COMPASS
www.compassis.com

Tdyn-CFD+HT - Validation Case 4

Conical Inviscid Nozzle

Tdyn

Version
15.1.0

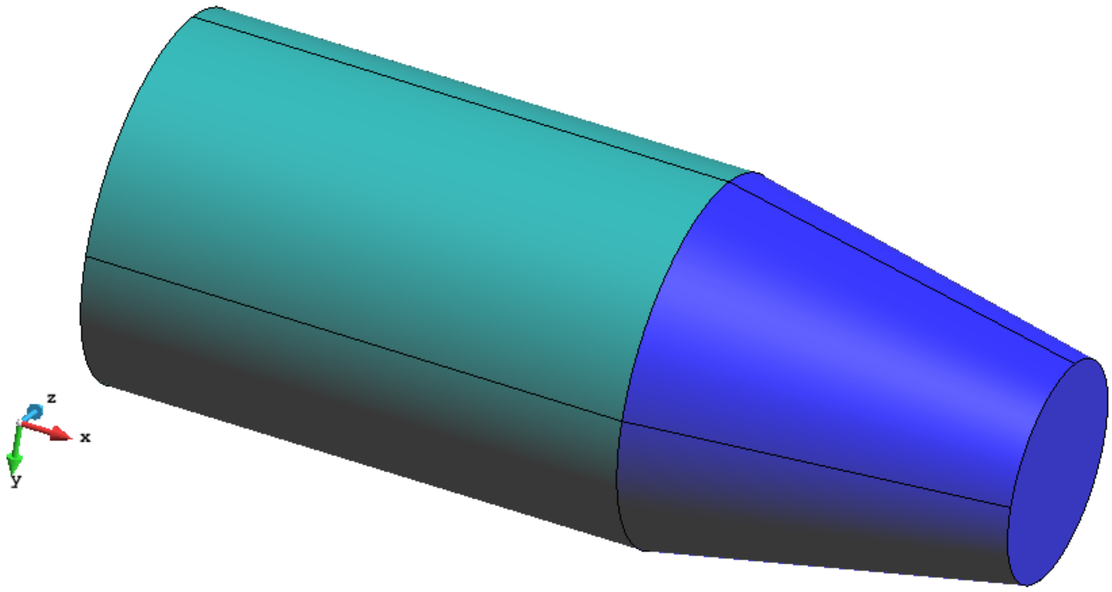
http://www.compassis.com
info@compassis.com
November 2018

Table of Contents

1 Validation Case 4 - Conical Inviscid Flow

This case studies the three-dimensional flow on a circular pipe, with an Inlet velocity to a conical converging nozzle. The radius at the start of the nozzle is $R1=D_1/2$ whereas the radius at the exit of the nozzle is $R_2=D_2/2$. The fluid exhausts at a pressure value P_2 and it is assumed to be incompressible (Mach number small enough), and inviscid.

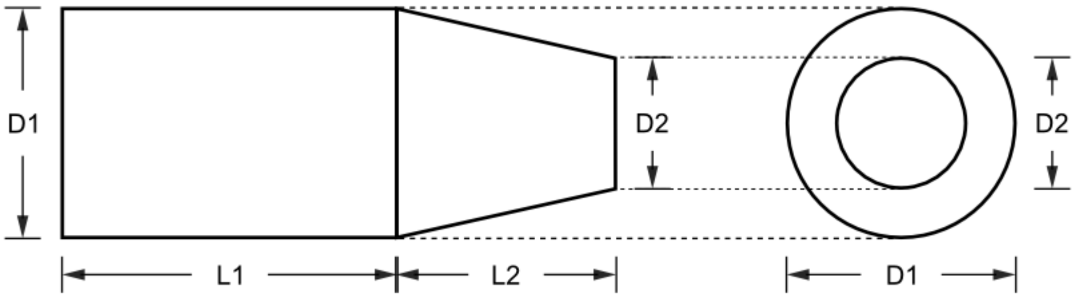
Under these assumptions an analytical solution exists that allows to calculate the force exerted by the fluid on the nozzle. Numerical calculations of the 3D incompressible inviscid flow are presented, and results are compared against the analytical solution.



General view of the 3D model

The table given below shows the dimensions of the three different geometries under study, by varying the Radius R_2 at the exit of the nozzle:

Geometry	$R_1 = D_1/2$ [m]	$R_2 = D_2/2$ [m]	L_1 [m]	L_2 [m]
Nozzle 1	0.127	0.0762	0.381	0.254
Nozzle 2	0.127	0.1072	0.381	0.254
Nozzle 3	0.127	0.1157	0.381	0.254



View of the xy-plane and yz-plane, respectively

The boundary conditions used in the problem are the following:

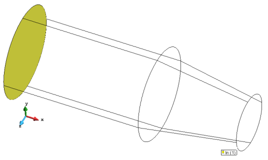
-An Inlet velocity condition is assigned at the boundary Γ_{Inlet}.

-An Outlet pressure condition is assigned at the boundary Γ_{Outlet}.

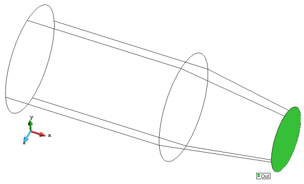
-An "Invis Wall" condition is applied at the contour of the nozzle $\Gamma_{Wall/Bodies}$ in order to simulate a non-viscous fluid with a frictionless boundary.

A brief summary of the boundary conditions that have been applied on the domain is given as follows:

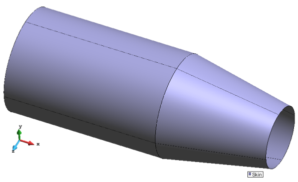
Condition	Boundary
Invis Wall	$\Gamma_{Wall/Bodies}$
Inlet velocity	Γ_{Inlet}
Outlet pressure	Γ_{Outlet}



Inlet velocity



Outlet pressure



Invis wall

Problem description

The problem consists of a conical inviscid flow, with the following characteristics:

- User defined problem
  Simulation dimension: 3D
  Multi-physics analysis: Fluid flow

- Geometry
  The geometry of the present analysis consists on a circular 3D pipe with a conical convergent nozzle at the end.

- Domain
  Steady-state.

- Fluid properties
  Incompressible inviscid fluid. Since the fluid is assumed to be perfect non-viscous, viscosity is completely neglected and hence set to 0.0. Consequently, the only relevant material parameter is the density.

- Fluid Model
  Laminar fluid.

- Material properties
  Density ρ=1000 kg/m^3
  Viscosity μ=0.0 kg/(m·s)

- Boundary Conditions
  Inlet: an inlet velocity condition is used to fix a constant velocity on the inlet cross-section of the pipe. Such an inlet velocity is directed along the pipe axis direction.

  Outlet: an outlet pressure condition is used to fix the pressure at the outlet cross-section of the pipe.

  Wall/Body: An "Invis Wall" condition has been used to simulate a non-viscous fluid with a frictionless boundary.

- Initial conditions
  Velocity: is initialized within the entire domain to the value specified at the inlet boundary V_{in} = 10 m/s.

  Pressure: automatically initialized to the operating pressure value P_o = 0.0 Pa.

- Solver parameters
  The simulation is run using the implicit fractional step solver.

  Assembling type: Nodal.

Time step: 0.001 s.

Non-symmetric solver: Bi-Conjugate Gradient (tolerance = 1.07E-07) with ILU preconditioner.

Symmetric solver: Conjugate Gradient (tolerance = 1.0E-07) with ILU preconditioner.

Analytical solution

The linear momentum conservation theorem states that the rate at which linear momentum contained in a given control volume increases with time, plus the net flow rate of linear momentum out through the control surface, is equal at every instant to the total external forces exerted on the material within the control volume.

$$\frac{d}{dt}\int_{cv} \rho \, \mathbf{v} \, dV + \int_{cs} \rho \, \mathbf{v} \cdot v \, dA = \mathbf{F}_{ext}$$

In the formula above cv states for control volume, cs states for control surface and ext concerns external contributions. The sketch below shows a suitable control surface that eases the calculation of the various contributions to the momentum conservation expression.

Our current analysis clearly concerns a stationary problem. Hence, the rate at which linear momentum contained in the control volume changes is zero.

$$\frac{d}{dt}\int_{cv} \rho \, \mathbf{v} \, dV = 0$$

Moreover, the only contributions to the flow rate of linear momentum out through the control surface take place at the inlet and outlet open cross-sections of the pipe-nozzle system under consideration. Taking into account that in both inlet and outlet sections, the velocity is perpendicular to the section area, and the flow rate of momentum term reduces to:

$$\int_{cs} \rho \, \mathbf{v} \cdot v \, dA = \rho \, v^2{}_2 \, A_2 - \rho \, v^2{}_1 \, A_1 = \rho \, Q \, (v_2 - v_1)$$

where Q states for the flow rate which, due to the mass conservation in the assumed incompressible fluid, must be equal at both sections.

$$Q = v_2 \cdot A_2 = v_1 \cdot A_1$$

Finally, for the given control volume, the resultant of the external forces is the following equation:

$$\mathbf{F}_{ext} = P_1 \cdot A_1 - P_2 \cdot A_1 + F_{nozzle} = \Delta P \cdot A_1 + F_{nozzle}$$

Note that in the formula above, both pressure contributions are multiplied by the same cross-section area A1 corresponding to the inlet. This is so because of the particular choice of the control volume. Such a choice makes the calculation easier since the oblique faces of the nozzle do not directly enter into the control surface integral neither into the external forces contribution.

After some straightforward algebra, the net force over the nozzle can be expressed as follows:

$$F_{nozzle} = \rho\, Q\, (v_2 - v_1) - \Delta P \cdot A_1$$

In order to determine the Pressure drop $\Delta P = P_1 - P_2$ in the previous formula, we appeal to the Bernouilli principle which describes the behaviour of an ideal fluid (frictionless and non-viscous) moving along a streamline. In such a situation the energy stored within the fluid remains constant along its circulation path.

$$v^2 \cdot \frac{\rho}{2} + P = \text{constant}$$

Applying the Bernouilli principle at the inlet and outlet sections of the system and rearranging terms we obtain:

$$\Delta P = P_1 - P_2 = -\frac{1}{2} \cdot \rho \cdot v_1^2 \left(\frac{R_1^4}{R_2^4} - 1 \right)$$

where we made use of the mass conservation relation in the form $v_2 = v_1 \cdot A_1 / A_2$.

Finally the force on the nozzle can be calculated as:

$$F_{nozzle} = \rho\, v^2{}_2\, A_2 - \rho\, v^2{}_1\, A_1 - \frac{A_1}{2} \cdot \rho \cdot v_1^2 \left(\frac{R_1^4}{R_2^4} - 1 \right)$$

which after some algebra reduces to:

$$F_{nozzle} = \frac{\rho \cdot v_1^2 \cdot \pi \cdot R_1^2}{2} \left(1 - \left(\frac{R_1}{R_2} \right)^2 \right)^2$$

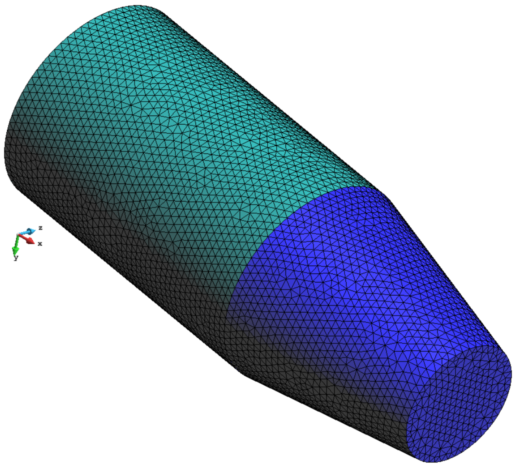
It should be emphasized that this analytical approach assumes that the velocity is parallel to the centerline, and constant over the cross sectional area. Therefore Fnozzle gives an approximate solution to the exact value.

Mesh

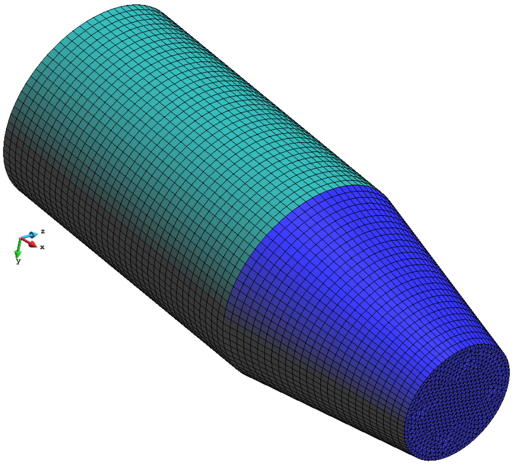
All the simulations has been performed with the same geometry. Two different meshes has been employed for this case of study. The first one consists of an unstructured mesh of linear tetrahedra elements. The second one is a semistructured mesh of linear prismatic elements.

Mesh characteristics can be summarized as follows:

Case	Mesh type	Number of elements	Number of nodes
Nozzle 1	Unstructured	289764	49924
Nozzle 1	Semistructured	78418	38220
Nozzle 2	Unstructured	316396	54498
Nozzle 2	Semistructured	78418	38220
Nozzle 3	Unstructured	324567	55917
Nozzle 3	Semistructured	78418	38220



Unstructured mesh corresponding to Nozzle 1



Semistructured mesh corresponding to Nozzle 1

Results

The following tables show the values of the total forces F over the body walls, the velocity values V2 and the pressure values P1 associated with the numerical solution for the three geometries, of Tdyn solver versus the approximate analytical solution, calculated as indicated previously.

These tables also show the difference in percentage between the approximate analytical value and the Tdyn result, for both unstructured and semistructured meshes.

Tdyn offers a good approximation to the analytical approximation. In all cases the disagreement between the simulations and the approximate analytical results is smaller than 9%.

Such a disagreement can be attributed to the assumption made in the analytical treatment that pressure and velocity are uniform all on the Outlet section of the nozzle. Although such an assumption is quite close to reality, it is not fully accurate. Moreover as can be appreciated in the simulations, the non-uniformity of both velocity and pressure at the outlet increases as the convergence degree of the nozzle also increases. Because of this reason, analytical predictions and simulated results become closer when the nozzle geometry presents a smaller convergence angle (larger values of the outlet radius).

Nozzle 1	F [N]	diff %	V2 [m/s]	diff %	P1 [Pa]	diff %
Analytical result	8007 *	--	27.78	--	335802	--
Unstructured	8759	8.59	28.30	1.84	358149	6.24
Semistructured	8586	6.61	28.02	0.86	351320	4.42

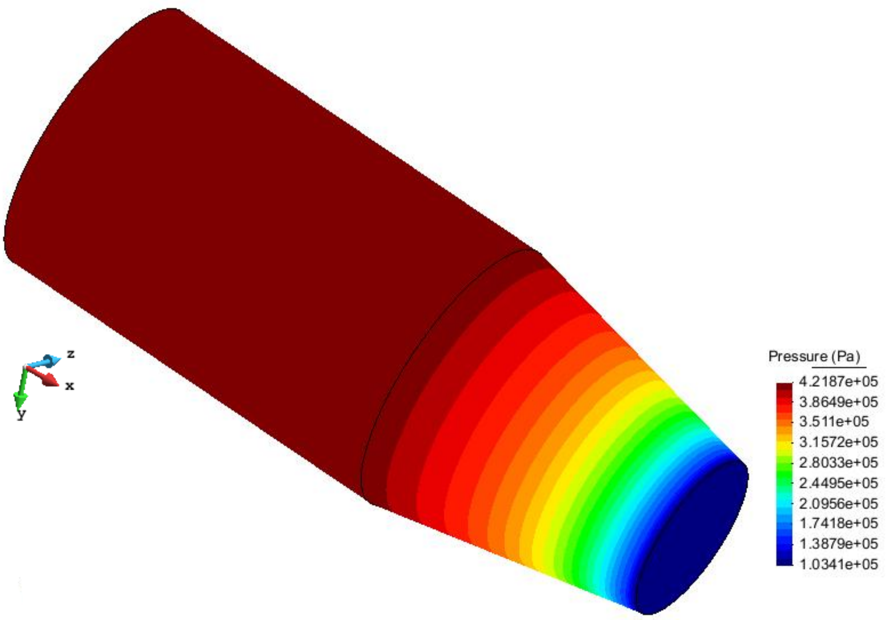
Nozzle 2	F [N]	diff %	V2 [m/s]	diff %	P1 [Pa]	diff %
Analytical result	414.86 *	--	14.05	--	48653	--
Unstructured	442.60	6.27	14.12	0.52	50664	3.97
Semistructured	437.50	5.17	14.08	0.19	49864	2.43

Nozzle 3	F [N]	diff %	V2 [m/s]	diff %	P1 [Pa]	diff %
Analytical result	105.71 *	--	12.04	--	22513	--
Unstructured	111.20	4.94	14.09	0.38	23266	3.24
Semistructured	110.70	4.51	14.06	0.14	22961	1.95

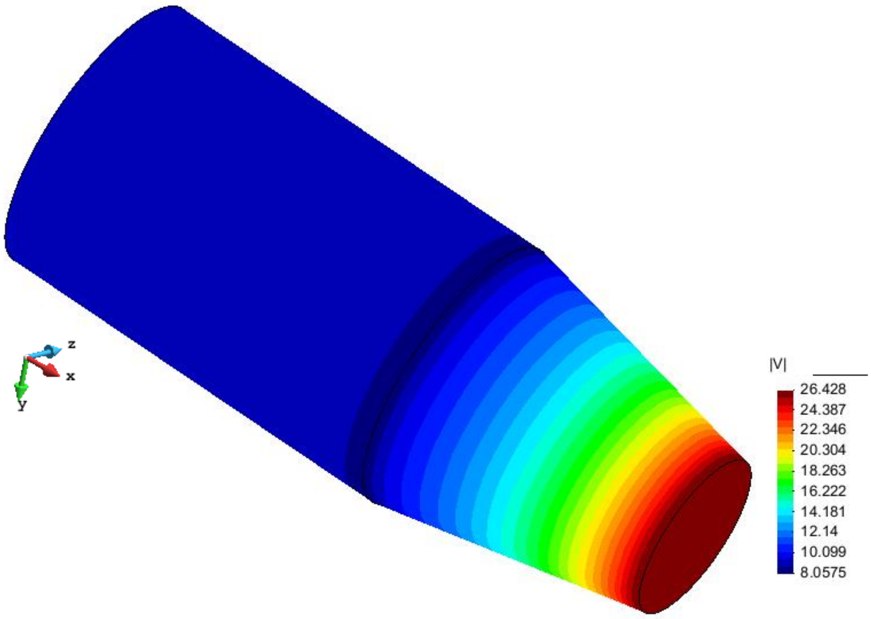
* Calculated applying the following formulation (see section Analytical solution -pag. 4-):

$$F_{\text{nozzle}} = \frac{\rho \cdot v_1^2 \cdot \pi \cdot R_1^2}{2} \left(1 - \left(\frac{R_1}{R_2}\right)^2\right)^2$$

The results given below correspond to the velocity and pressure distributions on the domain concerning the geometry nozzle 1, once the solution has reached the steady state (t=1s).



Pressure distribution at the last time step (t=1s) concerning case nozzle 1



Velocity distribution at the last time step (t=1s) concerning case nozzle 1

Verification

The following table shows the values (results of velocity |V| [m/s], pressure [Pa] and force on wall [N]) associated with the numerical solution for the given mesh, of the current Tdyn solver version versus the reference result of Tdyn, at various points of the domain.

It also shows the percent error, the difference between the current Tdyn value and the reference result, for each field. It must be noted that Tdyn result for the current version is exactly equal to the reference result.

- Nozzle 1

Nozzle 1 unstructured	Coordinates	Field	Ref. value	Curr. value	Err %
P1	0.635,-0.00175,-0.0045	Velocity	28.117	28.117	0.0
P2	0.0, 0.0, 0.0	Pressure	3.6E+05	3.6E+05	0.0
Time 1.0s	Global result	Force	8759	8759	0.0

Nozzle 1 semistructured	Coordinates	Field	Ref. value	Curr. value	Err %
P1	0.635,-0.0025,-0.0026	Velocity	27.888	27.888	0.0
P2	0.0,-0.0042,-0.0044	Pressure	3.5E+05	3.5E+05	0.0
Time 1.0s	Global result	Force	8586	8586	0.0

- Nozzle 2

Nozzle 2 unstructured	Coordinates	Field	Ref. value	Curr. value	Err %
P1	0.635,-0.0057,-0.0058	Velocity	14.114	14.114	0.0
P2	0.0,-0.0006,-0.0017	Pressure	50605	50605	0.0
Time 1.0s	Global result	Force	442.60	442.60	0.0

Nozzle 2 semistructured	Coordinates	Field	Ref. value	Curr. value	Err %
P1	0.635,0.0027,0.0020	Velocity	14.077	14.077	0.0
P2	0.0,-0.0042,-0.0044	Pressure	49864	49864	0.0
Time 1.0s	Global result	Force	437.50	437.50	0.0

- Nozzle 3

Nozzle 3 unstructured	Coordinates	Field	Ref. value	Curr. value	Err %
P1	0.635,0.0032,-0.0023	Velocity	12.078	12.078	0.0
P2	0.0,-0.0006,-0.0017	Pressure	23231	23231	0.0
Time 1.0s	Global result	Force	111.20	111.20	0.0

Nozzle 3 semistructured	Coordinates	Field	Ref. value	Curr. value	Err %
P1	0.635,0.0049,-0.0066	Velocity	12.056	12.056	0.0
P2	0.0,0.0054,-0.0073	Pressure	22961	22961	0.0
Time 1.0s	Global result	Force	110.70	110.70	0.0

Validation Summary

CompassFEM version	15.1.0
Tdyn solver version	15.1.0
RamSeries solver version	15.1.0
Benchmark status	Successfull
Last validation date	27/11/2018