The Christmas Community Challenge 2017

This PDF provides some information about the all 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 file and all necessary information can be found in the training screencast session number eight 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. In addition the case can be downloaded at the given website too.

__E__njoy and keep foaming

__G__enerated by LaTeX on November 12, 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 in order to build the geometry, smooth the surface and close all gaps.

The generation of the monkey head is an easy task. The surface refinement can be done by using different approaches. 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.

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®. Based on the background bounding box information, the background mesh was build by an pure hexaedral algorithm in the Salome® meshing module.

The set-up of the snappyHexMesh control dictionary and all related information are given in the training videos mentioned above. The meshing result which contain about 1.4 Million cells is depicted in the figures above.

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

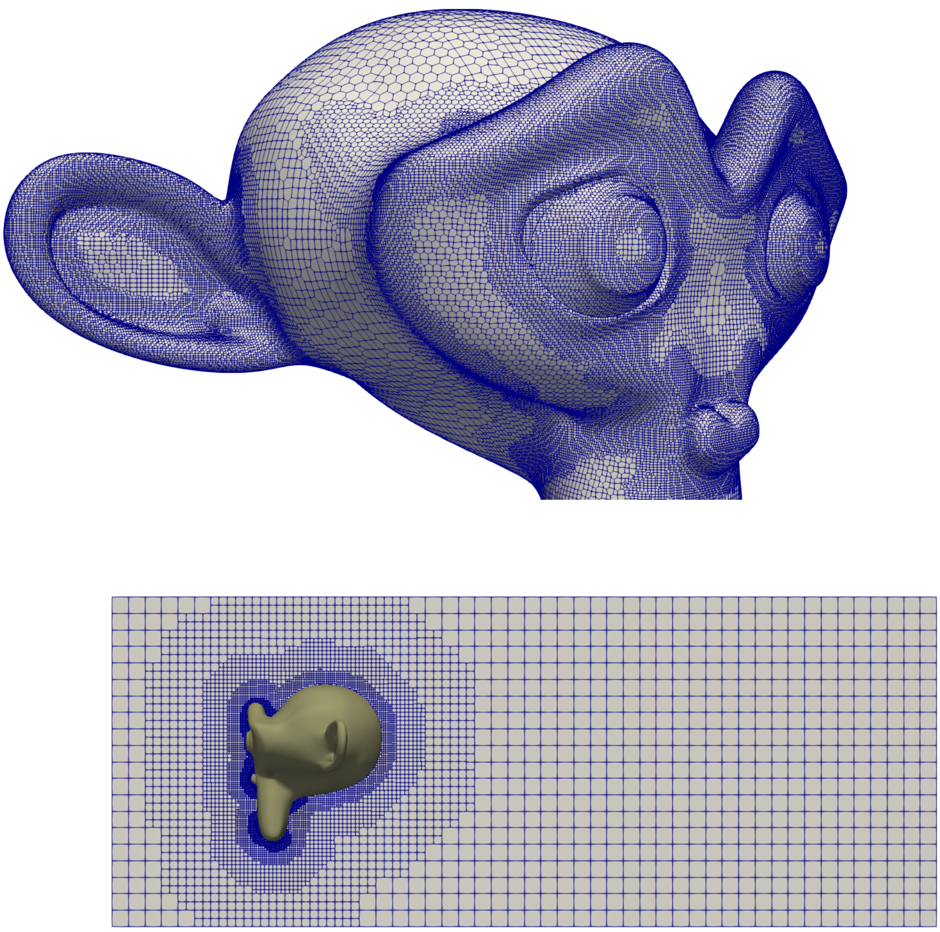

Figure 0.1: The numerical mesh generated by snappyHexMesh; no layers were generated

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 the hydraulic diameter of Suzannes head while a Reynolds number around 50 was assumed (I hope that I used 50 here :)). Based on the Reynolds number definition, the velocity was calculated. However, further information are needed such as the characteristic length and the fluid viscosity. For the characteristic length, the projected area of the monkey head has to be evaluated. The Blender® work flow about how to calculate the projected area can be found in the training videos. The characteristic length is calculated by the hydraulic diameter of Suzannes head.

The surrounding fluid is assumed to be air. The side walls are slip boundary conditions and the outlet has an fixed pressure while the inlet velocity of 0.02 m/s were applied (standard set-up). At the beginning a dynamic pressure driven flow was set but not further investigated based on lack of time.

In addition a passive scalar equation is solved by using the powerful objectFunction library. The passive scalar is set to a fixed value 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 matrix system is solved by using the PISO algorithm. The matrix systems are solved by a conjugated gradient solver using the incomplete lower upper preconditioner. For the pressure the AMG solver was used. The system is solved till the steady-state is reached. Based on the residual controls, the solver stops after the converged criteria are reached. In addition, the residual function is implemented to get the residual plot during the run. The residual plot is given below.

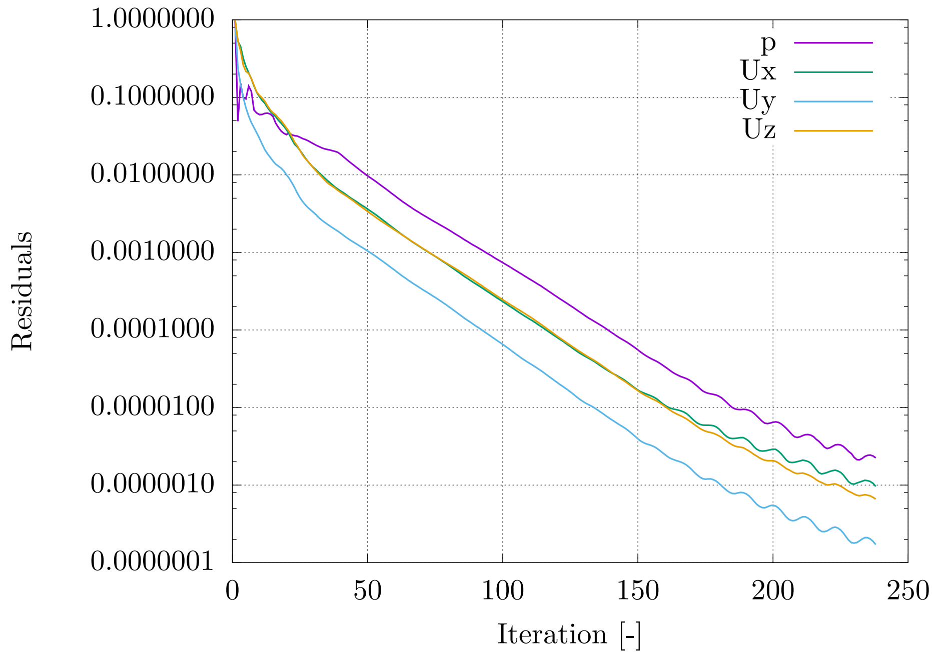

Figure 0.2: Residual behavior during the steady-state simulation

Drag Coefficient Calculation

The idea of the fun case was to build-up nice vortex shedding behind Suzannes head by using the low Reynolds number. Unfortunately a stable flow occur and no vortex appeared. However, the drag coefficient calculation was done by using the *forceCoeffs* library. The general formulation is well known and therefore not repeated here. Based on the fact that OpenFOAM® calculates only the forces at the surfaces, the other quantities that are necessary for the calculation of the drag coefficient have to be set-up manually. The quantities are the projected area of Suzannes head, the far field velocity and a reference length. The reference length was set to be the hydraulic diameter of Suzannes head. The var field velocity is set to the inlet velocity 0.02 m/s. As some further information: the projected area was calculated to be 0.02562 m^2 while the perimeter was 0.12743 m.

Results

The flow around the monkey head visualized by vectors and streamlines is given in the above pictures. Furthermore, the drag coefficient is given in the graphic. 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.

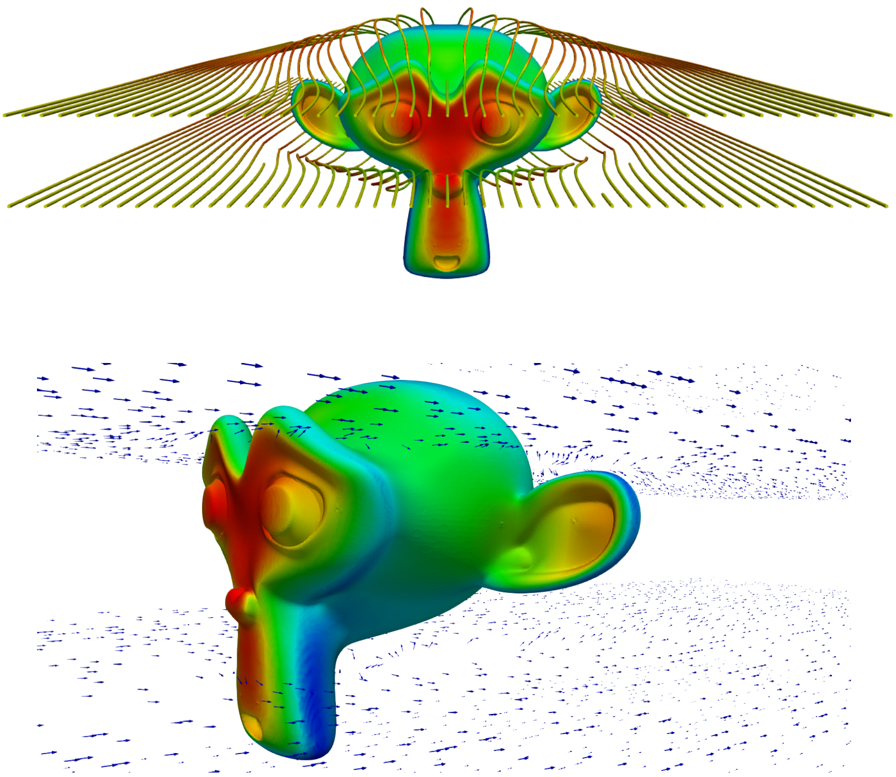

Figure 0.3: The pressure profile at Suzannes head and some vectors given in the top figure while a streamline profile is given below. The units are not given because it is a fun project and the case can be downloaded

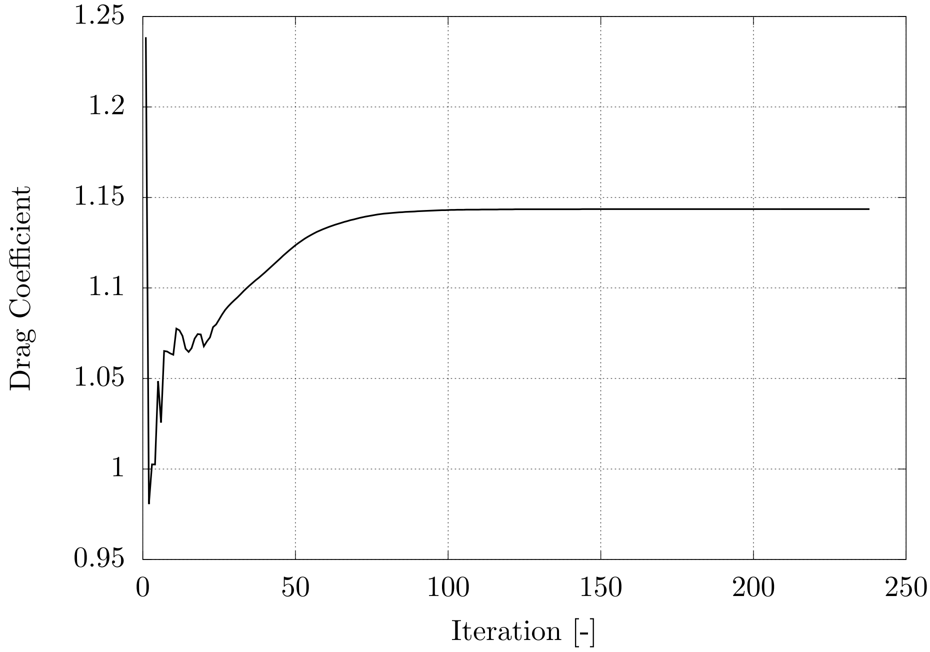

Figure 0.4: Suzannes drag coefficient evolution

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