The Christmas Community Challenge 2017

This PDF provides some information about the steps done in order to calculate the drag coefficient of our sweetheart named Suzanne; it is the monkey head of the open source Blender® 3D render/manipulation program.

Prologue

***C**ontributor: Tobias Holzmann*

***D**edicated to the OpenFOAM® community for further improvement in the field of numerical simulations by using open source toolboxes such as OpenFOAM®, Blender®, ParaView® and Salome®.*

***T**he case files and all necessary information can be found in the training video number seven accessible at Holzmann CFD. The description given in the next pages are kept short. To rebuild the case one is referred to the training videos and the corresponding training case.*

***E**njoy and keep foaming*

***G**enerated by LaTeX on November 23, 2017*

The Suzanne Case - 2018

The following document provides basic information that are required in order to rebuild the Suzanne case. As already mentioned, the description is kept short based on the available training videos and published case.

The Challenge

The Competition was invented by Jozef Nagy. The challenge is to calculate the drag coefficient of Suzannes head by using the open source toolbox OpenFOAM®. There are two constrains that has to be fulfilled. a) The flow direction has to be normal to Suzannes face and the geometry has to be the Blender® monkey head named Suzanne. Other quantities such as the flow type (laminar, turbulent), the velocity, the dimensions, the used fluid etc. are defined by the user.

The Geometry - Suzannes Head

The monkey head is generated by using Blender®. The main focus during the geometry generation is getting a smooth surface while taking care of the water proofness of Suzannes head. Thus, one has to be more familiar with the software tool Blender® or using different approaches to close the gaps. In this set-up, Blender® was used to generate the geometry, smooth the surface and close the gaps.

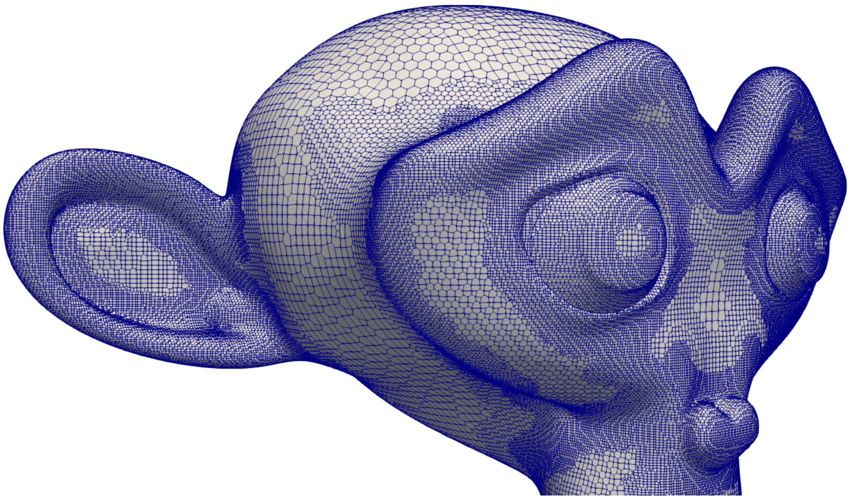
The generation of the monkey head is an easy task. The surface refinement can be done by using different approaches as already mentioned. However, in Blender® there are more possible ways to do the surface smoothing too. Three of them are discussed in the training videos published at Holzmann CFD. The given link provides all necessary information. Furthermore, if one is interested in the toolbox Blender®, you might find additional tips and tricks there especially in 7.2 where I demonstrate how to calculate the projected area of the monkey head.

The Numerical Mesh

The triangulated monkey head is exported as STL file and subsequently used by the OpenFOAM® meshing application snappyHexMesh. The background mesh was set-up by transforming information from Blender® to Salome®. The bounding box information in Blender® are used to build the 3D box in Salome®. The box is used in the meshing module to build the background mesh by applying the hexaedral algorithm.

The set-up of the snappyHexMesh control dictionary and all related information are given in the training videos mentioned above and can be checked in the published case. The meshing result which contains about 1.4 Million cells is depicted in the two figures above. There are no special treatment and investigation for the boundary layer (y^+) based on the fact that the simulation is run in an laminar flow. Thus the viscose sub-layer exist but has a smooth transition to the internal field. However, the mesh should be still to coarse around our monkey head to have a good approximation. The limitation of the mesh size (based on my personal computer) and the available time are the main reasons for neglecting the influence of the y^+ value. Furthermore, it is a fun case and one could be interested in how accurate the calculation of the drag coefficient is compared to the other contributors.

The mesh size (bounding box) is (-0.25 -0.25 -0.2) (0.25 0.75 0.2).



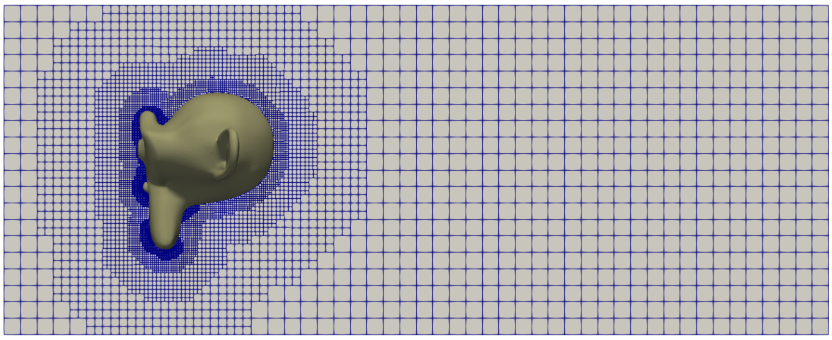


Figure 0.1: The numerical mesh generated by snappyHexMesh; no layers were generated

The Numerical Set-up

The set-up of the numerical simulation is as follows. The flow characteristic is laminar and the velocity was set to be 0.02 m/s. The velocity was estimated by calculating the hydraulic diameter (characteristic length scale) of Suzannes head while a Reynolds number around 255 was assumed. It follows:

$$\mathrm{Re} = \frac{U \mathrm{d}_{\mathrm{hyd}}}{\nu} \ ,$$

and thus the velocity is the one given above (the fluid medium is air). The characteristic length — or here the hydraulic diameter — was estimated by using the projected area of the monkey head and its perimeter. The Blender® workaround about how to get the quantities is given in the training video. The definition is given as:

$$d_{\text{hyd}} = \frac{4A}{P} ,$$

while A is the projected area and P is the perimeter of the projected area. The side walls are slip boundary conditions. The inlet velocity is set to 0.02 m/s while a fixed pressure at the outlet is set. This is a basic numerical set-up and lead to a nice matrix system.

In addition a passive scalar equation is solved by using the powerful objectFunction library. The passive scalar is set to a fixed value (0) at the inlet and zeroGradient elsewhere. The monkey head was prepared with some feature patch in order to set the passive scalar to a value of one there. More information about that can be found in the training tutorials. The passive scalar was introduced in order to visualise the flow behaviour in a nicer way; this was not done at the end.

The pressure-momentum coupling is solved by using the SIMPLE algorithm. The matrix systems are solved by conjugated gradient solvers while using a incomplete lower upper preconditioner. For the pressure the AMG solver was used. The system is solved till the steady-state is reached by checking the convergence criteria. In addition, the residual function is used to get the residual plot during the run which is depicted in the figure above.

Drag Coefficient Calculation

The idea of the fun case was to start building nice vortex shedding behind Suzannes head by using a low Reynolds number. Unfortunately a stable flow occur and no vortex appeared. However, the drag coefficient calculation was done by using the *forceCoeffs* library and the result matches with the one given for a sphere. The formulation of the drag coefficient is defined as:

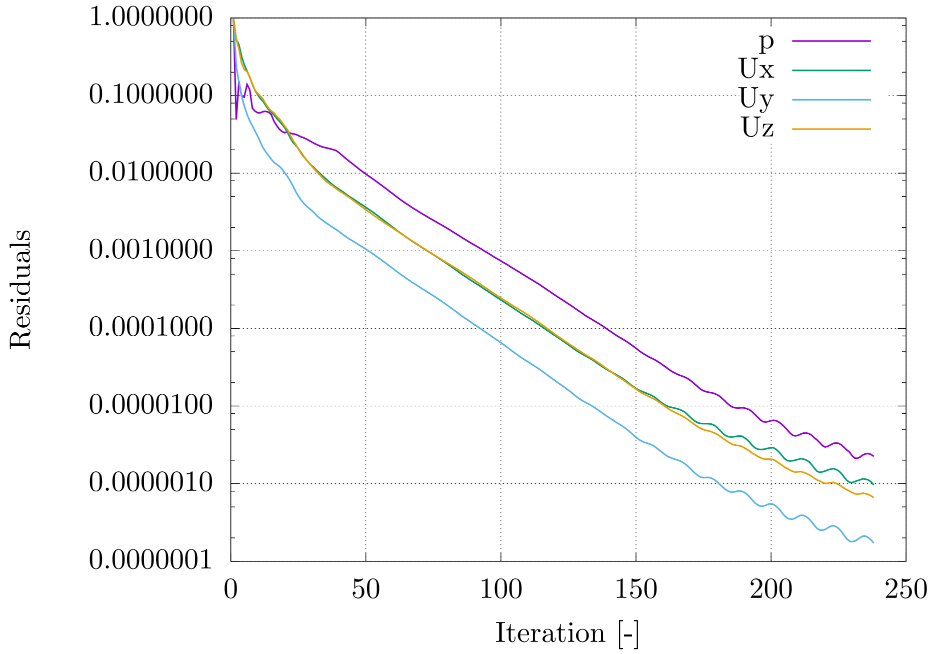


Figure 0.2: Residual behavior during the steady-state simulation

$$c_d = 2\frac{F}{\rho U^2 A_{proj.}}$$

Based on the fact that OpenFOAM® calculates only the forces at the surfaces, the other quantities that are necessary have to be set-up manually in the function definition (controlDict). The var field velocity is set to the inlet velocity 0.02 m/s and the projected area is 0.02562 m^2.

Results

The flow around the monkey head is visualised by vectors and streamlines in the pictures below. Furthermore, the drag coefficient (plotted against the iteration number) is given too. As one can see, after the stable flow field is reached, the drag coefficient keeps a constant level of $c_d \approx 1.14$. After 100 iterations the flow is almost stable. The results were achieved by using the OpenFOAM® Foundation version 5.x.

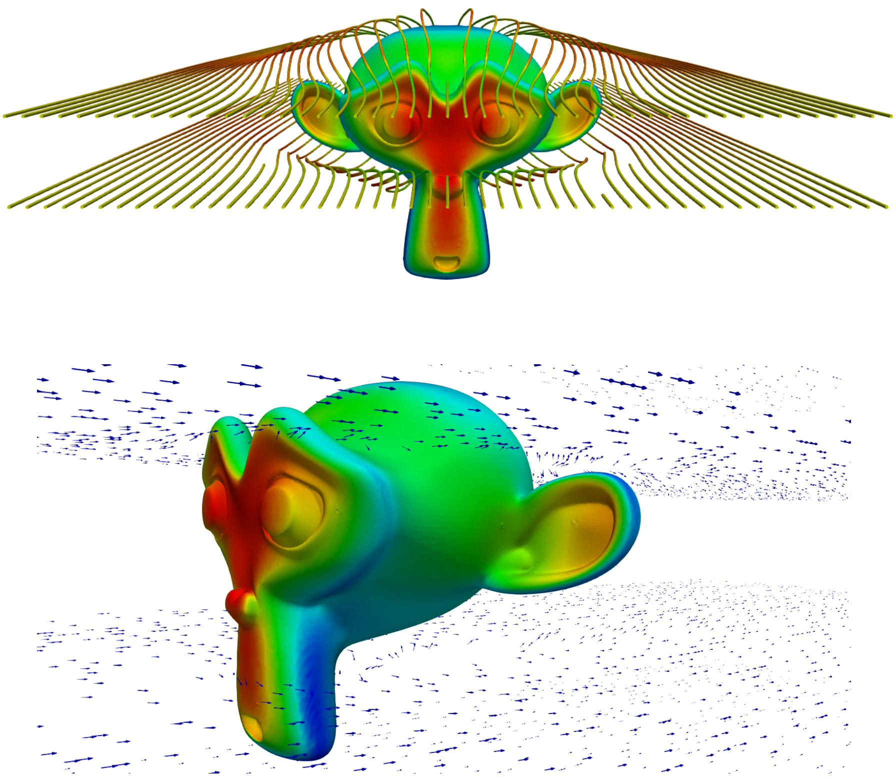


Figure 0.3: The pressure profile at Suzannes head and some vectors given in the top figure while a streamline profile is given below. The units are not given because it is a fun project and the case can be downloaded

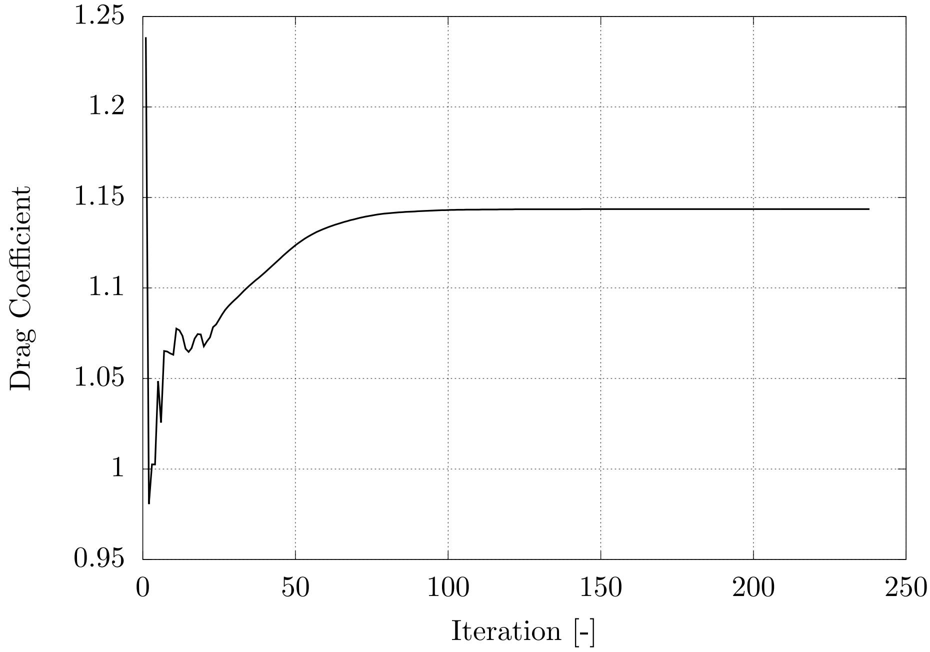


Figure 0.4: Suzannes drag coefficient evolution; The steady-state value is 1.41 for Re=255

Epilogue

The project was done just for fun without investigating to much time. However, I hope you enjoyed the training videos and the results that were achieved here. I am sorry that the description is kept very short but actually there is no time to describe things in more detail. Thus, the video training session were build. As already said, I was hoping to get some vortex shedding (no turbulent eddies) by using a low Reynolds number which should be around 50 here. However, this was not achieved and can be related to the very small domain (the side walls can influence the internal field), mesh resolution, wrong numerical set-up, a longer simulation time (in order to establish the vortex shedding or simply — there is no vortex shedding behind the Suzannes head such as the pitzDaily case that shows no vortex shedding but turbulent eddies. I refer to the tutorials of Jozsef Nagy.

Thank you for everything and keep foaming. Tobias Holzmann