Computational Fluid Dynamics of the Fuselage of a Model Speed Helicopter

Final Report

Project Group:

Philipp Bucher a1640637
Alexander Strbac a1634027
Michael Suetterlin a1641946

Table of Contents

1. Introduction

The project group consists of three exchange students from Germany. Two of them are aerospace students who are interested in Computational Fluid Dynamics (CFD) methods. The third member of the group flies model helicopters as a hobby. So, the group decided to do a project from this special field.

There is a great fascination in flying model helicopters all over the world because it is entertaining and fun. It is also a great activity to have contests with these model helicopters. One of the different challenges is to get the world speed record in flying with model helicopters. The latest world record in the class F5 Open (radio controlled flight) is held by the German team of BANSHEE Helicopters (***Banshee, 2013***). You can see an image of this helicopter in ***figure 1***.

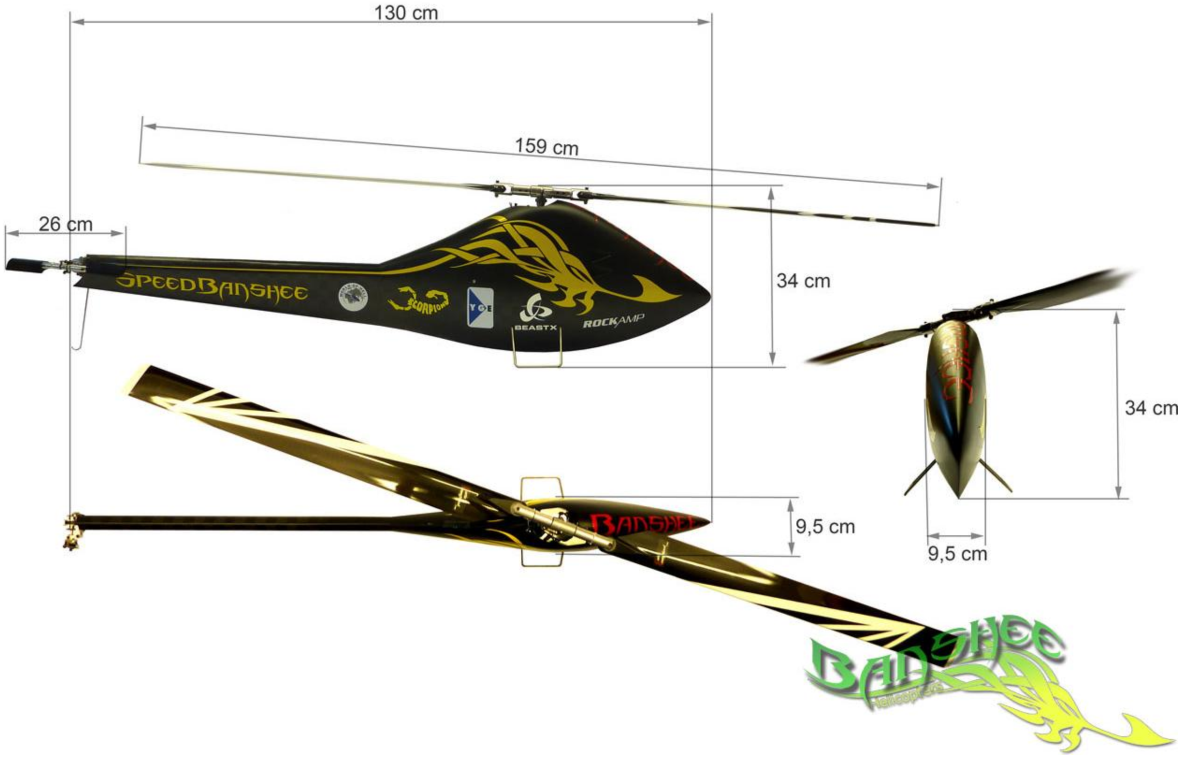


Figure 1: *Speedbanshee (three-dimensional side view)*
(http://www.banshee-helicopters.de/index.php/speedbanshee)

One group member found an article on the internet about a current similar project. They were on their way to building the fuselage of their model helicopter (see ***figure 2***). After asking them about some data of this fuselage the group designed a likely fuselage with ANSYS v14.0 DesignModeler. The reason for this was to easily vary the shape of the connection part of the tail boom of the fuselage. This project analysed the different shapes of the fuselage with CFD to find out which shape is best for obtaining the lowest drag coefficient and therefore the highest velocity. That will be a good starting position to build a model helicopter for breaking the speed world record.

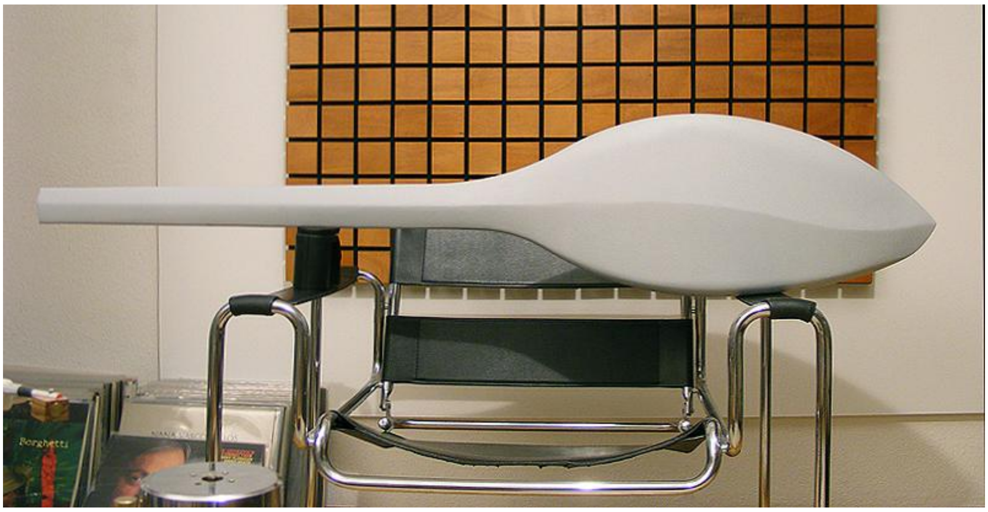


Figure 2: *Fuselage of 'Suzi Janis Speedprojekt'*
(http://www.rc-heli.de/board/showthread.php?t=221490&page=13)

Because of the complexity of the fluid flow around a whole helicopter we are only interested in the flow field around the fuselage. Therefore the main engineering problem of this project is to find the best shape of the fuselage related to breaking the world record for model helicopter. So, we analysed the drag force in relation to the geometry of the different shapes of the fuselage.

2. Numerical Methods

In this chapter we describe the numerical methods of our model and how we conceptualise the real-world problem into a simpler problem that can be solved using CFD.

2.1. Governing equations

CFD is fundamentally based on the governing equations of fluid dynamics. The governing equations describe mathematical statements of the conservation laws of physics (***TU, 2013***). The following physical laws are adopted:

- Mass is conserved for the fluid. (***continuity equation***)
- NEWTON's second law: The rate of change of momentum equals the sum of the forces on a fluid particle. (***momentum equations***)
- First law of thermodynamics: The rate of change of energy is equal to the sum of the rate of heat addition to and the rate of work done on a fluid particle. (***energy equations***)

All governing equations can be cast into a generic form in partial differential form (***TIAN, 2011***):

$$\underbrace{\boxed{\frac{\partial(\rho\Phi)}{\partial t}}}_{\text{Transient Term}}+\underbrace{\boxed{\frac{\partial(\rho u\Phi)}{\partial x}+\frac{\partial(\rho v\Phi)}{\partial y}+\frac{\partial(\rho w\Phi)}{\partial z}}}_{\text{Convection Term}}=\underbrace{\boxed{\frac{\partial}{\partial x}\left[\Gamma\frac{\partial(\Phi)}{\partial x}\right]+\frac{\partial}{\partial y}\left[\Gamma\frac{\partial(\Phi)}{\partial y}\right]+\frac{\partial}{\partial z}\left[\Gamma\frac{\partial(\Phi)}{\partial z}\right]}}_{\text{Diffusion Term}}+\underbrace{\boxed{S}}_{\text{Source Term}} \qquad (1)$$

One conservation law is that matter may be neither created nor destroyed (**Mass Conservation**). One can derive the ***continuity equation*** from the generic form with setting of $\Phi = 1$ and $S = 0$. Then from (1) it follows:

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \qquad (2)$$

For the ***momentum equations*** one considers NEWTON's second law of motion. It states that the sum of forces acting on a fluid element equals the product of its mass and the acceleration of the element. For the u momentum one can derive this equation from the generic form with setting of $\Phi = u, \Gamma = \mu$ (*dynamic viscosity*) and $S = -\frac{\partial p}{\partial x}$ (*pressure gradient*). Then from (1) it follows:

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho uu)}{\partial x}+\frac{\partial(\rho vu)}{\partial y}+\frac{\partial(\rho wu)}{\partial z}=\frac{\partial}{\partial x}\left[\mu\frac{\partial u}{\partial x}\right]+\frac{\partial}{\partial y}\left[\mu\frac{\partial u}{\partial y}\right]+\frac{\partial}{\partial z}\left[\mu\frac{\partial u}{\partial z}\right]-\frac{\partial p}{\partial x} \qquad (3)$$

In our special case with the speed model helicopter we have a steady state $\left(\frac{\partial}{\partial t}=0\right)$ because we are not interested in any behaviour according to time but in the absolute value of the drag coefficient. Furthermore this problem is 3-dimensional and incompressible. To verify the incompressibility we used the equation for the MACH number (***TIAN, 2011***) because one can assume an incompressible flow with $Ma < 0.3$ (***TU, 2013***):

$$Ma=\frac{u}{c}=\frac{u}{\sqrt{(\gamma RT)}} \qquad (4)$$

Here *u* is the velocity (*u = 42 [m/s]*), *γ* is the ratio of specific heats of air at 25 [°C] (*γ = 1.401*) (***EngineeringToolBox, 2013***), *R* is the specific gas constant of air (*R = 286.9 [J/(kgK)]*) (***Engineering ToolBox, 2013***) and *T* is the temperature (*T = 25 [°C] = 298.16 [K]*) (***EngineeringToolBox, 2013***). With all the values we calculate a MACH number of *Ma = 0.121*, i.e. incompressible flow ($\rho = const.$).

Overall we have steady state, incompressible and 3-dimensional flow at a constant temperature ($T = 25\,^{o}C$). That leads to the following continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \qquad (5)$$

For the momentum equations in all dimensions we got

$$\frac{\partial(uu)}{\partial x}+\frac{\partial(vu)}{\partial y}+\frac{\partial(wu)}{\partial z}=\nu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)-\frac{1}{\rho}\frac{\partial p}{\partial x} \quad (6a)$$

$$\frac{\partial(uv)}{\partial x}+\frac{\partial(vv)}{\partial y}+\frac{\partial(wv)}{\partial z}=\nu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)-\frac{1}{\rho}\frac{\partial p}{\partial y} \quad (6b)$$

$$\frac{\partial(uw)}{\partial x}+\frac{\partial(vw)}{\partial y}+\frac{\partial(ww)}{\partial z}=\nu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)-\frac{1}{\rho}\frac{\partial p}{\partial z} \quad (6c)$$

where v is the kinematic viscosity $\left(\upsilon=\frac{\mu}{\rho}\right)$.

2.2. Hand Calculations

For this problem of a speed model helicopter we assumed a velocity of 42 [m/s]. With this value we expected a turbulent flow field. To determine this exactly we used the equation for REYNOLDS number (*Re*) regarding the characteristic length of a surface (***TIAN, 2011***):

$$Re_L=\frac{\rho UL}{\mu} \quad (7)$$

In this case the characteristic length of the surface of the helicopter is 1.4 [m]. The values of the surrounding air are the density ρ with a value of 1.185 [kg/m^3] (***ANSYS v14.0***) and the dynamic viscosity μ with a value of 1.831*10^{-5} [kg/(m*s)] (***ANSYS v14.0***).

This results in a *Re* number of $Re_L = 3.8*10^6$. This value is greater than the given value for turbulent flow in external flows (***TIAN, 2011***) of $Re_x \geq 5*10^5$, so the flow around the speed model helicopter is turbulent.

For the mesh generation it is necessary to calculate the distance between the wall and the first grid node y_1, i.e. the thickness of the very first layer (***TIAN, 2011***). ANSYS CFX recommends using the following formula to estimate the y_1 value based on the y^+ value

$$y_1=L*y^+\sqrt{74}Re_L{}^{-13/14} \quad (8)$$

where L is the characteristic length and Re_L is the REYNOLDS number based on that characteristic length (see above). The y^+ value is a dimensionless wall distance with respect to the local conditions of the wall. It is part of a modelling procedure (*wall functions*) which is required for near-wall models to handle wall-bounded turbulent flow problems (***TU, 2013***). For the standard turbulent k-ε model *wall functions* are always used and ANSYS CFX recommend y^+ values of

$$11.63<y^+<300 \quad (9)$$

For reasons of time saving in mesh generation we choose a y^+ value of $y^+ < 300$ because we do not need a very high accuracy in the analysis of the turbulent boundary layer. With all these values we calculated the y_1 value to $y_1 < 2.8$ [mm]. For the mesh generation we halved this value to $y_1 = 1.4$ [mm] to be secure and to capture almost all different values.

2.3. Computational domain of the flow

For this project we used ANSYS v14.0 and the CFD solver ANSYS CFX. We modelled the fuselage of the speed model helicopter with ANSYS DesignModeler to have the ability to change some parameters to compare the different results to get the best solution (see ***figures 3, 4***).

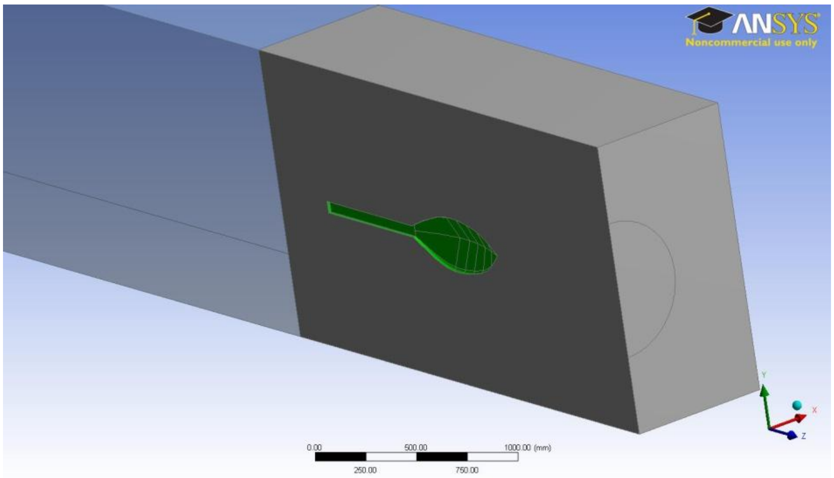


Figure 3: *Model of the fuselage of the speed model helicopter (short fuselage)*
ANSYS v14.0 DesignModeler

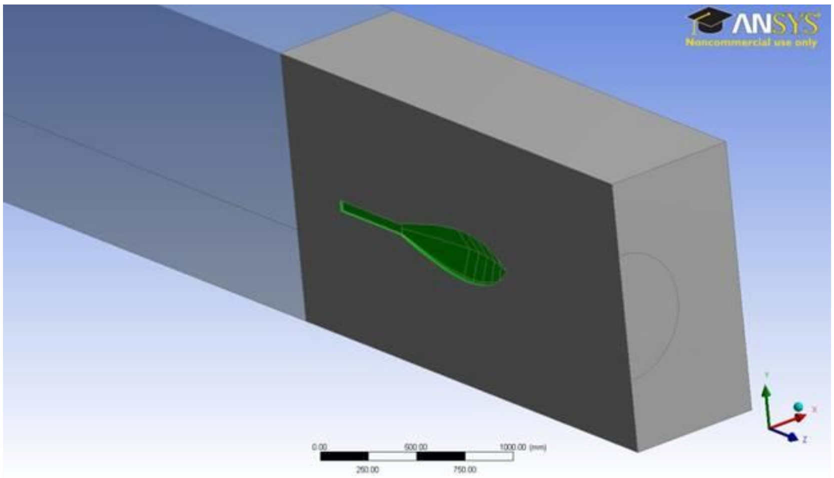


Figure 4: *Model of the fuselage of the speed model helicopter (long fuselage)*
ANSYS v14.0 DesignModeler

As shown in ***figure 3*** and ***figure 4*** we changed the length of the rear part of the fuselage in a range from 300 to 500mm.

The length of the fuselage is 1.4 [m]. So, we created a flow field around the helicopter with the dimensions 0.75 [m] x 1.5 [m] x 16.4 [m] (see ***figure 5***). We chose a computational domain of about ten times the length of the fuselage behind it because of the rule of thumb to consider all possible turbulent flow characteristics (***TIAN, 2011***).

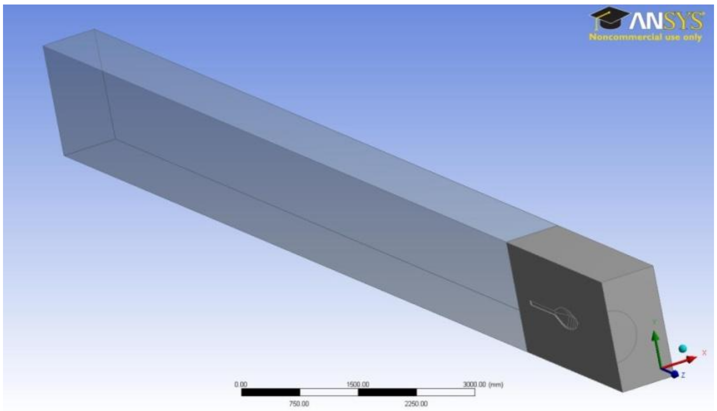


Figure 5: *Computational domain of the flow field and fuselage*
ANSYS v14.0 DesignModeler

For the numerical model we chose a steady state analysis type because we are interested in an absolute value of the drag coefficient and are not interested in time varying factors. For Material we chose Air at 25 [^{o}C] because that is a model helicopter that flies near the ground and not at a high altitude. The settings for the Fluid Domain we determined no heat transfer, non buoyant, initialisation: v = 42 [m/s], medium turbulence intensity of 5%, no combustion, no thermal radiation, turbulence wall function: scalable and no mesh deformation.

As boundary conditions we chose inlet, outlet, opening, symmetry for the flow field and wall condition (no slip wall) for the fuselage (see ***figure 6a, b***). We used a symmetry boundary condition because the fuselage is symmetrical and we expected a symmetrical flow field. We chose an opening boundary condition because the helicopter operates in the air and we are not interested in relations to any obstacles or side effects.

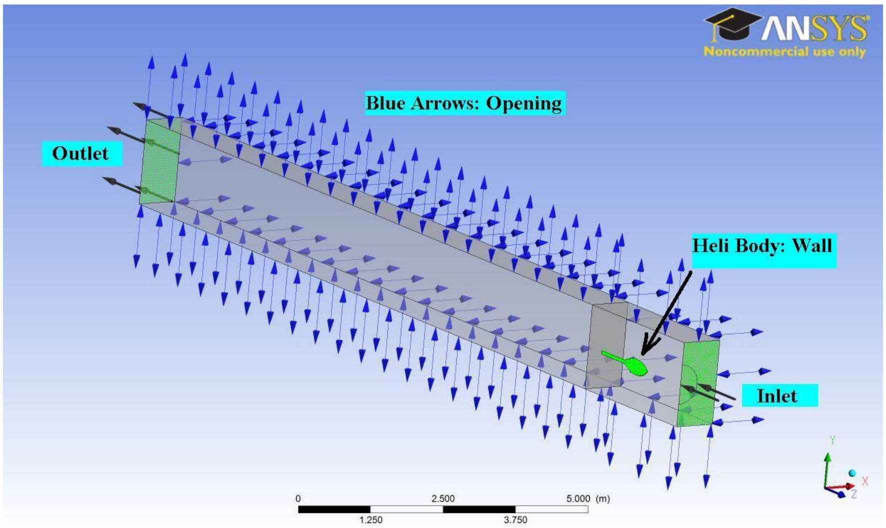


Figure 6a: *Boundary conditions, Part 1*
ANSYS v14.0 DesignModeler

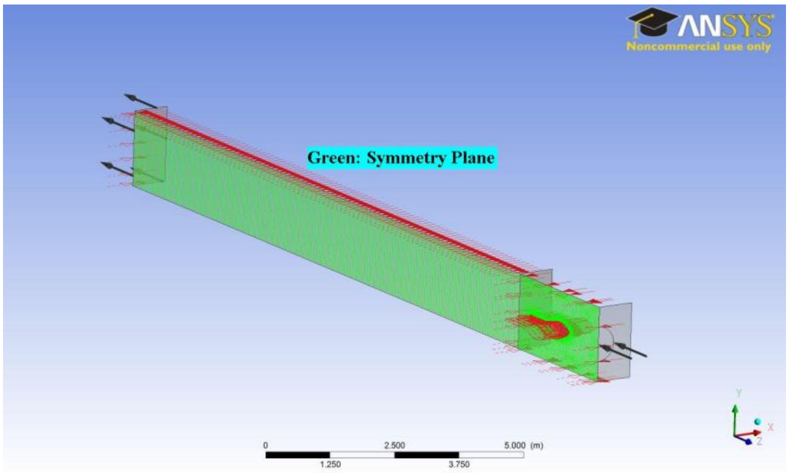


Figure 6b: *Boundary conditions, Part 2*
ANSYS v14.0 DesignModeler

To simulate the effects of the turbulence we chose the standard k-ε turbulence model with a medium turbulence intensity of 5%.

For convergence criteria we set the residual type to RMS and the residual target to $1*10^{-5}$ because this is a good convergence and usually sufficient for most engineering applications (***TIAN, 2011***). It is also important to define two numbers of iterations to have a control of the convergence. So, we needed a minimum number of iterations to prevent the solver from wrong results in case of the solver converging after only a few and obviously not enough iteration steps. We set the minimum number of iterations to 10. Also, we needed a randomly high number of iterations. Otherwise the solver may end the simulation before the results have converged. To improve convergence we changed the Fluid Timescale Control to physical timescale and set the value to 0.001[s].

The overall simulation converged after about 350 iterations (see ***figure 7a***) and in the drag coefficient there is no significant change after about 100 iterations (see ***figure 7b***).

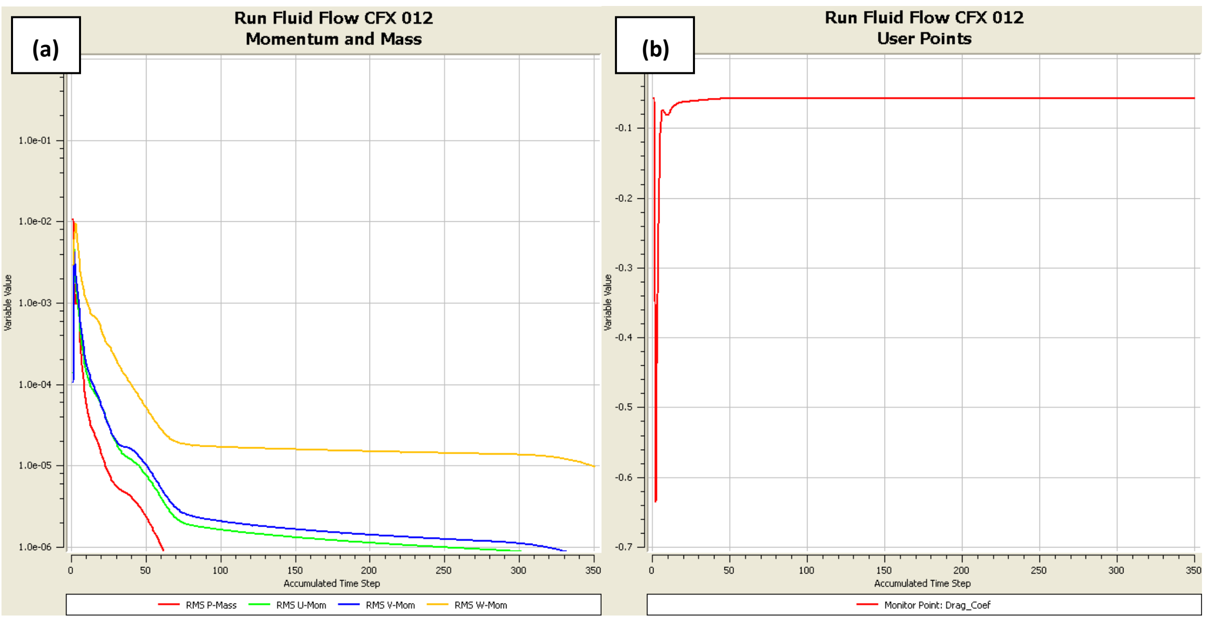


Figure 7: *Convergence plots of Momentum and Mass (**a**) and of Drag Coefficient (**b**)*
ANSYS v14.0 CFX CFD-Post

2.4. Mesh

2.4.1. Mesh generation and mesh refinement

For the entry region and the area around the helicopter fuselage we used mainly tetrahedron (Tet) elements to match the complex geometry. To achieve a structured hexahedron (Hex) mesh in the rest of the model, we separated the model into two bodies and formed one part again in the DesignModeler. The reason why we decided to separate the model is because the Hex-mesh is more suitable in the rear part of the model.

Because we are interested in the drag coefficient of the different shapes of the fuselage, we wanted to get the drag force as accurate as possible, because we needed it to calculate the drag coefficient. The results in the near wall region of the fuselage should be as accurate as possible. Therefore we modelled the boundary layer as a Hex mesh with inflation as well as the region around the fuselage with a very fine mesh, in general.

We chose the 'Fine' sizing method for the first mesh we want to create and tried to solve the model. But we could not achieve convergence, because the mesh was still to coarse. Therefore, the second step was to reduce the maximum element size of the whole model. In addition, we included a 'Face Sizing' in the area of the fuselage of the helicopter. We were now able to reduce the mesh size again in the area we were interested in, because the velocity gradient is very high there. After these refinements the solution converged the first time.

Because the mesh was still relatively coarse, we refined it three times to make sure that our model is mesh independent and to get more accurate results (see ***figure 8a, b***). We refined it by decreasing the maximum element size for the whole model and same for the 'Face Sizing' around the fuselage.

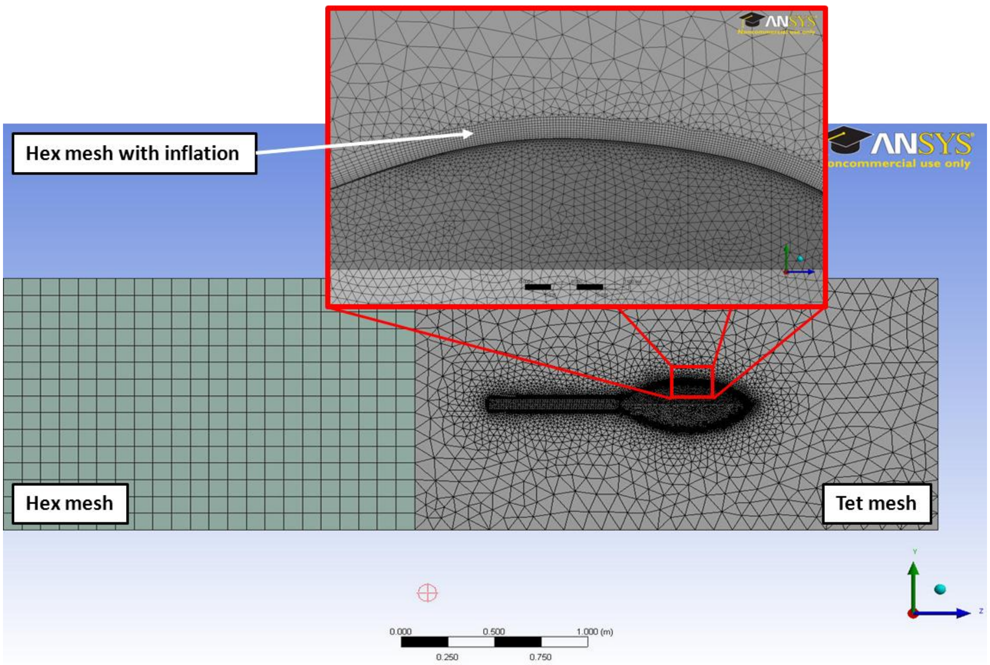


Figure 8a: *Mesh 1 (very coarse)*
ANSYS v14.0 Meshing

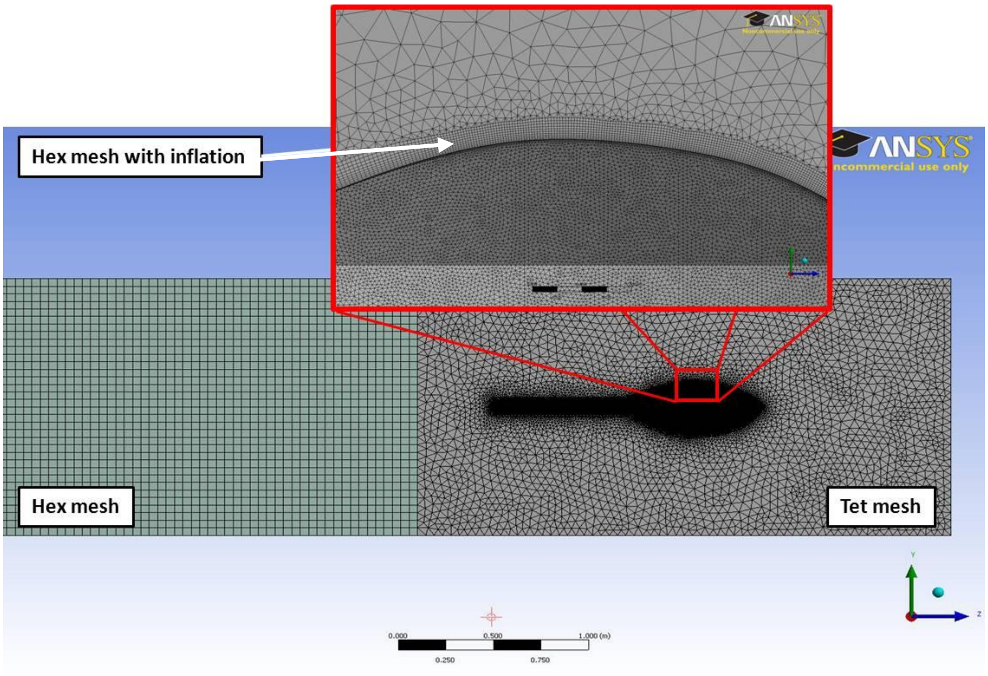


Figure 8b: *Mesh 4 (very fine)*
ANSYS v14.0 Meshing

After all these refinements we were able to do the calculations for the zero grid-space and the grid convergence index (GCI), which show that our mesh is good quality.

2.4.2. Grid convergence studies

We chose the drag coefficient as flow parameter for the grid convergence studies.

<u>*RICHARDSON extrapolation:*</u>

Because no computer is powerful enough to calculate a CFD model with zero grid space we used the equations of the RICHARDSON extrapolation to obtain the drag coefficient values at zero grid space (***TIAN, 2011***), where

f_0 = drag coefficient at zero grid space
f_1 = drag coefficient obtained from the third mesh refinement
f_2 = drag coefficient obtained from the second mesh refinement
f_3 = drag coefficient obtained from the first mesh refinement
f_4 = drag coefficient obtained from the first coarse mesh

r is the average refinement ratio
p is the order of convergence.

$$f_0 \cong f_1 + \frac{f_1 - f_2}{r^p - 1} \qquad (10)$$

$$p = \frac{ln\left(\frac{f_3 - f_2}{f_2 - f_1}\right)}{\ln(r)} \qquad (11)$$

We derived the values for the drag coefficient at zero grid space for every variation of the geometry (see ***table 1***).

Fuselage rear length	f_4	f_3	f_2	f_1	p	f_0
300	0.110126	0.10719	0.101656	0.100906	5.522	0.100788
350	0.110608	0.106928	0.10309	0.101722	2.850	0.100964
400	0.110776	0.108812	0.103484	0.103302	9.329	0.103296
450	0.112736	0.110108	0.104796	0.104656	10.046	0.104652
500	0.113816	0.111268	0.106794	0.105742	3.999	0.105419
Number of Nodes	109536	229680	363796	468696		infinite
	Refinement Ratio		r_{23} = 1.58	r_{12} = 1.29	r_{avg} = 1.44	

Table 1: *Values for the drag coefficient at zero grid space*

The results for the zero grid space are very close to the results from the finest mesh. Therefore, it can be said that we have reached the mesh independence and no more mesh refinement is required.

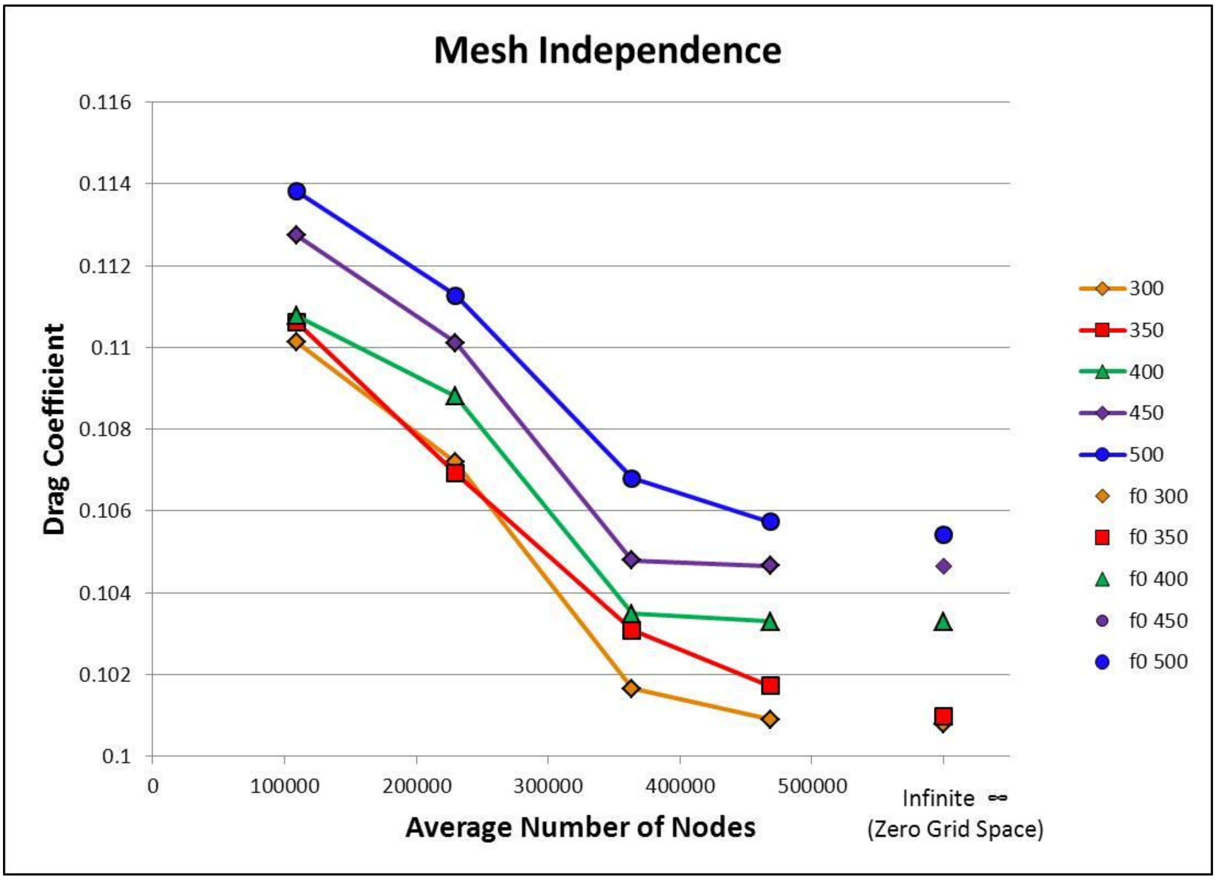


Figure 9: *Development of drag coefficient for different fuselage shapes*

Figure 9 shows the development of the drag coefficient over the several mesh refinements. It also shows the values for the zero grid space, for each fuselage length.

<u>*Grid Convergence Index (GCI):*</u>

"The Grid Convergence Index is a measure of the percentage the computed value is away from the value of the asymptotic numerical value" (***TIAN, 2011***). Therefore, it is a measure of the discretisation error of the model. To derive the GCI, we used the following equations (***TIAN, 2011***).

$$GCI = \frac{F_S\,|\varepsilon|}{r^p - 1} \qquad (12)$$

with

$$\varepsilon = \frac{f_2 - f_1}{f_1} \qquad (13)$$

and $F_S = 1.25$ for three or more grids.

As for the RICHARDSON extrapolation we derived the grid convergence index for each fuselage length (see ***table 2***). The results are, as expected, best for the finest meshes. The maximal value is 0.00931, i.e. 0.931%. But all other values are much less. Therefore, no further mesh refinement was needed. This is the same result as we obtained from the RICHARDSON extrapolation.

	Grid Convergence Index		
Fuselage rear length	**GCI 34**	**GCI 23**	**GCI 12**
300	0.00537	0.01067	**0.00146**
350	0.02383	0.02577	**0.00931**
400	0.00080	0.00228	**0.00008**
450	0.00081	0.00172	**0.00005**
500	0.00880	0.01610	**0.00382**

Table 2: *Grid Convergence Index for all shapes*

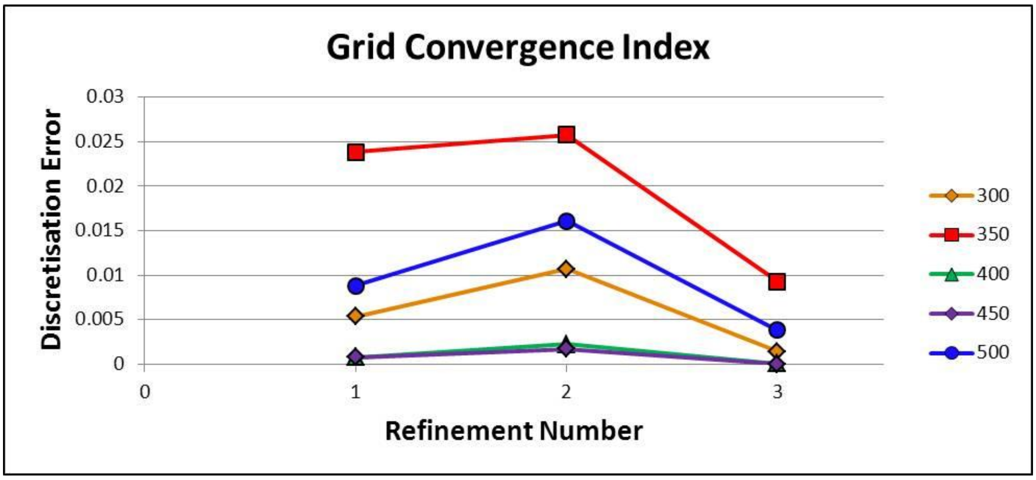


Figure 10: *Development of the Grid Convergence Index for different fuselage shapes*

Figure 10 shows the development of the grid convergence index over the three mesh refinements. It reaches its smallest values after the third refinement; for each fuselage length.

Asymptotic range of convergence:

It is important that each grid level solution is in the asymptotic range of convergence for the computed solution (***TIAN, 2011***). This can be checked by observing two GCI values as computed over three grids; the following equation should be valid:

$$\frac{GCI_{23}}{r^p * GCI_{12}} \approx 1 \qquad (14)$$

With this equation we obtained the following values for each fuselage length (see ***table 3***). The values were all very close to 1, i.e. the equation is valid. Therefore, the solutions are within the asymptotic range of convergence.

Fuselage rear length	GCI values over three solutions
300	0.9926
350	0.9867
400	0.9982
450	0.9987
500	0.9901

Table 3: *GCI values over three solutions for all fuselage shapes*

2.4.3. Mesh Statistics

Aspect ratio:

The aspect ratio should be between 0.2 and 5 (***TIAN, 2011***). The aspect ratio of our project is in average about **1.88** (maximum of 14.98 and minimum of 1.00). Therefore the average of the aspect ratio is in between the range for an optimal solution. The maximum value of 14.98 is quite high, but one can see in ***figure 11*** that most values of the aspect ratio are in the desired range.

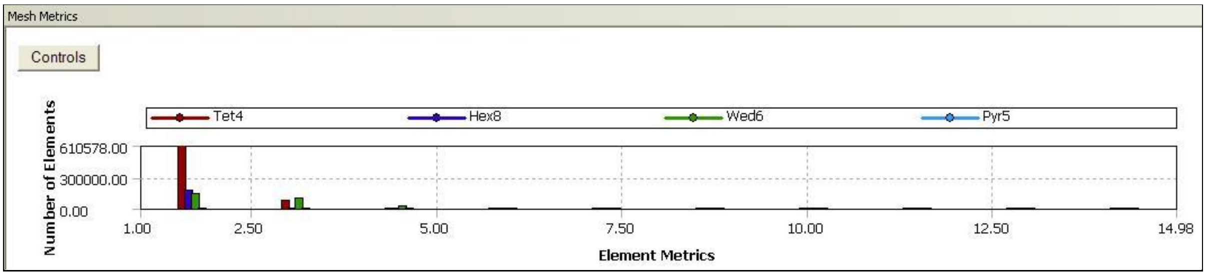


Figure 11: *Distribution of aspect ratio*
ANSYS v14.0 Meshing

Skewness:

The skewness should be as close as possible to 0. The worst possible value is 1 (***TIAN, 2011***). The average of the skewness is of about **0.20** (maximum of 0.99 and minimum of $1.31*10^{-10}$). This is a good value and also the distribution of the skewness values is in the desired range (see ***figure 12***).

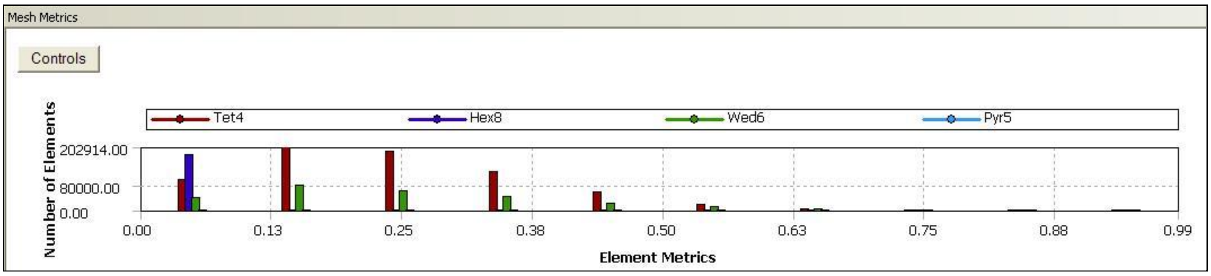


Figure 12: *Distribution of skewness*
ANSYS v14.0 Meshing

<u>Element quality:</u>

The element quality is a factor computed for each element of the model. It should be as close as possible to 1. The worst value is 0 (***TIAN, 2011***). The average of our model is **0.83** (maximum of 1.00 and minimum of 0.13). The average is a good value and the distribution of the element quality factor shows that most of the values are in the desired range (see ***figure 13***).

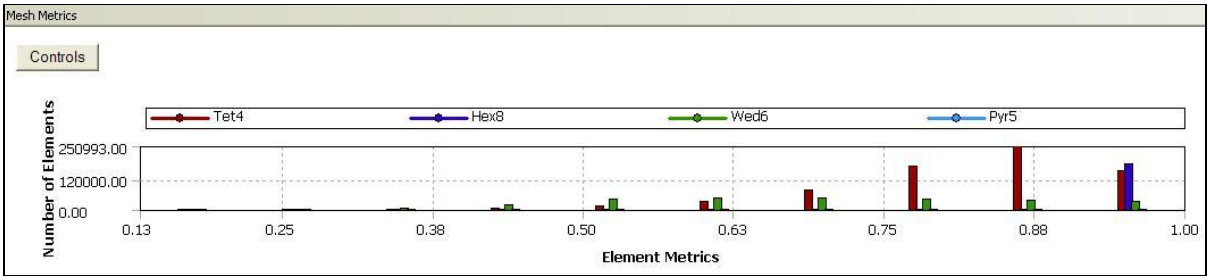


Figure 13: *Distribution of Element Quality*
ANSYS v14.0 Meshing

<u>Orthogonality factor:</u>

The orthogonality factor should be greater than 1/3 (***TIAN, 2011***). The average of this model is **0.89** (maximum of 1.00 and minimum of 0.19) which is much greater than 1/3. The distribution of the values also shows that most of the values are in this very good area (see ***figure 14***).

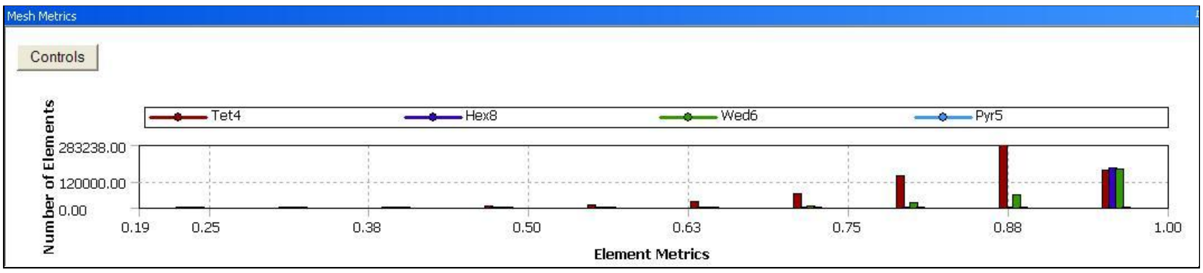


Figure 14: *Distribution of Orthogonality Factor*
ANSYS v14.0 Meshing

<u>Expansion factor and orthogonality angle:</u>

The orthogonality angle should be greater than 20°. The minimum of this model is **35.5°**, i.e. that our model falls within the desired range (see ***table 4***).

The expansion factor has a maximum value of 29. Less than 1% of the cells are critical and 99% of the cells are in the desired range (see ***table 4***), i.e. one can neglect the marginal amount of critical cells and the model is fine.

Mesh Statistics			
Domain Name	Orthog. Angle	Exp. Factor	Aspect Ratio
	Minimum [deg]	Maximum	Maximum
Airflow	35.5 ok	29 !	8 OK
	%! %ok %OK	%! %ok %OK	%! %ok %OK
Airflow	0 <1 100	<1 1 99	0 0 100

Table 4: *Mesh Statistics (ANSYS v14.0 CFX-Solver Manager)*

2.5. Validation/Verification

For the validation we did our own calculation based on knowledge as well as on experience, because one of the group members is a hobby pilot for model helicopter. Therefore he assumed some values from his own experience. We also got a part of the values from a speed cup for model helicopters (***POETING, 2010***). For further detail see ***Appendix A***.

The drag coefficient of our model is about $c_D = 0.1$ (see ***3.2.***). We compared this value with the data from the calculation of the estimated results of the speed cup one can see that they are in the same order of magnitude. However, the drag coefficients of the speed cup are in general larger than ours. This is reasonable, because in our CFD model we made a lot of simplifications. So, the CFD model only calculated the fuselage and neglected other important parts like the rotors or the landing gear.

As a result, our CFD model seems to provide reasonable values for the drag coefficient. But this is a very coarse validation due to the assumptions on the calculation of the drag coefficient (see ***Appendix A***). A proper validation would use data from a wind tunnel experiment of helicopter fuselages.

3. Results and Discussions

3.1. Flow field results

3.1.1. Velocity vectors

The velocity vectors are the same as what we expected. The velocity of the air is increasing along the main body of the fuselage and decreasing after the largest part of the helicopter until it becomes almost constant along the tail boom (see ***figure 15***).

The maximum velocity is about *46 [m/s]* which is about *4 [m/s]* above the free stream velocity.

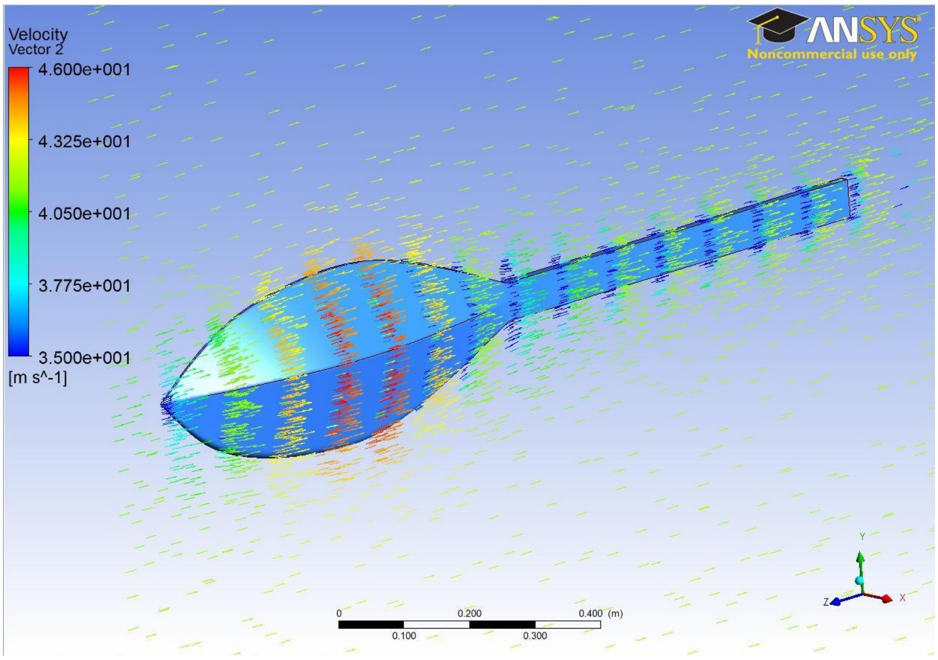


Figure 15: *Plot of velocity vectors*
ANSYS v14.0 CFD-Post

3.1.2. Streamline plots

Streamlines are parallel to the mean velocity vector of the flow. In CFD post-processing software a streamline is normally shown by the path that massless particles would take through the fluid domain (***TU, 2013***).

The streamlines flow very smooth around the geometry of the fuselage of the helicopter. One can see the acceleration of the air around the largest part of the fuselage (see ***figure 16***). This is similar to the plot of the velocity vectors.

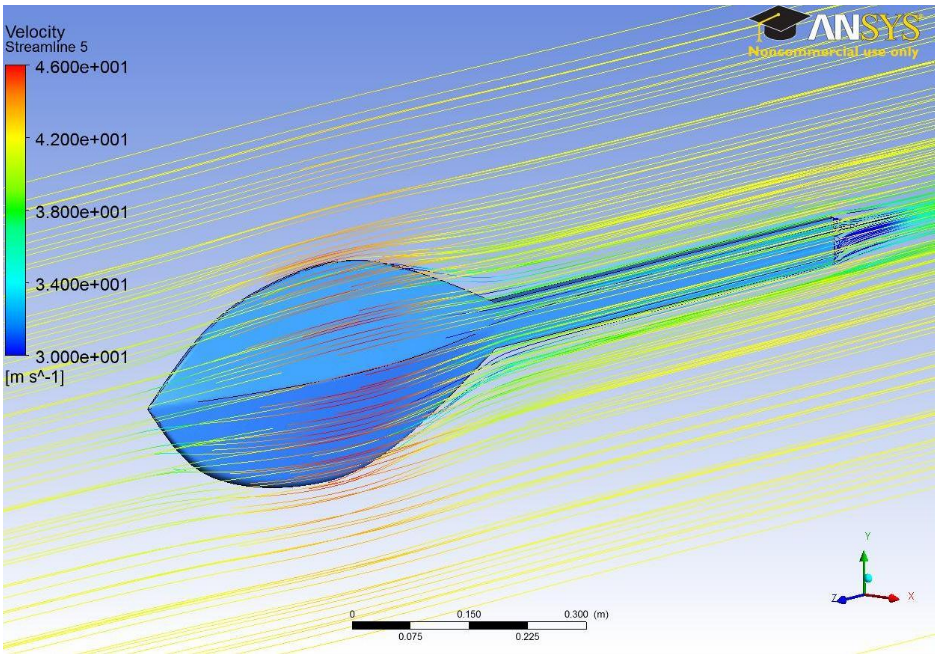


Figure 16: *Plot of streamlines*
ANSYS v14.0 CFD-Post

There is also a small recirculation zone at the end of the tail boom (see ***figure 17***). But this is only an issue of the CFD model. We simplified the geometry in order to reduce the computational effort. Real fuselages have very smooth contours in these areas to avoid recirculation to reduce the drag coefficient.

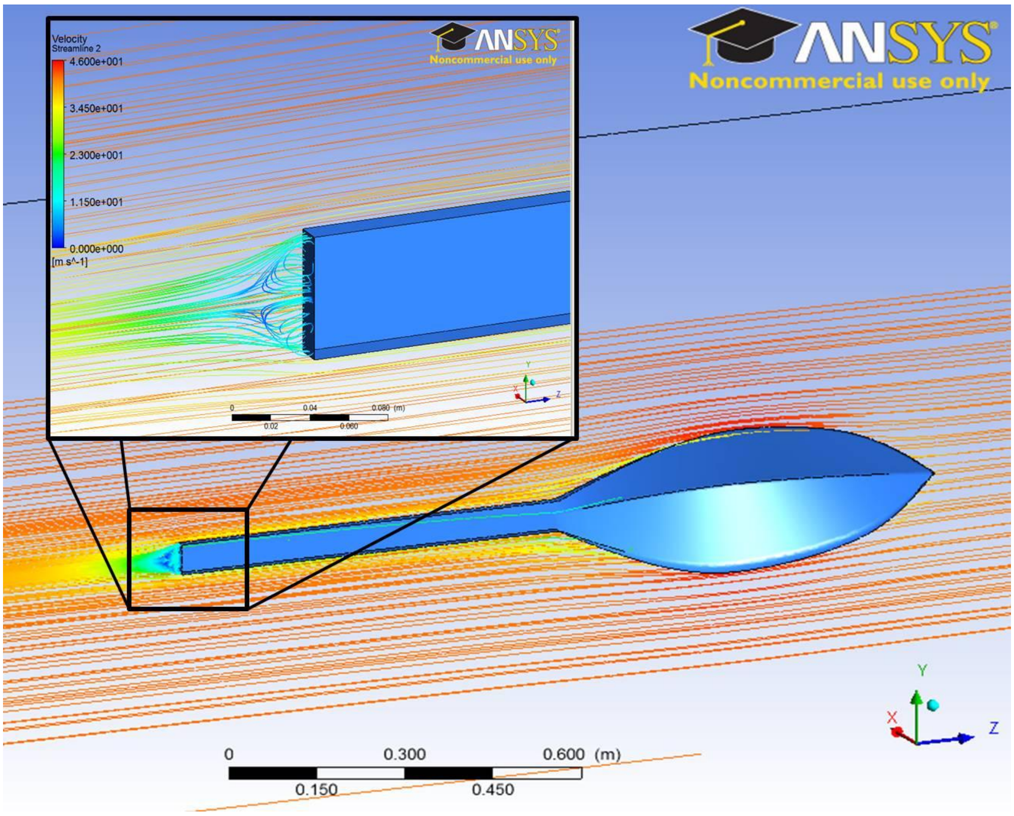


Figure 17: *Recirculation zone at the end of tail boom (streamline plot)*
ANSYS v14.0 CFD-Post

3.1.3. Y^+ contour

The y^+ values are very important for turbulent flows, because they are needed to check whether the first nodes in the mesh of the boundary layer are in the right location to resolve it. The k-ε model we used for our calculations resolves the boundary layer with wall functions. To make sure that the near wall nodes are in the right place to get accurate results, we use inflation with the first layer height we calculated. It is recommended that the y^+ values are in the range from $11.63 < y^+ < 300$ for the k-ε model. To ensure this, we plotted the y^+ values in the CFX-Post (see ***figure 18***) and our values were within this range (minimum value of 20 and maximum value of 205). So, we used the y_1 from the calculation for each mesh.

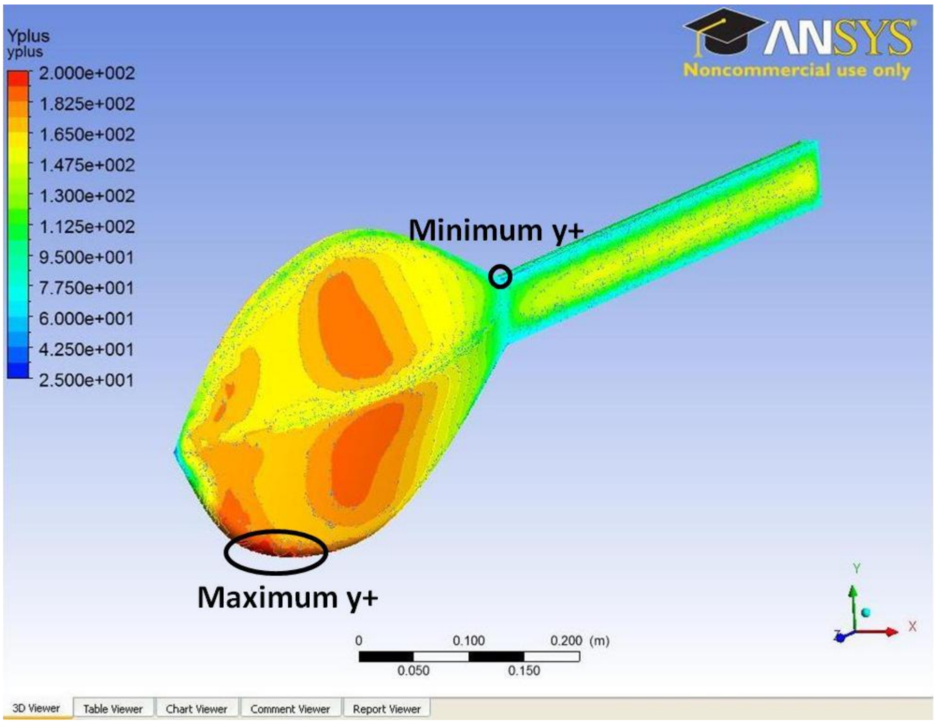


Figure 18: *Plot of Yplus (y^+)*
ANSYS v14.0 CFD-Post

3.2. Results

<u>*Drag coefficient:*</u>

To compare the different fuselage rear lengths, we needed to calculate the drag coefficient with the output from the CFD solver. To calculate the drag coefficients we obtained the value of the drag force from the solver.

We used this value to create an expression in the CFX-Preprocessing, which calculated the drag coefficient. Therefore, we used the equation for the drag coefficient:

$$c_D = \frac{F_D}{p_{dyn} * A} = \frac{2 * F_D}{\rho * v^2 * A} \qquad (15)$$

Because we used the symmetry boundary in the model, the drag force is only half of the real value. We considered this by only using half of the cross-section area of the model. We used the Computer Aided Design (CAD) program CREO Parametric to easily calculate the cross-section area of the fuselage.

The values of the density of air ρ and the velocity v are the same values as in the settings of the solver, i.e. $v = 42[m/s]$ and $\rho = 1.185[kg/m^3]$.

After we finished all of our calculations we obtained the following chart which shows the variation of the drag coefficient for the different fuselage rear lengths (see ***figure 19***). We included one curve for each mesh refinement we made.

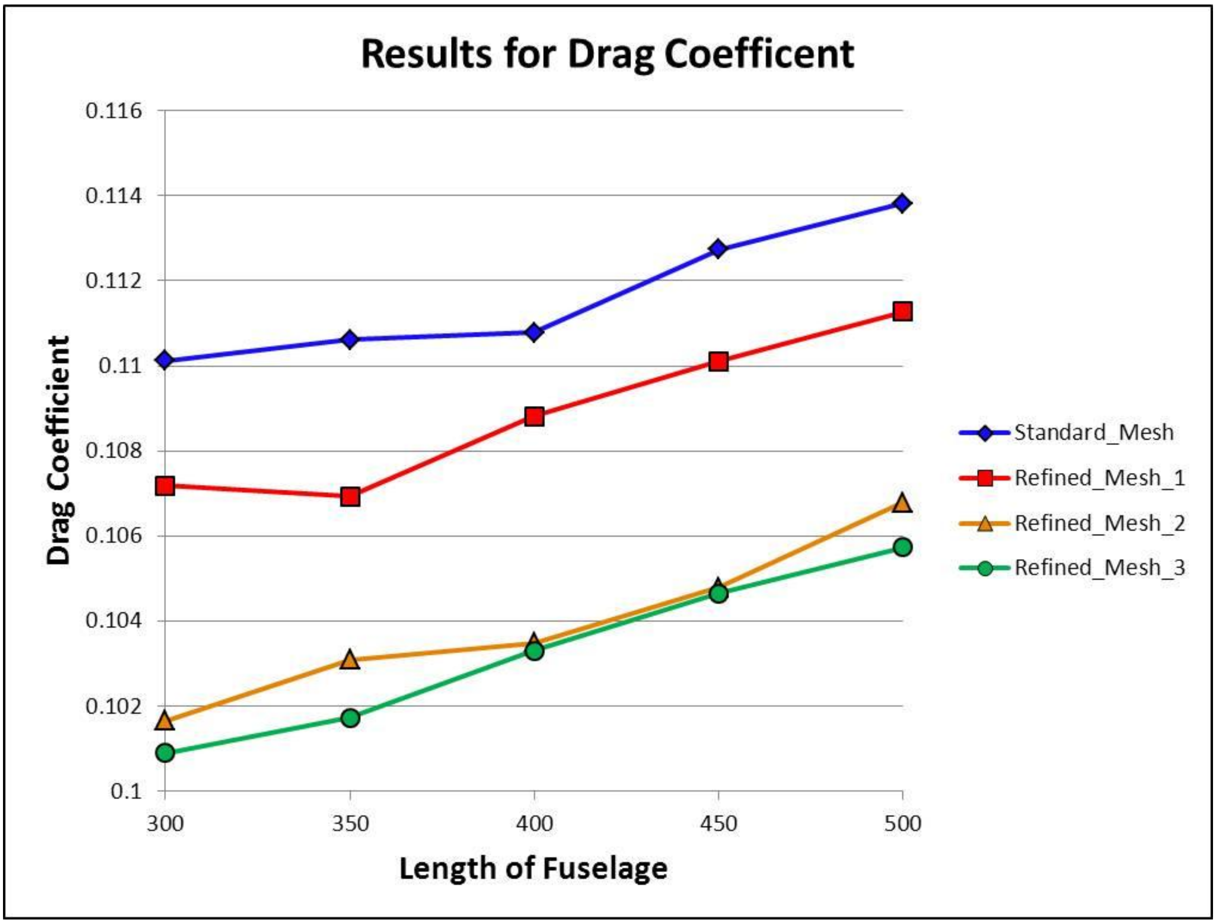


Figure 19: *Drag coefficients for different fuselage shapes and all meshes*

Although we expected that the drag coefficient would decrease with an increasing fuselage rear length, obviously the opposite has happened. The difference between the largest and the shortest fuselage rear length is small (about 5% for the finest mesh).

However, we can obtain from these results that an increase in the fuselage rear length is not the right way to decrease the drag coefficient. The larger fuselage rear length ends up with a higher drag coefficient and therefore a lower velocity of the model helicopter in reality.

Without the CFD we would have needed to build a prototype to test the influence of the fuselage rear length. Therefore, CFD is a great tool to test the behaviour of a physical system without actually spending time and effort building a prototype.

In our case the CFD might not deliver accurate absolute values, but this is not necessary for this analysis. We only conducted a sensitivity study on the behaviour of the drag coefficient in relation to the fuselage rear length.

3.3. Discussion of results

The total drag for aircrafts and helicopters consists mainly of form drag, skin friction and lift-induced drag. The fuselage produce almost no lift, therefore we can neglect the lift-induced drag. As a result, the total drag of our fuselage is dominated by form drag and skin friction.

Our original idea was to change the design of the fuselage to a shape that is more similar to a streamlined body ($c_D \approx 0.04$) (***EngineeringToolBox, 2013***). Thus, we studied the change of the total drag coefficient in dependence of the shape. The initial drag coefficient with a fuselage rear length of 300[mm] was about $\boldsymbol{c_D = 0.100788}$. We expected with rising fuselage rear length, which is equal to a more streamlined shape, a decreasing in the drag coefficient.

After further studies we have found out that the opposite was happening. The drag coefficient for our best shape is almost 5% higher than for our initial design.

A possible explanation for that behaviour needs a more detailed look on the composition of the drag, in this case the skin friction drag and the form drag. The driver of the form drag is the shape of the model helicopter. A non-optimal shape could lead to higher pressure differences between front and back. In worst case, recirculation occurs, which drops the static pressure significantly in that area and leads to an even higher pressure difference between front and back. Actually, our project mainly tried to reduce the form drag. The driver of the skin friction is the area of the model helicopter. An increase in the area would increase the skin friction of the model helicopter. Both types of drag are dependent on the design of the fuselage and in general are not independent of each other.

A streamlined body would be an almost optimal trade-off between skin friction and form drag. In our case, it seems that our shape changes improved the form drag, but on the other hand it probably increases the skin friction of the model helicopter.

4. Conclusion

As closing words it can be said that we have learned a lot by conducting this CFD analysis. We had to overcome many issues on the way to the correct CFD analysis.

One of the main issues was to get the model to converge. We achieved this through the optimisation of the mesh and the solver settings such as reducing the physical timescale. Another main issue was the conceptualisation of the whole model because a real helicopter is a very complex system with time dependent flow parameters around the rotor blades and very complex geometries. Because our model is flying at very high speed we did not have to include these complex behaviours and therefore we were able to run the simulation at steady state. We could also simplify our model to capture the main flow behaviour at the high speed. This was very helpful; we would not have been able to run a simulation on the entire helicopter due to the immense computational effort.

With the simplified model of the helicopter fuselage we ran a lot of simulation to see how the variation in the fuselage rear length affects the drag coefficient. We expected the drag coefficient to decrease with an increase in the rear fuselage length, but the results showed otherwise. In fact, the drag coefficient increased with the larger fuselage rear length. After several mesh refinements we were able to confirm our results. They had become mesh independent.

After all, the analysis of the helicopter fuselage with the CFD as an engineering tool was successful. We obtained the behaviour of the variation of the drag coefficient due to the variation of the fuselage rear length without building any physical model. In fact, we saved a lot of time and efforts through performing the CFD analysis. It is a very useful tool to deal with fluid problems, which we are now able to use for the upcoming tasks in our future engineering life.

References

Tian, Z., 2011, “Computational Fluid Dynamics (CFD) for Engineering Applications”, MECH ENG 4111 and 7045, *Lecture notes*

Tu, J., et al., 2013, “Computational Fluid Dynamics”, A Practical Approach

Banshee Helicopters, 2013
http://www.banshee-helicopters.de

Poeting Bernd – Ihre Modellflugschule aller Klassen, 2010
http://www.poeting1.de/p/54_57_Veranstaltungen-2--Speedcup.html

The EngineeringToolBox, 2013
http://www.engineeringtoolbox.com/

APPENDIX A: Validation of the Drag Coefficient

For this validation we used the velocity data provided by a speed cup for model helicopters in Kreuztal Littfeld, Germany in 2010. We calculated each special drag coefficient for the first ten helicopters.

First of all we need the velocity in [m/s]. Therefore we have to divide the velocity in [km/h] by 3.6 to get the value in [m/s]. Then we can calculate the dynamic pressure p_{dyn}

$$p_{dyn} = \frac{1}{2}\rho U_{max}^2 \qquad (16)$$

where ρ is the density of the surrounding air at 25[°C]. The value ist $\rho = 1.185[kg/m^3]$ (***ANSYS v14.0***).

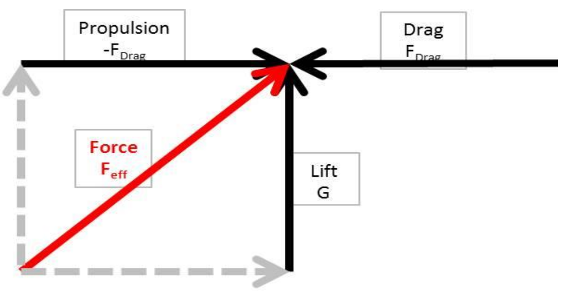
Because of the experience as a pilot for model helicopter we assumed the power of the engine to $P_{max} = 7[kW]$ and the mass of the fuselage to $m = 4.4[kg]$. The value of the thermal efficiency is assumed for a general combustion process to $\eta = 0.4$. So, we can calculate the effective power of the engine P_{eff}:

$$P_{eff} = \eta * P_{max} \qquad (17)$$

The values of the effective force F_{tot} are calculated by the following relationship:

$$P = u * F \quad \Rightarrow \quad F_{eff} = \frac{P_{eff}}{U_{max}} \qquad (18)$$

The values for the drag force F_{Drag} are calculated by the relationship in triangles (PYTHAGOREAN theorem). Therefore one has the following velocity triangle (see ***figure 20***):



$$F_{eff}^2 = F_{Drag}^2 + G^2$$
$$\Rightarrow \quad F_{Drag} = \sqrt{F_{eff}^2 - G^2} \qquad (19)$$

Figure 20: *Velocity triangle of forces at a flying system*

The values of the drag coefficient c_D then can finally be calculated by the relationship

$$F_{Drag} = c_D * p_{dyn} * A \quad \Rightarrow \quad c_D = \frac{F_{Drag}}{p_{dyn} * A} \qquad (20)$$

where A is the projected area of the fuselage. This value was calculated with the CAD program CREO Parametric for our fuselage as $A = 0.022004[m^2]$.

On the next page one can see the overall results of our validation data (see ***table 5***).

U_{max} [km/h]	U_{max} [m/s]	p_{dyn} [Pa]	P_{max} [W]	P_{eff} [W]	F_{eff} [N]	G [N]	F_{Drag} [N]	c_D [-]
230	63.89	2418.46	7000	2800	43.83	43.16	7.59	**0.14**
225	62.50	2314.45	7000	2800	44.80	43.16	12.00	**0.24**
210	58.33	2016.15	7000	2800	48.00	43.16	21.00	**0.47**
207	57.50	1958.95	7000	2800	48.70	43.16	22.54	**0.52**
199	55.28	1810.46	7000	2800	50.65	43.16	26.51	**0.67**
198	55.00	1792.31	7000	2800	50.91	43.16	26.99	**0.68**
192	53.33	1685.33	7000	2800	52.50	43.16	29.89	**0.81**
187	51.94	1598.70	7000	2800	53.90	43.16	32.29	**0.92**
181	50.28	1497.75	7000	2800	55.69	43.16	35.19	**1.07**

Table 5: *Calculation of our Validation data*