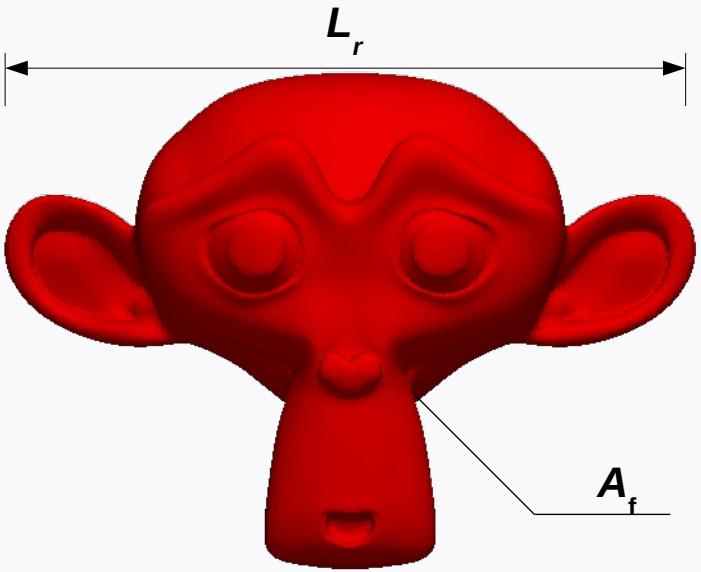
Suzanne immersed in an air flow

VELOCITY EFFECTS ON SUZANNE DRAG COEFFICIENT

Author: **Vincenzo Sammartano**, PhD
Affiliation: Independent Researcher
Email: v.sammartano@gmail.com

WorkFlow and CFD choises

- Removed the holes on Suzanne head by means of Blender;
- Estimation of the projected surface area A_f of Suzanne normal to the direction of flow (Paraview);
- Selection of the reference lenght L_r for Reynolds number estimation;



- **A_f = 2.4667 m^2 ;**
- **L_r = 2.7 m;**

- Selection of fluid type in its normal conditions (NTP):

Air in normal conditions

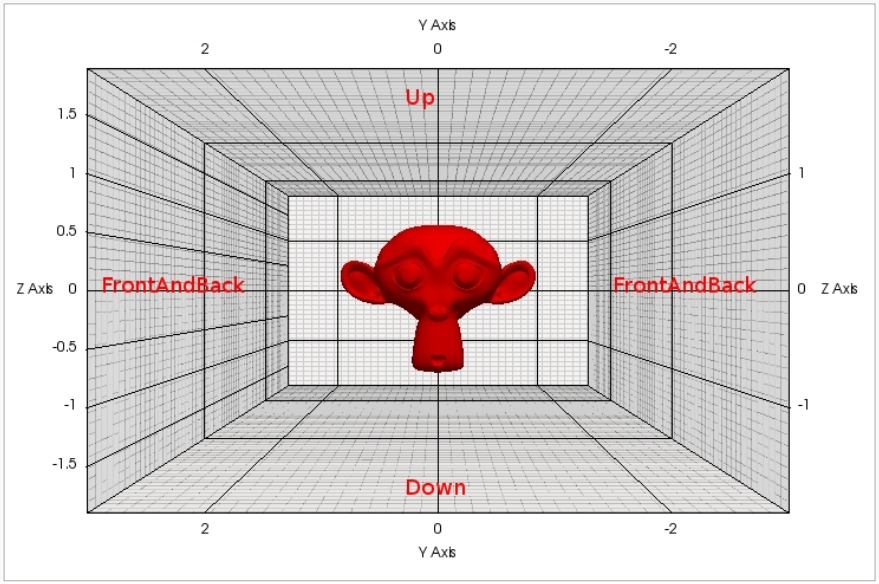
Variable	Value	Unit
Temperature - T	20.00	°C
Absolute pressure - p_{atm}	1.00	atm
Density - ρ	1.204	kg/m^3
Specific Volume - γ	11.81	N/m^3
Dynamic Viscosity - μ	1.72e^{-5}	Pa s
Kinematic Viscosity - ν	1.43e^{-5}	m^2/s

WorkFlow and CFD choises

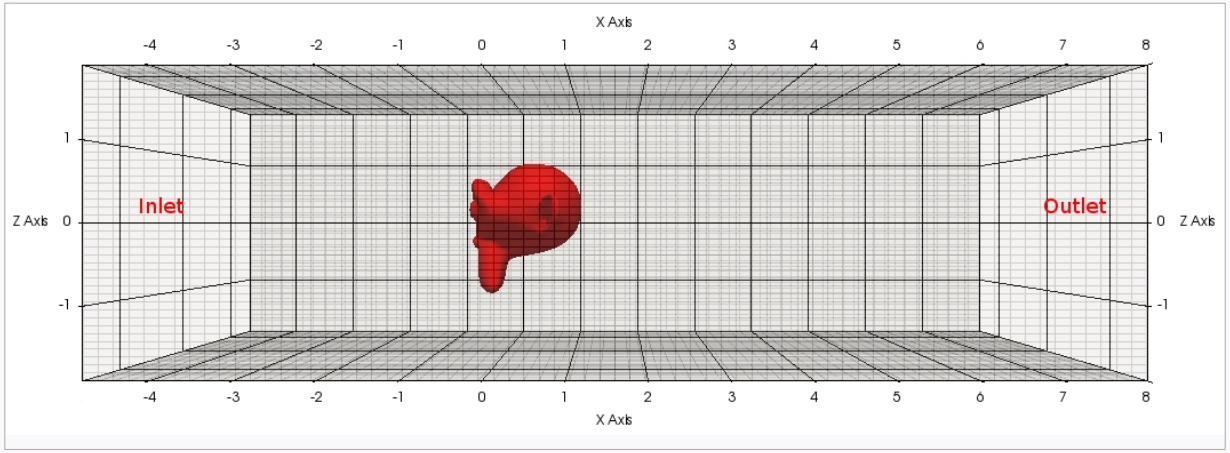
- Domain discretization by using blockMesh and snappyHexMesh:
  - ✔ **blockMesh** generates a base mesh with an **aspect ratio ≈ 1**;
  - ✔ **snappyHexMesh** discretizes the fluid domain around Suzanne's head;
  - ✔ Each run has been performed with a mesh such that **y+ < 5** over Suzanne's head;
  - ✔ Each mesh has a number of cells strictly dependent on the stream velocity value $\boldsymbol{U_\infty}$.

- The computational domain is a parallelepiped 12.8 m long, 6 m wide and 3.8 m tall:
  - Longitudinal direction : X Axis [-4.8, 8.0]
  - Vertical direction : Z Axis [-1.9, 1.9]
  - Transversal direction : Y Axis [-3.0, 3.0]

Transversal view

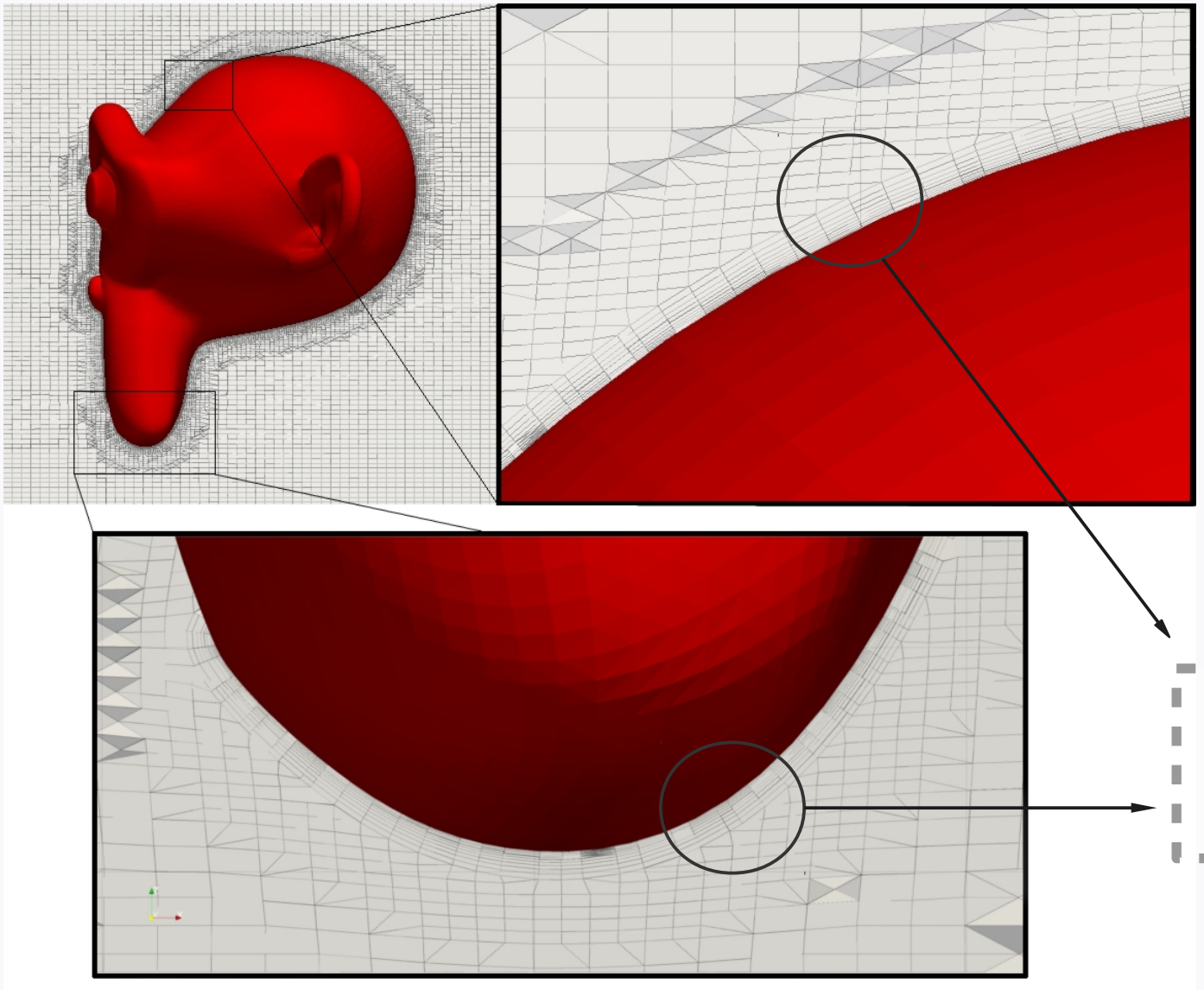


Longitudinal view



WorkFlow and CFD choises

- Example of a mesh generated with snappyHexMesh for the case study with U_∞ = **0.1 m/s**



Mesh Statistics:
- Cells: 2,770,230
- Max aspect ratio: 37.9
- Max skewness: 3.99
- **y+** min : 5.0e-7
- **y+** max: 3.191
- **y+** avg : 0.059

Multiple surface layers:
- Number of layes = 10
- Expansion Ratio = 1.2
- Final Layer Thickness = 0.4

CFD simulations set

- The drag coefficient C_d has been estimated for a wide range of the stream velocity U_∞ **= 1e^{-4} ÷ 5e^{+1} m/s**

- CFD simulations settings:
  - ✔ Category: **RANS**;
  - ✔ Closure model: **SST k-ω turbulence Model**;
  - ✔ steady-state solver: **simpleFoam**;
  - ✔ unsteady-state solver: **pimpleFoam**;

- For the considered stream velocity range the **drag coefficient** C_d was esistimated as follow:

$$C_d = \frac{(2 \cdot F_d)}{(\rho_{air} \cdot A_f \cdot U_\infty^2)}$$

being F_d the drag force.

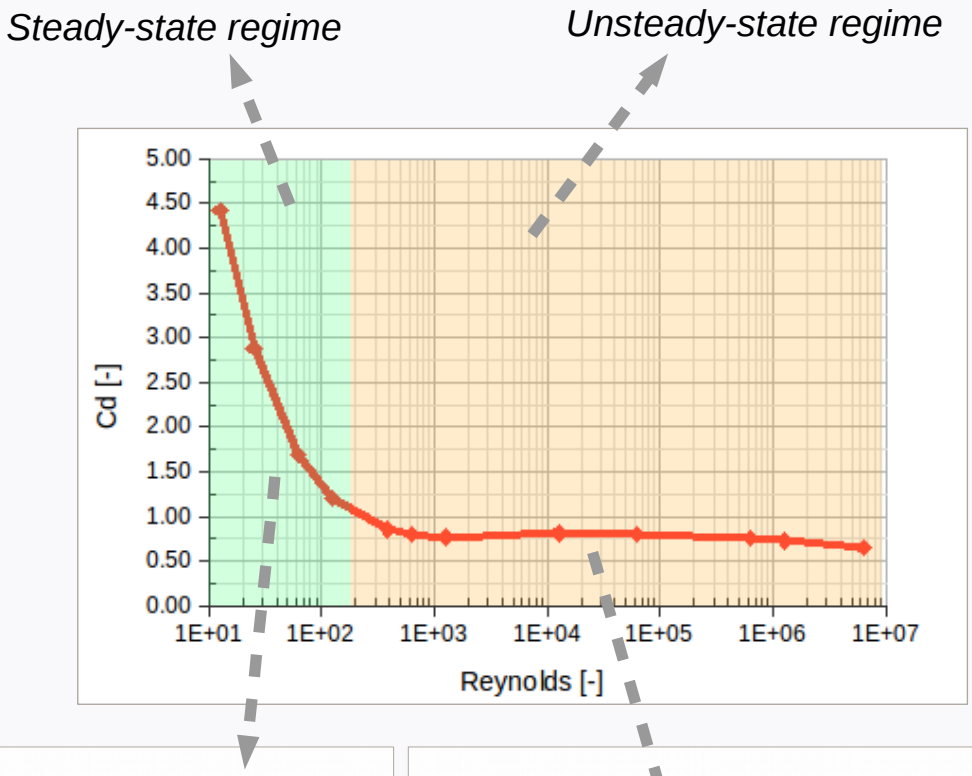
- Setting up the boundary conditions **BCs** according to the following table**:**

Patch	**Vars**	***U***	***p***	***k***	***nut***	***omega***
Inlet		freestream	freestreamPressure	fixedValue	calculated	fixedValue
Outlet		freestream	freestreamPressure	inletOutlet	calculated	inletOutlet
Face		noSlip	zeroGradient	kqRWallFunction	nutkWallFunction	omegaWallFunction
Down | Up		freestream	freestreamPressure	inletOutlet	calculated	inletOutlet
frontAndBack		freestream	freestreamPressure	inletOutlet	calculated	inletOutlet

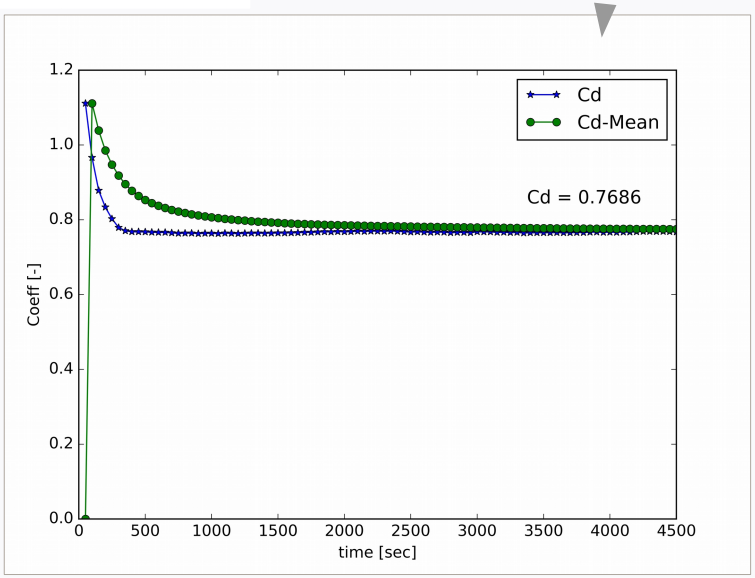
Results

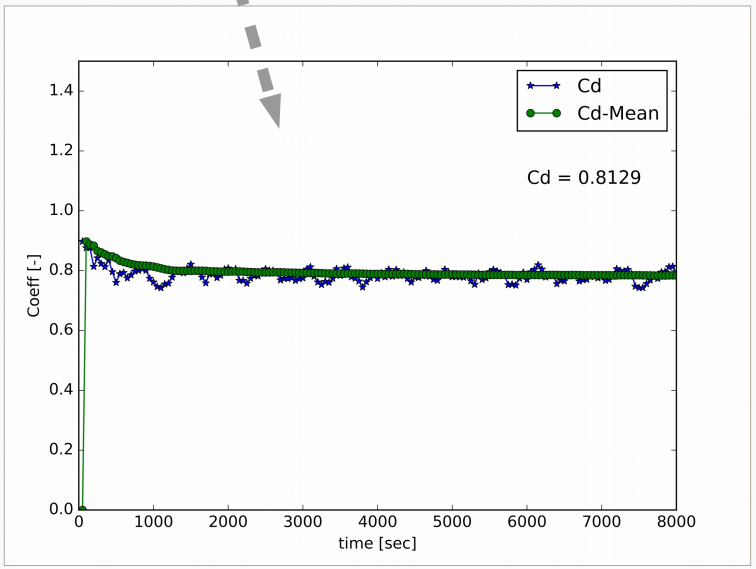
- Table and plot of the drag coefficient C_d values relative to different Reynolds number values.

U_∞ [m/s]	Reynolds [-]	C_d [-]
0.0001	1.89E+01	4.4211
0.0002	3.78E+01	2.8768
0.0005	9.45E+01	1.6932
0.001	1.89E+02	1.2085
0.003	5.67E+02	0.8547
0.005	9.45E+02	0.7937
0.01	1.89E+03	0.7686
0.1	1.89E+04	0.8129
0.5	9.45E+04	0.7949
5	9.45E+05	0.7604
10	1.89E+06	0.7288
50	9.45E+06	0.6558



- Time series of the drag coefficient C_d of two simulations in different flow regime.





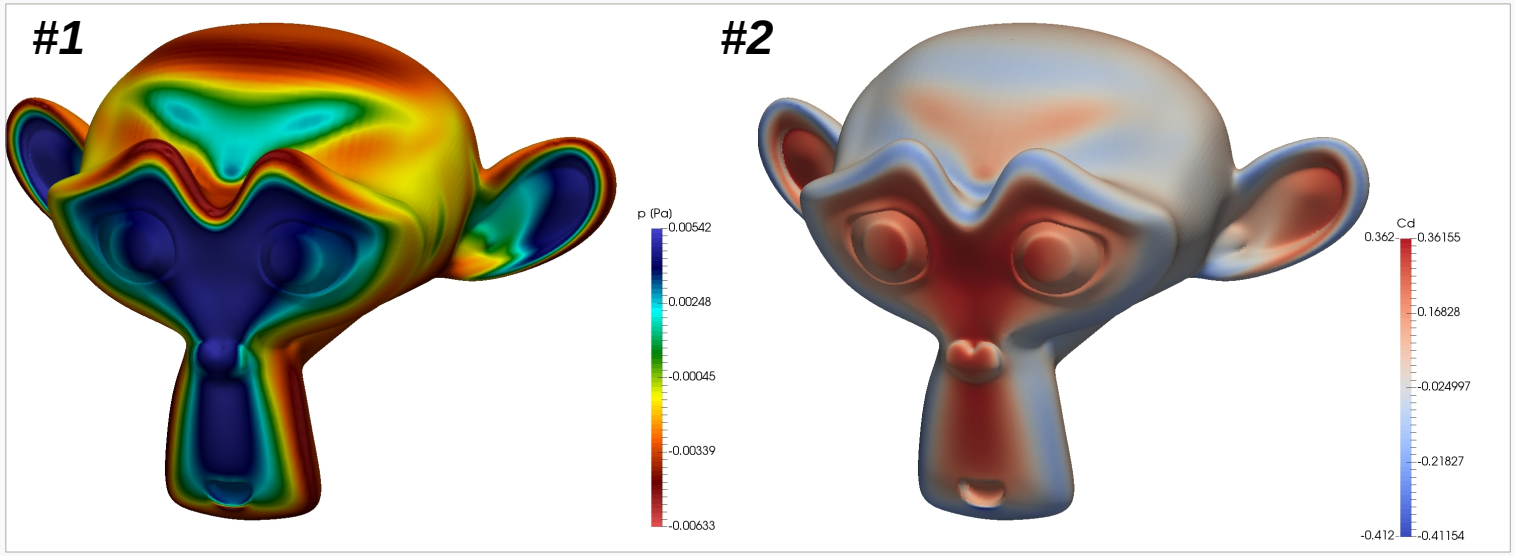
Results

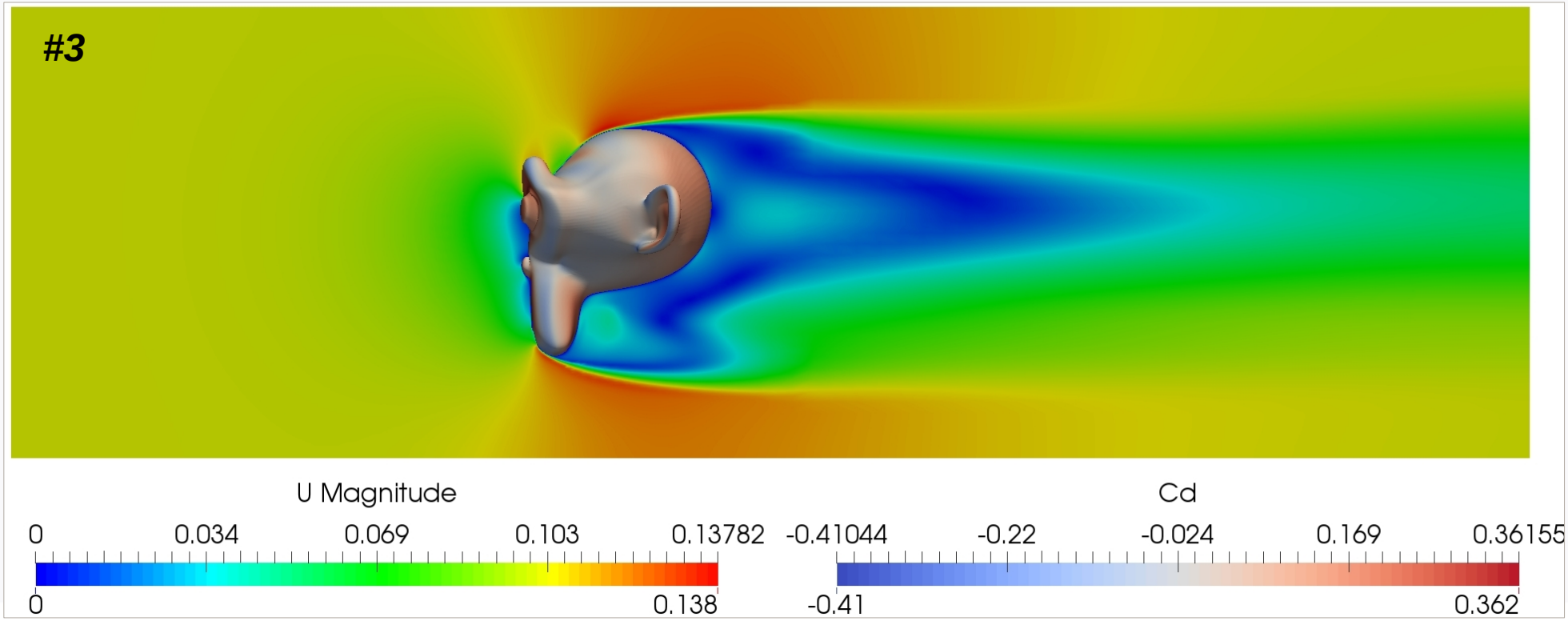
- Contour plots of pressure, drag coefficient and velocity magnitude for the case study with U_∞ = 0.1 m/s

#1 Pressure

#2 Cd

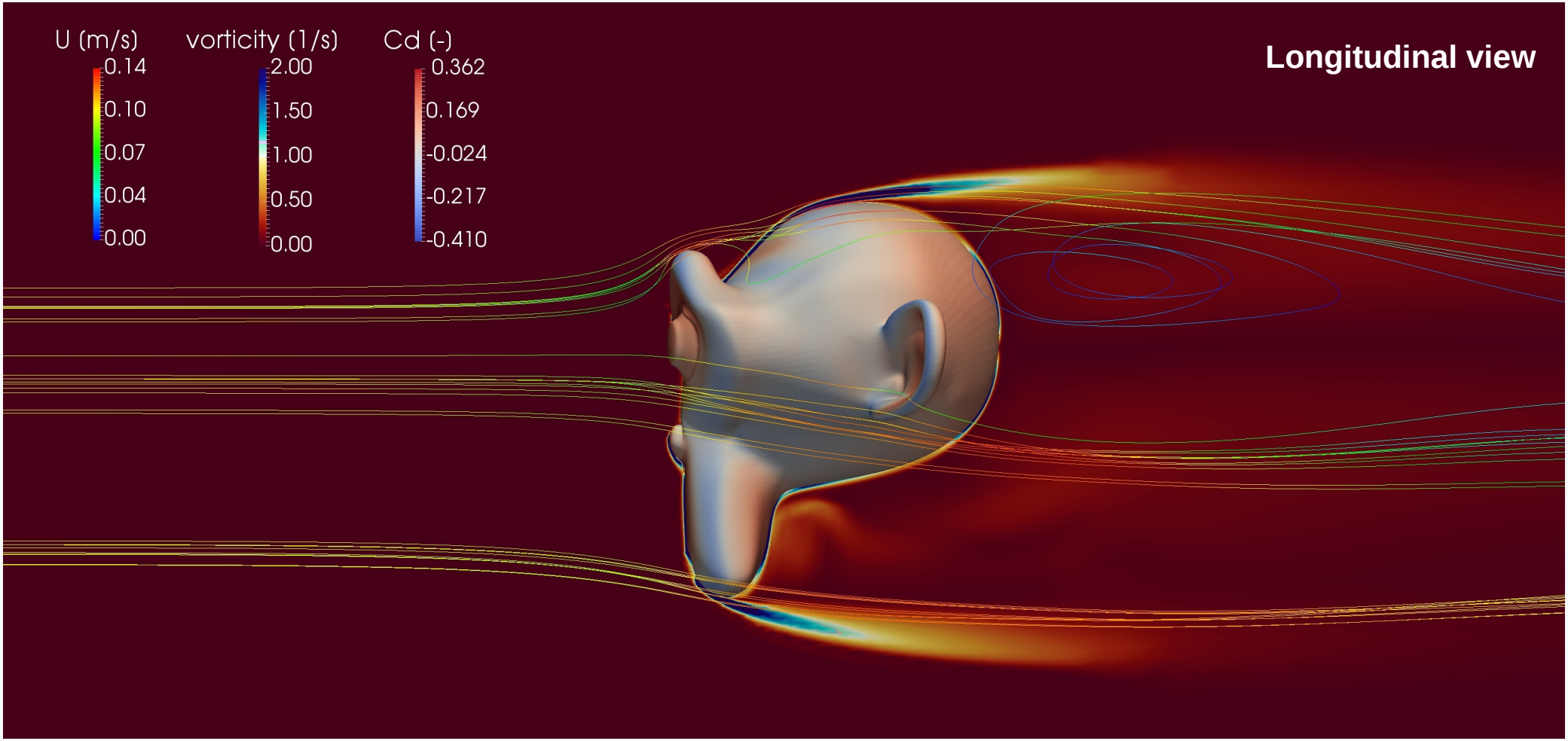
#3 U magnitude





Results

- Contour plot of vorticity and stream lines for the case study with $\boldsymbol{U_\infty}$ **= 0.1 m/s** - Longitudinal view



Results

- Contour plot of vorticity and stream lines for the case study with $\boldsymbol{U_\infty}$ **= 0.1 m/s –** Frontal view