Geek Hub

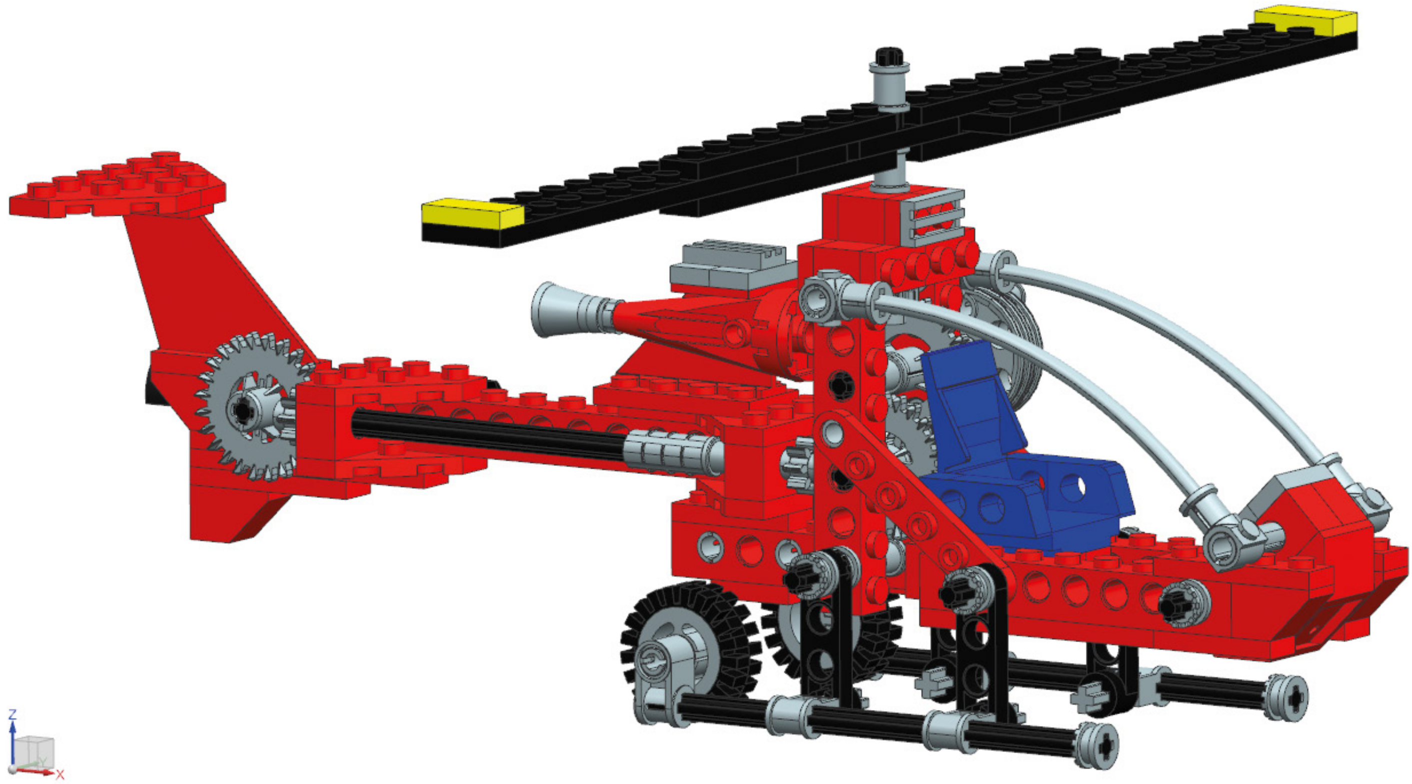


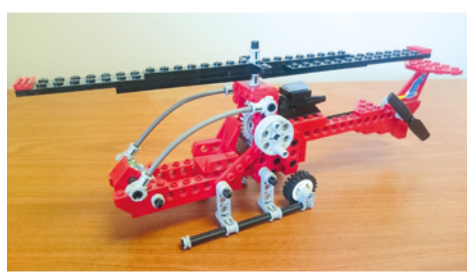
Figure 1. LEGO Technic Aero Hawk. The CAD model of the Aero Hawk was obtained from http://www.GrabCAD.com.

Ever wondered if a LEGO Aero Hawk Helicopter could actually fly?

By Karl du Plessis, Technical Specialist Esteq

Remember that LEGO Aero Hawk from your seventh birthday? Remember how it did loops, flew upside down and even reached warp speed? In the imagination of a seven-year-old anything is possible. But being all grown-up, combined with harsh reality and a degree in engineering, you know that anything is indeed possible, but sadly everything isn't. So you set out to apply your engineering knowledge to determine if there is even the slightest possibility of a LEGO Aero Hawk managing some form of flight. You dive into your vast engineering toolbox and pull out your traditional Computational Fluid Dynamics (CFD) tool.

At first glance, the geometry is quite complex. The LEGO parts being modeled in meticulous detail, which the designer in you can definitely appreciate, but knowing what you know about CFD simulation, immediately you start to think "How on earth am I going to build the mesh for this model?" You consider doing some serious CAD clean-up to reduce the complexity but consequently realise that you would lose too much of the authenticity of the LEGO model when simplifying the geometry down to the level typically required to perform CFD simulations. You are faced with a conundrum. It is time to revive and even stretch a little further that imagination that is long lost due to the doctrine of what is and what isn't possible in the world of CFD… Imagine a world in which one can open up a



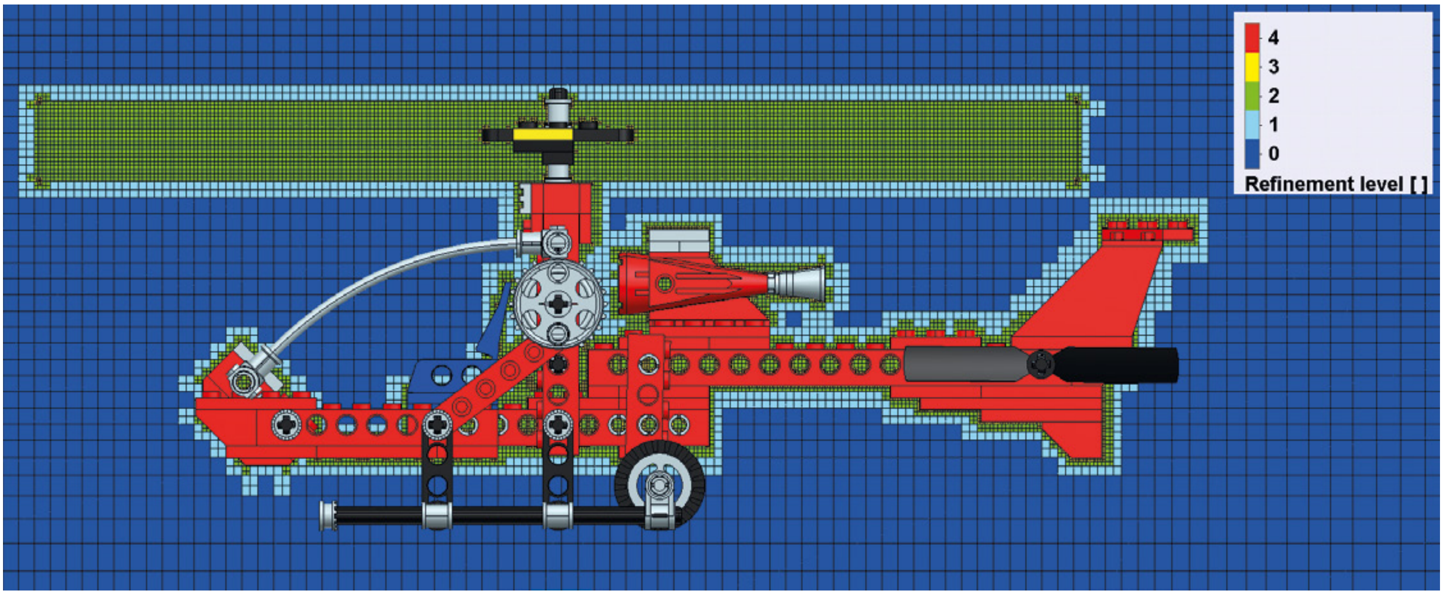


Figure 2. FloEFD Immersed Boundary Cartesian Mesh Cut-plot showing Level of Refinement

CAD model, specify the mesh settings, apply boundary conditions and start solving within a matter of minutes. Sounds like a bit of a stretch of the imagination, doesn't it? Enter FloEFD™!

FloEFD as integrated in Siemens NX was selected to analyse the LEGO Aero Hawk helicopter, making use of the new Sliding Mesh technology in FloEFD to simulate the rotation of the main rotor. This serves as a good test to see how FloEFD would handle the rotation of more complex geometry. You load the model into NX as-is, with only an additional rotating region part (simple cylindrical component) modeled to represent the rotation region that encapsulates the main rotor. You apply a rotor speed of 300rpm. You set the computational domain size and make use of a relatively coarse base mesh as a start with local mesh refinements to control the level of mesh resolution in the rotating region and on the helicopter. You are satisfied with the resulting mesh resolution as in the image below, with approximately one million cells being generated. You are completely amazed at the fact that FloEFD had no trouble creating the mesh even with all of the small geometric features and unnecessary narrow channels between the stacked LEGO blocks. This is obviously wasteful and one would spend some time cleaning up the CAD model in this regard. Nonetheless, you are curious as to how FloEFD will cope with this and proceed, leaving the model as-is in order to test the solver stability. When you are ready to solve you notice that you have access to all of

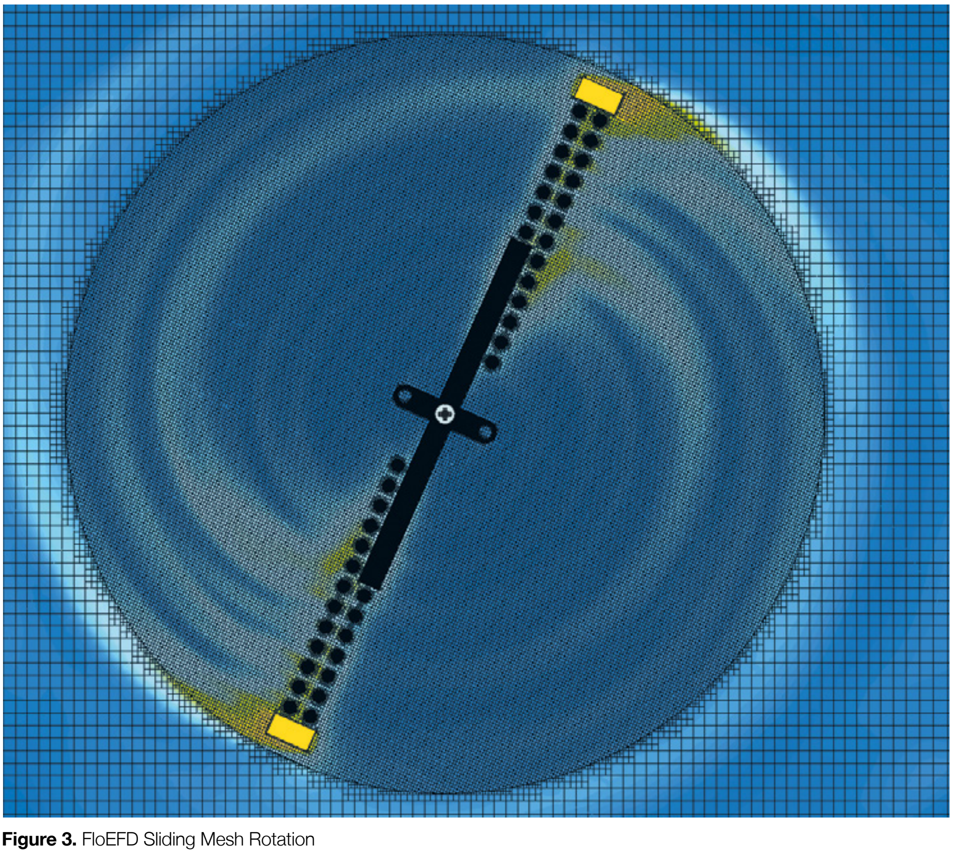


Figure 3. FloEFD Sliding Mesh Rotation

your CPU cores. You select all 16 cores to your availability (or four if you don't have that luxury) and press the "Solve" button and leave the simulation to run overnight. The sliding rotation option requires a transient analysis to be conducted and of course you do have to sleep at some point…

Come morning, the simulation has finished solving without error or divergence. Again you find yourself quite impressed. You play around with the animations post-processing feature and record videos of the velocity field in the plane of the propeller rotation as shown in figure 3, shows the rotation of the mesh within the rotating region just moments after the initial start-

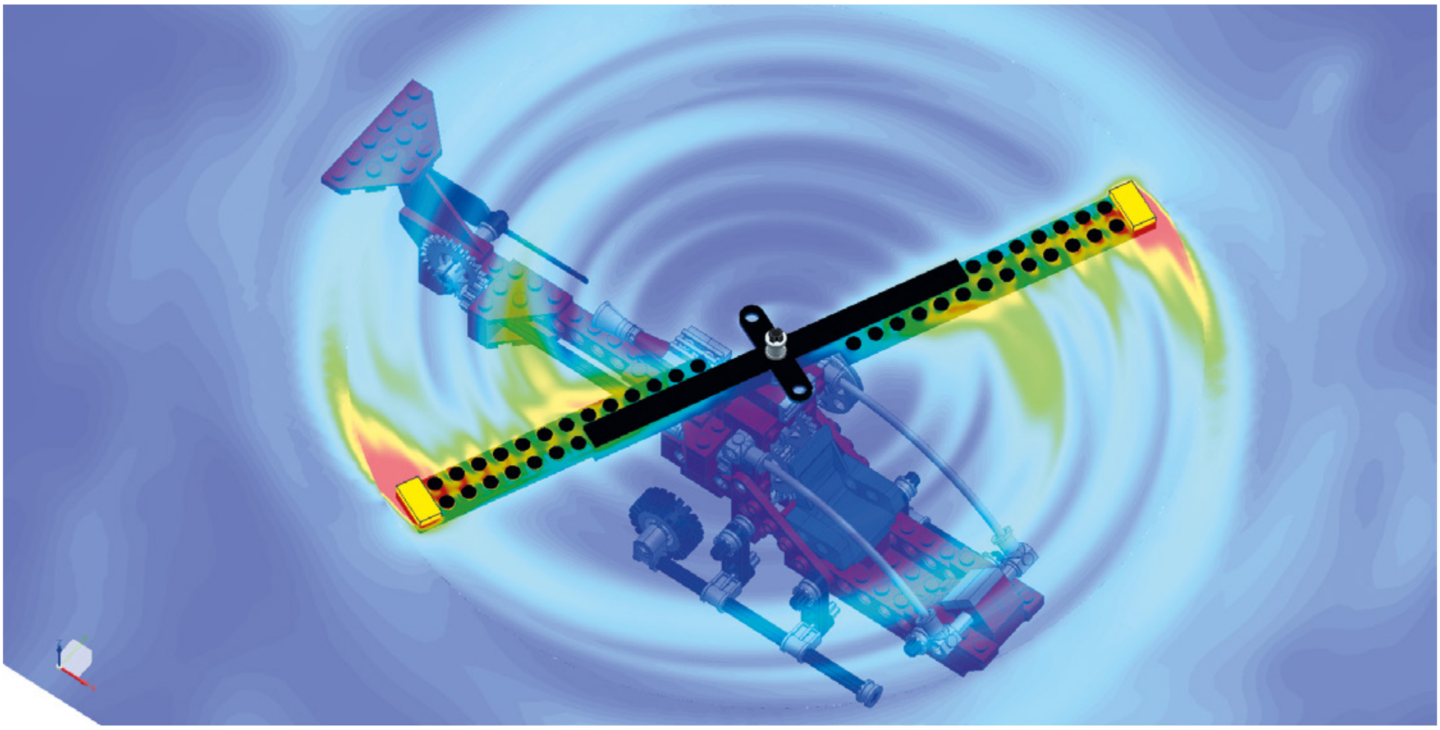


Figure 4. Velocity contour plot with rotor blade rotation

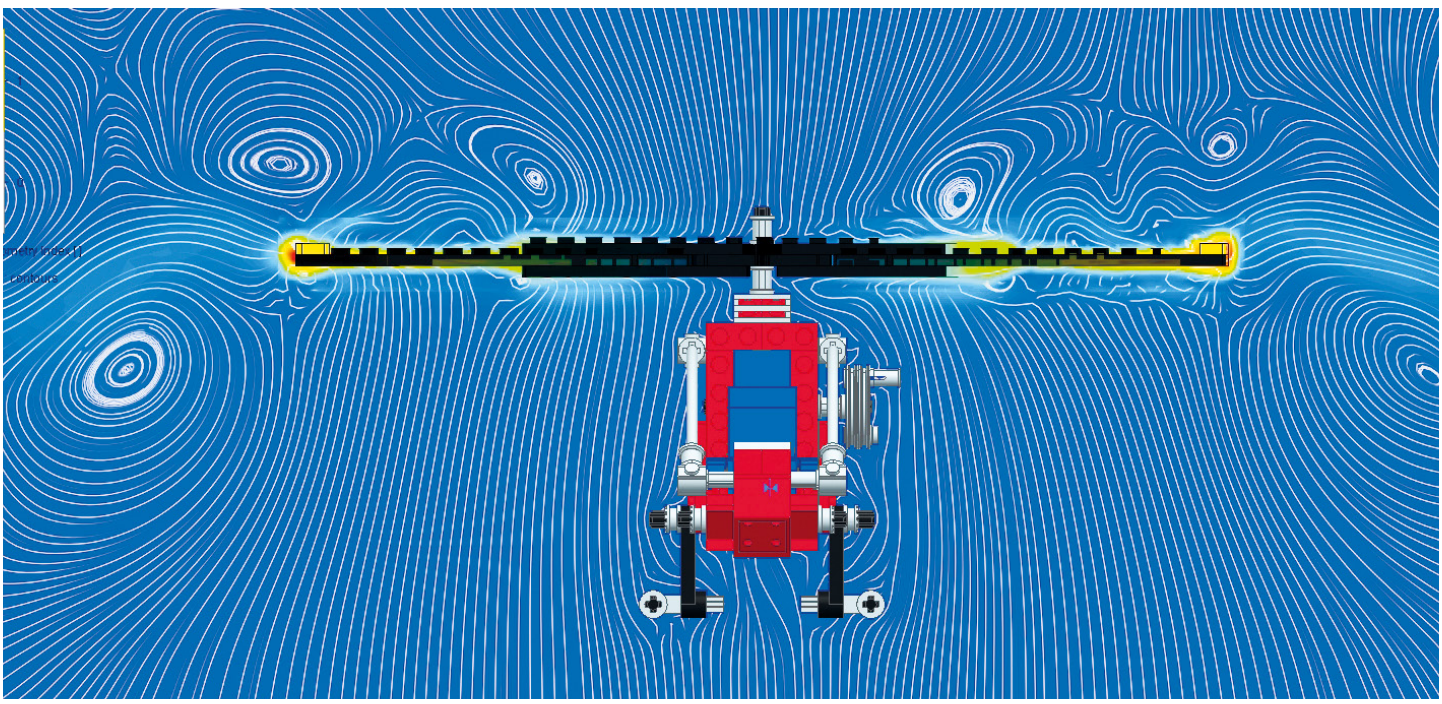


Figure 4. Velocity contour plot with rotor blade rotation

up. Figure 4 shows the velocity field as developed after a few rotations. You make cut-plots of the velocity over the propeller blades and you note in these images the development of the boundary layer at the propeller surfaces, one of the key technologies within FloEFD that make it possible to perform simulations with such complex geometry.

There is still the question – Can this LEGO Aero Hawk actually fly? You make a time-history plot of the propeller lift force. After close inspection you can see a definite positive offset in the average lift force, albeit fairly small, definitely not enough to lift the helicopter off the ground in the real world, but still a positive lift force nonetheless, enough to revive that youthful imagination that knows no bounds. You start to think like the seven-year-old you once were, thinking "if I can develop a strong lightweight material, lighter than anything known to man…or if I can spin the propeller at 3,000,000 rpm, then this LEGO Aero Hawk from my childhood just might actually be able to fly"! Thanks to FloEFD this whole new world of CFD has been unlocked. A world not confined to simplified geometry or constrained by meshes and convergence issues. What seemed hard to imagine in the past has become a reality, giving the engineer in you free reign to imagine more…(It goes without saying that the very low lift force is due to the fact that the LEGO propeller does not represent an airfoil very well)

FloEFD (Engineering Fluid Dynamics) is a highly sophisticated CFD tool aimed at Engineers. The main objective of FloEFD is to enable engineers to make design decisions as quickly and simply as possible, without having to ask the resident CFD specialist (if one should be so lucky to employ one of these very rare specimens) to perform the thermal-fluid analyses that traditionally required a lot of expertise and patience. FloEFD transcends almost all of the inherent obstacles of traditional CFD through innovative and ingenious technologies that truly stretch the imagination. Some of these technologies include,

- CAD Embedded
- Immersed Boundary Adaptive Cartesian Mesh
- Two-scale Modified Wall Functions
- Enhanced k-ε Turbulence Model.

These technologies are what make FloEFD the efficient and productive engineering tool that it is. Now hopefully the engineer in you is starting to imagine again a world of possibilities that seemed lost in translation and that it is easy to imagine a CFD tool that is easy to use, makes meshing a breeze, is highly tolerant of complex geometry and provides quick, stable, accurate and converged solutions. So in closing, if I may borrow the theme song from The LEGO Movie…

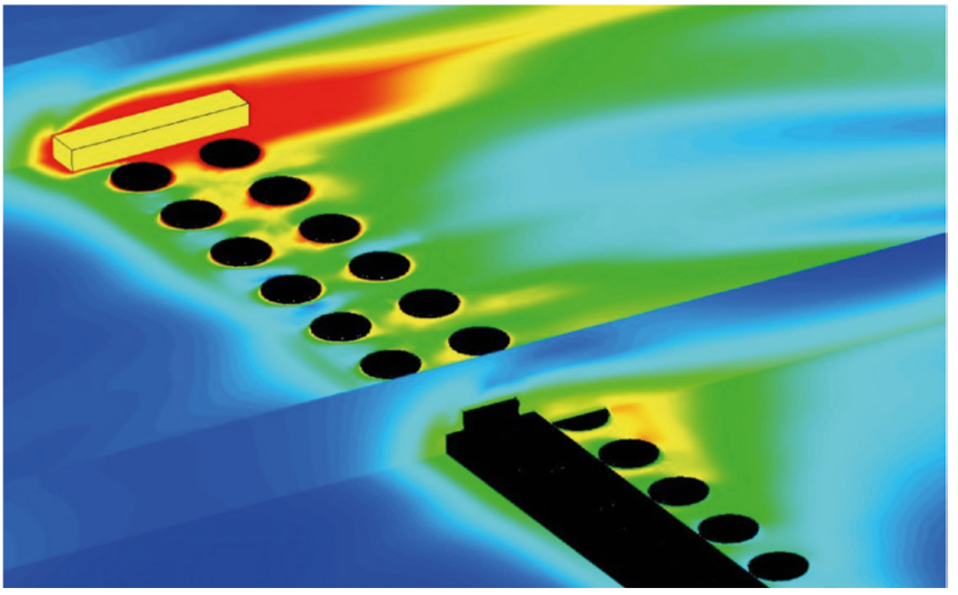


Figure 5. Velocity Cut-plots around rotor blade detail

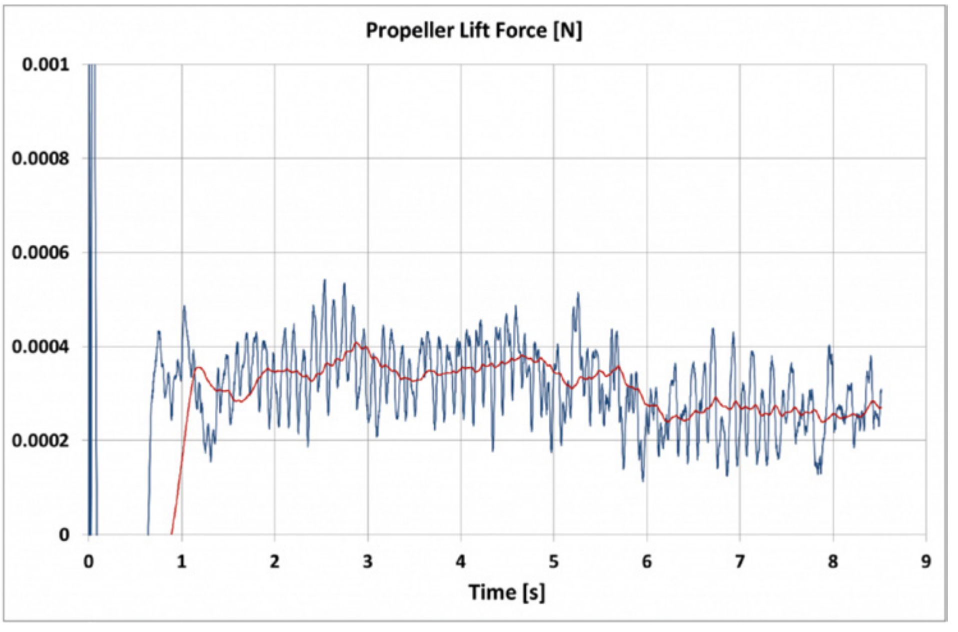


Figure 6. Time-history graph of rotor blade lift force