Tdyn-CFD+HT - Validation Case 6

Taylor-Couette Flow

Version
15.1.0

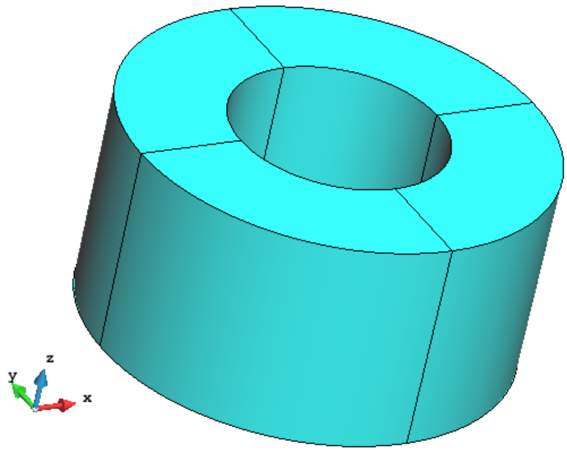
Compass Ingeniería y Sistemas http://www.compassis.com
Tel.: +34 932 181 989 - Fax.: +34 933 969 746 - E: info@compassis.com - C/ Tuset 8, 7o 2a, Barcelona 08006 (Spain)

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

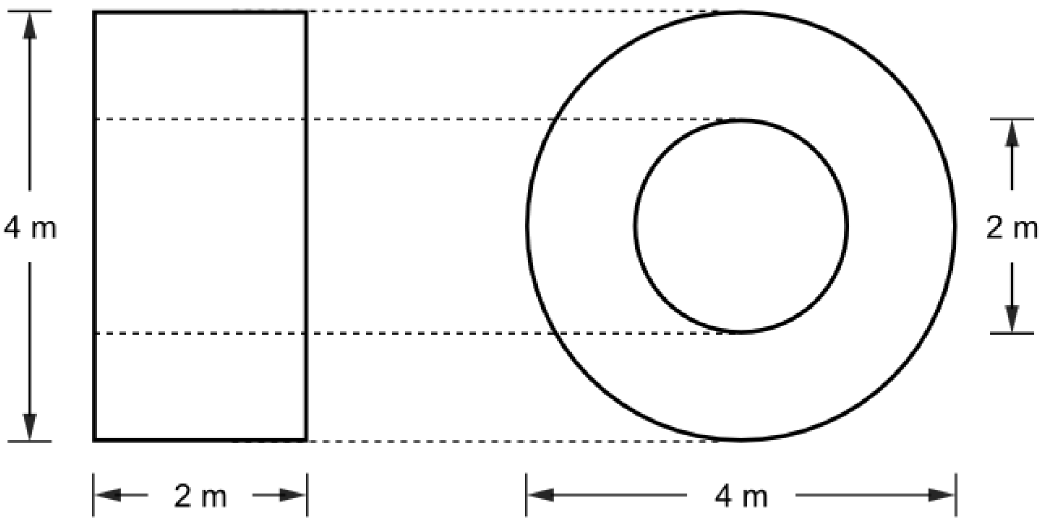
Table of Contents

1 Validation Case 6 - Taylor-Couette Flow

Taylor-Couette problem studies the three-dimensional flow of a Newtonian fluid contained within two concentric cylinders, one or both of which are rotating along their common axis.



General view of the 3D model



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

The main objective of this problem is to determine the velocity distribution of an incompressible viscous fluid with a kinematic viscosity ν and density ρ. It can be characterized by the amount of curvature of the system, defined by the radius ratio $\xi = R_1/R_2$, being R_1 and R_2 the radius of the inner and outer cylinders, and by their respective angular velocities Ω_1 and Ω_2. The aspect ratio is defined as $\Gamma = L/d$, being L the axial dimension of the domain and $d = R_2 - R_1$ the gap width between both cylinders. Several dimensionless parameters can be used to characterize the flow. Two of them are the geometrical parameters that have already been introduced previously (i.e. radius ratio ξ and aspect ratio Γ). In addition, the Reynolds number (Re) and Ekman number (Ek) can be used to assess the importance of the viscous effects.

Two different test cases have been considered, for the sake of comparison. For the first one, the inner cylinder rotates in the anti-clockwise direction while the outer cylinder remains at rest. It should be noted that Coriolis effect is not considered in this test case. For the second one, the inner cylinder velocity has been held to zero, while the outer cylinder rotates clockwise. In addition, the Coriolis effect is considered by introducing an acceleration term ($a_c = 2\Omega x v$) to the equations of motion. Under these conditions, both configurations under analysis are dynamically equivalent. Nevertheless, for the sake of comparison, the rotation of the reference system must be substracted from the total velocity field obtained in case 2.

An analytical solution exists for this problem, which allows to calculate the Taylor-Couette flow. Numerical calculations of the 3D Taylor-Couette flow are presented, and results are compared against the analytical solution.

Dimensional and non-dimensional parameters characterizing the problem are summarized in the following tables, for both cases under analysis:

Data	Description	Value
R_1	Radius of the interior cylinder	1.0 m
R_2	Radius of the exterior cylinder	2.0 m
L	Axial dimension of the cylinders	2.0 m
ρ	Fluid density	1.0 kg m^{-3}
ν	Kinematic viscosity	0.1 m^2/s
Re	Reynolds number	1.9635
Ek	Ekman number	0.509
ξ	Radius ratio	0.50
Γ	Aspect ratio	2.0

Case 1	Description	Value
Ω_1	Inner cylinder angular velocity	0.19635 $rad \cdot s^{-1}$
Ω_2	Outer cylinder angular velocity	0.0 $rad \cdot s^{-1}$
f	Coriolis parameter = $2 \cdot \omega \cdot \sin\Phi$	0.0

Case 2	Description	Value
Ω_1	Inner cylinder angular velocity	0.0 $rad \cdot s^{-1}$
Ω_2	Outer cylinder angular velocity	-0.19635 $rad \cdot s^{-1}$

f	Coriolis parameter = 2·ω·sinΦ	0.39267

The boundary conditions used in the problem are the following:

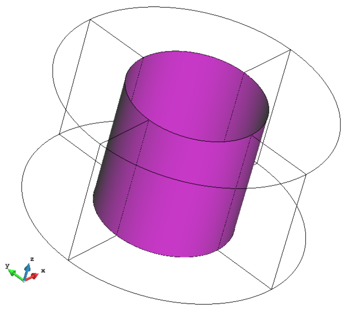
-An "Invis Wall" condition is used to simulate a non-viscous fluid at both the top and bottom surfaces of the domain.

-A "VFixWall" condition is used to enforce the no-slip condition at the vertical surfaces of the outer cylinder.

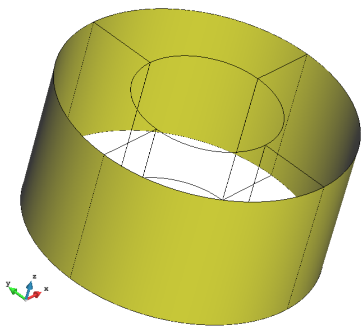
-An Inlet velocity condition is assigned at the vertical surfaces of the inner cylinder.

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

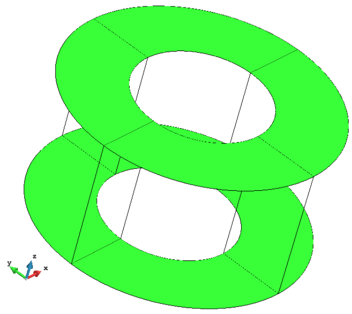
Condition	Boundary
Invis Wall	$\Gamma_{Wall/Bodies1}$
V FixWall	$\Gamma_{Wall/Bodies2}$
Inlet velocity	Γ_{Inlet}



Inlet boundary



V FixWall boundary



Invis wall boundary

Problem description

The problem consists of a Taylor-Couette flow, with the following characteristics:

- User defined problem
  Simulation dimension: 3D

  Multi-physics analysis: Fluid flow

- Geometry
  Concentric cylinders with radius ratio ξ=2.0 and aspect ratio Γ=2.0.

- Domain
  Steady state.

- Fluid Properties
  Incompressible viscous fluid flow.

- Fluid Model
  Laminar flow.

- Material properties
  Density ρ=1 kg/m^3

  Viscosity μ=0.1 kg/(m·s)

- Boundary Conditions
  Inlet: an inlet velocity condition is used to fix a constant velocity on the inner cylinder.

  Wall/Body: An "Invis Wall" condition has been used to simulate a non-viscous fluid with a frictionless boundary at the top and bottom surfaces of the domain. A "VFixWall" condition has been used in order to enforce the no-slip condition at the vertical surfaces of the outer cylinder.

- Initial conditions
  Velocity: initialized to the value 0.0 m/s for the whole model domain.

  Pressure: automatically initialized to 0.0 Pa.

- Time data and solver parameters
  Time increment: 0.1s

  Number of steps: 90

  Non-symmetric solver: Bi-conjugate gradient with ILU preconditioner

  Symmetric solver: Conjugate gradient with ILU preconditioner

Analytical solution

The governing equations of the problem start with the Navier-Stokes equations. An analytical solution of the momentum equation can be obtained assuming the velocity vector has azimuthal direction everywhere, and both velocity and pressure are independent of θ and z cylindrical coordinates.

Under these conditions, the continuity equation is automatically satisfied, and the azimuthal and radial components of the Navier-Stokes equation reduce to:

$$\frac{\partial^2 u_\theta}{\partial r^2} + \frac{1}{r} \cdot \frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r^2} = 0$$

$$\frac{\rho u_\theta^2}{r} = -\frac{dp}{dr}$$

Pertinent boundary conditions imply the assumption of the no-slip condition for the fluid layers attached to the surface of both cylinders. In these conditions, the steady-state laminar solution at small velocities would be the ideal Taylor-Couette flow, whose analytical form is given by the following equations:

$$u_\theta = A \cdot r + \frac{B}{r}$$

$$p_\theta = A^2 \frac{r^2}{2} + 2 \cdot A \cdot B \cdot \ln(r) - \frac{B^2}{2 \cdot r^2}$$

where u_θ and p are the azhimutal velocity and pressure, respectively. Constants A and B depend on the actual configuration of the rotating cylinders, and are given by:

$$A = \frac{\Omega_2 \cdot R_2^2 - \Omega_1 \cdot R_1^2}{R_2^2 - R_1^2}$$

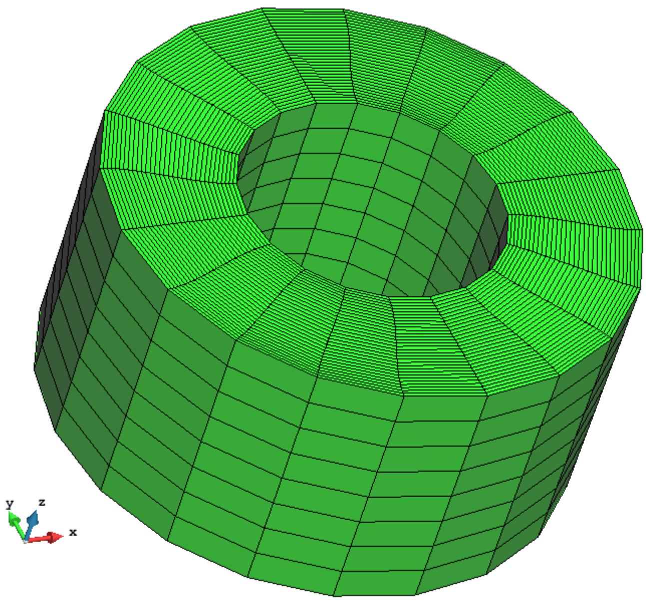
$$B = (\Omega_1 - \Omega_2) R_2^2 \cdot \frac{R_1^2}{R_2^2 - R_1^2}$$

Mesh

The domain for the present analysis is discretized by a structured mesh of hexahedral elements. The finite elements mesh has 5940 nodes, and 7744 elements (hexahedral and quadrilateral). Mesh characteristics can be summarized as follows:

Radial num. divisions	32
Angular num. divisions	20
Axial num. divisions	8

Number of elements	5120 hexahedral elem. + 2624 quadrilateral elem.
Number of nodes	5940

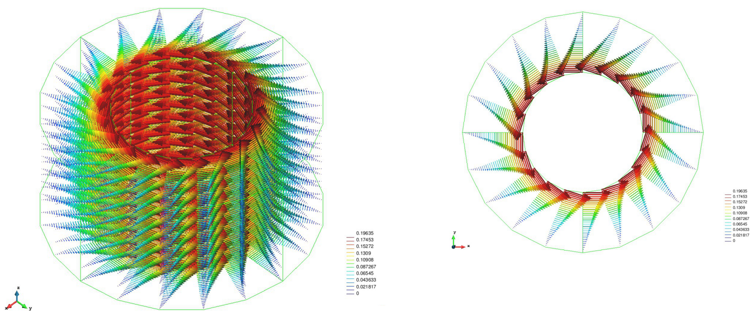


Mesh view and detail of mesh sizes.

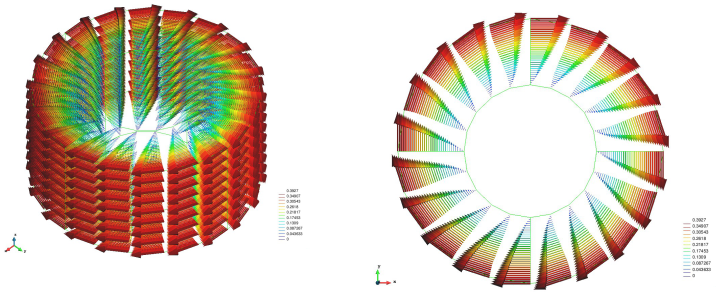
Results

As mentioned earlier in the present document, to compare the results of Case 1 and Case 2 it is necessary to substract the fluid velocity component due to the rotation of the reference system, from the total velocity field obtained in the second case. Such operation can be specified as a user defined function within the graphical user interface of Tdyn. Velocity fields of both test cases are compared in the following figures:

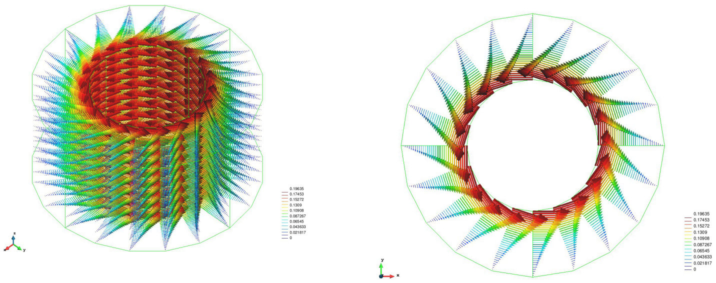
- Case 1: velocity vector field (isometric and top views)



- Case 2: total velocity vector field (isometric and top views)



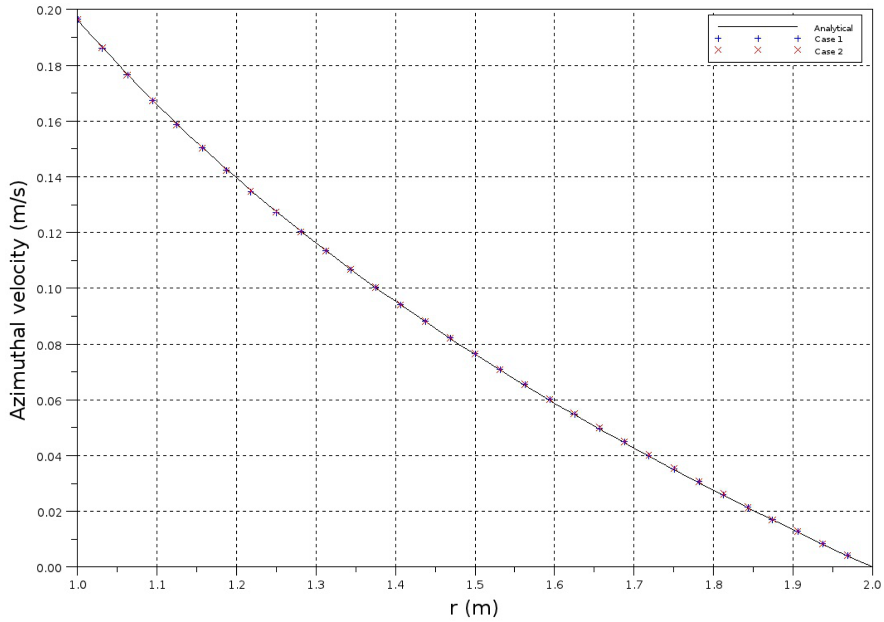
- Case 2: velocity vector field after substraction of the reference system rotation component (isometric and top views)



As expected, after substraction of the Coriolis effect in Case 2, the velocity field is completely equivalent between the two test cases under analysis.

The simulations were further validated by comparing Tdyn numerical results against the analytical solution presented previously.

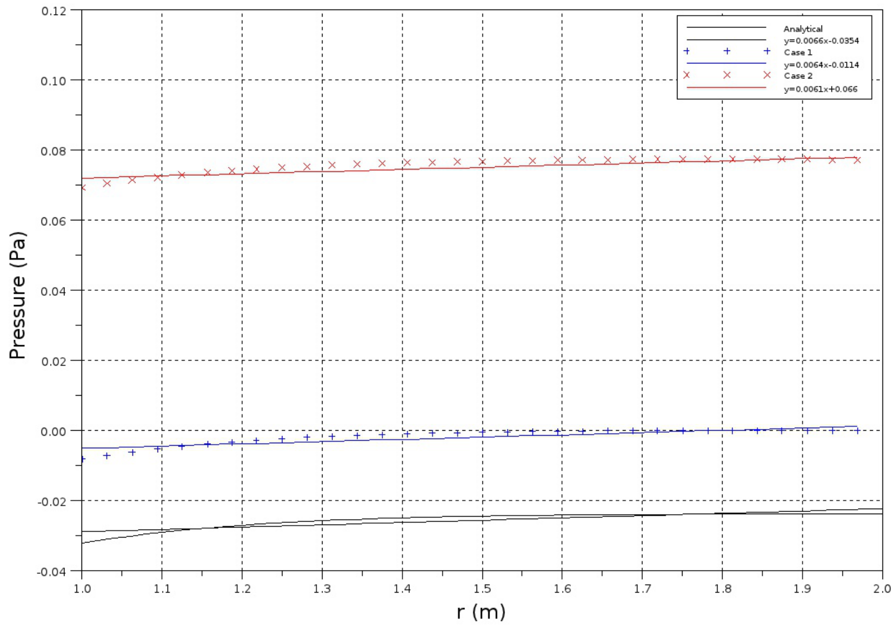
In this sense, the following graph shows the velocity profile along the radius at the top surface, with an azimuth coordinate θ=90º. Test cases 1 and 2 have been compared against the analytical solution, and it can be observed that the margin of error is negligible in both.

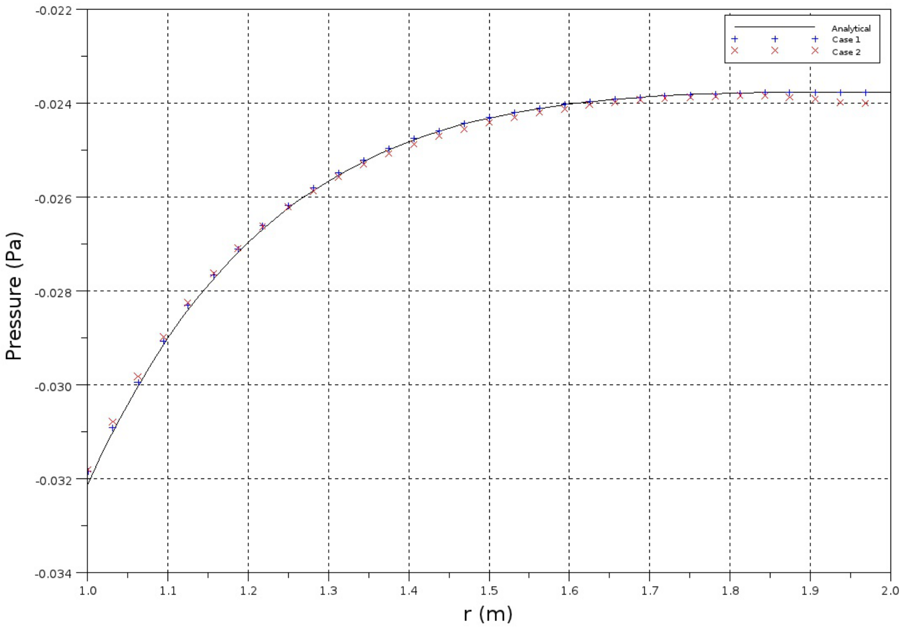


Pressure profiles along the same radius have been also compared to the analytical solution, for both cases (see the graphs below). It should be noted that the total pressure concerning Case 2 was obtained from the sum of the numerically calculated solution, and the term $1/2 \cdot \Omega^2 \cdot r^2$ that generates the centrifugal acceleration $\nabla (1/2 \cdot \Omega^2 \cdot r^2)$. Such a postprocess calculation can also be implemented as a new user variable function within Tdyn GUI.

In the figure below, the pressure profiles differ by an integration constant. In this way, a linear regression is performed for each data set to facilitate curves comparison.

In addition, these linear regressions have been used to evaluate the curve splitting at the radial midpoint (r=1.5m), which is further used to correct the pressure data sets for referring all of them to the same pressure origin.





Tdyn offers a good approximation to the analytical solution.

References

[1] D.J.Tritton, Physical Fluid Dynamics. Oxford Science Publications.

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