Tdyn-CFD+HT - Validation Case 5

Flow Through Porous Media

Tdyn

Table of Contents

1 Validation Case 5 - Flow Through Porous Media

This test case concerns the analysis of a gas flowing through a catalytic converter. Furthermore, it shows the capabilities of Tdyn for modelling the fluid flow through porous media.

The main objective is to determine the pressure gradient and the velocity distribution through the porous media that fills the catalytic element. The gas flows in through the inlet with a uniform velocity of 22.6 m/s, passes through a ceramic monolith substrate, and then exits through the outlet. While the flow in the inlet and outlet sections is turbulent, the flow through the substrate is laminar and is characterized by inertial and viscous loss coefficients in the flow X-direction. The substrate is impermeable in other directions, which is modeled using loss coefficients, whose values are three orders of magnitude higher than in the X direction. The viscous resistance and inertial resistance coefficients used in the three directions and reported in [1] are summarized in the following table:

Direction	Viscous resistance coefficient $(1/m^2)$	Inertial resistance coefficient (1/m)
X	3.846E+07	20.414
Y	3.846E+10	20414
Z	3.846E+10	20414

More details on the theoretical model and the implementation of viscous and inertial terms of the modified Darcy's law are presented in Flow through porous media -pag. 3-

Solving this kind of problem requires a modified formulation of the Navier-Stokes equations, which reduces to their classical form and includes additional resistance terms induced by the porous region.
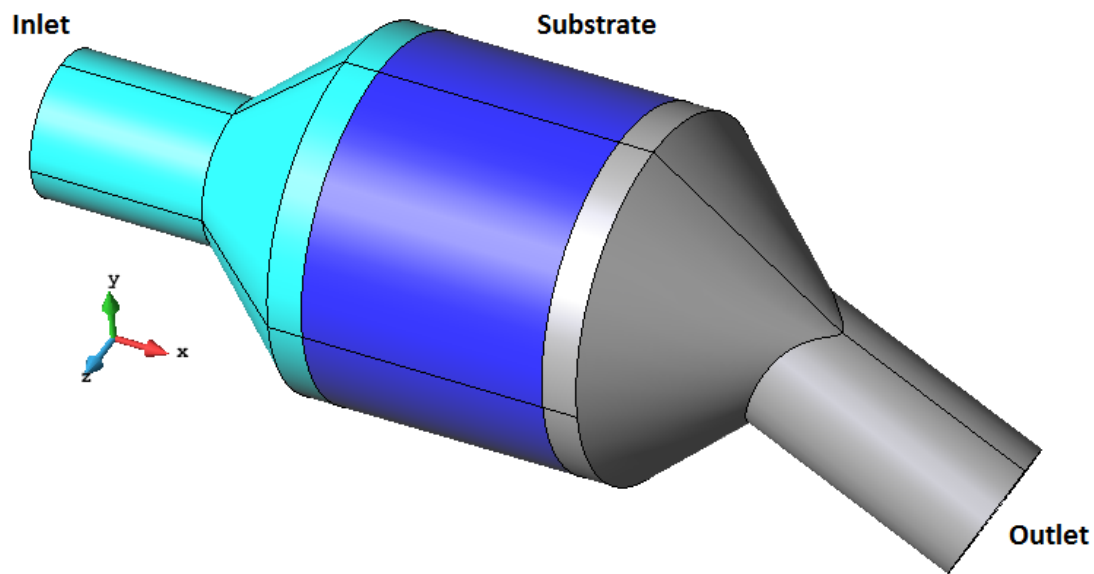
Geometry for flow modeling

The boundary conditions used in the problem are the following:

-A "Yplus Wall" condition is applied at the contour $\Gamma_{Wall/Bodies}$ of the catalytic element to simulate the Reichandt's law of the wall with prescribed value y + (yplus).

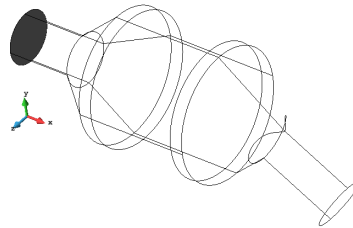-An Inlet velocity condition is assigned at the boundary Γ_{Inlet}.

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

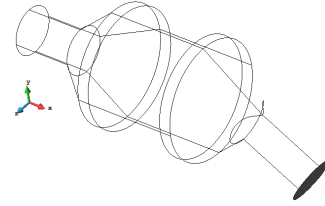A brief summary of the boundary conditions that have been applied on the domain is given as follows:

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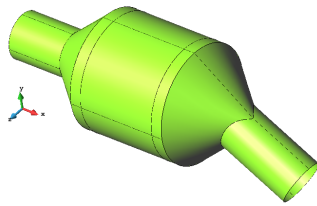
Yplus wall

Inlet velocity

Outlet pressure

A complete description of this problem is presented in reference [1].

Flow through porous media

The incompressible Navier-Stokes equations, in a given domain Ω and time interval $(0,t)$, can be written as:

$$\rho \left(\frac{\partial u}{\partial t} + (u \cdot \nabla)u \right) - \mu \nabla^2 u + \nabla p = f \;\; \text{on} \;\; \Omega \times (0,t)$$

$$\nabla \cdot u = 0 \;\; \text{on} \;\; \Omega \times (0,t)$$

where $u = u(x,t)$ denotes the velocity vector, $p = p(x,t)$ the pressure field, ρ the constant density, μ the dynamic viscosity coefficient and f represents the external body forces acting on the fluid (i.e. gravity).

In the case of solids, small velocities can be considered and the term $(u \cdot \nabla)u$ is neglected. Therefore, assuming an incompressible flow (constant density) in a certain domain Ω and considering the mass conservation equation, the Navier-Stokes equation can be written as follows:

$$\rho \frac{\partial u}{\partial t} - \mu \nabla^2 u + \nabla p = f \;\; \text{on} \;\; \Omega \times (0,t)$$

$$\nabla \cdot u = 0 \;\; \text{on} \;\; \Omega \times (0,t)$$

The general form of the Navier-Stokes equation is valid for the flow inside pores of the porous medium, but its solution cannot be generalized to describe the flow in porous region. Therefore, the general form of Navier-Stokes equation must be modified to describe the flow through porous media. To this aim, Darcy's law is used to describe the linear relation between the velocity u and the gradient of pressure p in the porous media.

It defines the permeability resistance of the flow in a porous media:

$$\nabla p = -\mu Du \text{ in } \Omega_p x(0,t)$$

where Ω_p is the porous domain, D the Darcy's law resistance matrix and u the velocity vector. In the case of considering an homogeneous porous media, D is a diagonal matrix with coefficients $1/\alpha$, being α the permeability coefficient.

Reynolds number is defined as:

$$Re = \frac{\rho UL}{\mu}$$

being U and L a characteristic velocity and a characteristic length scale, respectively.

In order to characterize the inertial effects, it is possible to define the Reynolds number associated to the pores:

$$Re_p = \frac{\rho U\delta}{\mu}$$

where δ is the characteristic pore size. Whereas Darcy law is reliable for values of $Re_p < 1$, otherwise it is necessary to consider a more general model, which accounts also for the inertial effects, such as:

$$\nabla p = -\left(\mu Du + \frac{1}{2}\rho Cu|u|\right) \text{ in } \Omega_p x(0,t)$$

where C is the inertial resistance matrix.

The modified Navier-Stokes equation in the whole domain results from considering the two source terms associated to the resistance induced by the porous medium (linear Darcy and inertial loss term). Hence, these source terms are added to the standard fluid flow momentum equation as follows:

$$\rho \frac{\partial u}{\partial t} - \mu\nabla^2 u + \nabla p - \mu Du - \frac{1}{2}\rho Cu|u| = 0 \text{ in } \Omega x(0,t)$$

Again, considering an homogeneous porous media, D is a diagonal matrix with coefficients $1/\alpha$, and C is also a diagonal matrix. Therefore, a modified Darcy's resistance matrix should be used in Tdyn, as follows:

$$\rho \frac{\partial u}{\partial t} - \mu\nabla^2 u + \nabla p = \mu D^* u \text{ in } \Omega x (0,t)$$

$$D^* = D + \frac{1}{2\mu} \cdot \rho C |u| I$$

being I is the identity matrix.

It should be noted that in laminar flows through porous media, the pressure p is proportional to the velocity u, and C can be considered zero ($D^* = D$). Therefore, the Navier-Stokes momentum equations can be rewritten as:

$$\rho \frac{\partial u}{\partial t} - \mu\nabla^2 u + \nabla p = -\mu D u \text{ in } \Omega x (0,t)$$

Remark: These momentum equations are resolved by Tdyn in the case of solids, where $(u \cdot \nabla)u$ term vanishes due to small velocities. In the case that $(u \cdot \nabla)u = 0$ can not be neglected in the modelization (i.e. high velocities), then Tdyn should resolve the following momentum equations in a fluid, instead of in a solid:

$$\rho \left(\frac{\partial u}{\partial t} + (u \cdot \nabla)u \right) - \mu\nabla^2 u + \nabla p = -\mu D u \text{ in } \Omega x (0,t)$$

Problem description

The problem consists of the analysis of a flow through porous media, 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 cylindrical catalytic converter with three regions (Inlet, Substrate, Outlet).

* Domain
  Steady-state.

* Fluid Properties
  Incompressible fluid.

- Material properties

  Density ρ=1.138 kg/m^3

  Viscosity μ=1.663·10^{-5} kg/(m·s)

  Darcy's law matrix:

  D_{11}: (3.846e7+0.5*20.414*dn*sqrt(vx^2+vy^2+vz^2)/vs)

  D_{22}: (3.846e10+0.5*20414*dn*sqrt(vx^2+vy^2+vz^2)/vs)

  D_{33}: (3.846e10+0.5*20414*dn*sqrt(vx^2+vy^2+vz^2)/vs)

  where dn is the density, v_i are the velocity components and vs is the viscosity.

- Turbulence model

  Spallart-Almaras.

- Boundary Conditions

  Velocity field: a fix velocity condition is applied at the Inlet surface of the model. The velocity vector is fixed to the value v = (22.6, 0.0, 0.0) m/s at the Inlet.

  Wall/Body: 'Y-plus Wall' law is applied to the whole contour.

  Pressure field: a pressure field condition is used to fix the dynamic pressure to zero at the Outlet surface of the domain.

- Initial conditions

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

  Pressure: automatically initialized to the operating pressure value P_0 = 0.0 Pa.

- Solver parameters

  The simulation is run using the implicit fractional step solver.

  Assembling type: Elemental.

  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.

Mesh

All simulations were performed with the same geometry and mesh size parameters.

The space domain is discretized by an unstructured grid of tetrahedral elements. The resulting finite elements mesh has 48.884 nodes, and 293.259 tetrahedral elements.

Results

The results given below correspond to the pressure and x-velocity distributions of the fluid flow on the domain, once the solution has reached the steady state (t=0.25s).
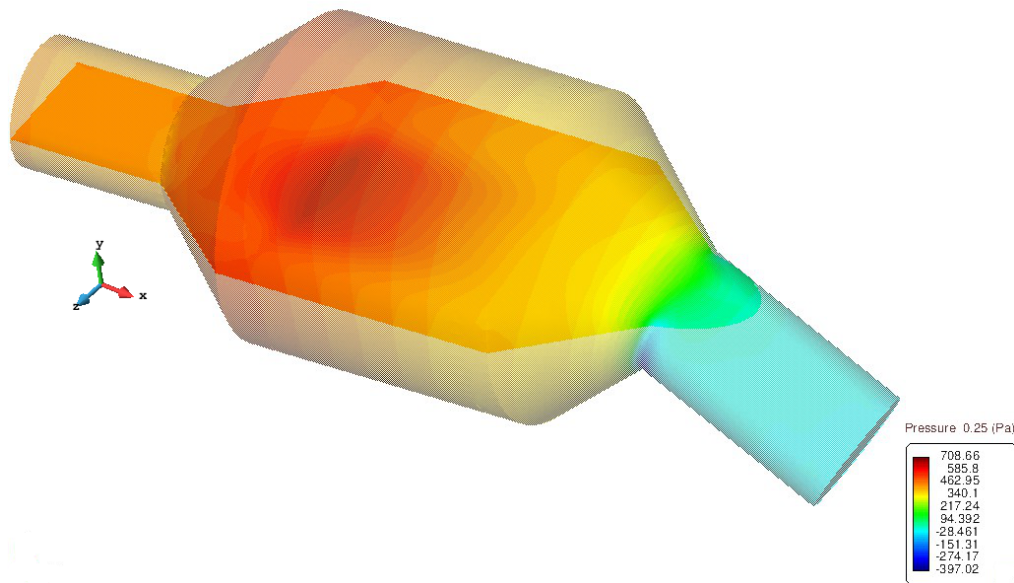
Pressure distribution of the fluid flow at the last time step (t=0.25s)

It can be observed that an important pressure drop occurs within the porous region. The pressure changes rapidly in the middle section, where the fluid velocity changes as it passes through the porous substrate. The pressure drop can be high, due to the inertial and viscous resistance of the porous media.
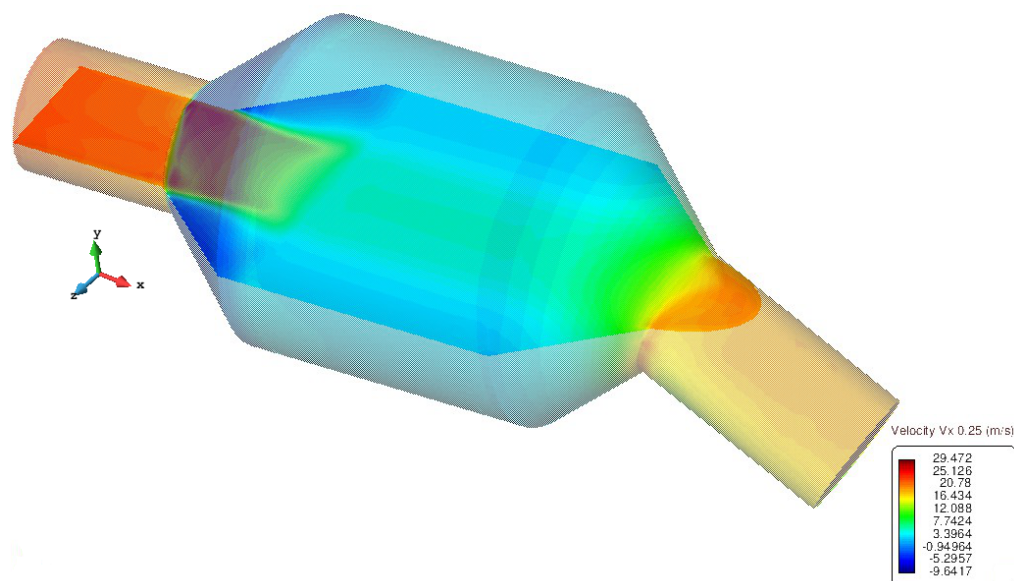
X-velocity distribution of the fluid flow at the last time step (t=0.25s)

It becomes evident how the fluid decelerates rapidly when entering to the porous region.

It can also be observed how the fluid recirculates before entering the central region of the catalyst (negative values of X-velocity, colored in dark blue) due to the resistance exherted by the porous media.

For the sake of validation, the pressure drop across the pororus media and along the centerline of the catalyst obtained with TDYN is compared against the result of the ANSYS/Fluent analysis reported in [1]. In can be observed in the following table that the pressure drop is in good agreement between both software packages, with a discrepancy of about 3%.

	ANSYS/Fluent	TDYN	Deviation
Pressure drop across the porous media	278.7 Pa	288.3 Pa	3%

References

[1] Modeling Flow through Porous Media (version Release 12.0). ANSYS Inc., 2009.

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