CD-adapco

FLOWHEAD

Fluid Optimisation Workflows for Highly Effective Automotive Development Processes

Workshop on industrial design optimisation for fluid flow, Varna, BG, 22-24 Sep 2010

Automated process for CFD simulation with Starccm+, from CAD surface to postprocessing

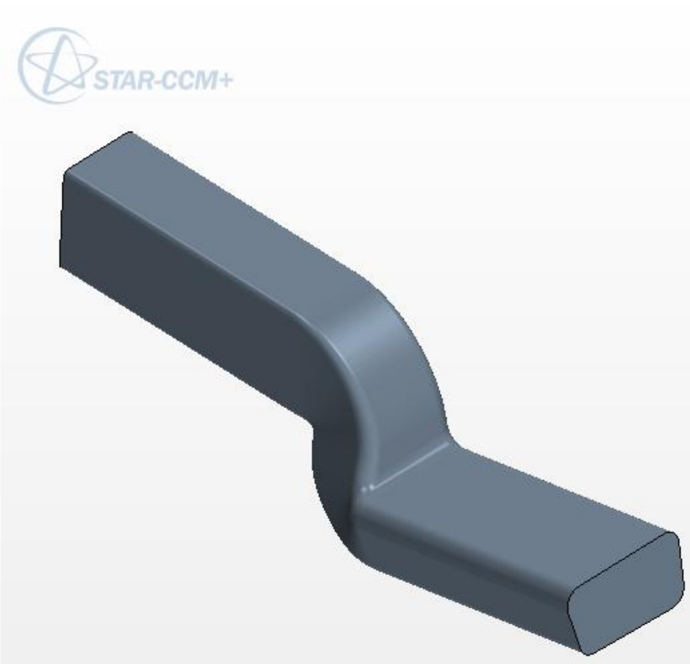
Alexandre CANO - Wednesday, 22 September

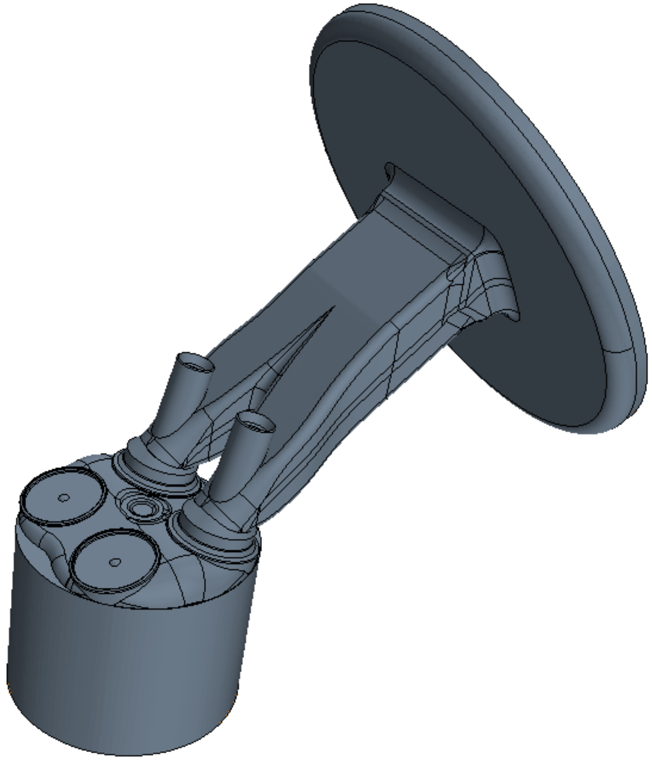
Overview

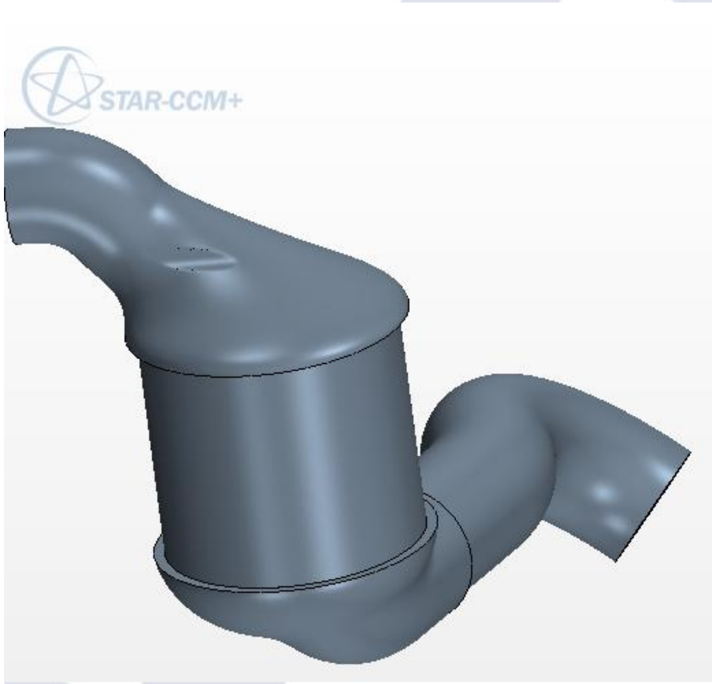
- CAD surface of the demonstration case
- The workflow Automated CFD process
- Automation – The meshing
- Automation – The boundary conditions
- Automation – The physics and solver setup
- Automation – The run
- Automation – The post processing
- Macro Java Parameters
- Conclusion

CAD surface of the demonstration case

- Full automated process performed on 3 test cases

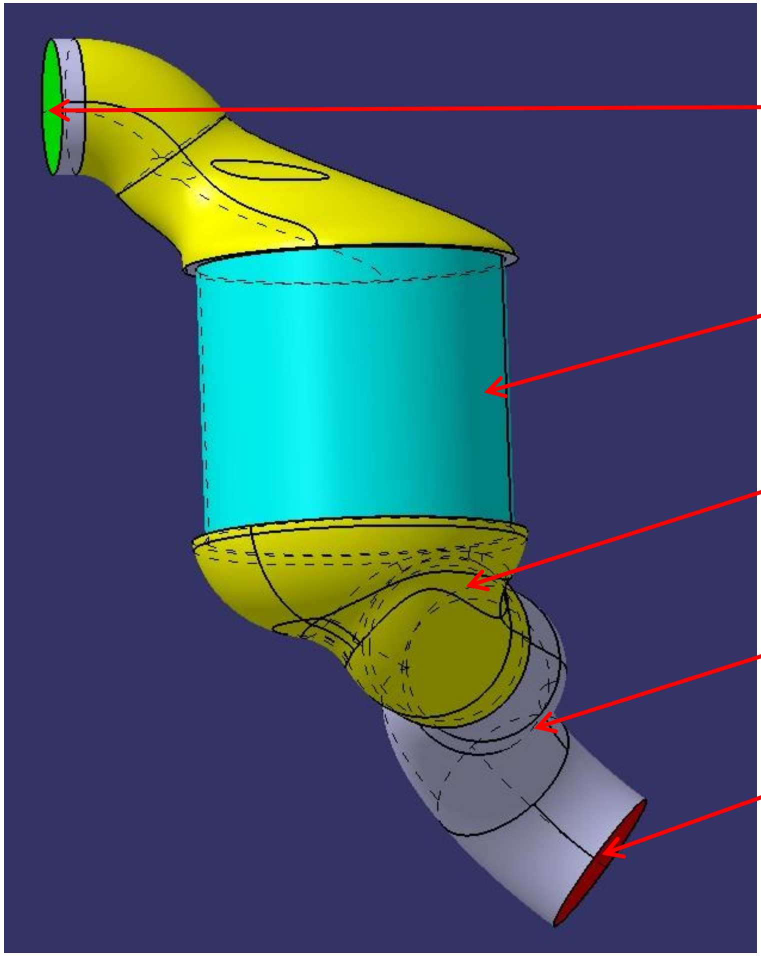






CD-adapco FLOWHEAD

CAD surface of the demonstration case : The Catalyst



Inlet

Porous meduim

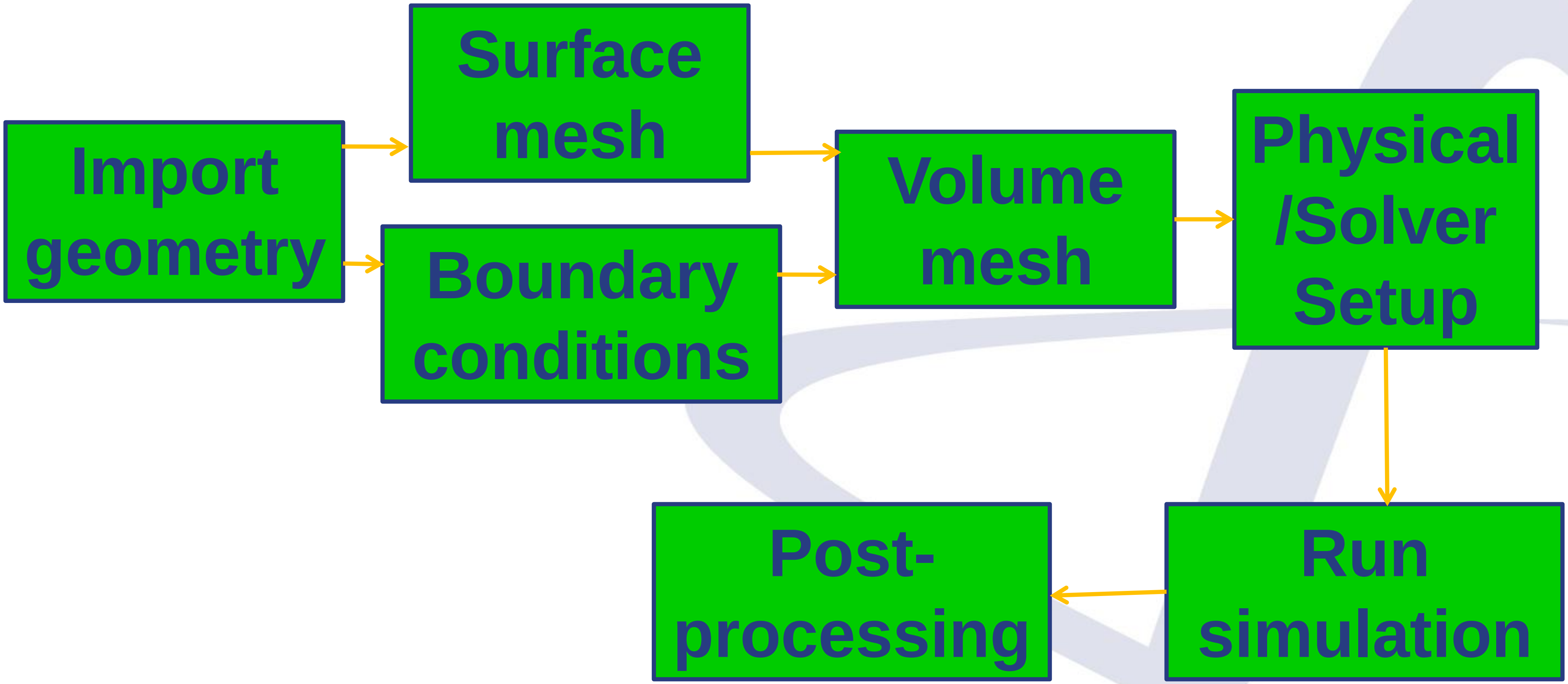
optimized zones (yellow parts)

Fixed zones (grey parts)

Outlet

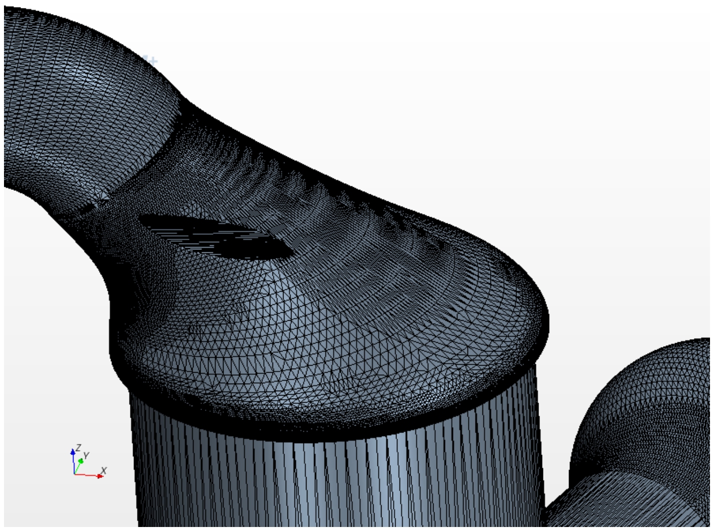
Objective functions : pressure drop and flow uniformity

The workflow Automated CFD process



Automation – The meshing

- First the import surface (CATPart in this example)
- Boundary condition surfaces: Split by angle, locations and areas



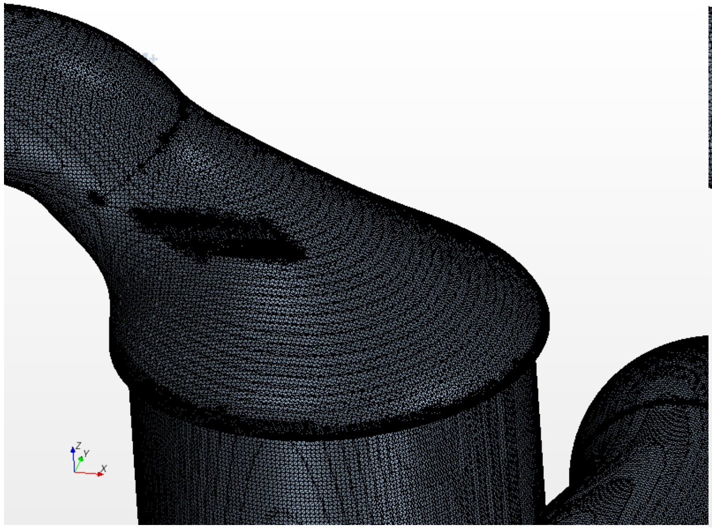
Import surface

Inlet, outlet, interfaces with porous region

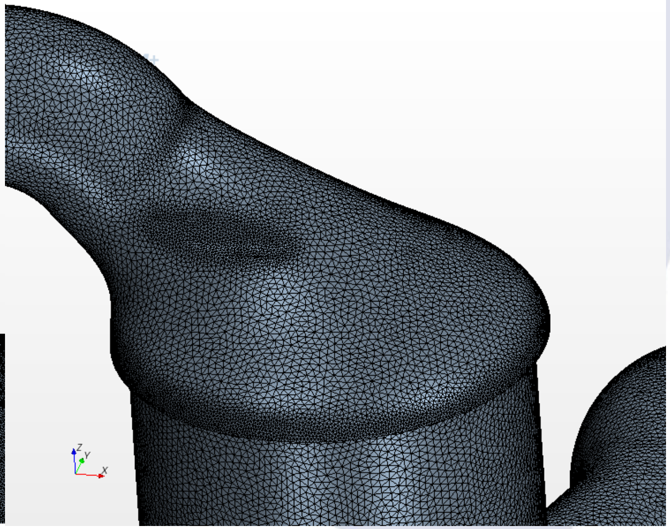
CD-adapco FLOWHEAD

Automation – The meshing

- The meshing generation is divided into 2 parts
  - Surface mesh : Wrap and remesh the surface
  - Volume mesh
- The base size is 3mm



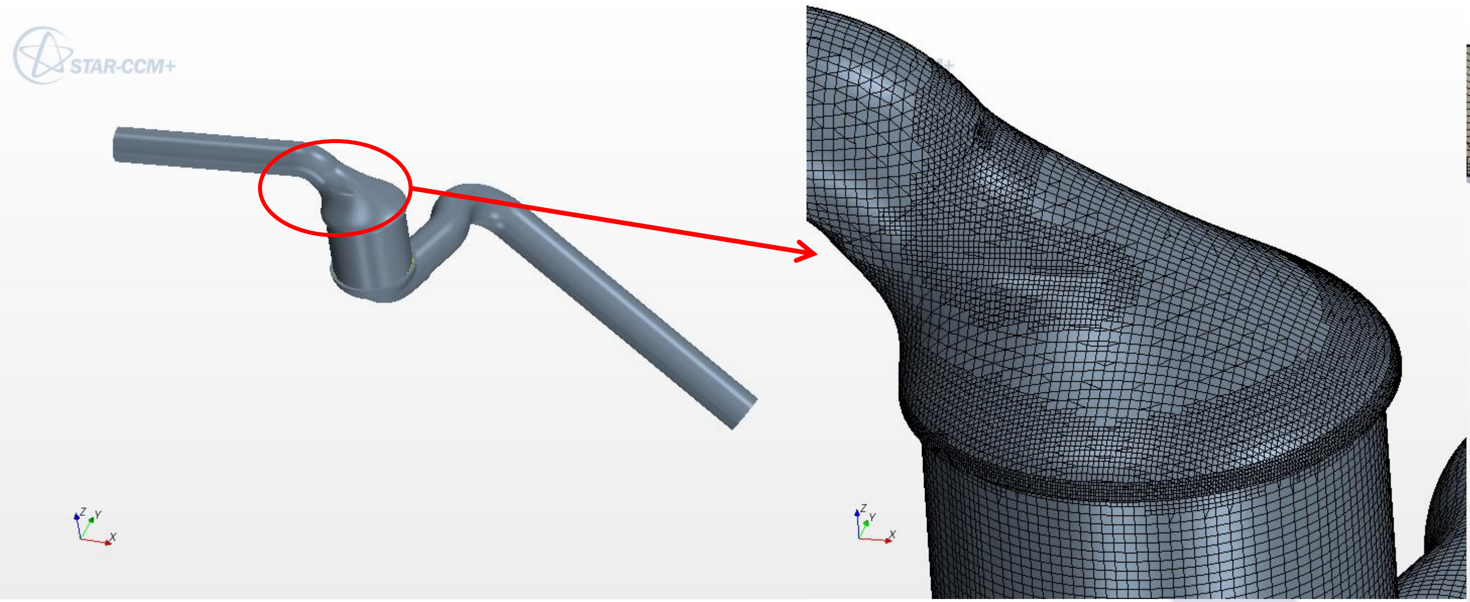
wrapping

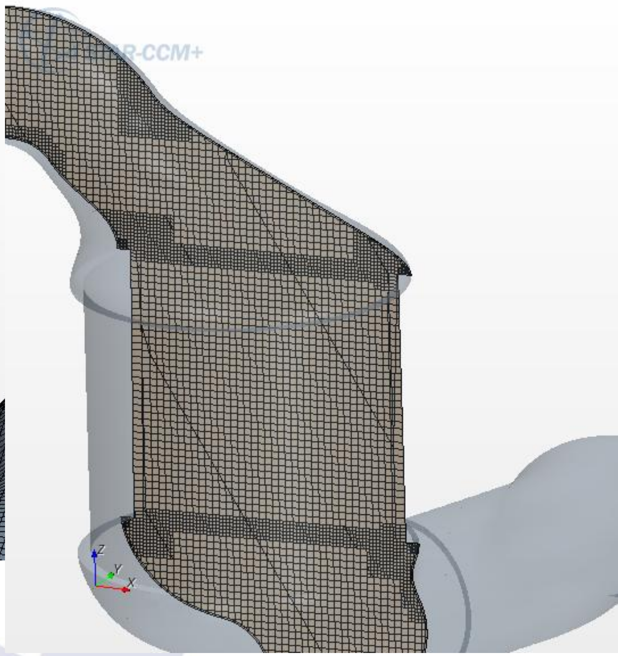


remesh

Automation – The meshing

- The volume mesh is an assemble of a core trimmed mesh and an extrusion layer.





Fluid cells number : ~ 400 000

Elapsed time mesh generation (surface + volume): 3 minutes

(laptop machine : Intel Core 2 Duo 2.59 GHz, 3.5Go de RAM)

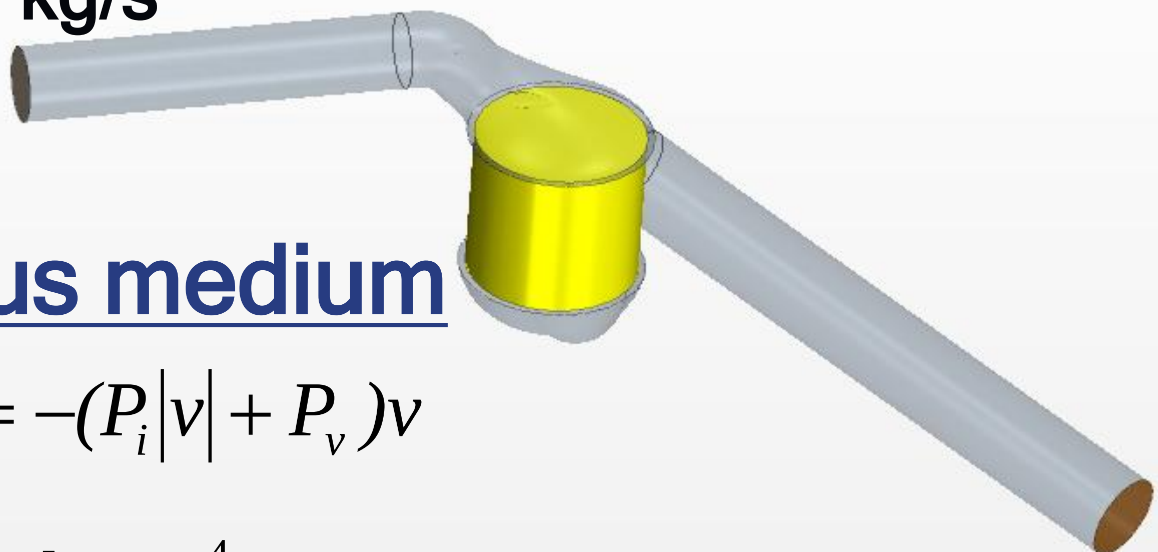
CD-adapco FLOWHEAD

Automation – The boundary conditions (physical values)

Mass flow Inlet

Qm = 0.26841 kg/s

T = 923.15 K



Porous medium

$$\frac{\Delta p}{L} = -(P_i|v| + P_v)v$$

$$P_i = 0 \text{ kg/m}^4$$

$$P_v = 2298.0 \text{ kg/m}^3$$

Pressure outlet

Ps = 0. Pa

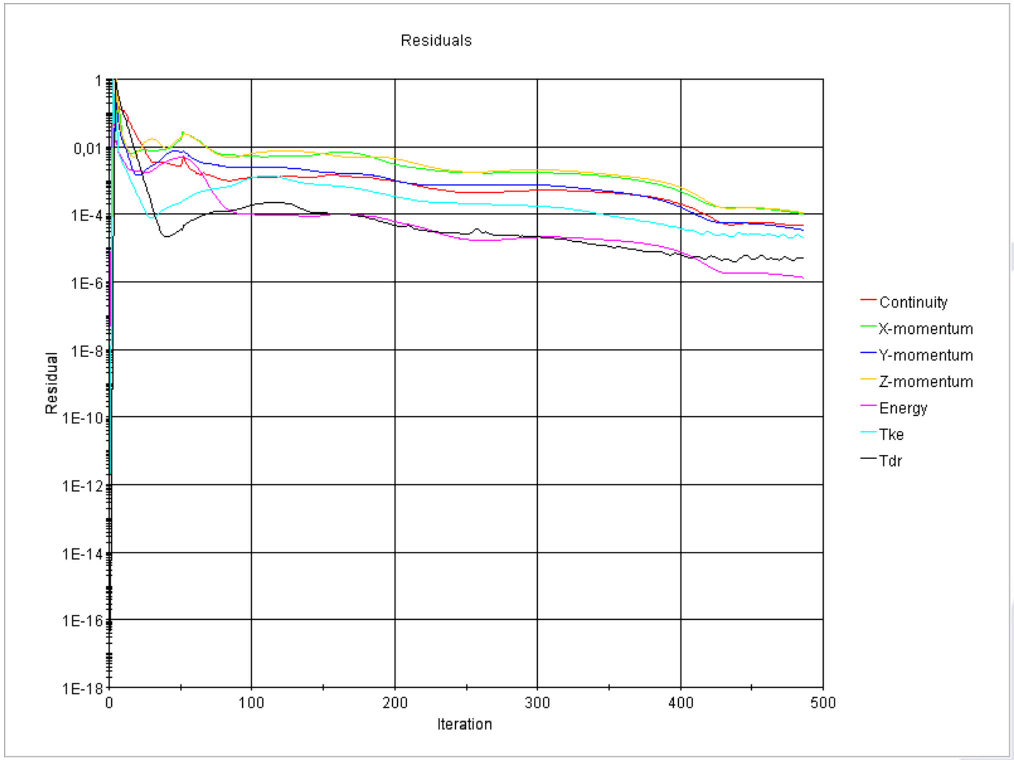
Automation – The physics and solver setup

- Physics setup
  - Steady simulation
  - Fluid properties : Air
  - Equation of state : Ideal gas
  - Viscous regime : Turbulent, K-Epsilon high Reynolds

- Solver setup
  - AMG solver
  - 2nd order : velocity, K-Epsilon, Temperature

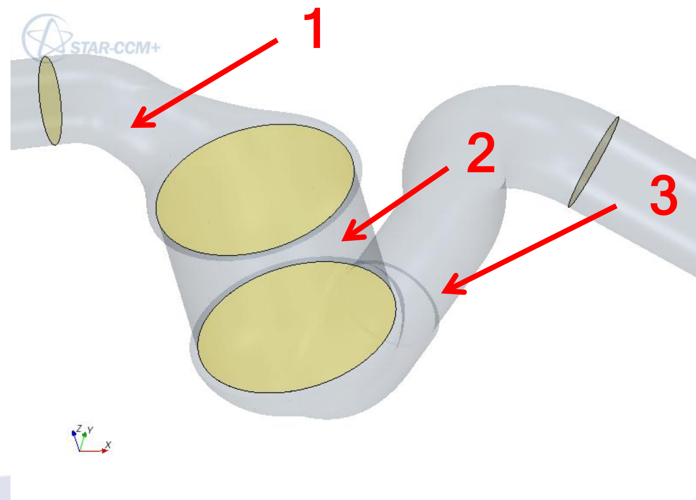
Automation – The run

- Less than 1 hour to converge on 2 CPU (100 iterations / 10 min) (laptop machine : Intel Core i7 Duo 2.67 GHz, 8Go RAM)



Automation – The post processing

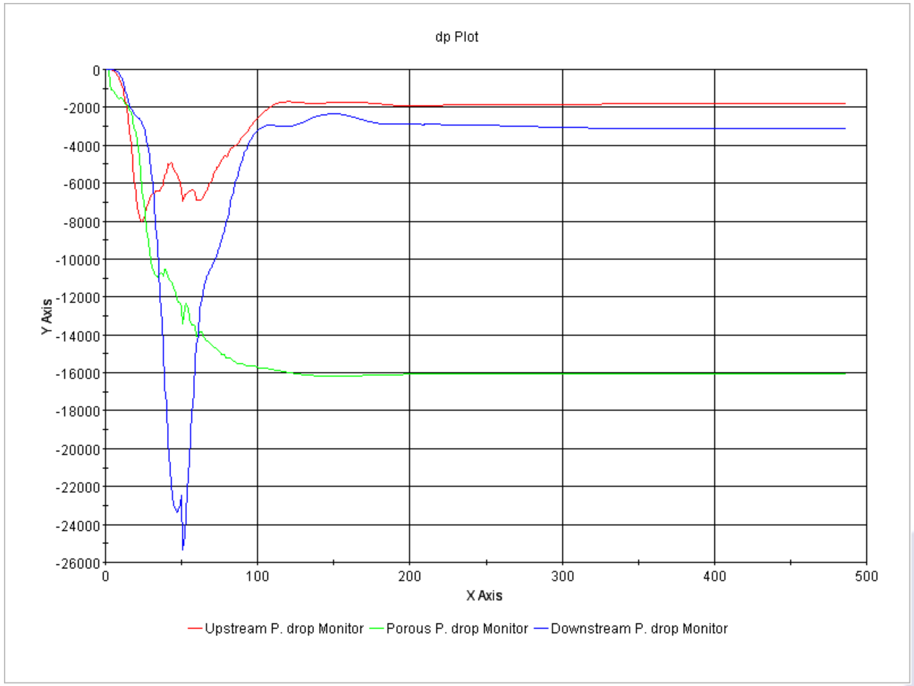
- Pressure drop
  - 1/ Upstream porous
  - 2/ Porous
  - 3/ Downstream porous

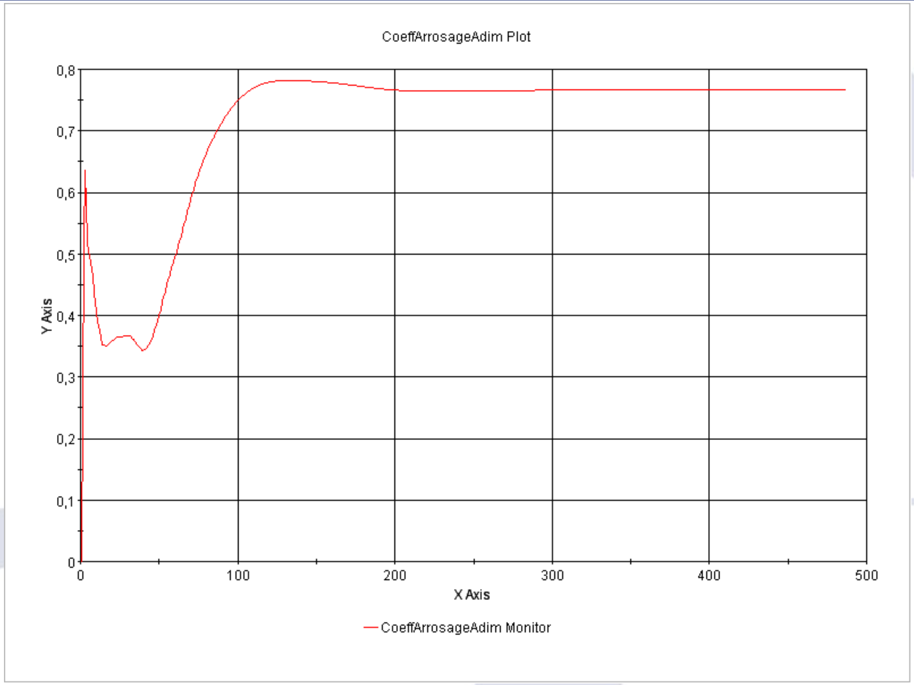


- Velocities uniformity normalized by the mean velocity

$$A = \frac{\sqrt{\frac{\sum (Vi - Vmean)^2}{Nbcells}}}{Vmean}$$

Automation – The post processing

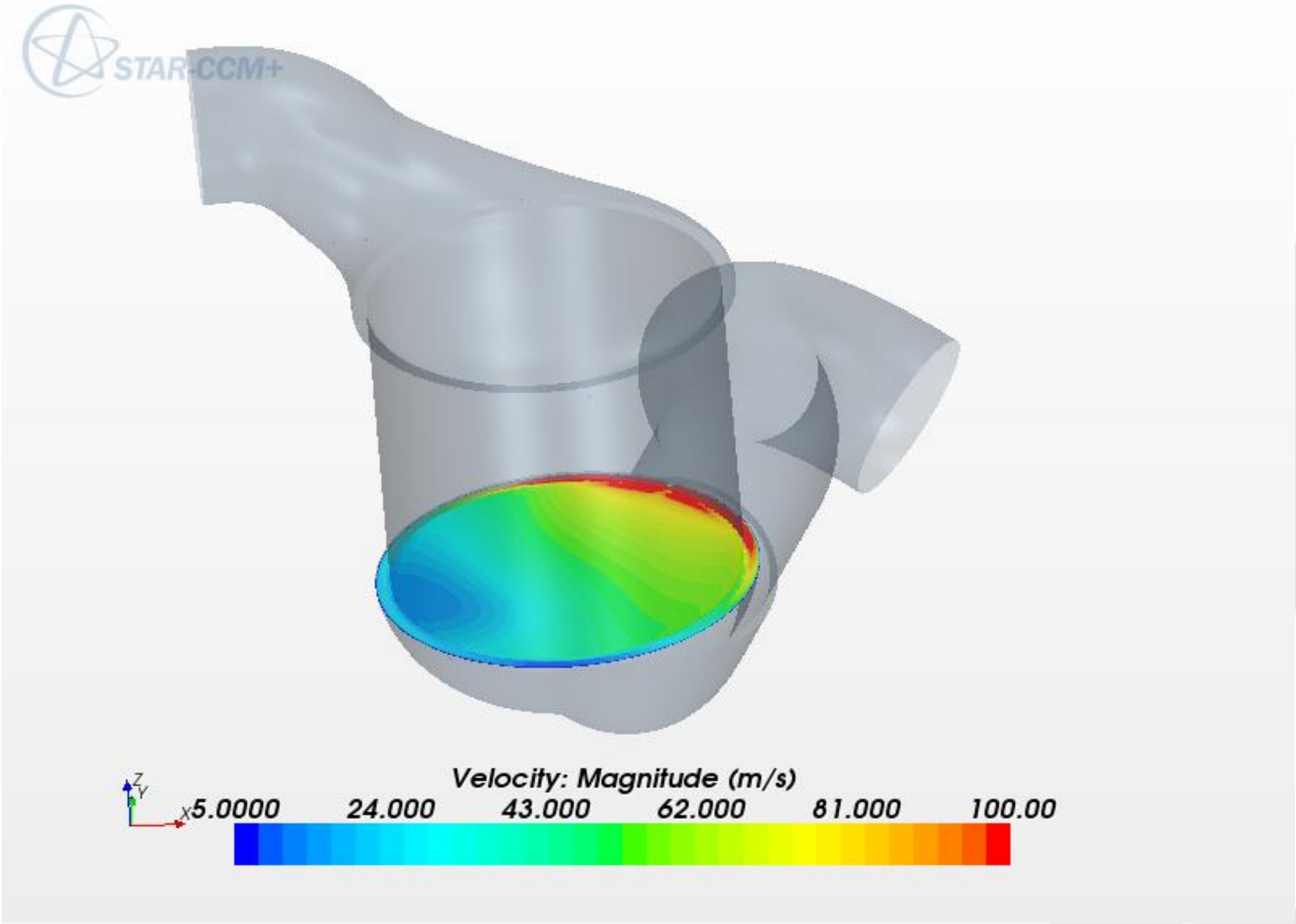




dp1 = 1810 Pa
dp2 = 16050 Pa
dp3 = 3090 Pa

A= 0.76

Automation – The post processing



Macro Java Parameters

```
// Mesh parameters  (size in m unit)
  double basesize = 3.E-3;
  double minsurfacesize = 33.3; // relative value (%)
  double max_cell_size = 5.E-3;
  double wrapping_factor = 50.0; // relative value (%)
  double prism_layer_thick = 0.75E-3;
  double prism_layer_thick_porous = 4E-3;
  int layer_numb = 1;
  //Upstream geometric extrusion
  double D_extr_Magnitude = 300E-3; // 5 times the inlet diameter
  int D_extr_NumLayers = 40;
  double D_extr_Stretching = 3.0;
  // Downstream geometric extrusion
  double Up_extr_Magnitude = 500E-3; // 8 time the outlet diameter
  int Up_extr_NumLayers = 70;
  double Up_extr_Stretching = 3.0;

  // Physic Parameters
  double T_in = 923.15; // K
  double V_in = 222.3; // m/s
  double Mass_flow_in = 0.26841; // kg.s-1
  double Cp_air = 1003.62; // J/kgK
  double Total_T_in = T_in + V_in * V_in / (2*Cp_air); // K
  double beta_axis = 2298.0; // kg.m-3.s in the Z local porous coordinate system
  double porous_conductivity = 0.02637; // W/mK

  // Maximum iterations
  int max_iter = 2000;
```

Conclusion

- The java macro creates the mesh, defines the physical and the solver setup and does the post-processing without user intervention.
- Any modifications on the moving zones surfaces are allowed.
- The automated process can be used to evaluate optimisation proposals

Do you have any questions ?