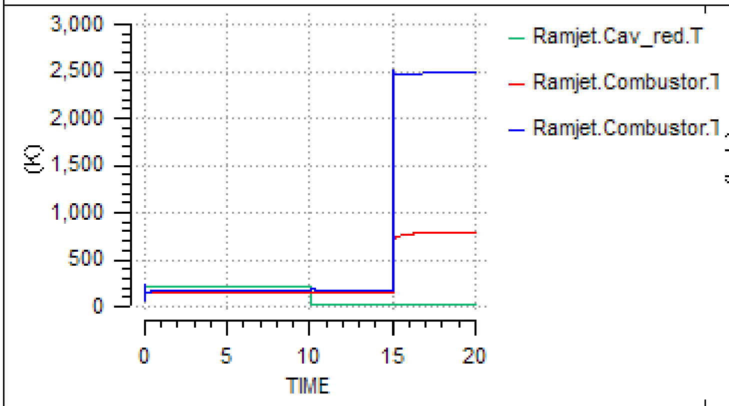




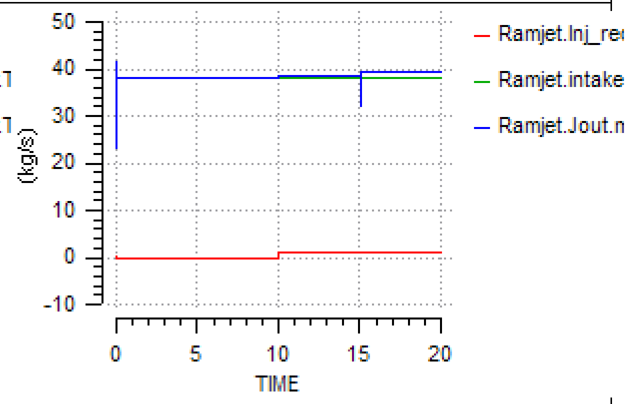


The time histories of the mass, flow thrust and other magnitudes as a response of the different engine conditions are presented below:

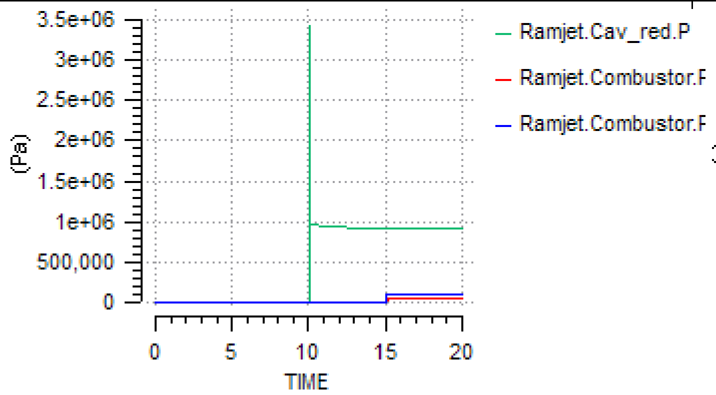
Flight Conditions: M =6, Alt = 20000 m
(Time histories)



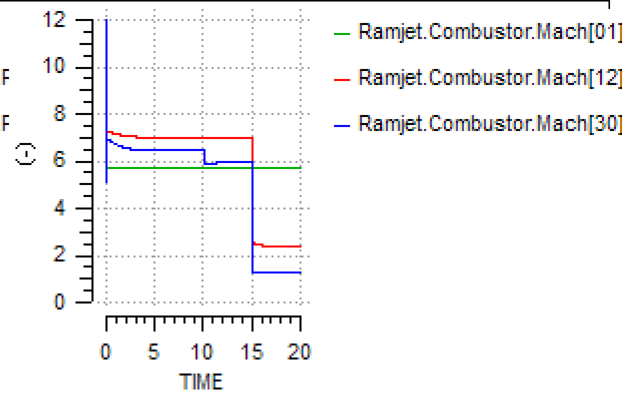
Temperatures



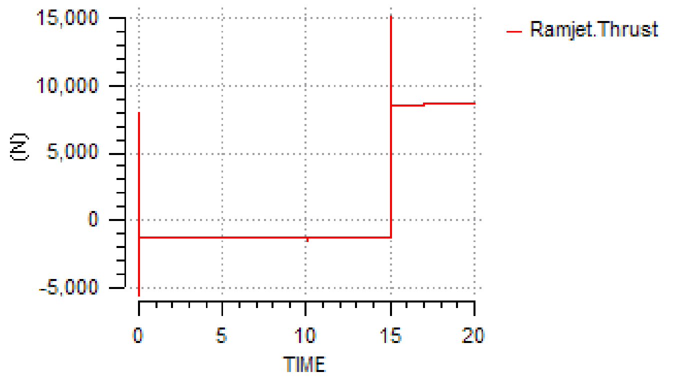
Mass flows



Pressures



Mach numbers



Thrust

It is interesting to see how the ESPSS model can simulate the influences of the friction, geometry, flight & LH2 injection conditions, etc. on the thrust, combustion efficiency and Mach evolutions.

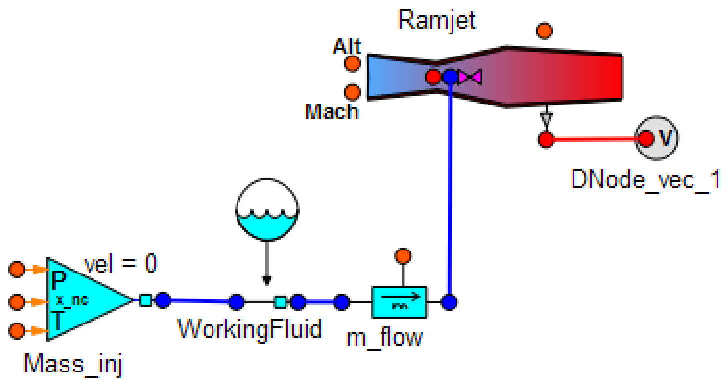
Of course, with no combustion, the thrust is negative. It becomes positive only with a good coupling of a supersonic combustion within a "small" range of LH2 injection. It can be tested for example that increasing the LH2 flow rate over a limit the combustor becomes subsonic with negative thrust.

2.4.8 Ramjet mass addition test case (T- CCN -008)

Library:	ROCKET_EXAMPLES
Model Name:	MP2
Partition Name:	default
Experiment Name:	exp1

2.4.8.1 Model description

This example validates the Ramjet capabilities concerning mass addition (injected fuel flow) under subsonic and supersonic conditions. Only the Centred scheme is available in the Ramjet component.



To be able to compare the results of this model with the analytical solution, the injection conditions are supposed to be the same in both cases (Air at different flight conditions). The combustor has constant flow area (0.05x0.03m, 1.2 m length) and no wall friction (k_f input data = 0). Total Pressure Recovery is supposed to be 1 (no pressure losses at the intake).

The boundaries and the flight conditions are given in the experiment file:

```
...
BOUNDS
    ...
    FLUID_FLOW_1D.Damp = 1
    FLUID_FLOW_1D.GRAV = 9.806
    FLUID_PROPERTIES.VDW_option = 0

    Ramjet.Combustor.IgnitFlag 0          -- no ignition
    Ramjet.s_alt.signal[1] = 5000              -- Altitude
    Ramjet.s_mach.signal[1] = 0.4467--3.161    -- Flight Mach number
    Mass_inj.s_pres.signal[1] = Ramjet.P_o     -- Injection conditions
    Mass_inj.s_temp.signal[1] = Ramjet.T_o
    Mass_inj.s_xNonCond.signal[1] = 0
    m_flow.s_massflow.signal[1] = 0.06 --0.15  -- Injected mass flow
    ...
```

The injected mass flow is supposed to be the same in all the combustor nodes (default input data for *Ramjet.Combustor.A_rel_red[i]).*

The inlet pressure of the combustor will be calculated according to the flow regime in the combustor and the flight Mach number and altitude. The discharge pressure is fixed to the static ambient pressure.

2.4.8.2 Analytical solution comparison

Document ACP7-GA-2008-21 1485 LAPCAT II. "Development of One-dimensional Propulsion Model" presents an analytical solution for supersonic/subsonic tubes with mass addition. This analytical solution (for constant

Cp, gamma =1.4) is coded under EcosimPro in the specific component "**analyticalMassAdd**" in the FLUID_FLOW_1D_EXAMPLES library.

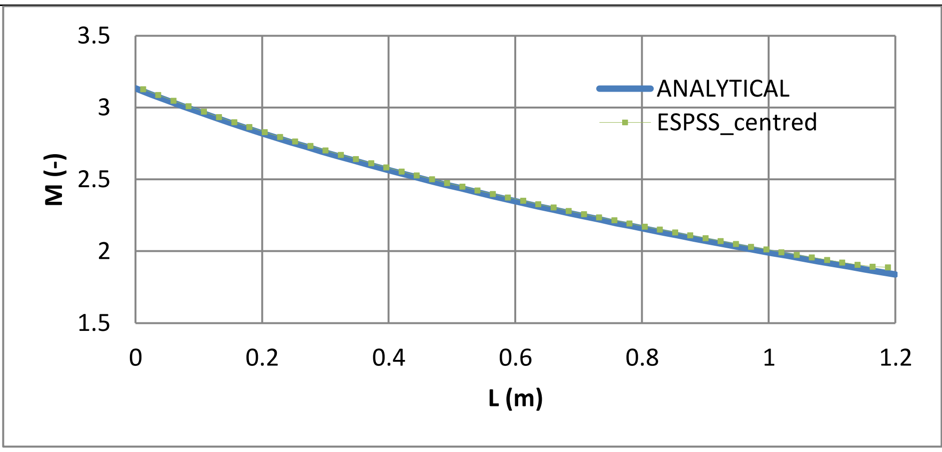
Plots have been automatically obtained using EXCEL files (attached to the experiment folder) by inserting the reports generated with EcosimPro. *There is nearly no differences between 50 and 200 nodes. Main conclusions are:*

- The Ramjet combustor is able to obtain good results in subsonic and supersonic cases, even though the small perturbation at the inlet in subsonic cases because of the intake equations.
- Present comparison validates the Ramjet 1D implementation of the governing equations in a combustor, that is a different more complex setup than the Pipe component because includes the mixture of the fluid propellants with the combustor products, the chemical equilibrium calculation according to CEA, etc.
- If combustion is activated in this test case where the injected fuel is Air, the results are remaining the same, because the equilibrium calculation does not change the composition.
- In the analytical solution the added mass has the same conditions as the local fluid in the tube, while the ramjet combustor calculates the injection conditions (temperature/pressure and speed) according to the injection system modelled. To be able to compare, the ramjet combustor was modified imposing the injected enthalpy equal to the local enthalpy. The following source code equation was modified only for this test case (use of `h[i]` instead of `s_red.signal[13]`, the injector enthalpy):

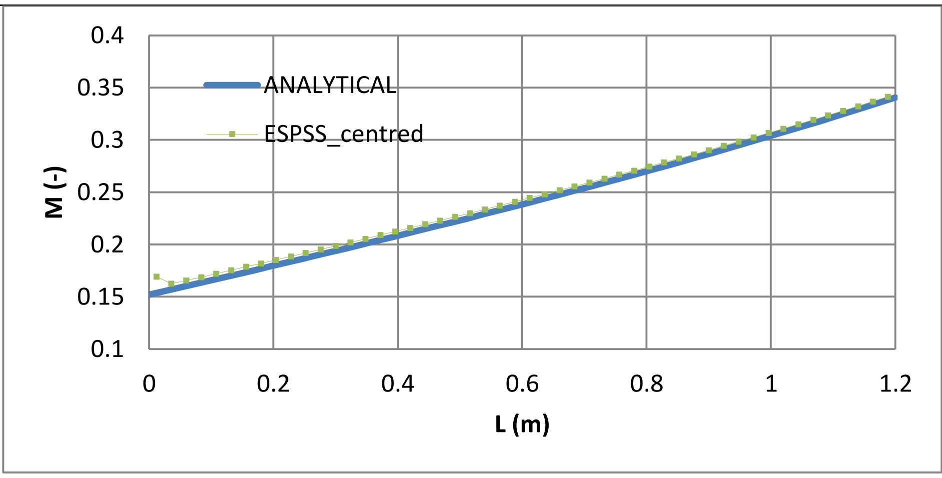
```
EXPAND (i IN 1, nodes) mh_red[i] = m_red*s_red.signal[13] → m_red*h[i]
```

<table>
<tr><th>Case</th><th>Ramjet Comparison (50 nodes)</th></tr>
<tr><td>SUPERSONIC mass addition

Alt = 5000 m
(P/T ambient = 0.54 bar / 255.7 K)

Mach = 3.161
m_in = 1.115 kg/s
m_inj= 0.15 kg/s</td><td>
</td></tr>
<tr><td>SUBSONIC mass addition

Alt = 5000 m
(P/T ambient = 0.54 bar / 255.7 K)

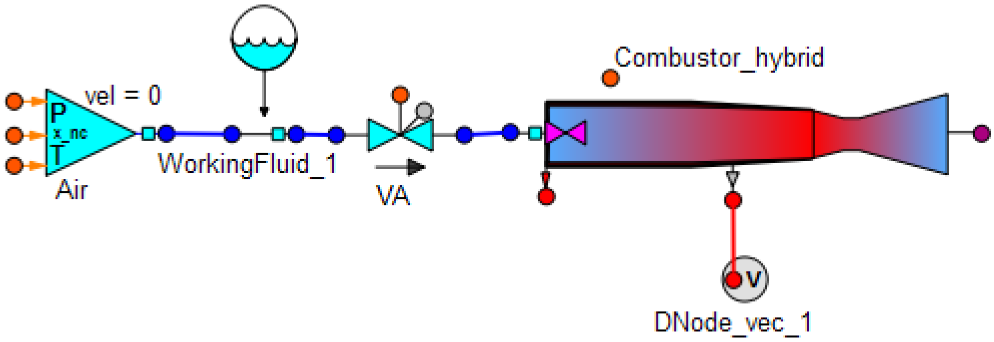
Mach = 0.4428
m_in = 0.063 kg/s
m_inj= 0.06 kg/s</td><td>
</td></tr>
</table>

2.4.9 Hybrid Engine mass addition test case (T- CCN -009)

Library:	ROCKET_EXAMPLES
Model Name:	MP3
Partition Name:	default
Experiment Name:	exp1

2.4.9.1 Model description

This example validates the Hybrid combustor capabilities concerning mass addition (mass flow related to the consumption of the solid propellant) under subsonic conditions. Only the Centred scheme is available in the Hybrid component.



The VA valve has been closed to allow the comparison between the results of this model with the analytical solution. This way, the perturbations caused in the results by the different air admissions (a direct entrance in the tube of the analytical solution and an injection plate in the hybrid combustor) can be avoided.

The mass injection corresponds to the consumption of the solid propellant at the walls. This propellant can be defined by the user. Its composition and characteristics are "artificially" set equal to those of the air at a gaseous state. Therefore, the solid propellant that is being injected in the chamber is in fact air.

The thickness of the solid propellant keeps constant to maintain the same effective area at any simulation time (there is no consumption of the walls, only the mass flow is considered). The combustor has a constant flow area (0.06 m effective diameter, 1.2 m length) and no wall friction (k_f input data = 0). The injected mass flow is supposed to be uniform in all the combustor nodes.

The boundaries and the solid propellant characteristics are given in the experiment file:

```
…
BOUNDS
    …
    Combustor_hybrid.Combustor.starter_m = 0
    Combustor_hybrid.Combustor.IgnitFlag = 1 -- ignition activated to account for the solid propellant mass flow
    Combustor_hybrid.Combustor.f_s[01] = 1
    Combustor_hybrid.Combustor.f_v[50] = 1
    Combustor_hybrid.np_out.P = 100000
    Combustor_hybrid.tp_inj.q[1] = 0
    FLUID_FLOW_1D.Damp = 1
    FLUID_FLOW_1D.GRAV = 9.806
    FLUID_PROPERTIES.VDW_option = 0
    VA.s_pos.signal[1] = 0                    -- inlet valve closed
BODY
    …
    -- Molar fractions of solid propellant constituents
    Combustor_hybrid.rubComp[RubUsr] = 1.
```

```
-- HTPB,IPDI,RubUsr, KNO3_a,Al_cr,S_a,NH4NO3_IV,NH4CLO4_I
Combustor_hybrid.rubUsrForm[Elem_Ar]= 0.0092
Combustor_hybrid.rubUsrForm[Elem_O] = 0.2096
Combustor_hybrid.rubUsrForm[Elem_N] = 0.7812 --- "RubUsr's formula within the following atoms list -H, O, S, N, C, Ar, He, Al, K, Cl- "
...
```

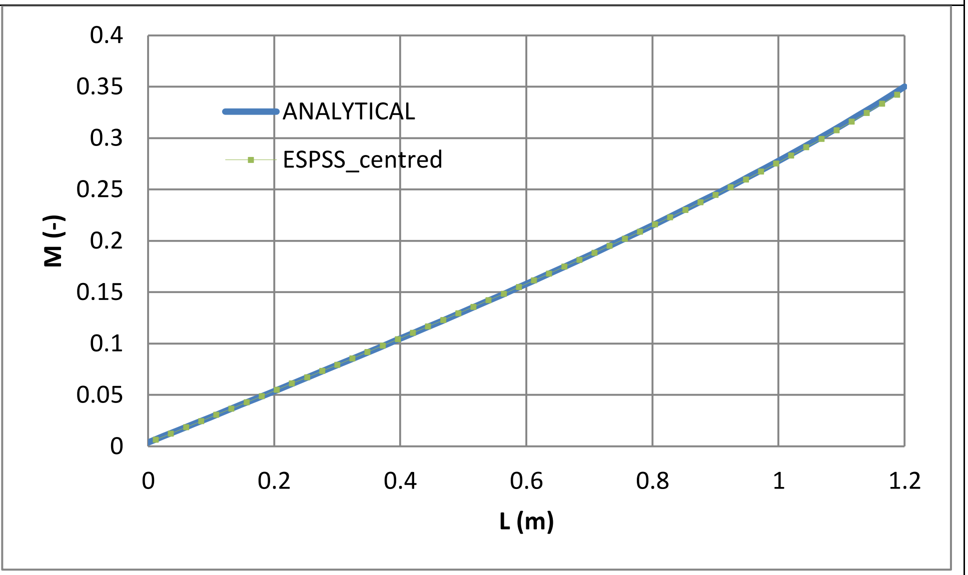
2.4.9.2 Analytical solution comparison

Document ACP7-GA-2008-21 1485 LAPCAT II. "Development of One-dimensional Propulsion Model" presents an analytical solution for supersonic/subsonic tubes with mass addition. This analytical solution (for constant Cp, gamma =1.4) is coded under EcosimPro in the specific component "**analyticalMassAdd**" in the FLUID_FLOW_1D_EXAMPLES library.

Plots have been automatically obtained using EXCEL files (attached to the experiment folder) by inserting the reports generated with EcosimPro. *There is nearly no differences between 50 and 200 nodes. Main conclusions are:*

- The Hybrid combustor is able to obtain good results in the subsonic case, even though the small perturbation at the inlet in subsonic cases because of the intake equations.
- Present comparison validates the Hybrid 1D implementation of the governing equations in a combustor, that is a different more complex setup than the Pipe component because includes the mixture of the fluid propellants with the combustor products, the chemical equilibrium calculation according to CEA, etc.
- The combustion is activated in this test case to allow the injection of a mass flow along the chamber caused by the release of the solid propellant, being actually this propellant Air. The combustion does not insert any perturbation in the results since the equilibrium calculation does not change the composition.
- The mass added in the analytical solution has the same conditions as the local fluid in the tube, while the hybrid combustor calculates the solid propellant consumption according to its saturation temperature and the local conditions of the fluid. To be able to compare, the hybrid combustor was modified imposing the saturation temperature of the solid propellant equal to the local temperature. The following source code equations were modified only for this test case (use of `Tg` instead of `Tsat`):

```
dH = dH + rubComp[j]*rub_x[j,i]*PfGas_h_vs_T(chem, Tsat) → PfGas_h_vs_T(chem, Tg)
dH = dH + rubComp[j]*PfGas_h_vs_T(j, Tsat) → PfGas_h_vs_T(chem, Tg)
```

Case	Hybrid Engine Comparison (50 nodes)
SUBSONIC mass addition Po_in = 1.317 bar P_ex = 1.1317 bar To_in = 271.78 K Mach = 0 m_in = 0 kg/s m_inj= 0.475 kg/s	

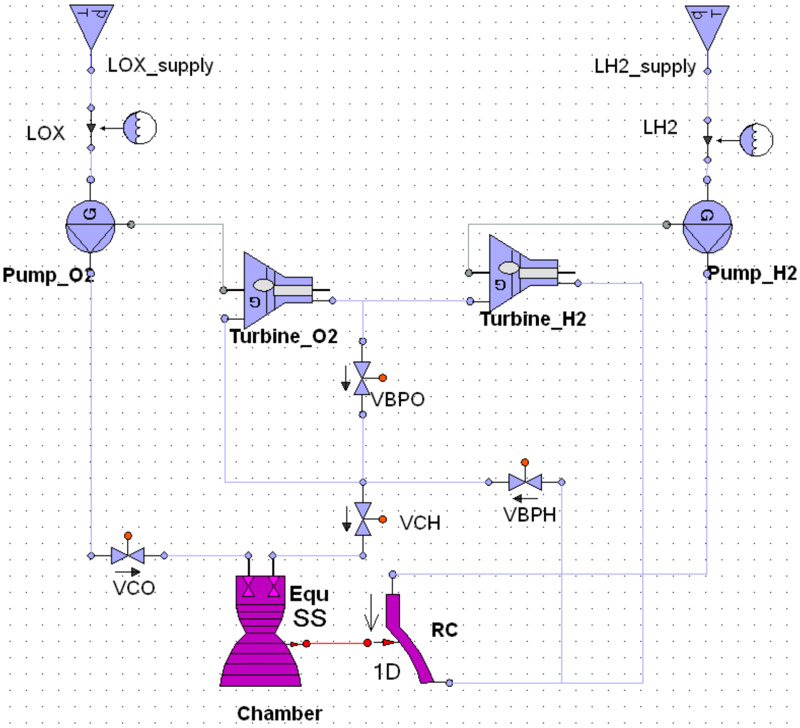
2.5 STEADY LIBRARY

2.5.1 Expander cycle in Design mode (T-STY-001)

Library:	STEADY_EXAMPLES
Model Name:	expander_cycle_design
Partition Name:	default
Experiment Name:	exp1

2.5.1.1 Model description

Let's us consider the "expander_cycle_design.eds" case of the STEADY_EXAMPLES library. It is a typical expander cycle with two serial turbines with by-pas valves driving two pumps:



Present case is a Design model that contains the design conditions as part of the input data of the components. All the components have activated design mode parameters: "type" = Design, known_PI_tt, known_Ns, etc, depending on the respective component, where "type" is a construction parameter.

Imposed/calculated design parameters are:

Component	Imposed parameters (design conditions)			Calculated variables (note 1)	
Turbines	PI_tt_o	eta (effic.)		rpm, (note 1)	mass flow
Pumps	Ns (spec. speed)	eta (effic.)		rpm (note 1)	mass flow
Chamber	Pc	MR (mixture ratio)	dP_per_oxy, dP_per_red	Inj_oxy.A (note 2)	Inj_oxy.A
Cooling Jacket	dP_design	Tw_throat (throat wall T)		t_ch_cal (note 3)	th_i_cal
Valves			VCO.dP_design, VCH.dP_design (note 4)	Valve areas (*.A)	

Notes

1. Calculated variables are internal to the respective component and include flows (m), power, torque, axial speeds, pressures and temperatures, etc
2. Inj_oxy.A, Inj_oxy.A are the injector areas
3. "t_ch_cal" and "th_i_cal" are the channel width and the internal wall thickness at throat position of the Cooling Jacket
4. Not all the junction / valve pressure drops are imposed, only the ones marked as "dp_type= UserGiven". The user must determine which valves are calculated "FromPorts" (normally, bypass valves where the pressure bounds are determined by the imposed turbine pressure ratio)

2.5.1.2 Default partition experiment

Default partition experiments automatically calculate the design values of the cycle for a design model. To do that, follow the following steps:

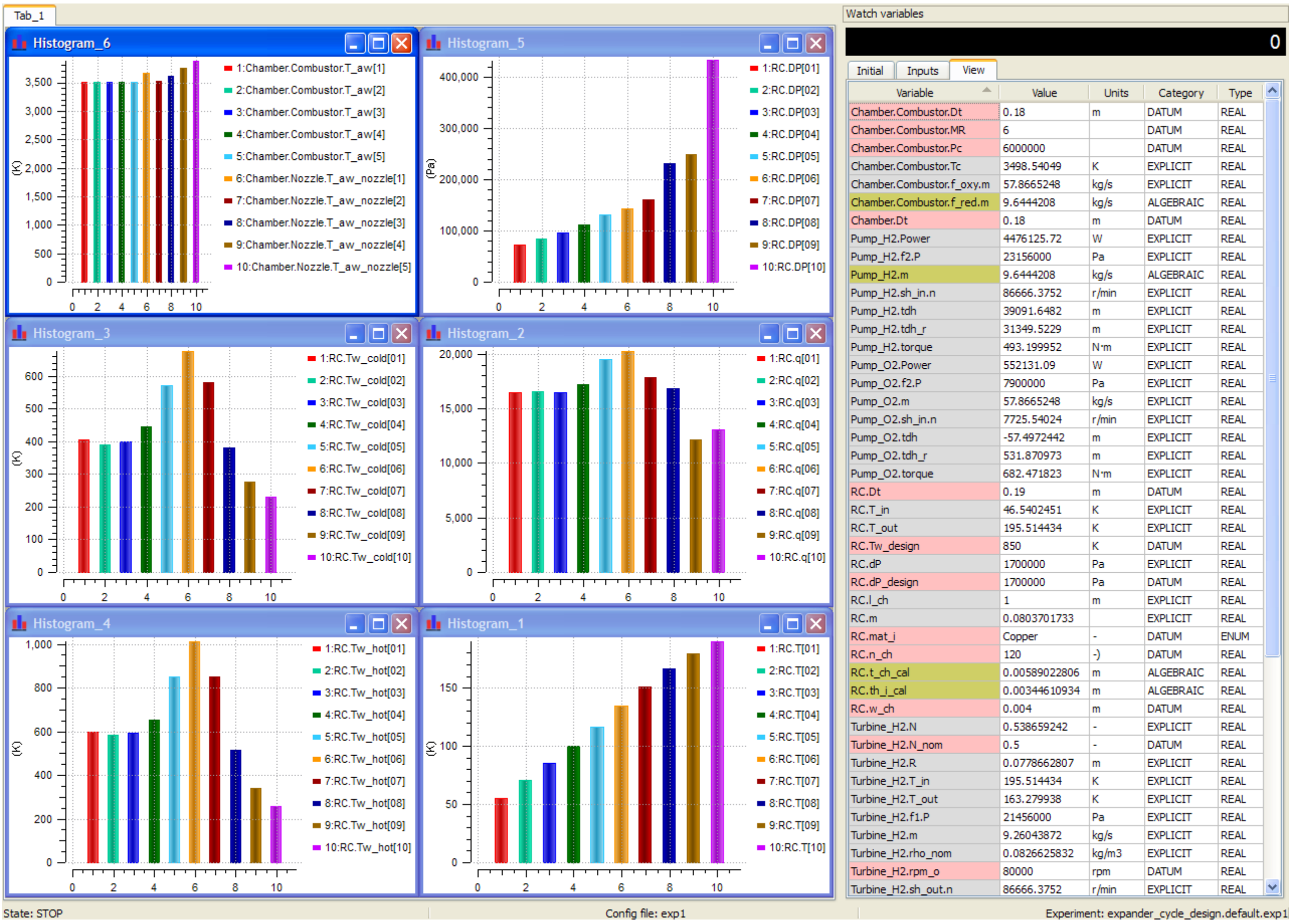
- Press "Generate model" button of the EcosimPro GUI. This compiles the graphical model and generates the default partition. (To be pointed out that in this moment EcosimproPro will internally find an automatic ordering of the whole model equations, searching for the appropriate algebraic variables to be solved).
- Generate default experiment. The INIT block of the experiment can be normally deleted because it is enough with the initial input data entered in the components.

Below the experiment used for the Expander" cycle model. Note that global variables as "m_fu" are used to initialize unknowns in several components:

```
EXPERIMENT exp1 ON expander_cycle_design.default
INIT
    -- all the initial values for algebraics have been deleted
    -- The INT block of components already provided the necessary initialization
BOUNDS
        FLUID_PROPERTIES.MinMolarFr = 1e-010
        STEADY.dp_lam = 3000
        VBPH.s_pos.signal[1] = 1
        VBPO.s_pos.signal[1] = 1
        VCH.s_pos.signal[1] = 1.
        VCO.s_pos.signal[1] = 1.
BODY
        Pc = 60e5
        MR = 6
        m_fu = 10    -- to init iterative calculations
        m_ox = 60    -- to init iterative calculations
        m_tu_ox = 5
        m_tu_fu = 6
        RC.t_ch = 0.006 – initial value for algebr. variable
        RC.th_i = 0.004 –       ".
        -- Below, possible redefinitions of the input data values (Input data control)
        Turbine_H2.PI_tt_o = 2.4
        Turbine_O2.PI_tt_o = 1.2
        Pump_O2.Ns = 10
        VCO.dP_design = 10e5
        VCH.dP_design = 5.5e5

        TIME = 0
        DEBUG_LEVEL = 3
        TOLTYPE= ABSTOL
        STEADY()
    END EXPERIMENT
```

2.5.1.3 Results

The plot of calculated values is presented below:



It can be remarked the following:

- Steady design conditions are reached in a few second of CPU, even though the initial conditions are far from the solution.
- Channel dP is nearly exponential with the axial length because the channels have constant flow area. A more homogeneous pressure drop can be obtained modifying the dimensionless channel widths and heights vs axial position (see the CoolingCircuit input data wc_vs_L and tc_vs_L).

2.5.2 Expander cycle in Off-Design mode (T-STY-002, Design conditions)

Library:	STEADY_EXAMPLES
Model Name:	expander_cycle_offDesign
Partition Name:	design
Experiment Name:	exp1

2.5.2.1 Model description

The Off-design model for the previous test is the "expander_cycle_offDesign. It is the same topology as before but with all the design parameters of the respective components set as Off_Design mode (type = Off_Design, Off_D_pump, Off_D_turb, etc).

Analysis mode needs a different model than the design one because of the design parameters that must be fixed before generating a partition, and *because an analysis model is using maps performance of turbines and pumps*.

2.5.2.2 Design partition experiment

A *design* partition in an *analysis* model is an alternative way calculating the design values of a cycle. With respect to the previous method (default partition in design models), this method has the following advantages:

a) the same model serves for Design and Analysis calculations because in both cases design parameters are set to "OffDesign" value,

b) the design values (operational point entered as input data in an analysis case) can be automatically loaded in an experiment by means of the RESTORE command of the design partition results.

To calculate the design values (operational point), follow the following steps:

- Define a *design* partition in the Analysis model. This is not very complicated and in the end this exercise should be always done to have a good knowledge of the variables being calculated designing the cycle. The rules that should be followed generating the design partition (use of the Design Partition wizard) are:

  a) Set the following Data as unknowns:

  1. Apply the filter "*.A*" to choose any Junction, Valve and injector area as unknown variable
  2. Apply the filters "*.m_o", "*.rpm_o" and "P_o" to choose operational mass flow, nominal speed and pressure rise of Pumps as unknown variables
  3. Apply the filter "*.rpm_o" to choose the operational speed of Turbines as unknown variable
  4. Apply the filter "*.Ain" to choose the characteristic inter-blade flow area of Turbine as unknown variable (only for generic Turbines)
  5. Apply the filter "*.N_nom" to choose the characteristic speed of Turbine as unknown variable (only for map Turbines)

  Note: Data to be converted to unknowns must not have been re-defined with model variables

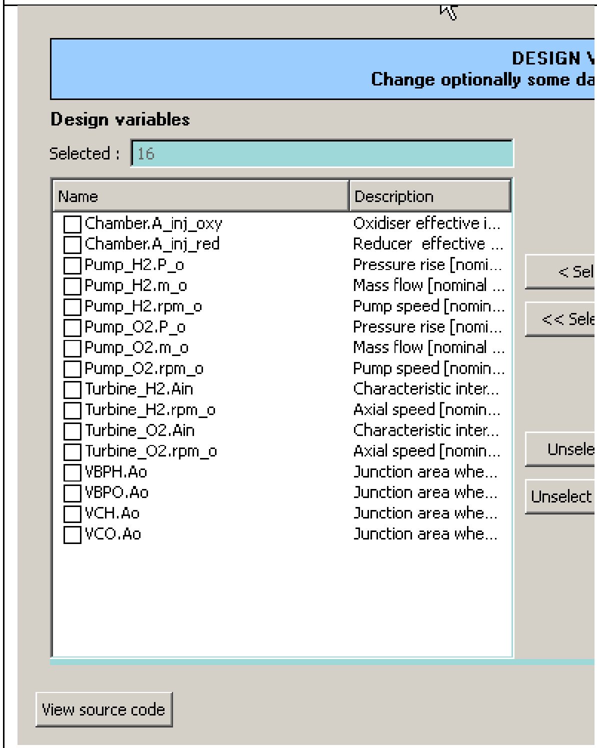
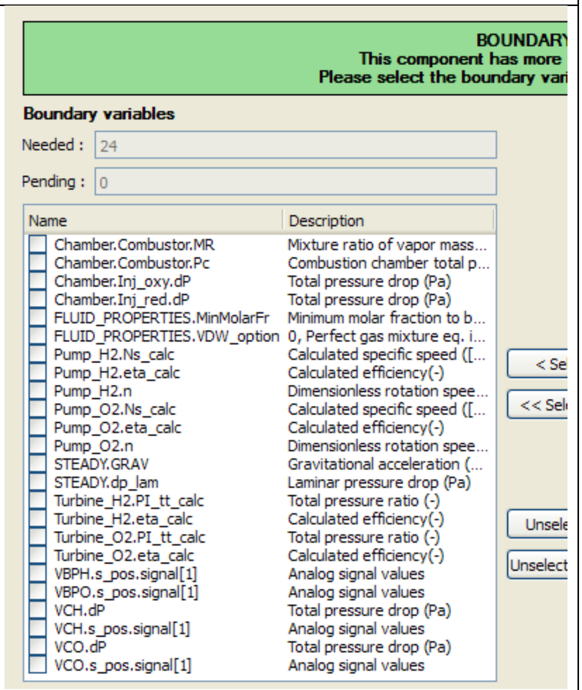
  b) Set the following variables as boundaries:

  1. Apply the filter *.dP to choose Junction, Valve and injector pressure drop as unknown variable (choose only the Junctions that in a design model would be "UserGiven" type)
  2. Apply the filter *.MR to choose imposed combustor mixture ratio (MR)
  3. Apply the filter *.Pc to choose imposed combustor chamber pressure
  4. Apply the filter *.eta_calc to choose imposed efficiency in turbines & pumps
  5. Apply the filter *.PI_tt_calc to choose imposed PI_tt in turbines
  6. Apply the filter *.n to choose imposed Ns in pumps (choose only the Pumps that in Design mode would be "KnowNs" type)
  7. Apply the filter *.signa[* to choose imposed valve positions

  Note: "*_calc" are local model variables with the actual value of the efficiency and pressure ratio depending on the performances map. Fixing to a value these variables is equivalent to fix the operational point in a particular point of the performances map.

  c) Select the proposed algebraic variables by the assistant wizard breaking the nonlinear boxes.

  Below a printout of the design partition wizards for the present case:

Input to be calculated	Associated boundaries
	

- Generate default experiment. The INIT block of the experiment can be normally deleted because it is enough with the initial input data entered in the components (a few set of pressures and mass flows). Below the experiment used for this model:

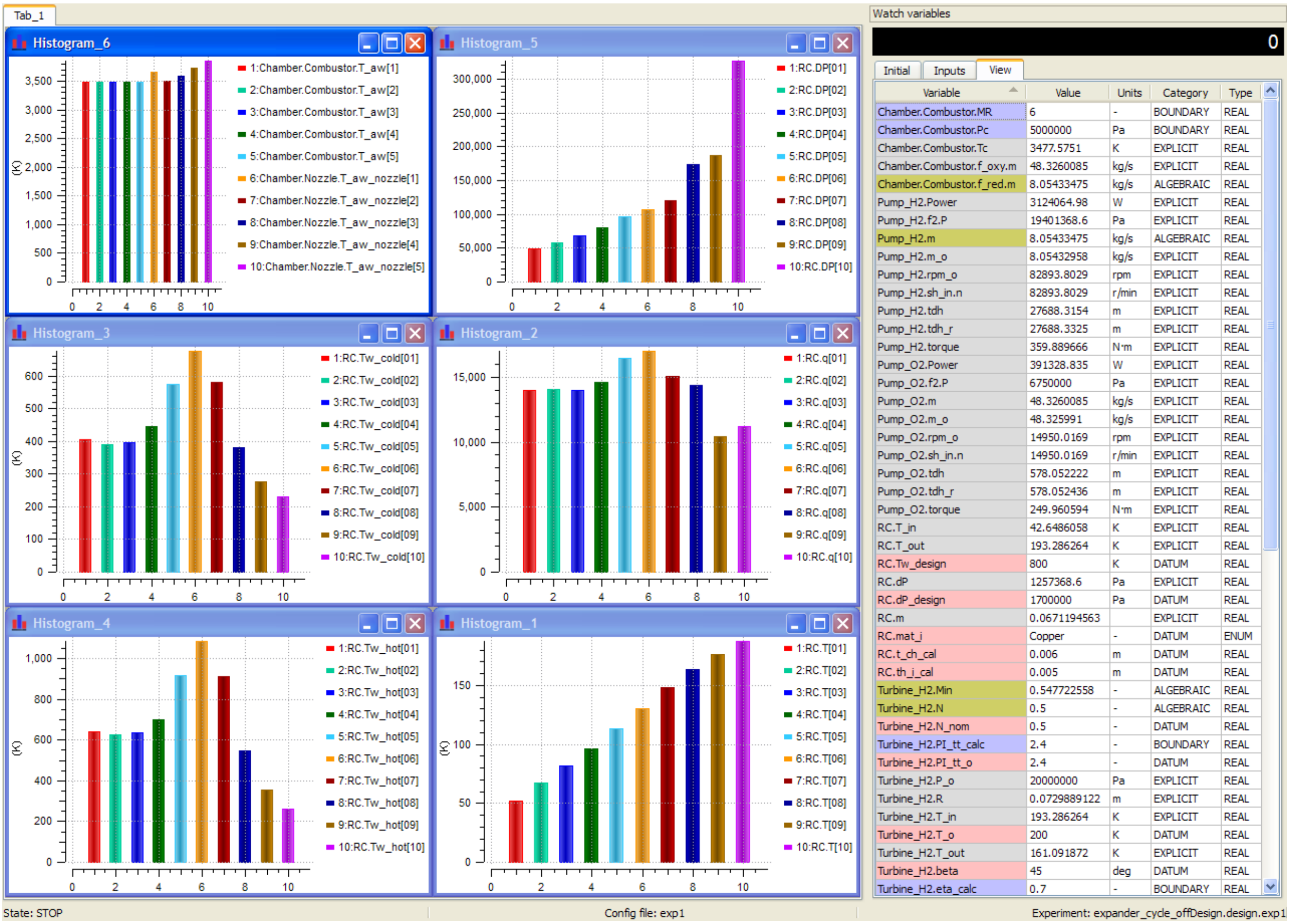
```
EXPERIMENT exp1 ON expander_cycle_offDesign.design
INIT
    -- all the initial values for algebraics have been deleted
    -- The INT block of components already provided the necessary initialization
BOUNDS
    -- Set equations for boundaries: boundVar = f(TIME;...)
    Chamber.Combustor.MR = MR
    Chamber.Combustor.Pc = Pc
    Chamber.Inj_oxy.dP = 7.5e5
    Chamber.Inj_red.dP = 7.5e5
    VCH.dP = 5.5e5
    VCO.dP = 10e5
    Pump_H2.Ns_calc = Pump_H2.Ns
    Pump_H2.eta_calc = 0.7
    Pump_H2.n = 1
    Pump_O2.Ns_calc = Pump_O2.Ns
    Pump_O2.eta_calc = 0.7
    Pump_O2.n = 1
    Turbine_H2.PI_tt_calc = Turbine_H2.PI_tt_o
    Turbine_H2.eta_calc = 0.7
    Turbine_O2.PI_tt_calc = Turbine_O2.PI_tt_o
    Turbine_O2.eta_calc = 0.7

    VBPH.s_pos.signal[1] = 1
    VBPO.s_pos.signal[1] = 1
    VCH.s_pos.signal[1] = 1
    VCO.s_pos.signal[1] = 1
    STEADY.GRAV = 9.806
    STEADY.dp_lam = 3000
    FLUID_PROPERTIES.MinMolarFr = 1e-008
```

```
BODY
    -- Below, possible redefinitions of the input data values (Input data control)
    Pc = 50e5
    MR = 6
    RC.t_ch = 0.006 -.
    RC.th_i = 0.005 -.
    RC.mat_i = Copper -- SS_321 --
    RC.dP_design = 17e5  -- not used, fixed channel area
    RC.Tw_throat = 800 -- not used in this case
    RC.n_ch = 120 --300
    RC.w_ch = 0.004 -- 0.004
    ...
    TIME = 0
    DEBUG_LEVEL = 3
    TOLTYPE= ABSTOL -- FRACTOL --
    STEADY()
    SAVE_STATE("design")
END EXPERIMENT
```

The SAVE_STATE command will produce an ASCII file in the experiment folder with the variables values.

2.5.2.3 Results

The plot of calculated values is presented below:



Calculated operational values are saved with the same name as the one used for the input data: for example, valve areas are saved in the *.Ao variables (input data), calculated nominal axial speeds are saved in *.rpm_o variables, etc. The local variables *.A, sh_in.n, etc. will have same values as the nominal ones for the design partition.

It can be remarked the following:

- As before, the steady design conditions are reached in a few second of CPU, even though the initial conditions are far from the solution.
- The design conditions are now slightly different than before because the design partition didn't impose a particular pressure drop in the cooling circuit, nor a wall temperature.
- The map performances and operational values are correctly calculated. For example, it can be observed that "rpm_o" value is equal to "sh_in.n" (current value), "N" is equal to N_nom, etc

2.5.3 Expander cycle in Off-Design mode (T-STY-003, Analysis case)

Library:	STEADY_EXAMPLES
Model Name:	expander_cycle_offDesign
Partition Name:	default
Experiment Name:	exp1

2.5.3.1 Default Partition experiment

Default partition experiments in an off-design model will calculate the off design operation of a cycle. As mentioned before, a design partition must be performed before over the same model.

The following rules apply generating experiments for off design analysis:

- Generate the *default* partition in the Analysis model. The default partition is automatically generated by EcosimPro that will internally find the ordering of the whole model equations, searching for the appropriate algebraic variables to be solved
- Generate default experiment. The INIT block of the experiment can be deleted. It is normally enough with the initial input data entered in the components (a few set of pressures and mass flows).

Below the experiment used to calculate the Expander" cycle off design behavior opening and closing the VCH/VCO valves:

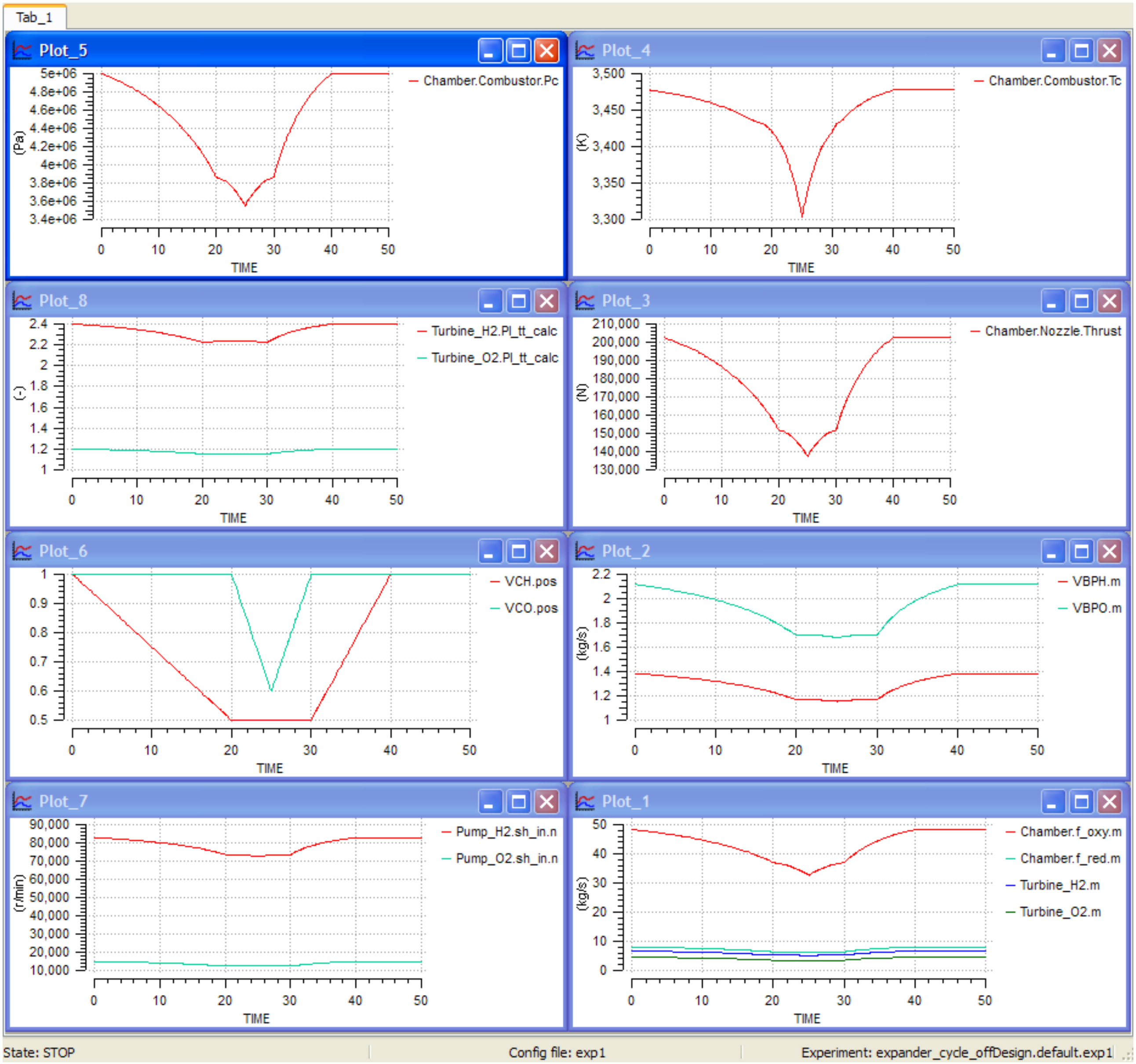
```
EXPERIMENT exp1 ON expander_cycle_offDesign.default
DECLS
    -- opening / closing law for VCH valve
    TABLE_1D law1 = { {0, 20, 30, 40, 50} ,     {1,0.5,0.5, 1, 1} }
    -- opening / closing law for VCO valve
    TABLE_1D law2 = { {0,20, 25, 30, 50} ,     {1, 1,0.6, 1, 1} }
INIT
    -- initial values for algebraics (not needed)
BOUNDS
    -- Set equations for boundaries: boundVar = f(TIME;...)
    FLUID_PROPERTIES.MinMolarFr = 1e-008
    STEADY.GRAV = 9.806
    STEADY.dp_lam = 3000
    VBPH.s_pos.signal[1] = 1
    VBPO.s_pos.signal[1] = 1
    VCH.s_pos.signal[1] = timeTableInterp(TIME,law1)
    VCO.s_pos.signal[1] = timeTableInterp(TIME,law2)
BODY
    -- restore a previous state
    SET_INIT_ACTIVE(FALSE)    --deactivateINIT blocks because the complete stae will be read from a file
    RESTORE_STATE("@STEADY_EXAMPLES@/experiments/expander_cycle_off+design.design/exp1/design")
```

```
        TSTOP = 50
        TIME = 0
        CINT = 1
        REPORT_MODE = IS_STEP
        REL_ERROR = 1e-5
        ABS_ERROR = 1e-5
        INTEG()
    END EXPERIMENT
```

Note how the design parameters have been loaded using the RESTORE_STATE command from the design partition experiment.

2.5.3.2 Results

Calculated transient is:



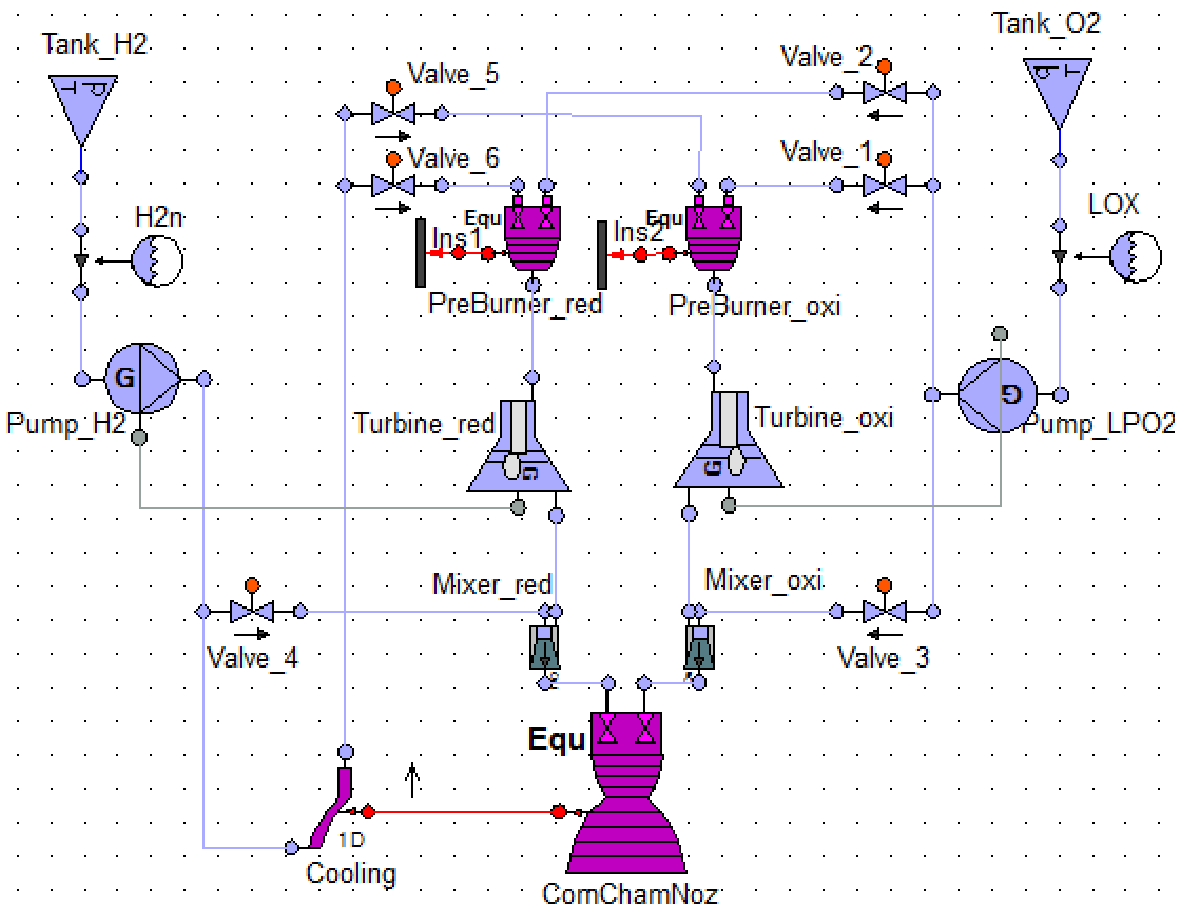
2.5.4 Staged cycle in Design mode (T-STY-004)

Library:	STEADY_EXAMPLES
Model Name:	ssme
Partition Name:	default
Experiment Name:	exp1

2.5.4.1 Default Partition experiment

The "ProdPF_Mixer" permits the simulation of a mixture of combusted gases and pure fluids. A combustion chamber with more than 2 injectors can be then modelled placing this component upstream one of the chamber injector and collecting two flows, one, for example, feeding combusted gases coming from a pre-burner and another feeding a pure fluid coming from a pump.

The following example ("ssme" of the STEADY_EXAMPLES library) shows a model of a simplified SSME engine with a main chamber of 4 injectors ("ProdPF_Mixer" component):

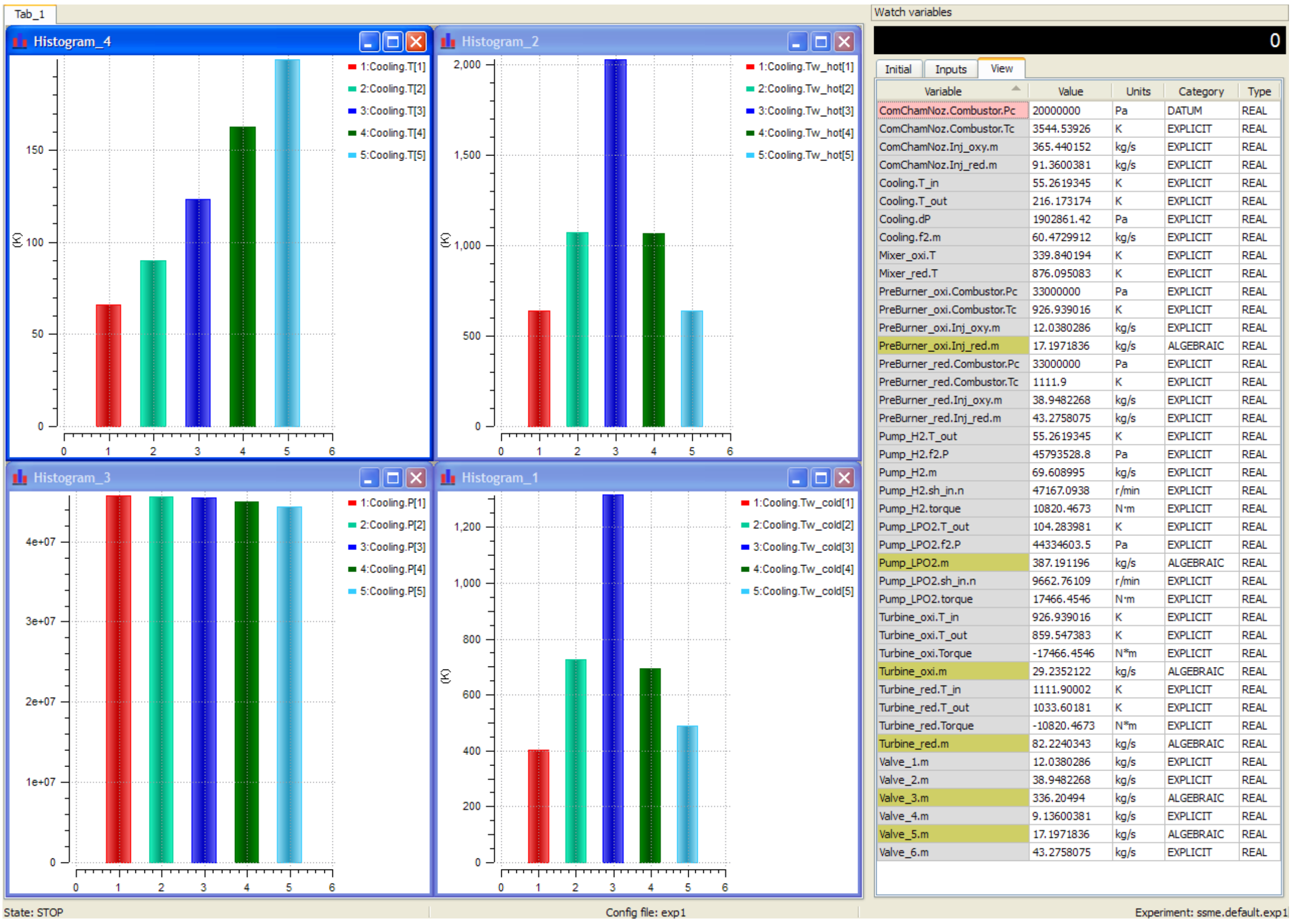


Below it is showed the experiment of this model:

```
EXPERIMENT exp1 ON ssme.default

DECLS
INIT
BOUNDS
    -- Set equations for boundaries: boundVar = f(TIME;...)
        FLUID_PROPERTIES.MinMolarFr = 1e-008
```

```
        STEADY.GRAV = 9.806
        STEADY.dp_lam = 3000
        Valve_1.s_pos.signal[1] = 1
        Valve_2.s_pos.signal[1] = 1
        Valve_3.s_pos.signal[1] = 1
        Valve_4.s_pos.signal[1] = 1
        Valve_5.s_pos.signal[1] = 1
        Valve_6.s_pos.signal[1] = 1
        -- Valve_7.s_pos.signal[1] = 1
        -- Valve_8.s_pos.signal[1] = 1
BODY
        Cooling.Dt = 0.3
        Cooling.n_ch = 200
        Cooling.t_ch = 0.015
        Cooling.f1.m = 30

        PreBurner_oxi.MR_o = 0.7
        PreBurner_red.MR_o = 0.9
        ComChamNoz.MR_o = 4 -- 4.5
        Mixer_red.mflow_ratio = 0.1 -- 0.2
        Mixer_oxi.mflow_ratio = 0.92

        DEBUG_LEVEL = 3
        STEADY()

END EXPERIMENT
```

Results can be seen below:



2.6 SATELLITE LIBRARY

2.6.1 Function JD test (T-SAT-001)

Library: SATELLITE
Model Name: testJD
Partition Name: default
Experiment Name: expDefault

2.6.1.1 Model description

A simple model has been designed to use the function "JulianDay" computing the Julian Day number.

2.6.1.2 Results

The Reference of PROGRAM PLANEPH 4.1, "G. Francou, J. Chapront, Bureau des Longitudes - France, Group : Dynamics of Solar System, December 1996", provides two values of JD

Beginning : Jan. 1 1900 0h (JD2415020.5).
End : Jan. 1 2099 0h (JD2487704.5).

The function of the Satellite library "JulianDay" confirms those values

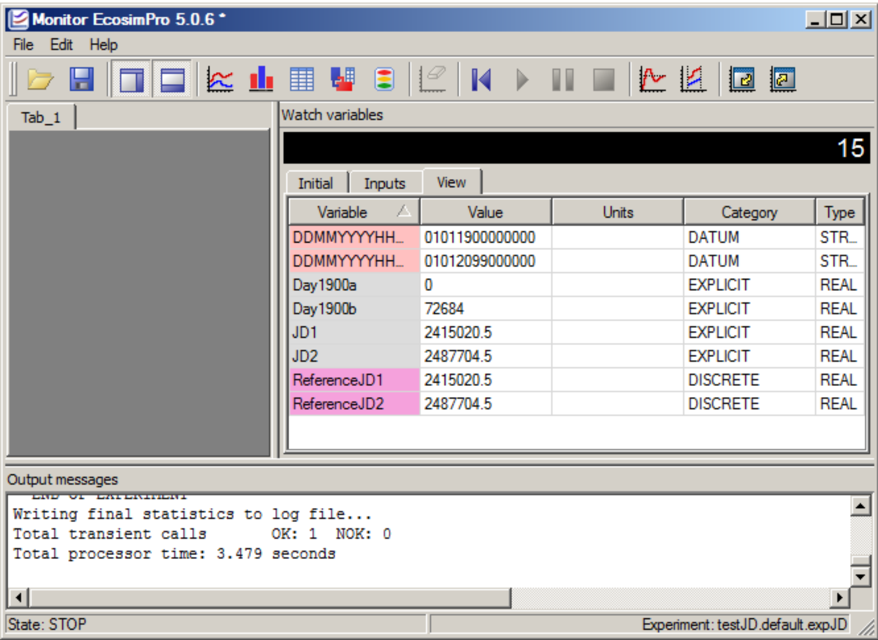


Figure 4 Output of Julian Days computations

In conclusion the simulation provides corrects results.

2.6.2 Function MoonSunECI test (T-SAT-002)

Library: SATELLITE
Model Name: testMoon
Partition Name: default
Experiment Name: expGEO

2.6.2.1 Model description

A simple model has been designed to use the function "MoonSunECI" computing the location of the Moon and Sun. This function is used in the component frame, so the model comprises only the component Frame.

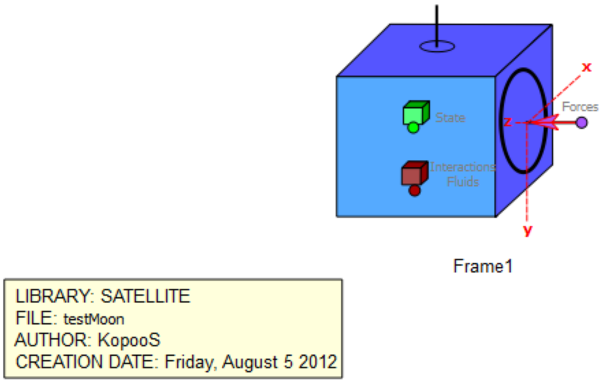


Figure 5 Model for test of Moon Sun orbit and perturbations induced

2.6.2.2 Results

The locations of Sun and Moon with respect to the Earth in the ecliptic geocentric frame is given by simple routine function of the date. Further the locations are set into the equatorial ECI frame.

The output of the function "MoonSunECI" is given in the figure below for 364 days, starting at equinox 2011 (21 March).

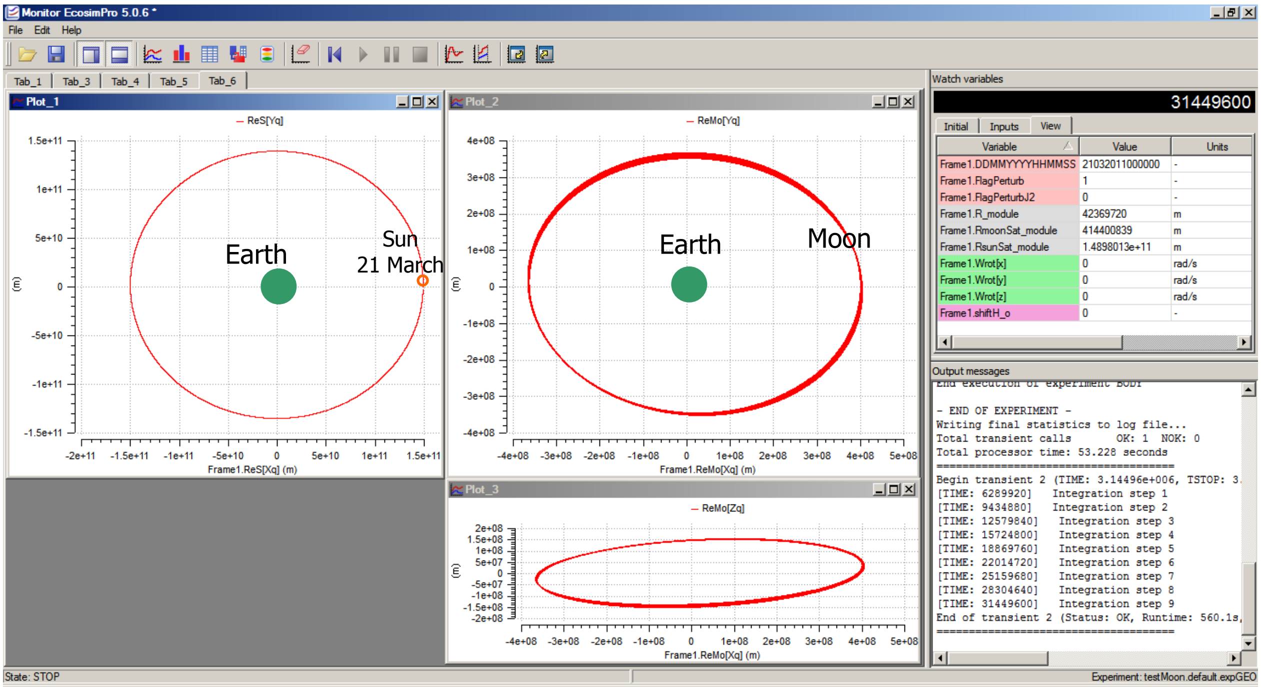


Figure 6 Sun and Moon orbits in the Frame ECI equatorial

The location of the Sun on the 21 March is at 1 AU along the X axis of ECI as expected. The orbit is quasi circular as expected. The Moon orbit is also quasi circular and inclined with respect to the equator.
In conclusion the simulation provides corrects results.

2.6.3 Moon Sun perturbations test (T-SAT-003)

Library:	SATELLITE
Model Name:	testMoon
Partition Name:	default
Experiment Name:	expGEO

2.6.3.1 Model description

Idem T-SAT-002

2.6.3.2 Results

The well-known effect on a GEO orbit coming from the perturbation from Moon and Sun is an increase of the inclination by about 1°/year.

This is confirmed with the inclination in degrees (parameter incd in the figure below). Moreover, the node axis or RAAN in those conditions does not vary too much around +90°.

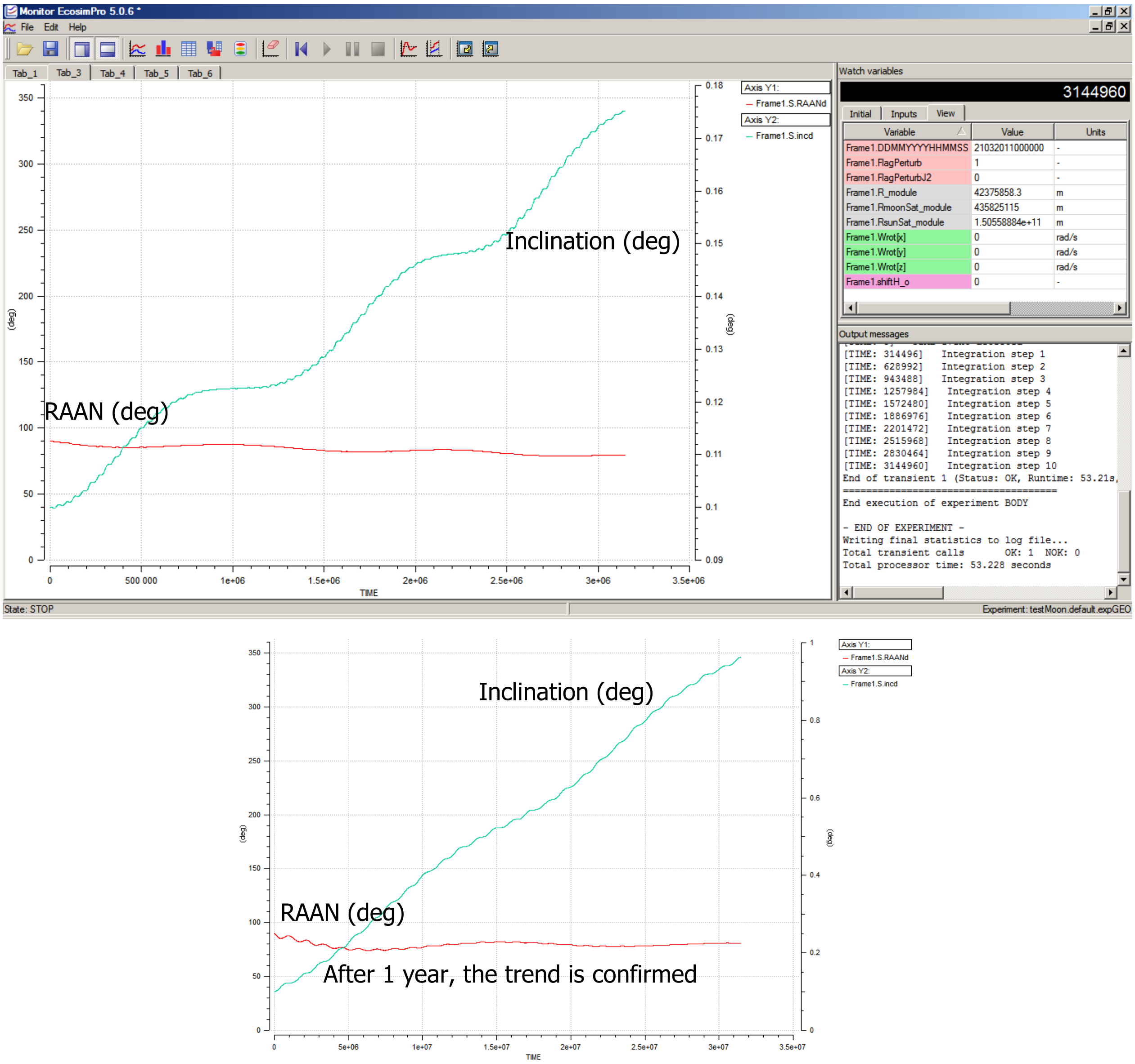


Figure 7 Moon and Sun perturbations on a GEO satellite (inclination increase +1° in 1 year)

In conclusion the simulation provides corrects results.

2.6.4 Earth flatness (J2) perturbations test (T-SAT-004)

Library: SATELLITE
Model Name: testSat
Partition Name: default
Experiment Name: exp_spot

2.6.4.1 Model description

A generic model of satellite with all the components of the library SATELLITE is used.

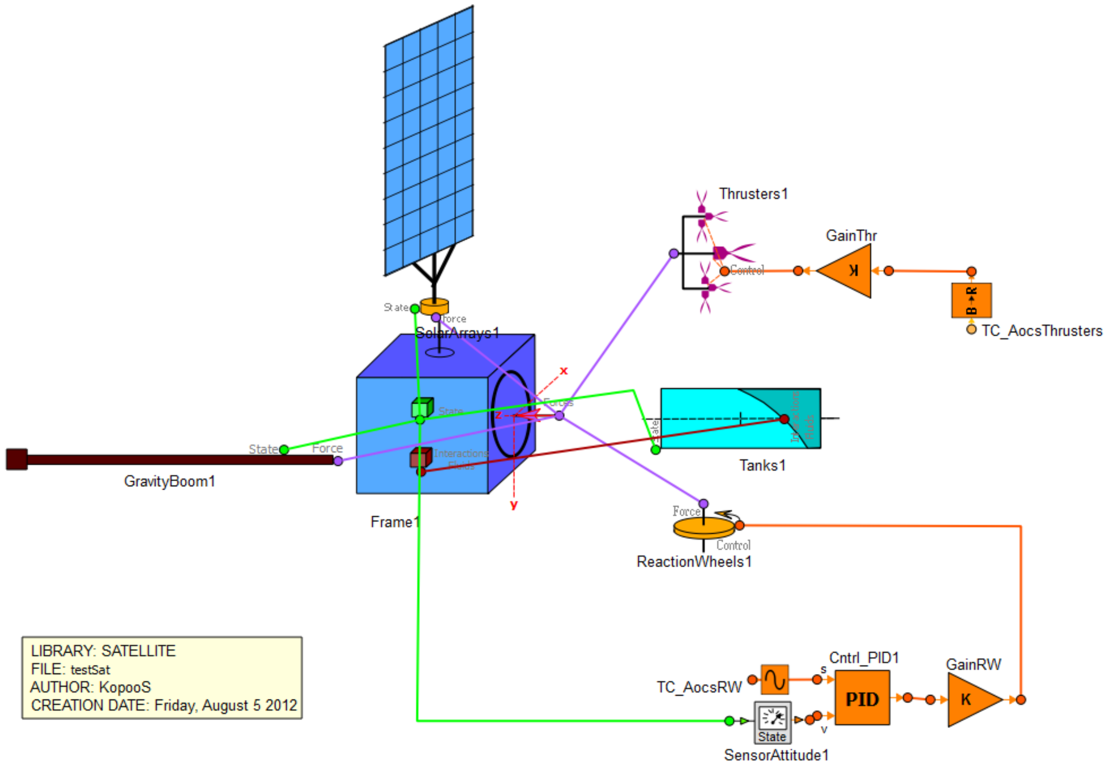
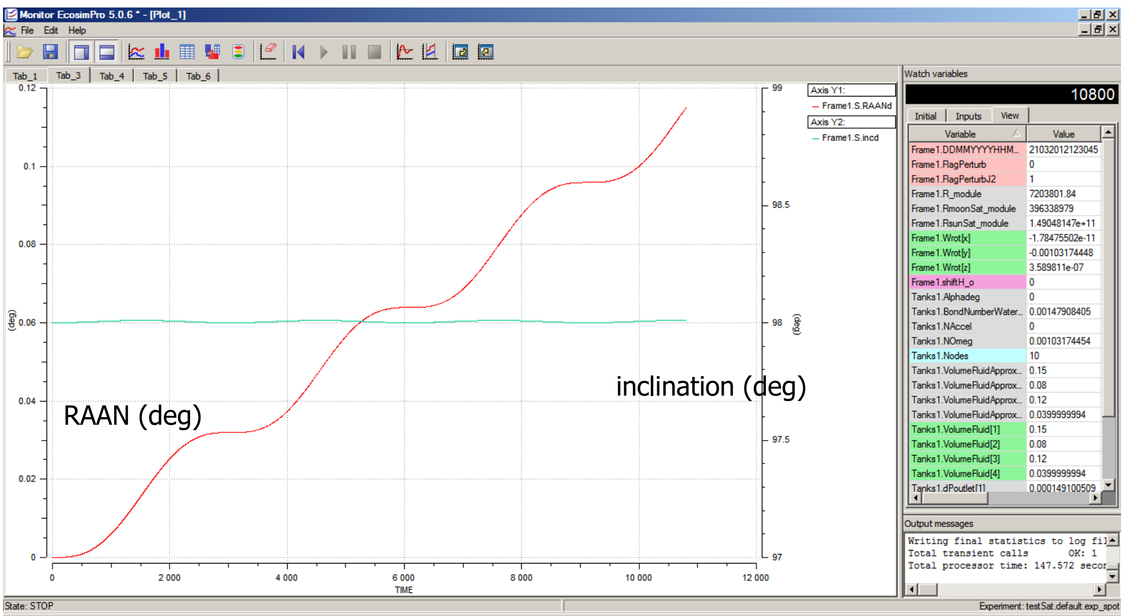


Figure 8 Generic Satellite Model

2.6.4.2 Results

For a low earth orbit (830 km altitude) inclined at 98°, the major behaviour is the heliosynchronism: that means that the RAAN is growing at the rate of about +1° per day in order to always keep the same configuration with respect to the Sun. Moreover, the inclination in those conditions does not vary too much around 98°.

This behaviour is correctly represented in the results of the simulation.



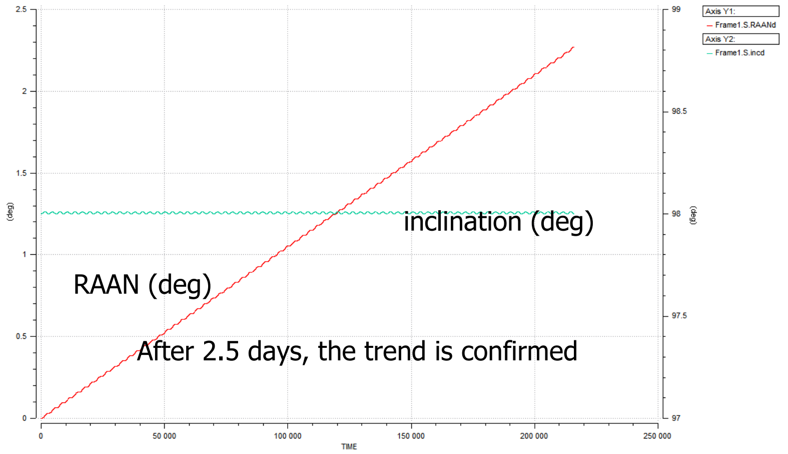


Figure 9 Heliosynchronism with the J2 effect (inclination 98°; altitude 830 km)

In conclusion the simulation provides corrects results.

2.6.5 Limit values Inclination Null(T-SAT-005)

Library: SATELLITE
Model Name: testSat
Partition Name: default
Experiment Name: exp_geoInc0

2.6.5.1 Model description

This example use the same model as above.

2.6.5.2 Results

The tool accepts a very small inclination for the initial orbit (any values >0.001°). The value 0 is not always accepted.

2.6.6 Orbit manoeuvre (T-SAT-006)

Library: SATELLITE
Model Name: testSat
Partition Name: default
Experiment Name: exp_geoNS

2.6.6.1 Model description

This example use the same model as above.

2.6.6.2 Results

The reference case is given by a freeware "TriaXOrbitaL" available on the web. Thus the initial mass, the thruster performances and the timing of the maneuvers is provided by this reference. The initial orbit with the trajectory followed by the satellite after the thrust periods is shown in the output plot of the reference (J2, Moon-Sun perturbations deactivated)

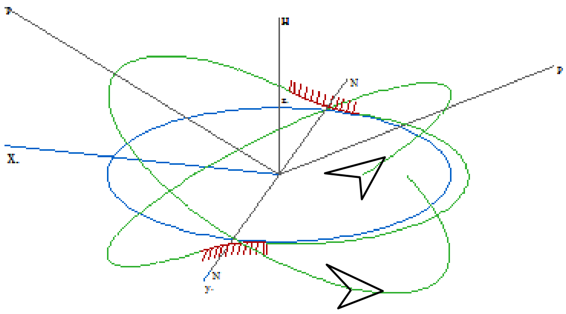
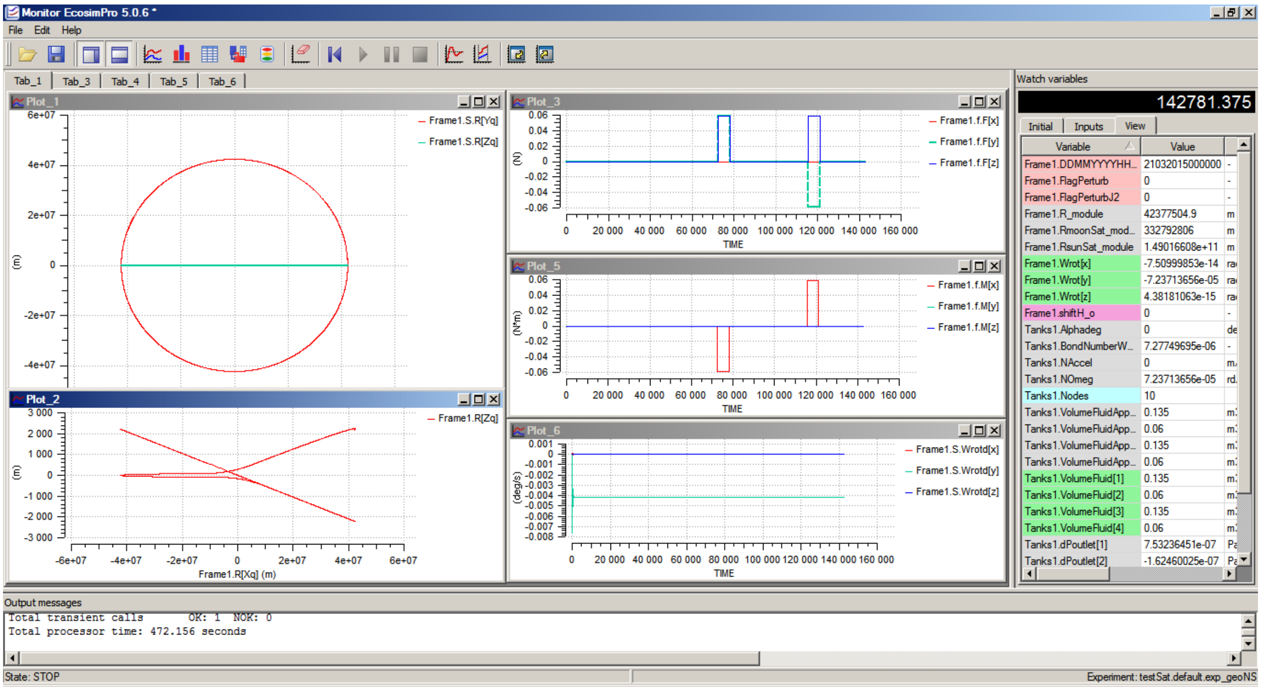


Figure 10 North-South station keeping with Reference Tool **(altitudes with respect to GEO +-20 km with zoom amplification in inclination)**

The results on the Monitor EcosimPro are comparable to the reference according to the following plot.



Note : after each thrust pulse, at the transition time where the thrust is turned to zero, the tool report a safety message because during the transient it found a vector null for the acceleration « *ERROR Normalise NULL vector for in Tanks eAvalid at Time=78155.7 X=0* » and « *ERROR Normalise NULL vector for in Tanks eAvalid at Time=121240.5 X=0* ». Those two messages are coming from to the tolerances values set in some ZONE statements to catch a null acceleration case. It has been checked that the vector *eAvalid is* used solely for making a unit vector *eA*, and that the unit *eA* is really a unit vector after the transient according to the management set by the ZONE statements. In order to keep the advantage of relevant messages in case of very wrong behaviour of the tool, this safety messages has not been removed.

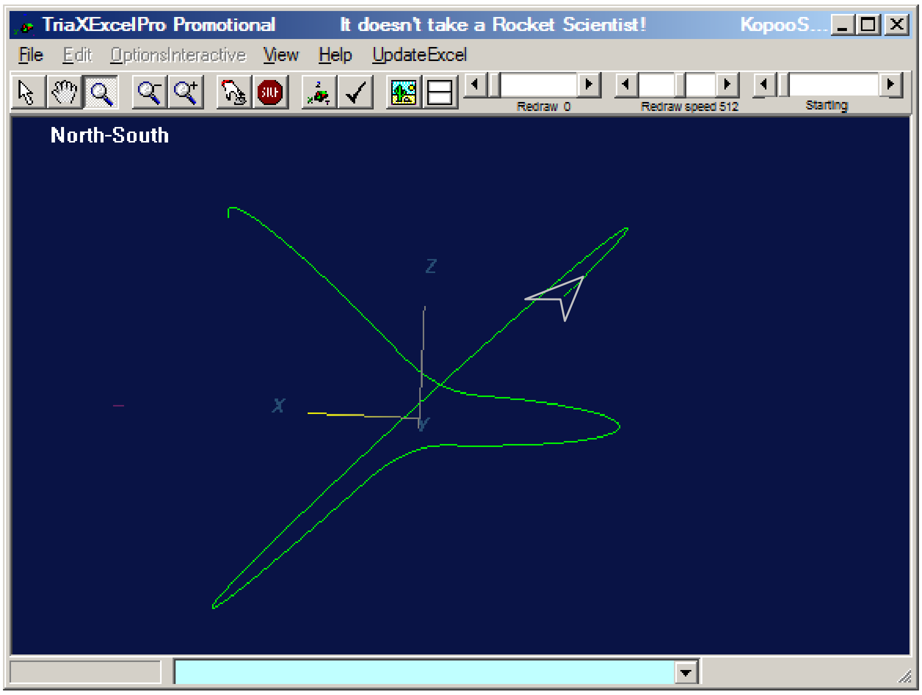


Figure 11 North-South station keeping with Satellite Library and 3D tool view

In conclusion the simulation provides corrects results.

2.6.7 Archimedes pressure under force (T-SAT-007)

Library: SATELLITE
Model Name: satCheckArchimedes
Partition Name: default
Experiment Name: exp1ForceZmX

2.6.7.1 Model description

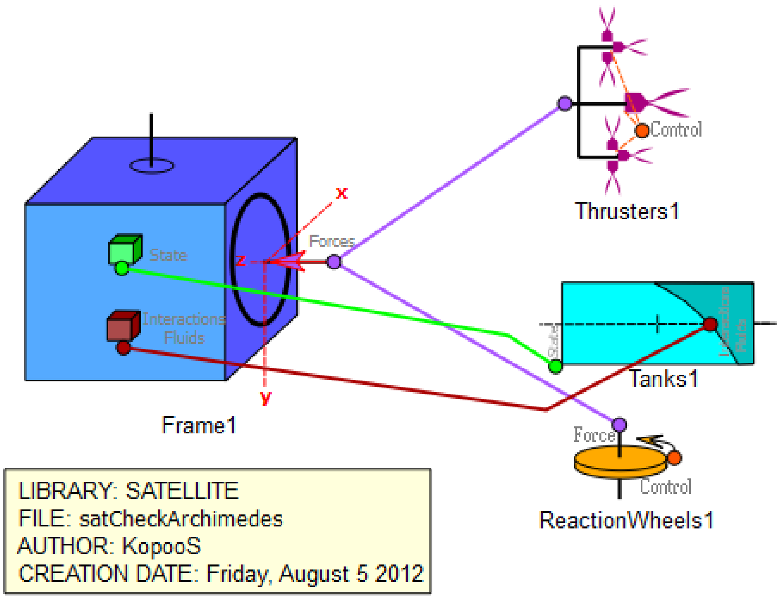


Figure 12 Model for tests of Archimedes pressure in the tank

2.6.7.2 Results

The thrust is inclined at 45° in the plane Z X (positive along Z, negative along X) and acting directly at the centre of mass. The expected shape of the free surface is thus a flat surface also inclined at 45°. In order to have a fixed COM, the mass change is avoided by setting a very high Isp for the thruster (1e5 s).

The tank n°1 with 40 litres of liquid into a total volume of 196.35 litres is along the Z axis and its centre is located at 0,1,1 in the frame X,Y,Z.

It is not obvious to check the shape of the surface in the Monitor. Thus a 3D view using the data saved in the report has been used. The shape of the surface is a flat surface inclined at 45° as expected. To be noted: the free surface is rectangular (even square) due to the method of discretisation of the cylindrical volume by a rectangular box of same volume and length.

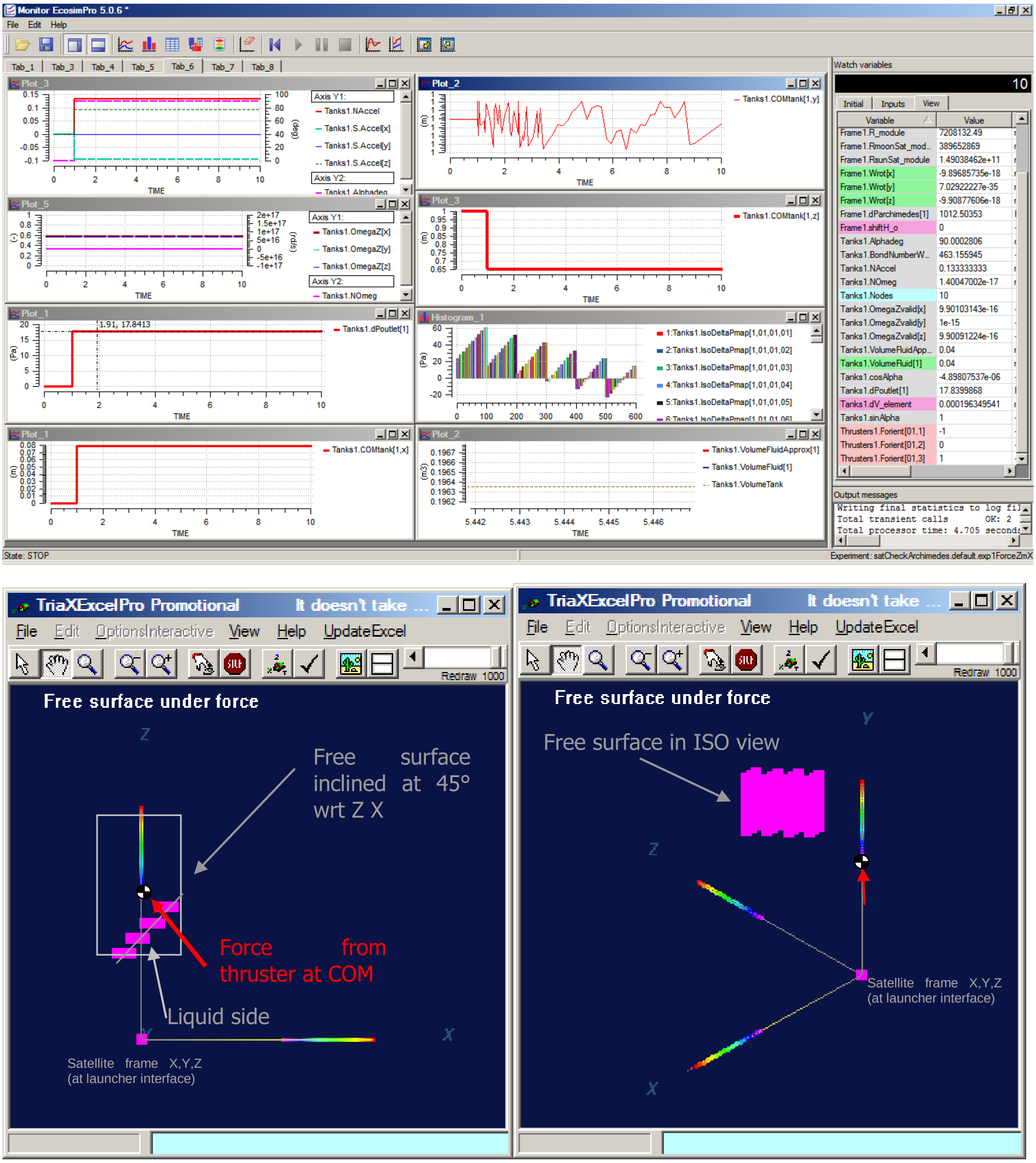


Figure 13 Output of the Monitor and 3D views of the free surface in the tank

The pressure increase due to Archimedes at the outlet of the tank is plotted below: for the tank n° 1 the deltaP=17.84 Pa.

With the acceleration of 0.133 m/s² (=3000 kg/400N), a density of 1000 kg/m^3, the Archimedes pressure being "rho.gamma.height" induces a height above the tank outlet= 0.1338 m which is the right order of magnitude for a tank of 0.5 m diameter and 1 m long with only 40 litres of fluid (without acceleration the height would be 0.203 m, but with inclined acceleration the height is lower because only a part of the base is wetted by the fluid).

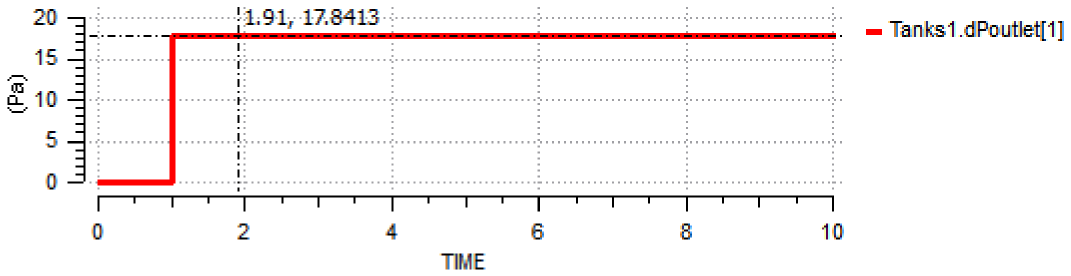


Figure 14 Pressure increase due to Archimedes at the tanks outlets

In addition to this test, the Frame includes a feature to compute the Archimedes pressure coming from each tank for one given point.

The point set in the experiment is along the axis of the tank n°1, and at location -10 m along Z. That is a coordinate with respect to the free surface of about 0.1338+0.5+10=10.6338 m where 0.5 m is the distance between the centre of the satellite axis (at launcher interface) and the outlet of the tank.

The result given by the simulation is "Frame1.dParchimedes[1]=1012.5 Pa". Once again this deltaP is equivalent to a height orthogonal to the free surface of 7.5938 m =1012.5Pa/(1000kg/m^3*0.133m/s^2). With a free surface inclined by 45°, the z coordinate is 7.5938/cos (45°) = 10.739 m which confirm (at +-1%) the forecasted value.

In conclusion the simulation provides corrects results.

2.6.8 Archimedes pressure under force and rotation (T-SAT-008)

Library:	SATELLITE
Model Name:	satCheckArchimedes
Partition Name:	default
Experiment Name:	exp1ForceZmXrotationZ

2.6.8.1 Model description

Idem previous model

2.6.8.2 Results

The thrust, applied at time 1 s, is inclined at 45° in the plane Z X (positive along Z, negative along X) and acting directly at the centre of mass. In addition 0.5 seconds after the thrust, a strong rotation occurs around Z axis. The expected shape of the free surface is more driven by the rotation and centrifugal force than by the force from the thruster, thus a curved surface is expected. In order to have a fixed COM, the mass change is avoided by setting a infinite Isp for the thruster (300*1e10 s).

The tank n°1 with 40 litres of liquid into a total volume of 196.35 litres is along the Z axis and its centre is located at 0,1,1 in the frame X,Y,Z.

It is not obvious to check the shape of the surface in the Monitor. Thus a 3D view using the data saved in the report has been used. The shape of the surface is a flat surface inclined at 45° as expected. To be noted: the free surface is rectangular (even square) due to the method of discretisation of the cylindrical volume by a rectangular box of same volume and length.

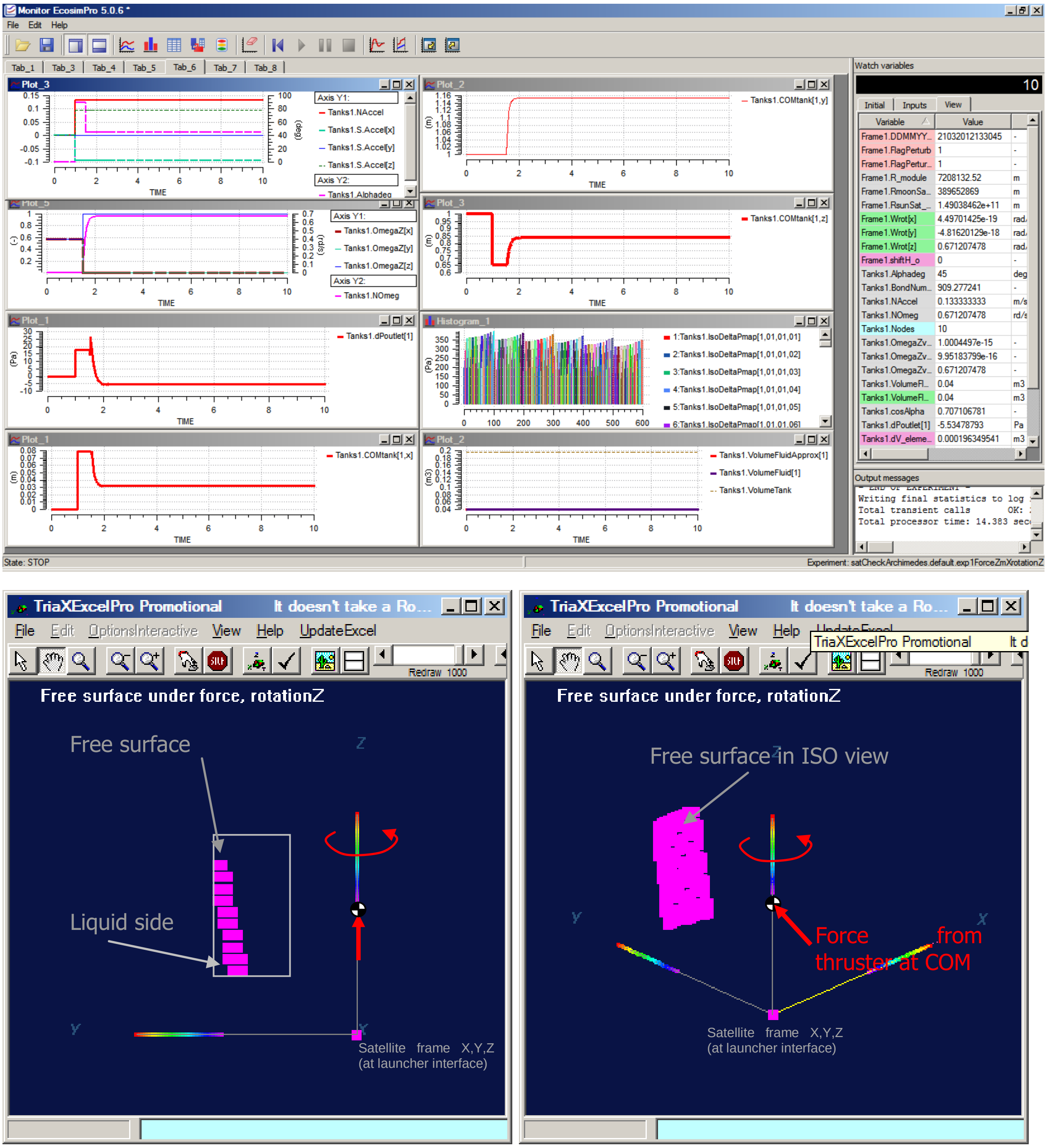


Figure 15 Output of the Monitor and 3D views of the free surface in the tank

For this case the outlet of the tank n°1 is no more covered by the liquid. The Archimedes pressure is thus negative.

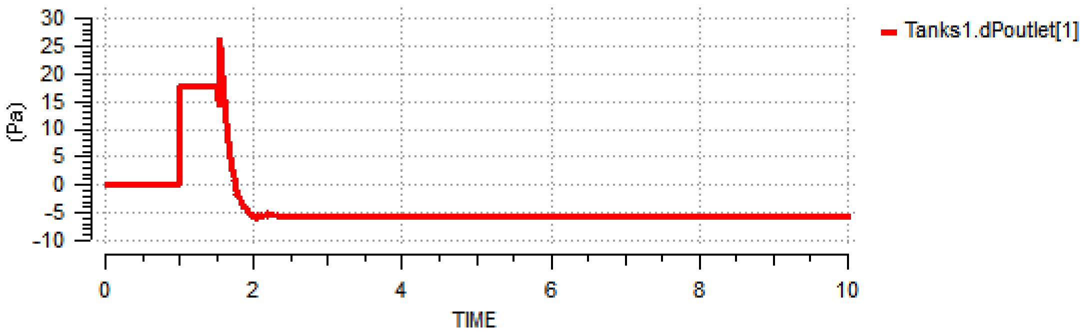


Figure 16 Pressure decrease due to Archimedes under rotation at the tanks outlets

In conclusion the simulation provides corrects results.

2.6.9 Flight dynamic with force (T-SAT-009)

Library: SATELLITE
Model Name: satCheckFlightDynamic
Partition Name: default
Experiment Name: expThrust

2.6.9.1 Model description

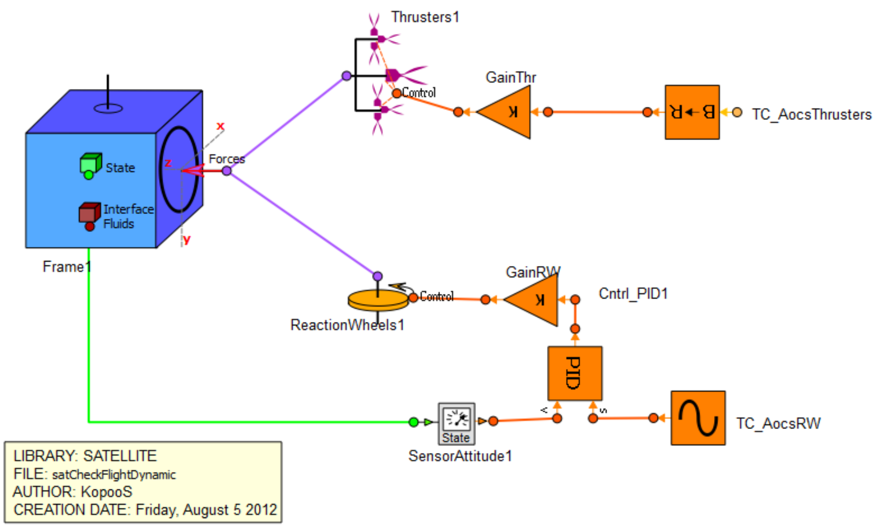


Figure 17 Model for tests of flight dynamic

2.6.9.2 Results

The well-known Edelbaum relation for continuous thrust spiral orbit –without inclination change—states simply that the delta V equal the difference between initial circular velocity and final circular orbital velocity.

$$\Delta V = V_{init} - V_{final}$$

The results of the simulation are shown below.

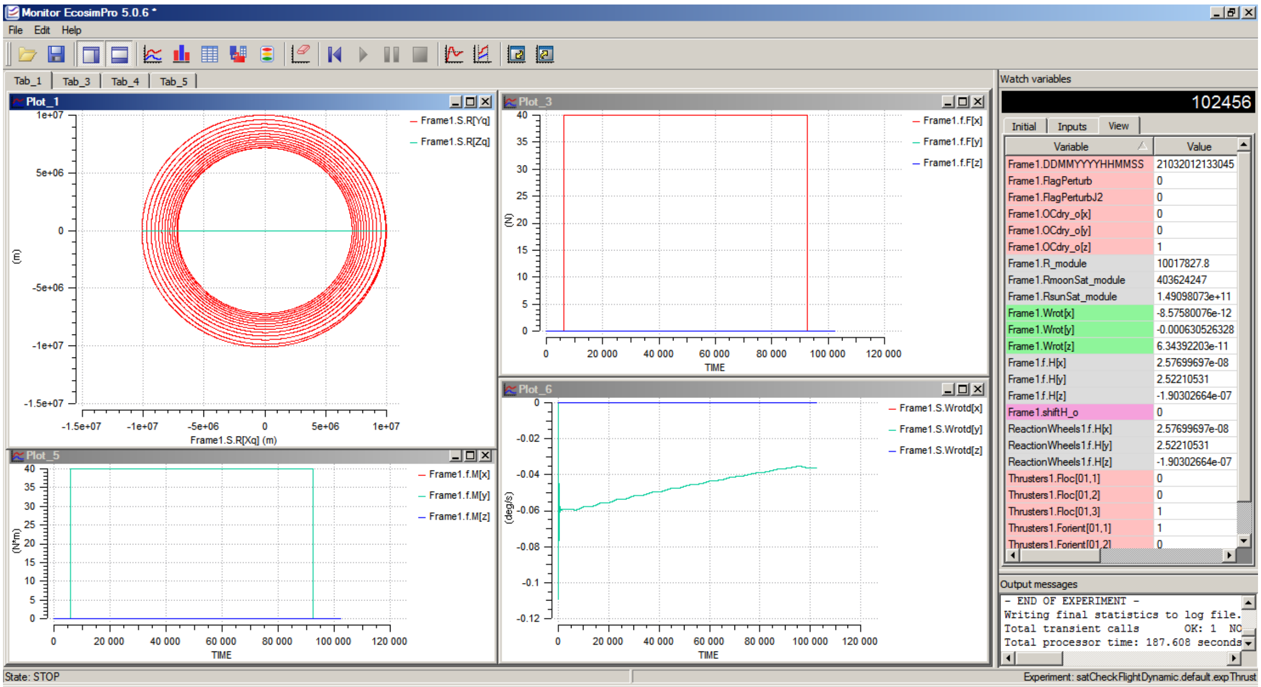


Figure 18 Results of the simulation during a continuous spiral thrust

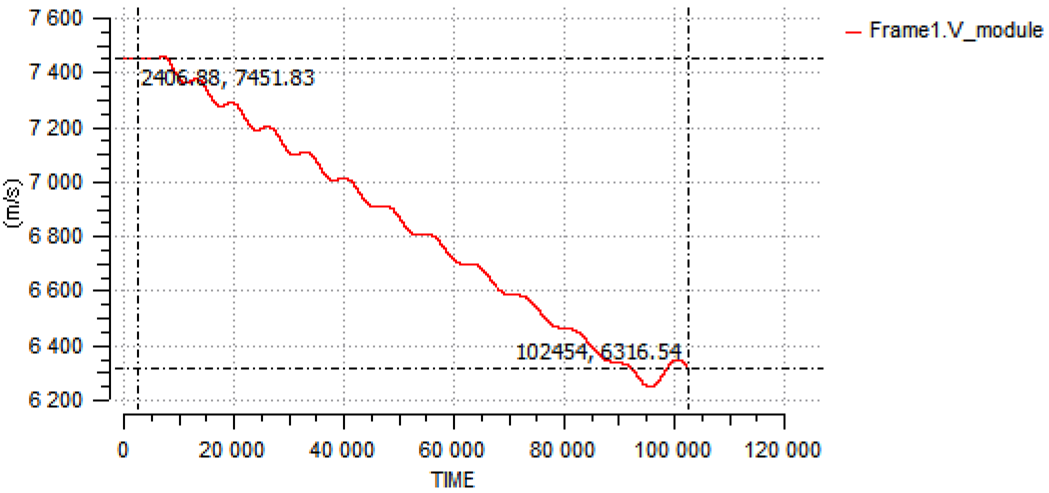


Figure 19 Orbital velocity during a continuous spiral thrust

The difference in orbital velocities modules is 7451.83-6316.54=1135.43 m/s by Edelbaum

The mass evolution of the satellite is 3000-2996.48=3.52 kg for a thruster specific impulse of 100 000 s

Thus the delta V can also be checked with the delta V equation

$$\Delta V = go.Isp.LN(M_{init} / M_{final})$$

That gives delta V=1151 m/s which is very near (1.35%) of the Edelbaum value 1135.43 m/s.

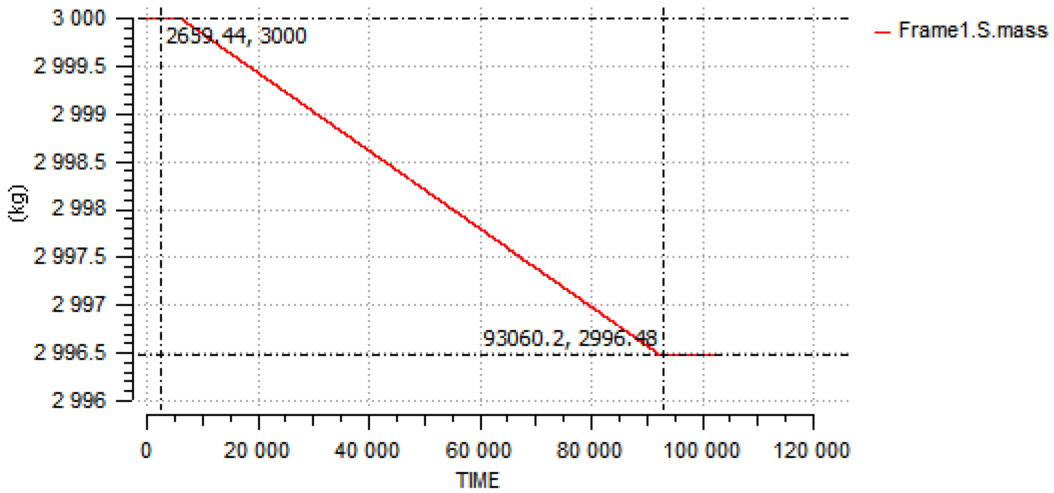


Figure 20 Mass during a continuous spiral thrust

In conclusion the simulation provides corrects results.

2.6.10 Limit values Eccentricity Null(T-SAT-010)

Library:	SATELLITE
Model Name:	testSat
Partition Name:	default
Experiment Name:	exp_geoExc0

2.6.10.1 Model description

This example use the same model as above.

2.6.10.2 Results

The tool accepts a null eccentricity for the initial orbit.

In conclusion the simulation provides corrects results.

2.7 EP LIBRARY

2.7.1 Default parameters acceptance (T-EP-001)

Library: EP
Model Name: EPsystem
Partition Name: default
Experiment Name: expDefault

2.7.1.1 Model description

This example shows that using all the EP components for building a system, the default settings are compliant for all the components.

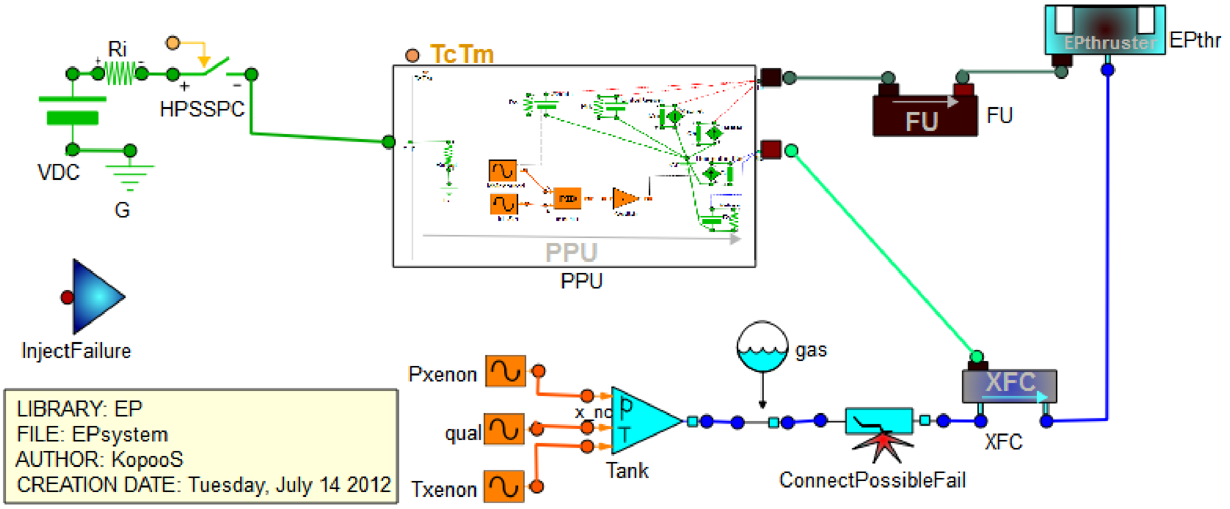


Figure 21 Model of EP system

2.7.1.2 Results

The test consists of the evaluation of the system answer to every command (direct tele-commands or digital TC) step by step. In addition the setting of the discharge current is changed when the thruster is running and the modification of the performances is reproduced.

- The TC PPU.TcTm.DTC=1 AFTER 3 is sent at time 3 s.
- Into the PPU the switch to PPU ON is done after 1 s the event occurs. And the state Stand-by is done after 0.1 s the event PPU ON occurs. So the consumption of the standby mode of the PPU is expected at time=4.1 s. This is shown in the plot below.
- The TC PPU.TcTm.TC=TCAutomatic AFTER 8 3 is sent at time 8 s
- Into the PPU the switch to Automatic is done after 0.1 s the event occurs. So the consumption of the PPU is expected to change at time=8.1 s. This is shown in the plot below.

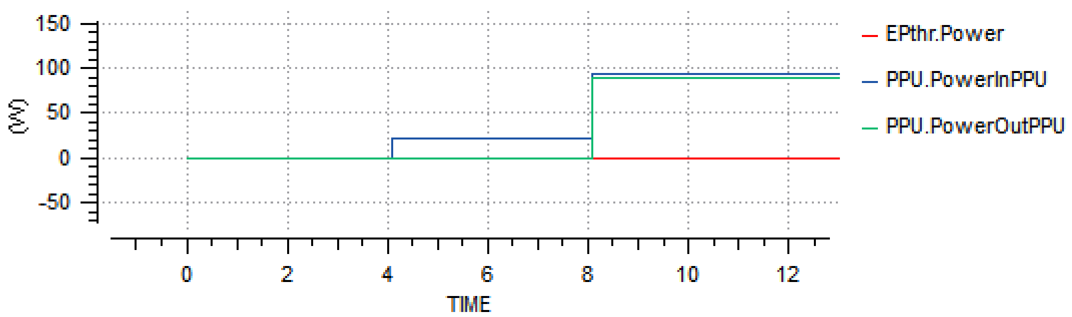


Figure 22 Time sequence of events

- In the Automatic mode the heating of the cathode starts immediately and last for Theating of 120 s and for Tflow of 5 s: that is a total of 125 s

- This is well done as shown in the following 2 plots: the start of heating (Iheating) occurs at T=8.1 s; the xenon flow (EPThr.m) occurs 120 s later at T=128.1 s and the end of heating occurs 5 s later at T=133.1s.

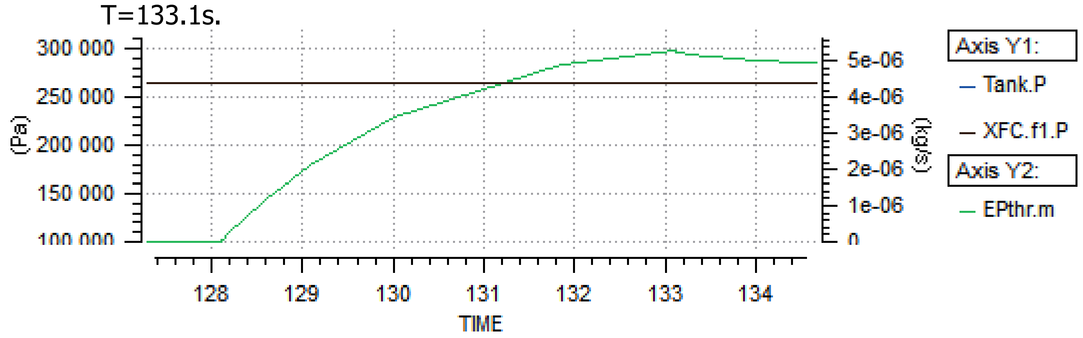


Figure 23 Time sequence of mass flow

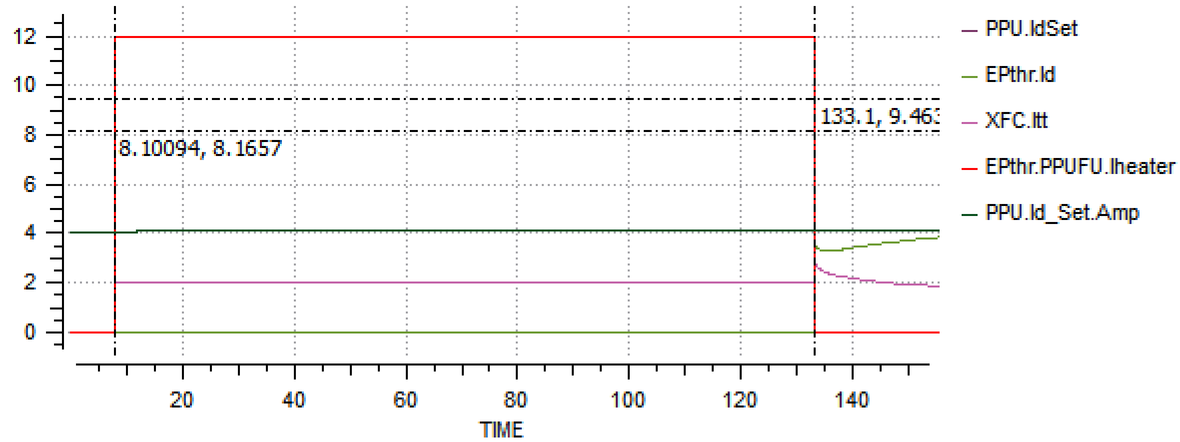


Figure 24 Time sequence of heater

- The ignitor/keeper pulse (duration set to in the equations 4 ms) is automatically generated by the PPU in Automatic mode immediately after the heating is off. This is well done as shown in the following plot: pulse VignitorKeeper from T=133.1 to T=133.105 s

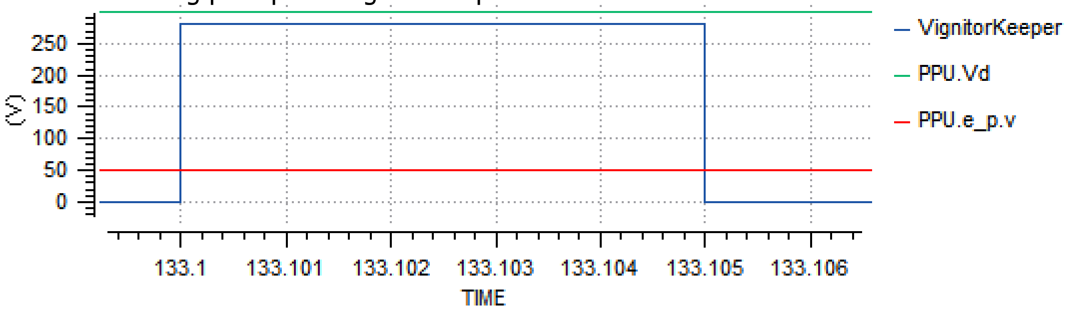


Figure 25 Time sequence of the ignitor voltage (VignitorKeeper)

- The Loop controlling the discharge current Id starts one the thruster is ignited with a minimum of xenon flow. This is represented on the plot below where Itt jump from its initial value of 2 A up to a value given by the PID (hence a maximum to Ittmax at 4 A then decreasing according to the non-saturated PID output).

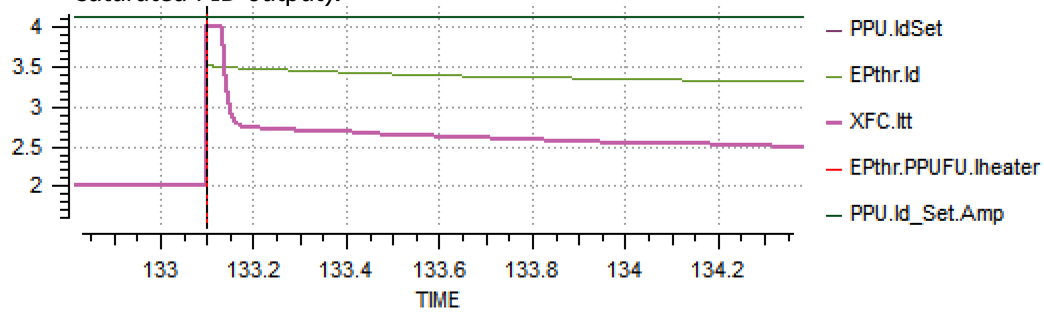


Figure 26 Time sequence of thermothrottle current (Itt)

- At time=410 s successively the following TC are sent to increase the discharge current:
    1. PPU.TcTm.TC_Value=5.125 AFTER 410
    2. and PPU.TcTm.TC=TCIdSet AFTER 410.
- The immediate expected action from the PPU is to act to increase the mass flow rate (Epthr.m) by decreasing sharply the Itt and then slowly stabilizing to a lower value and synchronously, the discharge current increase toward the higher value wanted (IdSet).

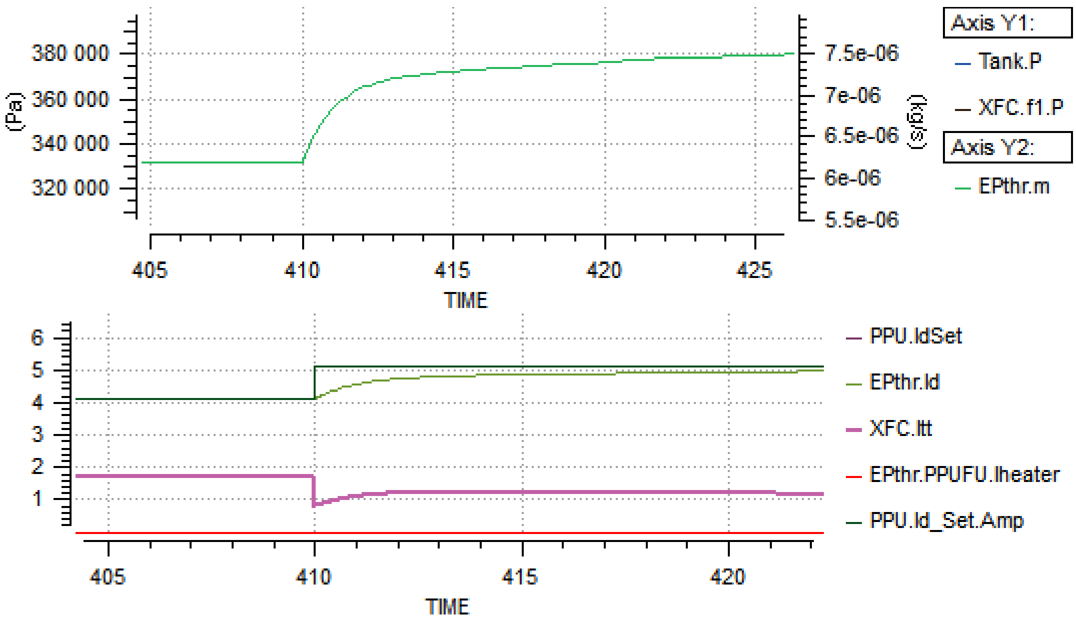


Figure 27 Time sequence of EP parameters

The whole sequence of results over a period of 750 s is shown below (computation duration 7 s)

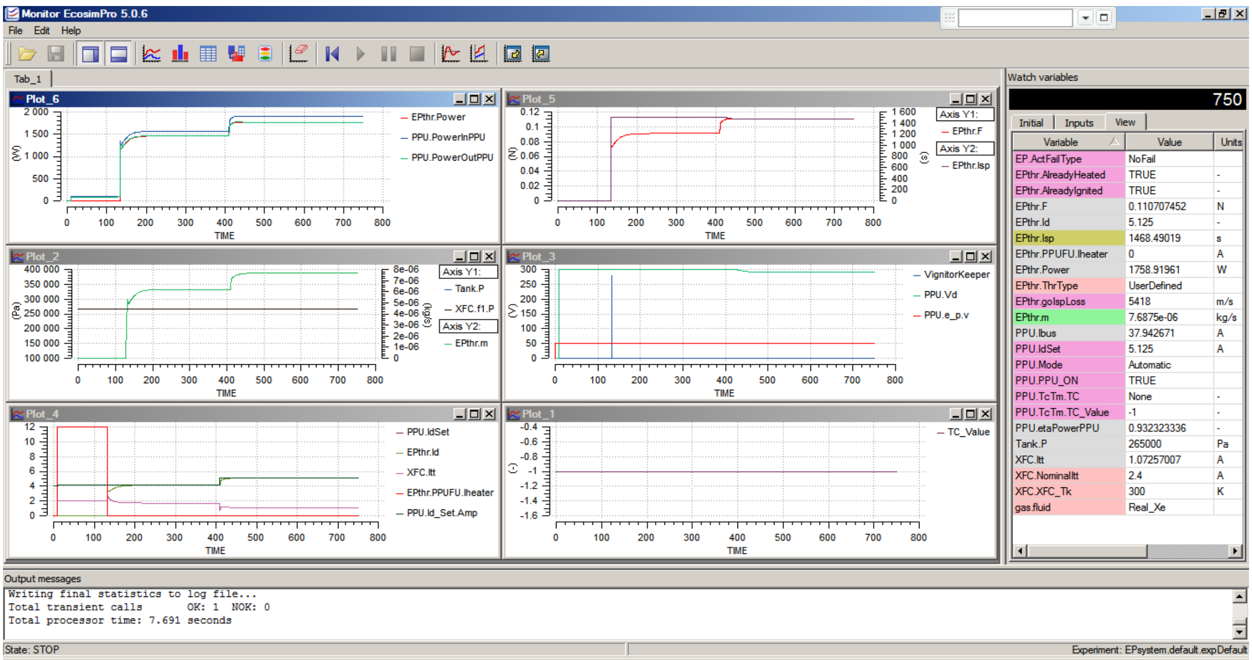


Figure 28 Whole time sequence of EP parameters

2.7.2 Characteristic of the Anode power supply (T-EP-002)

Library: EP
Model Name: EPsystem
Partition Name: default
Experiment Name: expVanode

2.7.2.1 Model description

This test shows that the characteristic of the anode power supply follow accurately the one wanted by the design.

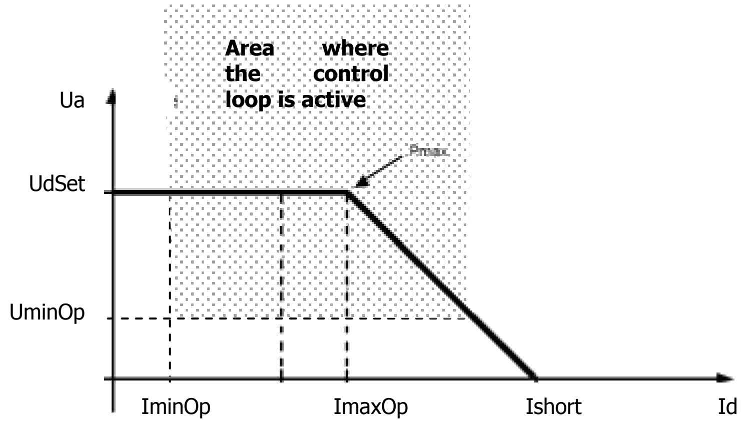


Figure 29 PPU voltage-current characteristic as designed

2.7.2.2 Results

The test consists in the watch of the plot Vd vesus Id for increasing values of Idset (by step of 1 A every 50 s after Time=400 s. The plot below follows accurately the wanted design.

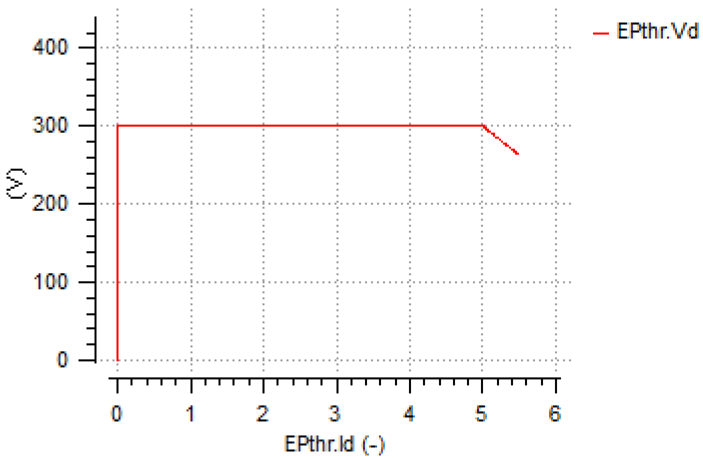


Figure 30 Characteristic Ud versus Id

Note: the whole characteristic cannot be obtained with this experiment because the xenon pressure is not increased when higher mass flow rate are needed to fulfil the increase of the Id as wanted by the increase of IdSet.: the thermothrottle current decrease as wanted by finally reached 0 A at Time=550s and no further change occurs. The current Id maxi is about 6 A as shown on the above characteristics

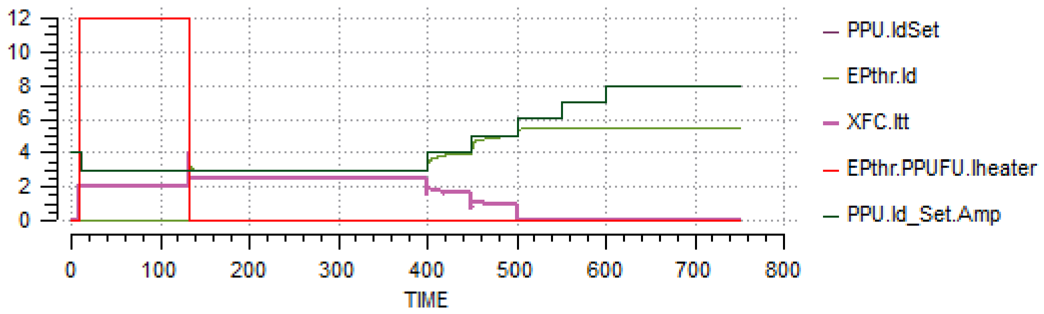


Figure 31 Time sequence of thermothrottle current (Itt) and Id

The whole sequence is shown on the simulation plots below.

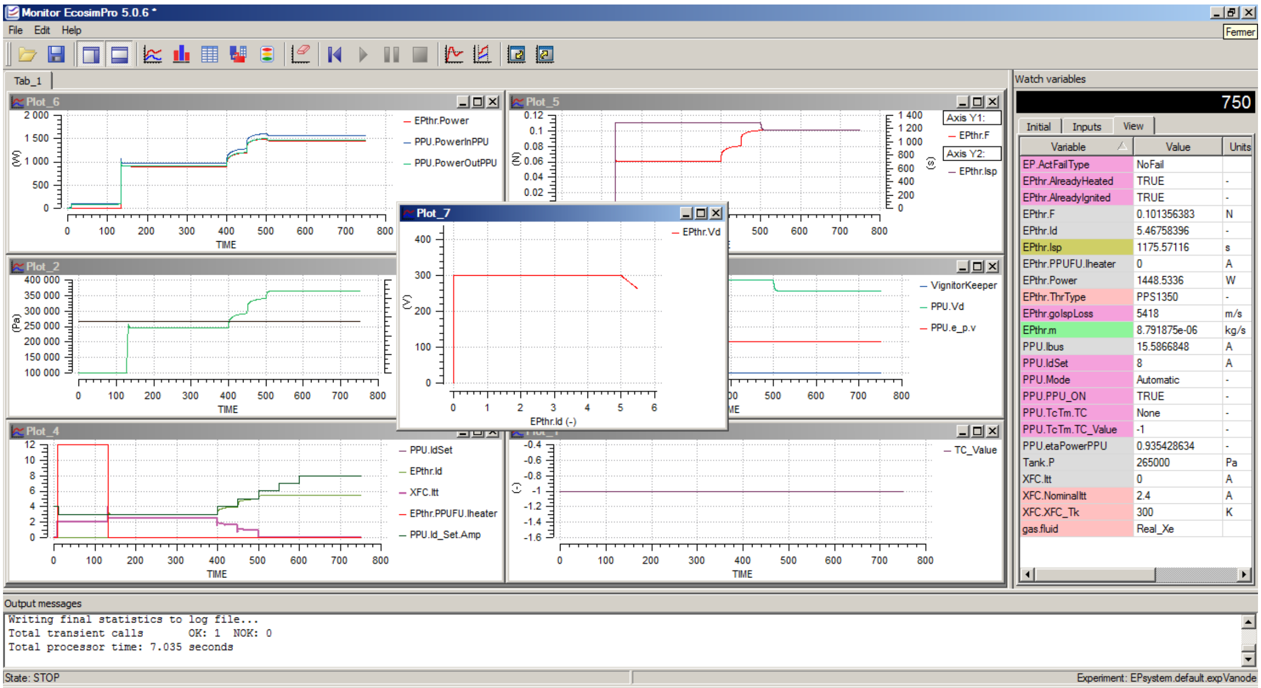


Figure 32 Whole time sequence of EP parameters for the characteristic Ud versus Id

2.7.3 Failure case management (T-EP-003)

Library: EP
Model Name: EPsystem
Partition Name: default
Experiment Name: expGIEwithFail

2.7.3.1 Model description

This example check the occurrence of a failure injected into a system.

2.7.3.2 Results

The test consists in sending a failure case (Fail_XeLeakage_1) at T=200s then clear that failure case at T=250s and then send a stronger failure case (Fail_XeLeakage_1) at T=300s
This is performed in the experiment with the following commands

1. InjectFailure.rams.event=**Fail_XeLeakage_1** AFTER 200
2. InjectFailure.rams.set=FALSE AFTER 250

3. InjectFailure.rams.event=**Fail_XeLeakage_2** AFTER 300

The analysis of the plot Vd vesus Id show that the pressure (XFC.f1.P) decreases between T=200s and T=250s) then reaches again its nominal value then at T=300s a shap decrease occurs. The mass flow rate (m) follows smoothly the pressure variation.

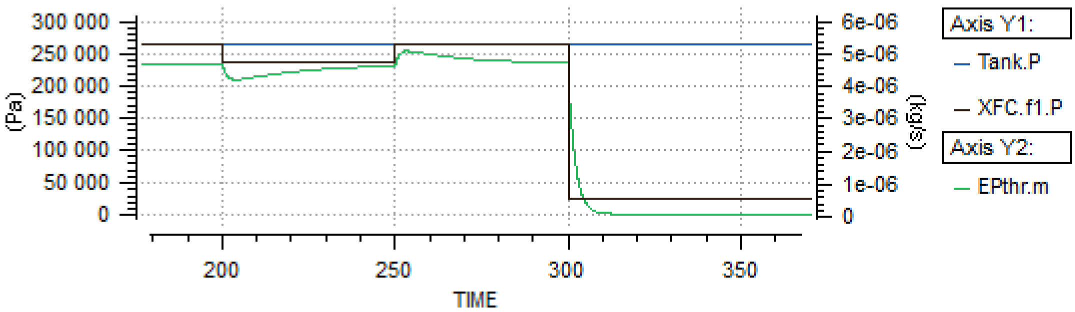


Figure 33 Time sequence of the mass flow rate

The plots of the simulation below show that the second stronger failure case lead to decrease the mass flow rate below the minimum mass flow rate to sustain the discharge, thus the performances of the thruster are turned to zero once this even is reaches.

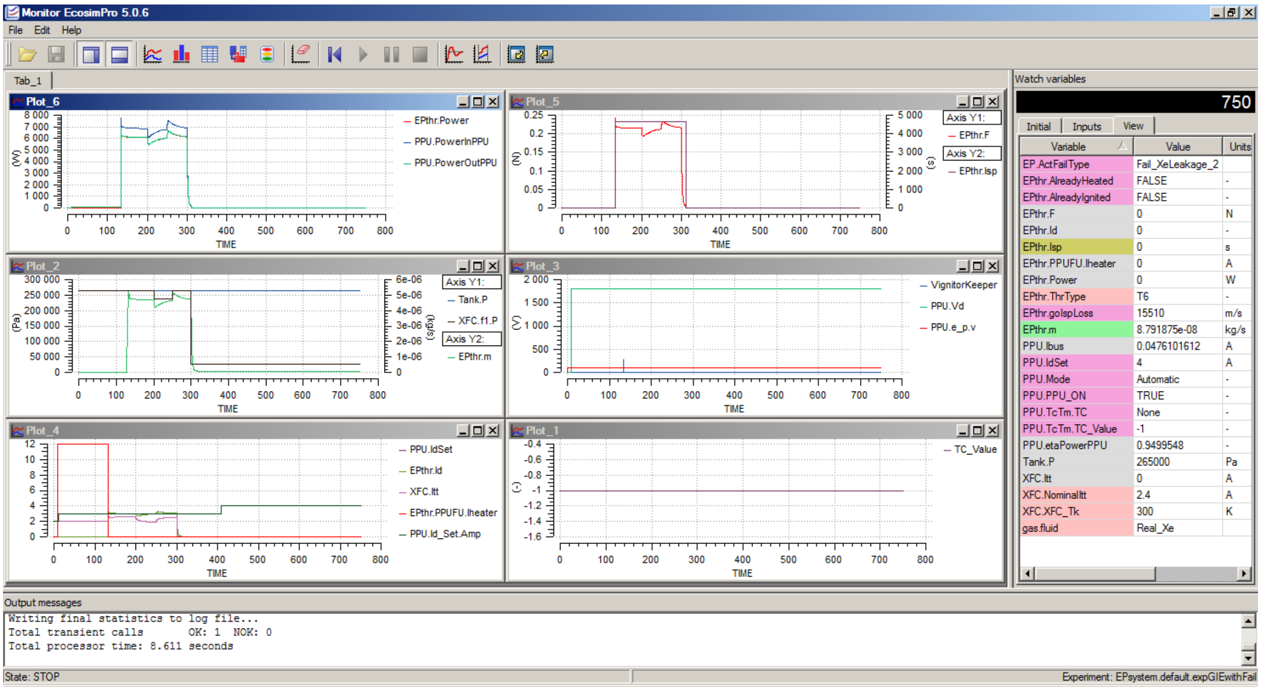


Figure 34 Whole time sequence of the EP system

2.7.4 Thruster case T6 (T-EP-004)

Library:	EP
Model Name:	EPsystem
Partition Name:	default
Experiment Name:	expGIE

2.7.4.1 Model description

This example use the thruster type *T6*.

2.7.4.2 Results

Correct.

2.7.5 Thruster case Alternate model (T-EP-005)

Library:	EP
Model Name:	EPsystem
Partition Name:	default
Experiment Name:	expALTERNATE

2.7.5.1 Model description

This example use the thruster type *AlternateModel*.

2.7.5.2 Results

Correct.

2.7.6 Thruster case HET (T-EP-006)

Library:	EP
Model Name:	EPsystem
Partition Name:	default
Experiment Name:	expHET

2.7.6.1 Model description

This example use the thruster type *PPS1350* model.

2.7.6.2 Results

Correct.

3. LOG OF ACCEPTANCE TESTS

3.1 ESPSS Version 1.0

Next table logs the tests performed at Empresarios Agrupados for the first version of the ESPSS libraries

TEST ID	PERFORMED BY	RESULT	DATE
T--FF-001	FRJ	OK	10-12-07
T--FF-002	FRJ	OK	10-12-07
T--FF-003	FRJ	OK	10-12-07
T--FF-004	FRJ	OK	10-12-07
T--FF-005	FRJ	OK	10-12-07
T--FF-006	FRJ	OK	10-12-07
T--TNK-001	FRJ	OK	10-12-07
T--TNK-002	FRJ	OK	10-12-07
T--TM-001	FRJ	OK	10-12-07
T--TM-002	FRJ	OK	10-12-07
T--CCN-001	FRJ	OK	10-12-07
T--CCN-002	FRJ	OK	10-12-07
T--CCN-003	FRJ	OK	10-12-07

Some minor problems have been found running EcosimPro 4.4:

- When generating new experiments, the initial values of the dynamic PORT variables (included the molar fractions of the "combustor_rate" components) will be collected in the INIT block. All these initializations can be deleted because they are already done in the INIT block of the corresponding component.
- Using the Monitor tool, for some plots, it is necessary to resize the scale just after the monitor has been opened. To do this it is enough to click on the plot area and press an "r".
- Graphical disposition of gauge and output areas are not correctly saved when quitting the monitor tool.

These problems should be fixed for the next version of EcosimPro.

3.2 ESPSS VERSION 1.4.1

Next table logs the differences of Version 1.4.1 with respect to Version 1.0.1 on the tests performed at Empresarios Agrupados for the ESPSS libraries:

TEST ID	RESULT	Comments
T--FF-001	OK	There are differences in the pressure regulator position. This is due to the upgrades performed in this component: Indeed, depending on the new stroke characteristics, we obtain same flow area for a different stroke position
T--FF-002	OK	During the bubble collapse, the two-phase temperatures are better calculated
T--FF-003	OK	During the bubble collapse, the vapour temperature increase is better calculated showing a more physic behaviour. Other small differences (pressure surge of 235 wrt 240 in previous version) are probably due to the new two-phase friction correlations used in Version 1.4
T--FF-004	OK	Very small differences in the liquid temperatures

T--FF-005	OK	No appreciable differences
T--FF-006	OK	This case has been run with Damp = 1 to smooth numerical pressure peaks during bubble collapse
T--TNK-001	OK	No appreciable differences
T--TNK-002	OK	Results on this test case are completely different due to the correction of an error in the calculation of the sub cooled critical mass flow under supercritical conditions (which is the case pressurizing with H2 critical valves. To get again same results than in version 1.0, the orifices of valves has been multiplied by 1.28 There are some other small differences in the tank temperatures and in the evaporation rate due to the upgrade in the heat exchange coefficient at the gas/liquid interface.
T--TM-001	OK	No appreciable differences
T--TM-002	OK	No appreciable differences
T--CCN-001	OK	The new model of the solid propellant starter produces some differences in the gas generator temperatures and in the axial speed of turbines: In Version 1.0, the starter gases was supposed to have the same composition as the current combustor gases (H2 in this case). Now, the solid propellant gases composition is that of the non-condensable gases (He). The consequence in this case is that the turbines will have lesser power for the same mass flow and temperature of the solid propellant gases The VCH opening time has been delayed from 1.5 to 1.7 seconds to avoid passing throughout the H2 critical point in the regenerative circuit, making the integration very unstable
T--CCN-002	OK	There are some differences in the combustor Mach number. It seems than in previous version, Mach numbers were calculated according with to Version 1.0 and not V1.0.1
T--CCN-003	OK	The new model of droplet vaporisation incorporated in the “combustor_rate” component produces important differences even though the tendencies are similar. On one hand, the new model better calculates pressure surges in the chamber during the start-up. On the other way, the new model (using the default vaporisation parameters) seems to calculate greater differences between the successive combustor stations

General comments for Version 1.4.1 are:

- CPU time is greatly reduced with respect to the original version 1.0. In normal cases the CPU time is divided by 2 or more. Dealing with vapor plus non-condensable gas mixtures (Tanks pressurization circuits), the CPU reduction is still greater
- Combustor models are upgraded and perform faster and more robust simulations during the ignition and start up processes than in Version 1.0.
- Even though the improvements made in Version 1.4, the mathematical iterative process for the calculation of the Homogeneous Equilibrium Model seems to be at the limit of its possibilities for liquid priming cases with non-condensable gases at very low initial pressure.

3.3 ESPSS VERSION 2.0

Next table logs the differences of Version 2.0 with respect to Version 1.4.1 on the tests performed at Empresarios Agrupados for the ESPSS libraries:

TEST ID	RESULT	Comments
T--FF-001	OK	No appreciable differences
T--FF-002	OK	No appreciable differences
T--FF-003	OK	No appreciable differences for Water cases Similar results with MMH, except for the higher pressure peaks, now being 25% greater. This is due to the upgrade of the MMH properties file
T--FF-004	OK	No appreciable differences
T--FF-005	OK	No appreciable differences
T--FF-006	OK	Only differences in the “unstable” behaviour in the boiling pipe region (now being more stable)
T--TNK-001	OK	The static pressures of non horizontal pipes are slightly different because the actual version corrects an error calculating the hydrostatic contribution when the total pipe length is not coherent with the x, y,z coordinates of the pipe’s end junctions: Now, the Δz contribution prevails in case of non-coherent data, even thought the total length as input data (L) will be considered for the pressure drops and inertia effects. No appreciable differences in the rest of the results
T--TNK-002	OK	No appreciable differences
T--TM-001	OK	No appreciable differences
T--TM-002	OK	No appreciable differences
T--CCN-001	OK	Same general behaviour. Performances engine are slighter degraded in correspondence with the inclusion of the combustion efficiency (98%) Version 2.0 still upgrades the model of the solid propellant starter. Now, the composition of the solid propellant gases (H2O,CO,CO2,N2,H2,He) is an input data. Results are more coherent than before, the chamber temperature being the starter gases (boundary) when the starter is working
T--CCN-002	OK	Same general behaviour. Performances engine are slighter degraded in correspondence with the inclusion of the combustion efficiency (98%)
T--CCN-003	OK	No appreciable differences

Multiple software upgrades concerning robustness were needed to validate the code against experimental data. Most of them were collected in the intermediate Version 1.4.1. Version 2.0 incorporates specific user requirements and stabilizes preliminary versions of the code:

- Simulations of pipe networks are more robust and faster than in previous versions, especially dealing with two-phase (priming), two-fluid systems.
- Models of tanks (boiling process) and pressurization systems (real gas properties for the pressuring gas) are improved with faster simulations
- Combustor models are upgraded in many respects permitting more successful simulations during the ignition and shut down processes
- Some non useful or confusing output variables (used for programming reasons) are now hidden to the user, so the plot variables selecting menus are simpler and faster

The minor problems detected under EcosimPro 4.4 have been solved using EcosimPro 4.6

3.4 ESPSS VERSION 2.2

Version 2.2 accounts for the upgrades and experience acquired since Version 2.0 was released mainly derived from the user feed-backs. New useful tips and comments have been added to the User manual helping the user defining the models. Main improvements of Version 2.2 with respect to version 2.0 are:

- New components for ambient P/T/Mach boundary conditions
- More robust simulation of Cold thrusters and in general dealing with supersonic/subsonic transitions
- Valves are provided with user-defined characteristics
- The Heat Exchanger component is upgraded for Cross Flow dispositions
- Fluid Cavities and Tanks account for the volume expansion due to wall compressibility
- Pumps: Table of loss of head vs NPSH and compressibility effects due to pump cavitation have been included
- 1D Tanks can work with supercritical conditions and simplified fluids, and they provide more robust simulations due to the use of better protected correlations concerning the gas/liquid interface
- Combustors can be initialized under ignited conditions. Transitions from the cold liquid injection to the burned gases is in general more robust than in Version 2.0
- Geometrical calculations are included in the continuous block of the components, so parametric studies can be done in a geometrical design without need to restart the simulation. Moreover, the exit to throat nozzle area ratio is now an input data

With Version 2.2 some components (Junctions, Valves, Pressure Regulators, Pumps, HeatExchanger, CoolinJackets and Combustors) are provided with new input data. It must be considered that:

- Default values (automatically loaded editing the schematic file) would normally reproduce Version 2.0 results
- Old models (under Version 1.0 or 2.0) must be compiled under schematic view, so that the new input default values will be automatically loaded

Next table logs the differences of Version 2.2 with respect to Version 2.0 on the tests performed at Empresarios Agrupados for the ESPSS libraries:

TEST ID	RESULT	Comments
T--FF-001	OK	No appreciable differences, only in the relative position (not on the flow area) of the pressure regulator valve due to the simplification on the code
T--FF-002	OK	No appreciable differences
T--FF-003	OK	No appreciable differences
T--FF-004	OK	No appreciable differences
T--FF-005	OK	No appreciable differences
T--FF-006	OK	Differences in the “unstable” behaviour in the boiling pipe region (now being more stable)
T--FF-007 to T--FF-011	OK	New test cases
T--TNK-001	OK	No appreciable differences
T--TNK-002	OK	No appreciable differences in case of default heat exchange option. Better results in case of “advanced” option due to the automatic limitation in the

		correlations
T-- TNK -003	OK	New test case
T--TM-001	OK	No appreciable differences
T--TM-002	OK	No appreciable differences
T--CCN-001	OK	Same general behaviour. The small differences are due to fact of better simulation of the non condensable gases during ignition. The pressure level in the chamber is greater because of the consideration of flow acceleration from the injection plan to the subsonic area
T--CCN-002	OK	Same comments as before
T--CCN-003	OK	Same comments as before
T--CCN-003	OK	Same comments as before
T--CCN-004	OK	New test case

The test cases have been run in different compilers: VC6, VC2003, VC2008 and GCC. The reference is VC6. The conclusions comparing different compilers are:

- Under quasi-steady conditions results are basically the same, even though the number of Jacobean's' calculations and time steps can be different
- Under very fast transients (combustion startups/shutdown) with implicit methods calculating the properties, it has been observed some differences on the stability depending on the compilation platform (GCC, VC6, VC2003 or VC2008)

3.5 ESPSS VERSION 2.4

Version 2.4 gains in robustness (especially starting up combustion chambers) and upgrades the TANKS library accounting for film boiling phenomena and better simulation of the generalized boiling process.

Moreover:

- ➢ The Tee volume calculation is changed accordingly to a right Tee. Added new input data for Y and User defined Tee types.
- ➢ New components (for pipes and Tees down-stream a pre-burner) are available to compute the delay in the transport of the combustion products also permitting the simulation of a mixture of combusted gases and pure fluids (chamber with more than 2 injectors).
- ➢ New option in Pumps (*tdh_correction* = TRUE) so that the pump head is modified to account for high compressible liquids.

Version 2.4 also includes new libraries for direct STEADY calculation designing cycles:

- ➢ The *STEADY* library contains a complete set of components (combustor, nozzle, turbines, pumps and valves) able to calculate the performances of any cycle type under design and off-design conditions.
- ➢ The STEADY_EXAMPLES library contains a set of cycle's types with examples helping the user building models.

Most of the components (Pipes, Junctions, Valves, Pressure Regulators, Pumps, HeatExchangers, Tanks, CoolinJackets and Combustors) are provided with new input data. *It must be considered that:*

- ➢ Old models (under Version 1.0 or 2.0) **must** be edited and compiled under schematic view, so that the new default input values will be automatically loaded.

- Default values (automatically loaded editing the schematic file) would normally reproduce Version 2.0 results.
- Tank mass exchange options at the gas/liquid IF are renamed with more logic names (advanced → Diffusion; default → EvapPool). These new options should be loaded manually in the attributor editor of the schematic view for any model containing 1D tanks.
- Re_lam is no longer a global parameter, but a particular input data for Junctions and valves. Re_lam must be deleted from the experiment files boundary variables.
- CAV_DAMP is no longer an input for Pipes because it slows down the simulation with low improvement of the results.
- Most of the port variables are made PRIVATE because they are redundant with the local variables of the components. Old plot configurations should probably be modified choosing component variables instead of port variables.
- Currently, the spin and lateral acceleration effects in TANKS are deactivated because the corresponding Astrium FORTRAN function does not work with GCC compiler.

Next table logs the differences of Version 2.4 with respect to Version 2.2 on the tests performed at Empresarios Agrupados for the ESPSS libraries:

TEST ID	RESULT	Comments
T--FF-001	OK	No appreciable differences
T--FF-002	OK	No appreciable differences
T--FF-003	OK	Without non-condensable gases, the temperature variation in the priming pipe (10 K) can be appreciated in this version, which seems more physic
T--FF-004	OK	No appreciable differences
T--FF-005	OK	No appreciable differences
T--FF-006	OK	No appreciable differences
T--FF-007 to T--FF-011	OK	No appreciable differences
T--TNK-001	OK	No appreciable differences
T--TNK-002	OK	No appreciable differences in case of default (EvapPool) heat exchange option. Results in case of “advanced” option (now call Diffusion) do not limit negative vaporisation flows. Diffusion approach seems not to be adapted when the vapour mass fraction in the ullage part of the Tank is important. A new case is presented with the film boiling option activated. The bubble production and the corresponding pressurisation is showed
T-- TNK -003	OK	No appreciable differences
T--TM-001	OK	No appreciable differences
T--TM-002	OK	No appreciable differences
T--CCN-001	OK	No appreciable differences
T--CCN-002	OK	No appreciable differences
T--CCN-003	OK	The pressure level in the chambers is slightly greater because of the new default value for the vaporisation factors f_v[*] of the droplets, this producing a higher

		efficiency that seems more realistic
T--CCN-004	OK	No appreciable differences
T--CCN-005	OK	New test case
T--STY-001 to T--STY-004	OK	New test cases

The number of Jacobian evaluations of the test cases can be different depending on the compiler

3.6 ESPSS VERSION 3.0

This version 3.0 *gains in calculation speed, accounts for important upgrades in the calculation of the combustor gas properties, includes new options and components (scramjet & solid/hybrid combustors) and upgrades the ESPSS libraries with the absorption/desorption of gases. The STEADY library has gained in robustness after months of tests*:

- FLUID_PROPERTIES Library changes:
  - a/ Molar fractions arguments changed to mass fractions in CEA thermo routines.
  - b/ A complete set of Van der Waals properties functions has been added.
  - c/ Properties functions explicitly depend on the burned gases chemical compositions. Perfect gas or Van der Walls state of equation is optional.
  - d/ Typical solid propellants constituents and their products are included in the list of chemicals and in the combusted gases properties calculation.
- FLUID_FLOW_1D Library changes:
  - a/ Diluted non-condensable gases are transported in volumes and Pipes.
  - b/ Convection and mixing of the chemical constituents are calculated dynamically (using the transport equations) for any component downstream of a combustor. These components will calculate properties *in accordance with the current chemical compositions, not with the combustor properties*. Thus, ESPSS components must be initialized with the **burnerGasesOption** parameter, which has to options:
    - *burnerGasesOption = FLUID_PROPERTIES.Chemicals* if the component is placed downstream of a combustor. The orking fluid is treated as a mixture of variable chemical composition calculated with the Perfect gas or the Van der Walls state of equation.
    - *burnerGasesOption = FLUID_PROPERTIES.noBurnGases* if the component is not downstream of a combustor. Mixture of a (real) fluid with a non-condensable gas.
  - c/ Initialization of the dynamic state variables is simpler, with no dependence on the order of the components instance and no need to use DISCRETE blocks.
  - d/ The Roe scheme in Pipe components has been upgraded to work with supersonic flows transitions in capacitive Pipes.
  - e/ Including non-condensable gas absorption - desorption effects.
- TANKS Library:
  - a/ Including non-condensable gas absorption - desorption effects.
- COMB_CHAMBERS Library changes:
  - a/ Combusted gases properties functions are re-built with a simpler and faster code. *Perfect gas or Van der Walls state of equation is optional*.

- b/ The *ProdMixer_Tee* component has been rebuilt to model a mixture of a pure fluid with combusted gases. Mixture of two combusted gases is now simulated by a standard Tee component. *ProdMixerPipe* does not longer exists because a simple Pipe replace it.
- c/ *Combustor* components are upgraded with new options: **GasLiqOption** concerning the liquid propellant vaporization models and **rateOption** to choose equilibrium or reaction delay method:
  - *GasLiqOption = Advanced*: convection of liquid is calculated assuming the same speed as in the gas side. Liquid droplets size is fine-tuned in time and position through the "f_v[]" boundary variables multiplying a reference size. Vaporization is modeled depending on combusted gas conditions, latent heat, etc.
  - *GasLiqOption = UserDefined*: no convection of liquid is calculated. Vaporization is assumed to be within a delay time just after the injection plate if the ignition order is given (boundary *IgnitFlag*=1), or null if no ignition order is given.
  - *rateOption = TRUE*: this is a first approach of a non-equilibrium chamber based on a time-delay between the equilibrium and the actual burned gases composition.
  - *rateOption = FALSE*: all the vapors will react instantaneously.
- d/ Combustor components have the option **liquidExitAllowed** to include (or not) the amount of liquid exiting a chamber in the gas mixture passing through turbines and other FLUID_FLOW components.
- e/ New solid/hybrid combustor components including an "evaporation" model for the solid (and liquid) propellants together with a 1D reaction delay model for the combustor core.
- f/ New ramjet/scramjet combustor components including an intake, shock capture, a 1D model for the liquid fuel evaporation together with a 1D reaction delay model.

➢ *STEADY* library changes:

- a/ The automatic initialization of ALG variables is more complete with no dependence on the combustors instance order and no need of DISCRETE blocks. Improving the initialization of enthalpy and mass flows unknowns.
- b/ New design option "known_pressures" for Turbines.
- c/ Combusted gases properties function are re-built with a simpler and faster code.
- d/ The calculation of dh_ise in pumps is modified by an approximate expression depending on the inlet density and deltaP.
- e/ Orifices flow area in design calculations takes into account critical flow conditions.

➢ *NEW APPROACH INITIALIZING STATE VARIABLES:* A new powerful approach initializing state (dynamic) variables has been included. Previous versions did not assure proper initial values for more than 1 pre-burner, and could fail depending on the components instance order in the schematic of a model.

The new initialization (also for the STEADY library) is done thanks to the following changes:

- Better use of the PRIORITY 200 directive in the INIT block of the WorkingFluid, ProdPF_Mixer and pre-burners to be sure that any new fluid is defined (and the properties file read) before any other component.
- Components working with combusted gases must be initialized with a non-condensable gas (which is on the other hand more physical) because it would be practically impossible for the user to define all the combusted gases compositions at the initial time.
- No need to transfer the pre-burners gases constants (R and Cp) to the downstream components because these properties are calculated in any ESPSS component at the current P/T/chemicals mass fractions conditions, thanks to another important upgrade in Version 3 including convection equations for chemicals.

This new initialization was impossible in previous versions because the chemical composition of the burner gases was not properly "transported" downstream of the combustor. It was assumed to be the same as in the combustor.

With respect to Version 2.0, most of the components (Pipes, Junctions, Valves, Pressure Regulators, Pumps, Heat Exchangers, Tanks, CoolingJackets and Combustors) are provided with new input data and calculation options. *It must be considered that:*

- Default values (automatically loaded editing the schematic file) would normally reproduce Version 2.0, 2.4 results. *The following exceptions are noted*:

  FLUID_FLOW_1D Library:

  - Re_lam is no longer a global parameter, but a particular input data for junctions and valves. Re_lam must be deleted from the experiment files boundary variables.
  - CAV_DAMP is no longer an input for Pipes because it slows down the simulation with little improvement of the results.
  - Any component working with combusted gases (therefore, placed downstream of a combustor) will be automatically initialized with non-condensable gases at the initial values of Po/To.
  - The numerical parameters using Roe scheme have to be defined by the user. Using the Roe scheme, the "Case" input data has been suppressed and a new input data, "Isent_Correl", allows supersonic flow transitions.

  COMB_CHAMBERS Library:

  - *"xxx_Rate"* and *"xxx_eq"* combustor types have become deprecated components because their characteristics are now included in new components without extension names.

    Models using the old *xxx_Rate"* and *"xxx_eq"* combustor types will use automatically the formulation contained in the new combustor types without extension name, but the user should change them. *To do this, the user can edit (outside EcosimPro) the corresponding *.eds files and to replace the old type with the new one without extension _rate or _eq.*
  - *starter_y* input data contain the *mass* fractions of the starter gases, not the *molar* fractions.
  - The default value of vaporization factors f_v[] in combustor models is set back to 1.
  - The ProdMixer_Tee component has been rebuilt to model a mixture of a pure fluid with combusted gases. The mixture of two combusted gases is now simulated by a standard Tee component.

  TANKS Library:

  - Tank options for the gas/liquid interface are renamed with more logical names (advanced → Diffusion; default → EvapPool). These new options should be loaded manually in the attributor editor of the schematic view for any model containing 1D tanks.
  - Currently, the spin and lateral acceleration effects in TANKS are deactivated because the corresponding Astrium FORTRAN function does not work with the GCC compiler.

  STEADY Library (Changes from preliminary version 2.6):

  - The input data "*eta*" (nominal efficiency) of Turbines and Pumps components has been changed by "*eta_o*". Similarly, the input data "*Tw_throat*" (design wall temperature) of CoolingJackets components of the STEADY library have been changed by "*Tw_design*".
  - Junctions & Valves can be initialized in mass flows and temperature in case of collection of flows upstream the valve.

- Most of the port variables are made HIDDEN because they are redundant with the local variables of the components. Old plot configurations should probably be modified choosing component variables instead of port variables.
- *The lists of arguments of some properties functions have changed to allow the calculation of both pure fluid and combusted gases properties. Combustion functions and components no longer work with molar fractions but with mass fractions*.

Next table logs the differences of Version 3.0 with respect to Version 2.4 on the tests performed at Empresarios Agrupados for the ESPSS libraries:

TEST ID	RESULT	Comments
T--FF-001	OK	No appreciable differences
T--FF-002	OK	No appreciable differences
T--FF-003	OK	No appreciable differences
T--FF-004	OK	No appreciable differences
T--FF-005	OK	No appreciable differences
T--FF-006	OK	No appreciable differences
T--FF-007 to T--FF-011	OK	No appreciable differences
T--FF-012	OK	**New test case for subsonic/supersonic Fanno tubes**.
T--FF-013	*NOK*	**New test case for subsonic/supersonic Rayleigh tubes.** Capacitive boundary conditions with the *resistive Pipe* can induce unforeseen solutions. The capacitive Roe Pipe *does not converge in the subsonic heat rejection case.*
T--TNK-001	OK	No appreciable differences
T--TNK-002	OK	The fixing of a bug concerning the lower dome wall where the thermal port temperatures were calculated in reverse order does not produce appreciable differences because these temperatures (mainly corresponding to the liquid side) are similar.
T-- TNK -003	OK	No appreciable differences
T-- TNK -004	*NOK*	**New test case for absorption/desorption in Tanks**. There is an important influence of the number of nodes in the time needed to stabilize the tank conditions (pressure and amount of dissolved gas). **More studies should be done to assess about this problem.**
T--TM-001	OK	No appreciable differences
T--TM-002	OK	No appreciable differences
T--CCN-001	OK	*The CPU time and the number of residues evaluation have increased. This change is due to the new option to calculate dynamically the chemical constituents for any component downstream a combustor.* *ESPSS 3.0 allows the simulation of the engine shutdown*
T--CCN-002	OK	No appreciable differences
T--CCN-003	OK	Same comment as for T-CCN-001
T--CCN-004	OK	No appreciable differences
T--CCN-005	OK	New options liquidExitAllowed = FALSE/TRUE have been tested. In contrast with previous ESPSS versions, chemicals and density values are correctly calculated. With liquidExitAllowed = TRUE, the simulation can fail due to the difficulty to deal "combusted gases" with liquids at low temperatures.

T--CCN-006	OK	**New test case for Ramjets**
T--CCN-007	OK	**New test case for solid/hybrid combustors**
T--CCN-008	OK	**New test case for mass addition in Ramjets**
T--CCN-009	OK	**New test case for mass addition in solid/hybrid combustors**
T--STY-001 to T--STY-004	OK	No appreciable differences

The number of Jacobian evaluations of the test cases can be different depending on the compiler

SATELLITE Library is a **new library** with 10 tests cases:

TEST ID	PERFORMED BY	RESULT	DATE
Function JD test (T-SAT-001)	**C. Koppel**	**OK**	**2/08/2013**
Function MoonSunECI test (T-SAT-002)	**C. Koppel**	**OK**	**2/08/2013**
Moon Sun perturbations test (T-SAT-003)	**C. Koppel**	**OK**	**2/08/2013**
Earth flatness (J2) perturbations test (T-SAT-004)	**C. Koppel**	**OK**	**2/08/2013**
Limit values Inclination Null(T-SAT-005)	**C. Koppel**	**OK**	**2/08/2013**
Orbit manoeuvre(T-SAT-006)	**C. Koppel**	**OK**	**2/08/2013**
Archimedes pressure under force (T-SAT-007)	**C. Koppel**	**OK**	**2/08/2013**
Archimedes pressure under force and rotation (T-SAT-008)	**C. Koppel**	**OK**	**2/08/2013**
Flight dynamic with force (T-SAT-009)	**C. Koppel**	**OK**	**2/08/2013**
Limit values Eccentricity Null(T-SAT-010)	**C. Koppel**	**OK**	**2/08/2013**

EP Library is a **new library** with 6 tests cases:

TEST ID	PERFORMED BY	RESULT	DATE
Default parameters acceptance (T-EP-001)	**C. Koppel**	**OK**	**2/08/2013**
Characteristic of the Anode power supply (T-EP-002)	**C. Koppel**	**OK**	**2/08/2013**
Failure case management (T-EP-003)	**C. Koppel**	**OK**	**2/08/2013**
Thruster case T6 (T-EP-004)	**C. Koppel**	**OK**	**2/08/2013**
Thruster case Alternate model (T-EP-005)	**C. Koppel**	**OK**	**2/08/2013**
Thruster case HET (T-EP-006)	**C. Koppel**	**OK**	**2/08/2013**

APPENDICES

A1. FANNO FLOW

4.1 Fanno Flow

4.1.1 Properties

adiabatic:	h_{tot} = constant along pipe
with friction:	$P+\rho v^2 \neq$ constant
no change in mass:	$\dot{m}$ = const

Adiabatic flows through a constant area duct with friction and without change in mass flow can be referred to as Fanno flows. The flows are assumed to be steady and one-dimensional. The process is irreversible due to viscous friction effects. Therefore the properties change along the duct.

For subsonic inlet conditions the flow accelerates. Dependent on the exit pressure and pipe length, area and wall roughness the flow can accelerate to a maximum speed of M=1. If a Mach number of 1 is reached at the outlet the flow is choked and cannot accelerate any further. Supersonic Fanno flows are decelerated along the pipe. For high exit pressures and sufficiently long pipes a sonic shock develops at the end or within the pipe. Downstream the shock, the flow is subsonic.

For thermally (cp=f(T)) and calorically perfect (c_p=const) gases an analytical solution can be found and used for comparison with the ESPSS results. The analytical solution has been created with GNU Octave, a program for performing numerical analysis, which is very similar and mostly compatible with Octave.

4.1.2 Analytical Solution – General Formulation for Perfect Gases

For a constant area duct the following equation can be used to asses the length of a pipe which is necessary to achieve an acceleration or deceleration from M_1 to M_2 [1]. To calculate the distribution of Mach number along the pipe either the inlet or the outlet Mach number has to be known.

$$\zeta = f\frac{L}{D} = \frac{M_2^{\,2} - M_1^{\,2}}{\gamma M_2^{\,2} M_1^{\,2}} + \frac{\gamma+1}{2\gamma}\ln\left[\frac{M_1^{\,2}\left(1+\frac{\gamma-1}{2}M_2^{\,2}\right)}{M_2^{\,2}\left(1+\frac{\gamma-1}{2}M_1^{\,2}\right)}\right] \qquad \text{Eq. 61}$$

The friction coefficient f is a function of the local Reynolds number Re, the relative wall roughness k and the pipe diameter D. In ESPSS the friction coefficient is calculated by an empirical formulation function “hdc_fric”, which is valid for laminar, turbulent and transient flow [5] . The same formulation has been used for the analytical solution (function “func_fric.m”). In Annex B the correlation used in ESPSS has been validated with another empirical formulation for f by Miller [2]. Empirical Correlation for friction factor f as used in ESPSS

$$a = \left[2.457 \quad \ln\left(\frac{1}{(7/\mathrm{Re})^{0.9} + 0.27k/D} \right) \right]^{16}$$ *Eq. 62*

$$b = \left[\frac{37530}{\mathrm{Re}} \right]^{16}$$ *Eq. 63*

$$f = 8\left[\left(\frac{8}{\mathrm{Re}} \right)^{12} + \frac{1}{(a+b)^{1.5}} \right]^{0.08333}$$ *Eq. 64*

The total temperature is constant along the pipe for Fanno flows. The temperature can be calculated for a specific Mach number with the total temperature given by the inlet boundary conditions.

$$T = \frac{T_{tot,in}}{1 + \frac{\gamma - 1}{2} M^2}$$ *Eq. 65*

The pressure can be calculated by the distribution of Mach number. Equations Eq. 66 and Eq. 67 are valid for a calorically perfect gas under assumption of constant mass flow and constant flow area. The inlet pressure Pin can be calculated from the total inlet pressure, given as boundary condition and the inlet Mach number by the isentropic correlation (Eq. 68).

$$\frac{\dot{m}}{A} = \rho v = \frac{P}{RT} M \sqrt{\gamma R T} = const.$$ *Eq. 66*

$$\frac{P_1}{P_2} = \frac{M_2}{M_1} \sqrt{\frac{T_1}{T_2}}$$ *Eq. 67*

$$P_{in} = \frac{P_{tot,in}}{\left(1 + \frac{\gamma - 1}{2} M_{in}{}^2 \right)^{\frac{\gamma}{\gamma - 1}}}$$ *Eq. 68*

<u>Subsonic Inlet Flow</u>

Static pressure, temperature and Mach number are sufficient to define the complete thermodynamic state of a fluid. With equations Eq. 61, Eq. 65 and Eq. 67 a complete subsonic flow along the pipe can be defined if the inlet Mach number is known.

The Inlet Mach number depends on the total pressure $P_{tot,\ in}$, total temperature $T_{tot,\ in}$ at inlet, the back pressure P_{ex} and the friction coefficient f=func(Re,k,D). It can not be calculated by an explicit function. Following implicit set of equations must be solved. The regarding GNU Octave function "sub_sub.m" is attached in Annex B.

Fanno Equations	$L = func(M_{in}, M_{ex}, f)$	compare Eq. 61
	$P_{ex} = func(P_{in}, M_{in}, M_{ex})$	compare Eq. 67
Isentropic Correlations for P_{in} and T_{in}	$P_{in} = func(P_{tot,\,in}, M_{in})$	compare Eq. 68
	$T_{in} = func(T_{tot,\,in}, M_{in})$	compare Eq. 65
Correlation for Friction Coefficient f	$f = func(M_{in}, P_{in}, T_{in})$	compare Eq. 62 - Eq. 64

These are 5 independent functions with 5 unknown variables M_{in}, M_{ex}, P_{in}, T_{in} and f. The friction parameter depends on roughness k, diameter D and Reynolds number RE (see Eq. 62 - Eq. 64). The parameter k, D and L are defined by the pipe properties and geometry. The Reynolds number depends on the diameter D, local velocity and viscosity. Therefore it can also be assessed as function of local Mach number, pressure and temperature.

Total pressure and total temperature at the inlet can be calculated by the user defined inlet boundary condition including static pressure, static temperature and velocity.

The exit boundary pressure is also defined by the user. Although for choked conditions, the pressure at the pipe outlet is equal or higher than the imposed pressure. In this case the pressure at the interface is equal to the critical pressure and the flow expands behind the outlet. Therefore the critical exit pressure has to be assessed in a previous step (function "sub_L_crit.m"). The critical exit pressure depends on the total inlet pressure, total temperature and the pipe length. It can be assessed by the same set of equations as above. The exit Mach number of a choked pipe is equal to one $M_{ex}=1$. In this case the solution of the set of equations is equivalent to the conditions for a choked pipe flow: $P_{ex,\,cr}$, $M_{in,\,cr}$, $P_{in,\,cr}$, $T_{in,\,cr}$ and f_{cr}.

The pipe is choked in the regarding experiment if the imposed pressure $P_{ex,\,imposed}$ is lower than the critical pressure $P_{ex,\,cr}$. In this case the inlet Mach number M_{in} is equal to the critical Mach number $M_{in,\,cr}$. Otherwise for a pure subsonic flow ($M<1$), the exit pressure P_{ex} is equal to the imposed pressure $P_{ex,\,imposed}$ and the inlet Mach number can be assessed by solving the set of equations.

Finally, a distribution of Mach numbers with the according position in the pipe can be generated by Eq. 61. With these pressure and temperature distributions can be calculated applying Eq. 65 and Eq. 67.

<u>Supersonic Inlet Flow</u>

1. Calculate critical pressure to check if shock occurs

Supersonic flows depend on the inlet conditions only. If the imposed exit pressure is higher than the critical pressure, a sonic shock occurs at the exit of the pipe. With rising exit pressure the shock move upstream into the pipe. The flow downstream the shock is subsonic. The critical pressure can be assessed directly by applying equation Eq. 65 and Eq. 67 in addition with the user defined inlet pressure $P_1=P_{in,imposed}$ and the inlet Mach number $M_1=M_{in,imposed}$ derived by the user defined inlet velocity and temperature $M_{in,imposed}=v_{in}\,(\gamma R T_{in})^{1/2}$. The critical exit Mach number is equal to one $M_2=M_{ex,cr}=1$.

$$\frac{P_1}{P_2}=\frac{M_2}{M_1}\sqrt{\frac{1+\frac{\gamma-1}{2}M_2^2}{1+\frac{\gamma-1}{2}M_1^2}}$$ Derived by Eq. 65 and Eq. 67 *Eq. 69*

$$P_{ex,cr}=P_{in}M_{in}\left[\left(\frac{2}{\gamma-1}\right)\left(1+\frac{\gamma-1}{2}M_{in}^2\right)\right]^{1/2}$$ *Eq. 70*

2. Calculate Mach number distribution

   a. Critical pressure higher than imposed exit pressure: $P_{ex,cr} < P_{ex,imposed}$ → no sonic shock

      $P_{in} = P_{in,imposed}$

      $M_{in} = M_{in,imposed}$

      $M_{ex} = func(L, M_{in})$ by solving Eq. 61

   b. Critical pressure lower than imposed exit pressure: $P_{ex,cr} < P_{ex,imposed}$ → sonic shock in pipe

      If the exit pressure is higher than the critical pressure a shock occurs in the pipe. A loss of total pressure has to be considered. Additionally the position of the shock has to be assessed.

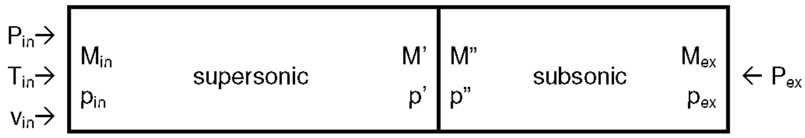


Fig. 17: Fanno Flow with sonic shock

Therefore two additional shock equations must be considered. M' and P' define the conditions directly in front of the shock. They are used to calculate the conditions behind the shock M" and P". (function "super_shock.m"). The fanno flow equations have to be calculated for the super- und subsonic part separately.

2 sonic shock equations

$$M''^2=\frac{1+\frac{\gamma-1}{\gamma+1}\left(M'^2-1\right)}{1+\frac{2\gamma}{\gamma+1}\left(M'^2-1\right)}$$ *Eq. 71*

$$\frac{p''}{p'}=1+\frac{2\gamma}{\gamma+1}\left(M'^2-1\right)$$ *Eq. 72*

2 fanno equations for supersonic part	$L_{supers.}$	= func (M_{in}, M', f)	compare Eq. 61
	P'	= func (P_{in}, M_{in}, M')	compare Eq. 67
2 fanno equations for subsonic part	$L_{subs.}$	= func (M", M_{ex}, f)	compare Eq. 61

	P_{ex}	= func (P", M", M_{ex})	compare Eq. 67
1 equation for pipe length	L	= $L_{supers.} + L_{subs.}$	

This is a set of 7 equations. The user defined boundary conditions M_{in}, P_{in} and P_{ex} are known as well as the pipe length L. The friction parameter depends on fluid properties, which are all constant for calorically perfect gases and on the mass flow, which is constant as well and defined by the inlet conditions. Therefore 7 unknown variables (M', P', M", P", M_{ex}, L_{supers}, L_{subs}) are left, which can be assessed by solving the set of equations.

3. Mach number, pressure and temperature distribution along pipe length

   Finally, a distribution of Mach numbers with the according position in the pipe can be generated by Eq. 61. With these pressure and temperature distributions can be calculated applying Eq. 65 and Eq. 67.