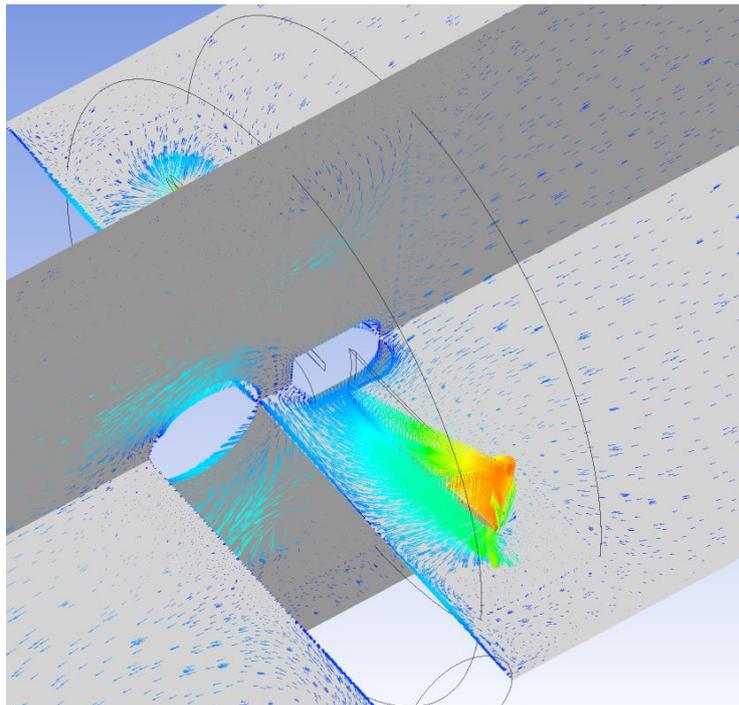


RESEARCH REPORT

VTT-R-01447-15



Fluid-structure interaction simulation utilising MpCCI

Authors: Vesa Nieminen

Confidentiality: Public

Report's title	
Fluid-structure interaction simulation utilising MpCCI	
Customer, contact person, address	Order reference
Tekes, Matti Säynätjoki Kyllikinportti 2, P.O. Box 69, FI-00101 Helsinki, FINLAND	Tekes: 40204/12
Project name	Project number/Short name
Computational methods in mechanical engineering product development, Task 1.4	78634 / SIMPRO
Author(s)	Pages
Vesa Nieminen	20/
Keywords	Report identification code
Co-simulation, Fluid-structure interaction, MpCCI	VTT-R-01447-15
Summary	
<p>The goal of this study was to demonstrate two-way fluid-structure interaction (FSI) co-simulation utilising MpCCI. Two-way FSI co-simulation with Fluent CFD-code and Abaqus structural FE-code utilizing MpCCI was applied successfully. In an FSI simulation, a structure deforms due to forces caused by a fluid flow while the deformation changes the fluids boundary. So called weak coupling was used which means that each problem is solved separately and during each time step some variables are exchanged and inserted into the equations of the other problem (pressure and nodal position in this case). Example structure in this case study was a rotating propeller and a non-rotating cylinder located at the wake of the propeller. The cylinder represents for example the body of the azimuthing thruster.</p> <p>The pressure and velocity fields obtained seemed to be reasonable. It was found that most dominating component of axial displacements of the propeller is the blade frequency. This is caused by the cylinder structure located at the wake of the propeller. For transversal and vertical displacements of the hub the dominating component is rotating frequency.</p> <p>The goal of this study was to demonstrate two-way FSI co-simulation utilising MpCCI, and therefore the CFD-model was not optimized for propeller calculations. In the future work, the results obtained should be compared with results with one-way co-simulation in order to find out impact of two-way coupling for this type of problems. Also computing times could be compared between these simulations.</p>	
Confidentiality	Public
Espoo 21.4.2015	
Written by	Reviewed by
Vesa Nieminen, Senior Scientist	Janne Keränen, Research Scientist
	Accepted by
	Johannes Hyrynen, Head of Research Area
VTT's contact address	
VTT Technical Research Centre of Finland, P.O. Box 1000, FI-02044 VTT, FINLAND	
Distribution (customer and VTT)	
Tekes/Matti Säynätjoki, 1 copy VTT/archive, 1 copy	
<p><i>The use of the name of the VTT Technical Research Centre of Finland (VTT) in advertising or publication in part of this report is only permissible with written authorisation from the VTT Technical Research Centre of Finland.</i></p>	

Contents

Contents.....	2
1. Introduction.....	3
2. Goal.....	4
3. Code Testing Cases	4
3.1 Elastic Flap in a Duct.....	4
4. Case Study	7
4.1 Structural Model.....	7
4.2 CFD Model	8
5. Methods.....	10
6. Results.....	11
7. Conclusions and Summary	19
References.....	20

1. Introduction

MpCCI has been developed at the Fraunhofer Institute SCAI in order to provide an application independent interface for the coupling of different simulation codes. The purpose of MpCCI is to perform multi-physical simulations. The systems under consideration are known as coupled systems. A coupled system consists of two or more distinct systems. Each system is governed by a characteristic set of differential equations, but both systems share some variables.

To find a solution for a coupled problem, all governing equations, which can be combined in a large system, must be solved. The solution in this way is called strong coupling. However, solving a system with strong coupling is often difficult as different approaches are necessary to solve the sub-problems. An alternative approach is through weak coupling. Here each problem is solved separately and during each time step some variables are exchanged and inserted into the equations of the other problem. This procedure usually yields a less exact solution compared to strong coupling. The advantages of the weak coupling are that the sub-problems can be solved faster than the complete system and that specialized solvers can be used for each.

Typical multi-physics simulations are fluid-Structure Interaction (FSI), thermomechanical coupling and electromechanical coupling. This study was concentrated on FSI coupling. Overview of the simulation process is presented in Figure 1.

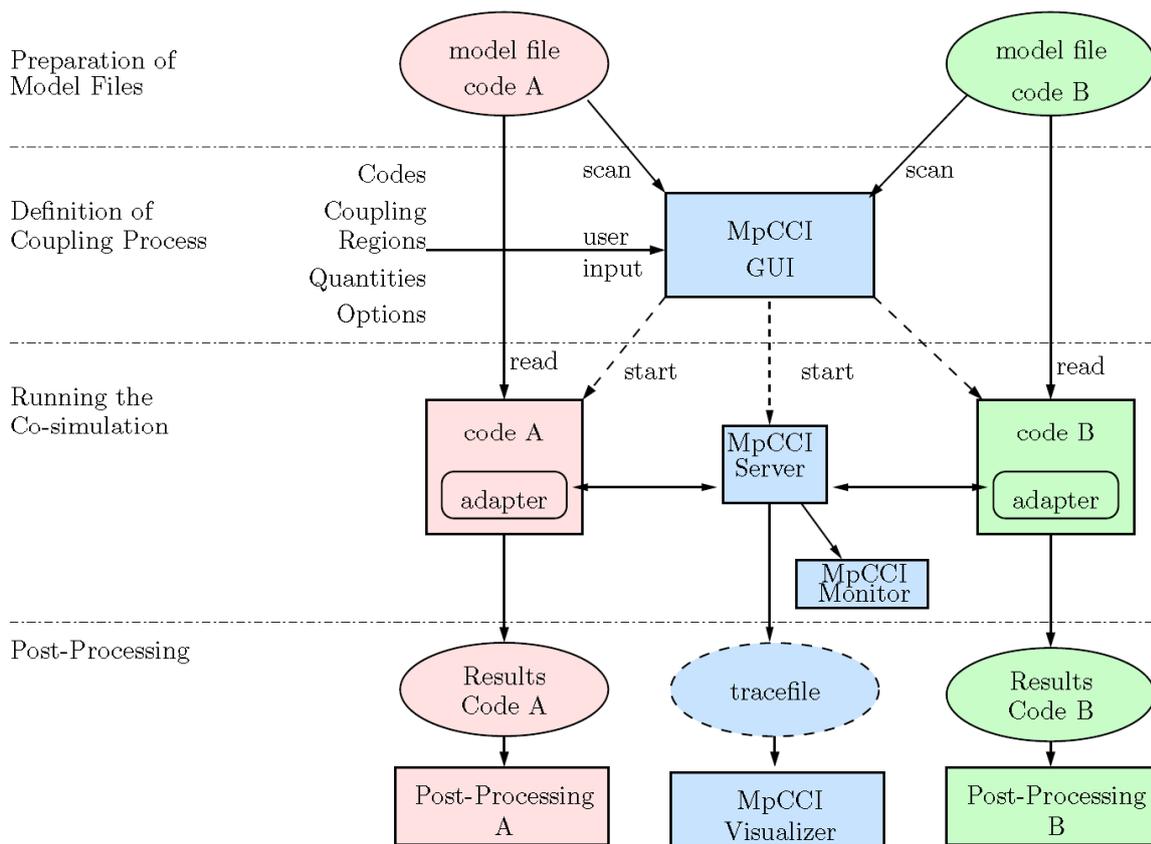


Figure 1. Overview of the simulation process /2/

In an FSI simulation, a structure deforms due to forces caused by a fluid flow while the deformation changes the fluids boundary. The deformation must be transferred to the fluid mechanics code, which corresponds to the quantity "Nodal position", while forces are sent from fluid mechanics code to the structural code, e. g. "Pressure". MpCCI supports the data

exchange between non-conforming meshes, i. e. the meshes of each subsystem can be defined to optimize the solution of the subsystem. A shape function mapping is used for data exchange between two non-matching grids, which simply interpolates a variable field using the shape functions. /2/

Solid mechanics code used was Abaqus (6.13.2). Deformation, stresses and strains of structures are computed using the Finite Element Method (FEM). The governing equations of solid mechanics problem are the mechanical equilibrium and Newton's laws of motion.

Fluid mechanics code used was Fluent (14.5.7). The numerical simulation of fluid mechanics is commonly referred as Computational Fluid Dynamics (CFD). Fluent uses Finite Volume Method for solving governing equations (Navier-Stokes equations and the continuum hypothesis).

2. Goal

Goal of this study was to test two-way co-simulation using MpCCI (version 4.3.1) with Fluent (version 14.5.7) CFD model and dynamic Abaqus (6.13.2) structural FE-model.

3. Code Testing Cases

In order to ensure that all codes work as intended, some testing cases provided by MpCCI were simulated before actual case study /2/. One of the cases is presented below. It was found that e.g. some dynamic link libraries (dll- files) must be updated in order to get MpCCI working properly with this current software version (4.3.1).

3.1 Elastic Flap in a Duct

The simulation couples a solid mechanics model (rectangle elastic flap) with a fluid mechanics model, see Figure 2. FE-model mesh consists of $20 \times 20 \times 2$ brick elements with 20 nodes each. All nodes at the top are fixed, i. e. they cannot move in any direction. The fluid domain (air) comprises the whole rectangular area in Figure 2 except for the flexible elastic flap structure. In this case also time steps of the two solvers are coupled, whereas the CFD-code will send the required time step to the FE-code.

Velocity streamlines are shown in Figure 3. Motion of elastic flap (displacement time history) is shown in Figure 4.

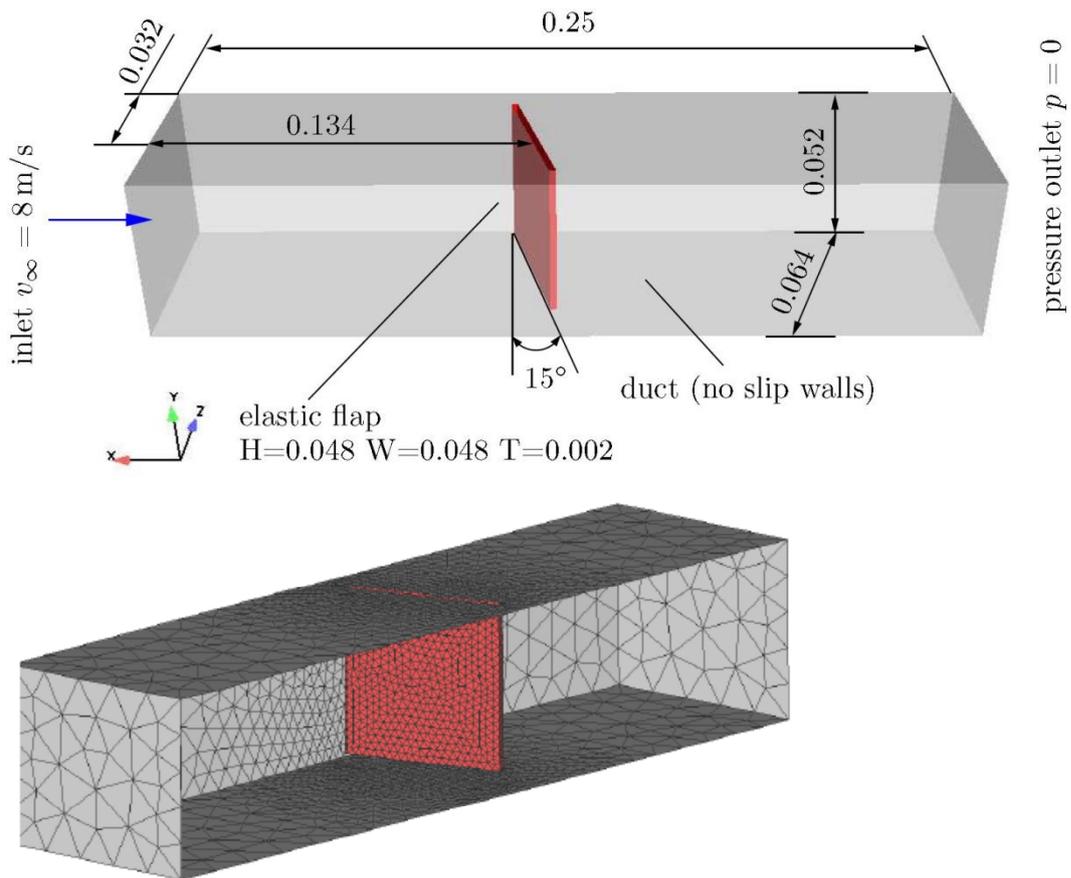


Figure 2 Elastic flap: partial mesh of the fluid domain

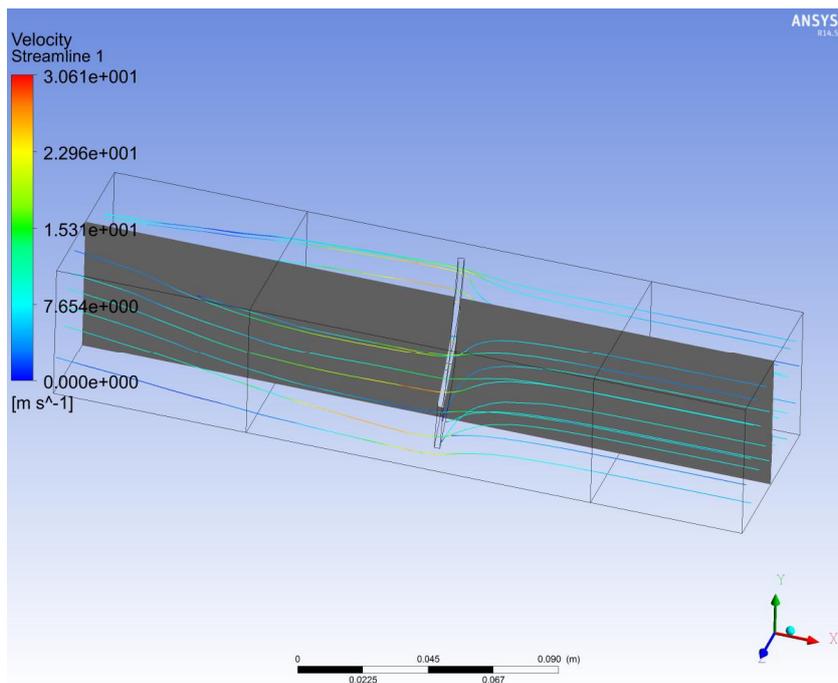


Figure 3 Velocity streamlines

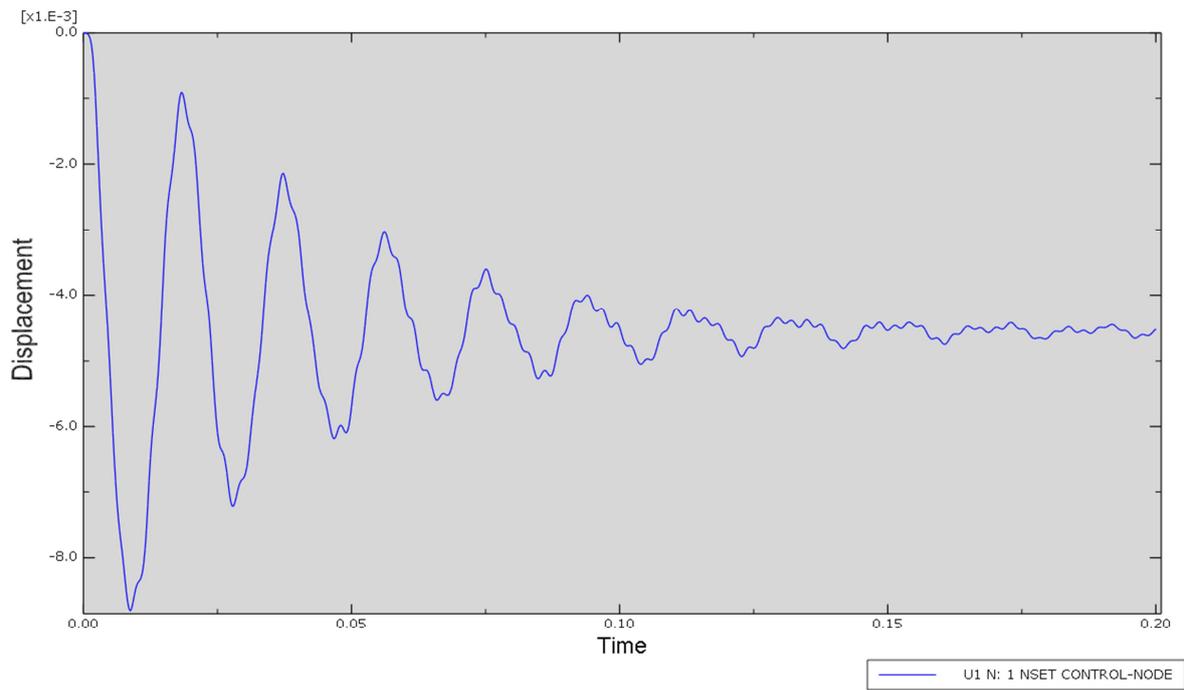


Figure 4 Displacement of the elastic flap in x-direction

4. Case Study

Case study structure was a rotating propeller and a non-rotating cylinder located at the wake of the propeller. The cylinder represents for example the body of the azimuthing thruster. Both structural parts were surrounded by a tube, which formed a cavity for the fluid (water in this case). The example case is shown in Figure 5.

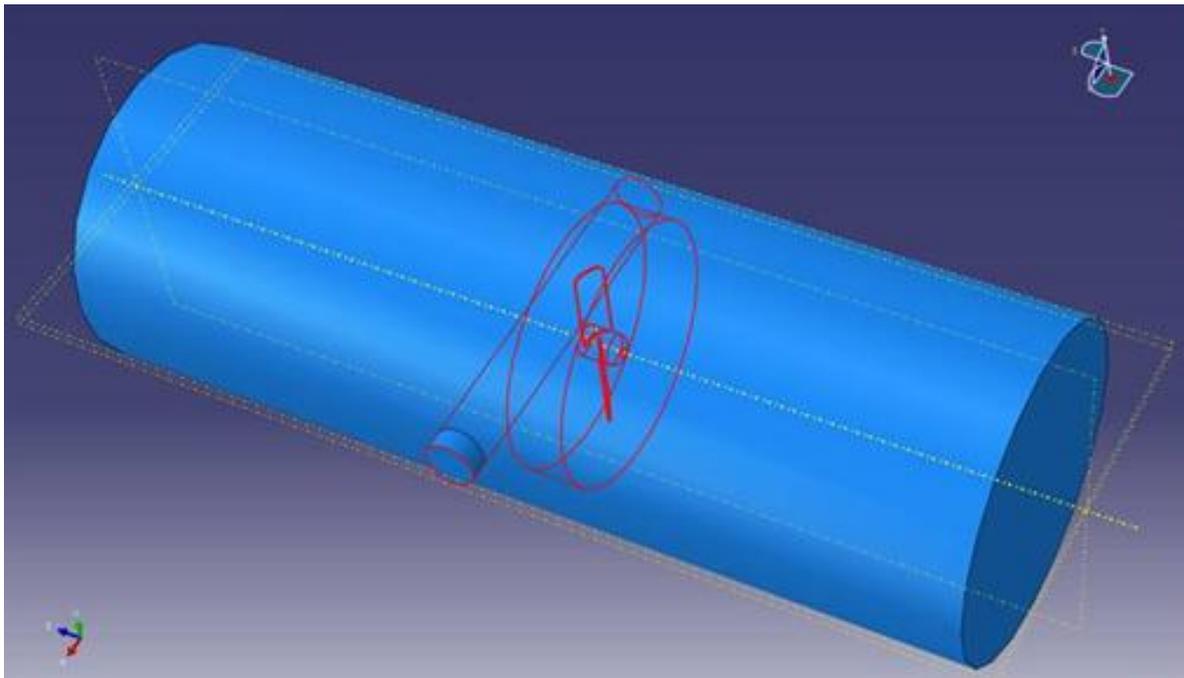


Figure 5 Example case geometries

4.1 Structural Model

The propeller was supported by ground springs at both ends of the hub (supporting point at rotating axis). Dashpots were used in parallel with the spring elements in order to introduce damping for lowest rigid body modes, see Figure 6. Cylinder was also supported by ground springs/dashpots. Material of the propeller and the cylinder was steel. Rayleigh damping model was applied for the material model with coefficients $\alpha=34.3$, $\beta=3.47e-5$.

Main dimensions of the model were:

- Propeller diameter: 600 mm
- Cylinder diameter: 150 mm
- Fluid tube diameter: 960 mm

The propeller was modelled as solid elements and the cylinder with shell elements. The structural FE-model included 29 269 elements and 26 352 nodes.

Constant rotating speed boundary condition was applied for the propeller during simulation. Rotating speed used was 300 rpm.

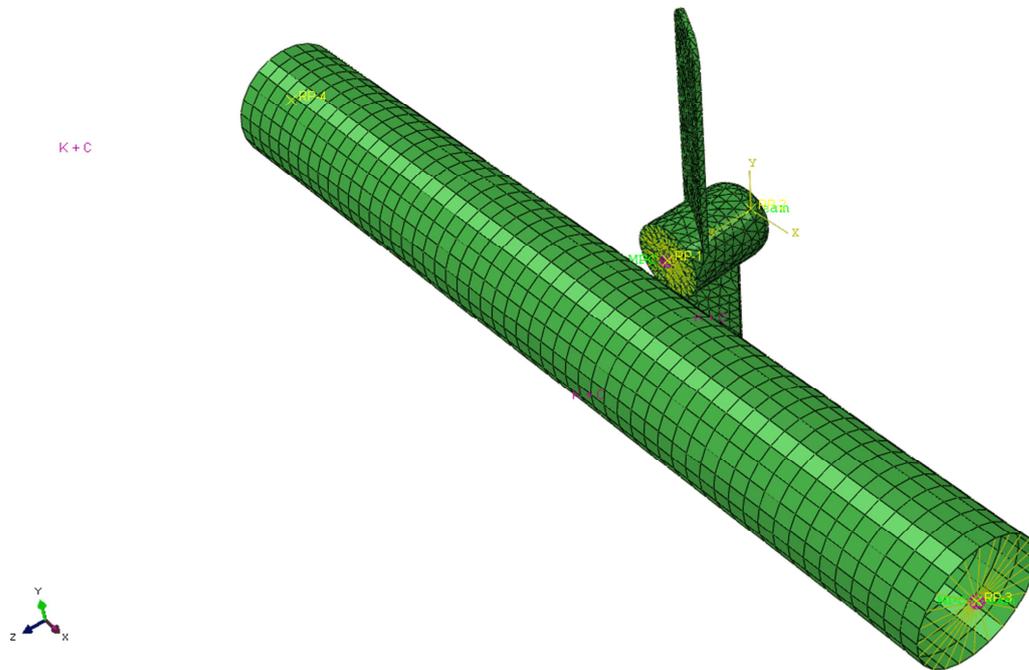


Figure 6 Abaqus FE-model

4.2 CFD Model

The water domain was modelled as a viscid fluid. Cavitation has not been taken into account in these analyses. The CFD model included 963 959 cells. The CFD mesh was generated by ANSYS Meshing. The model is presented in figures Figure 7- Figure 9.

Thickness of the first layer of the boundary layer was 50 μm and element size was 5 mm. The boundary layer included 15 layers. Turbulence model used was K-epsilon model with standard wall functions.

Yplus-parameter on the propeller boundary layer is shown in Figure 10. Yplus is a non-dimensional variable based on the distance from the wall through the boundary layer to the first node away from the wall. It is therefore dependent on the size of the mesh in the wall region. If the value of Yplus is too large, then the wall function will impose wall type conditions further from the wall than would normally be physically appropriate. In this case smaller values than approximately 20 should be avoided /1/.

A sliding mesh model for the rotating cell zone around the propeller was applied. The sliding mesh zone is shown in the figures Figure 7 and Figure 8 by different colour. Pressure-inlet and pressure-outlet boundary conditions (zero pressure) were used in inlet and outlet surfaces of the tube and rigid wall boundary condition for the inner surface of the tube.

Mesh smoothing was used to adjust the mesh of a zone with a deforming boundary (propeller). The mesh smoothing allows the interior nodes of the mesh to move, but the number of nodes and their connectivity does not change. In this way, the interior nodes "absorb" the movement of the boundary. Spring-based smoothing method was used in this case.

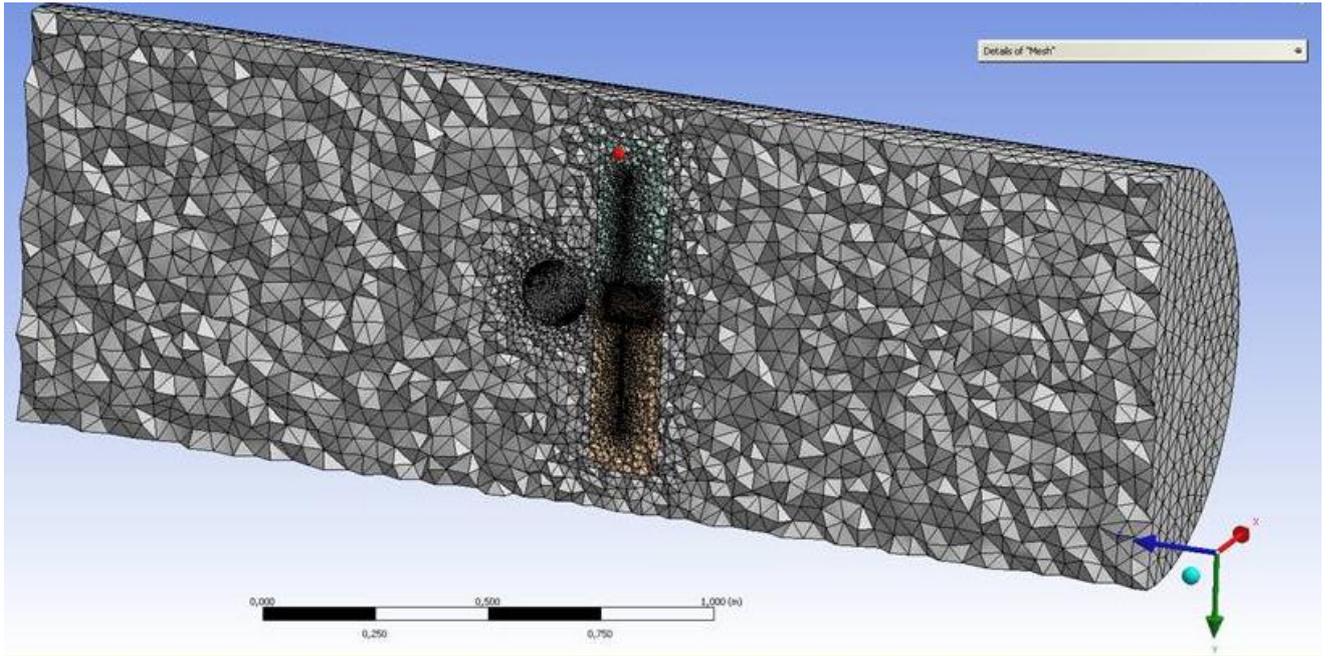


Figure 7 Mesh of the CFD model. Rotating and stationary mesh regions are seen in different colours.

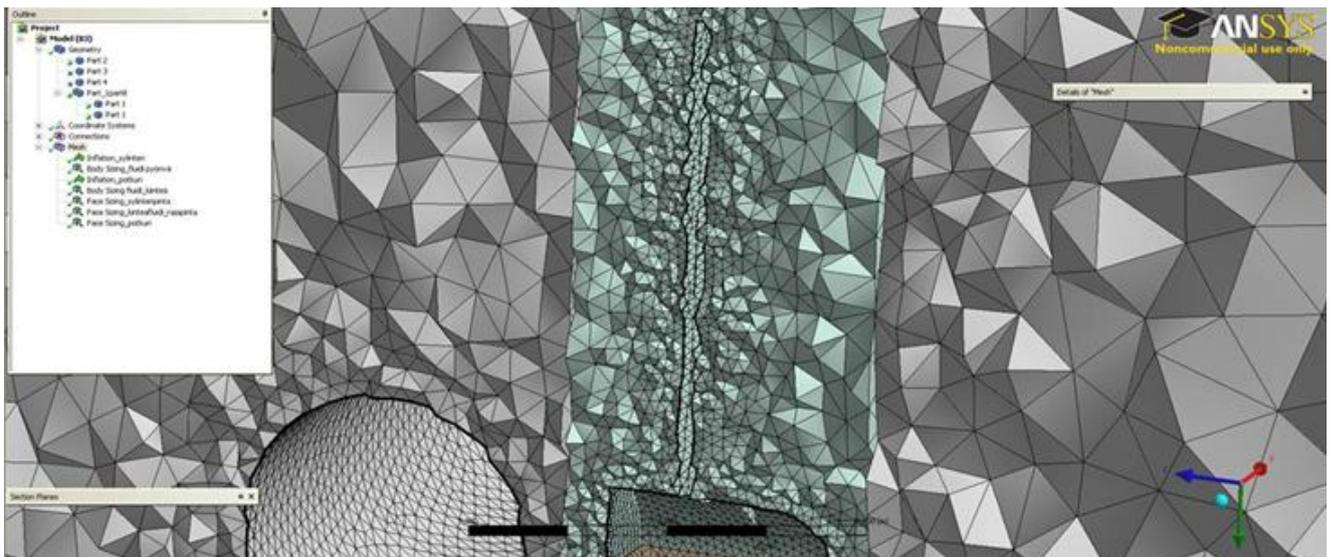


Figure 8 Boundary layers

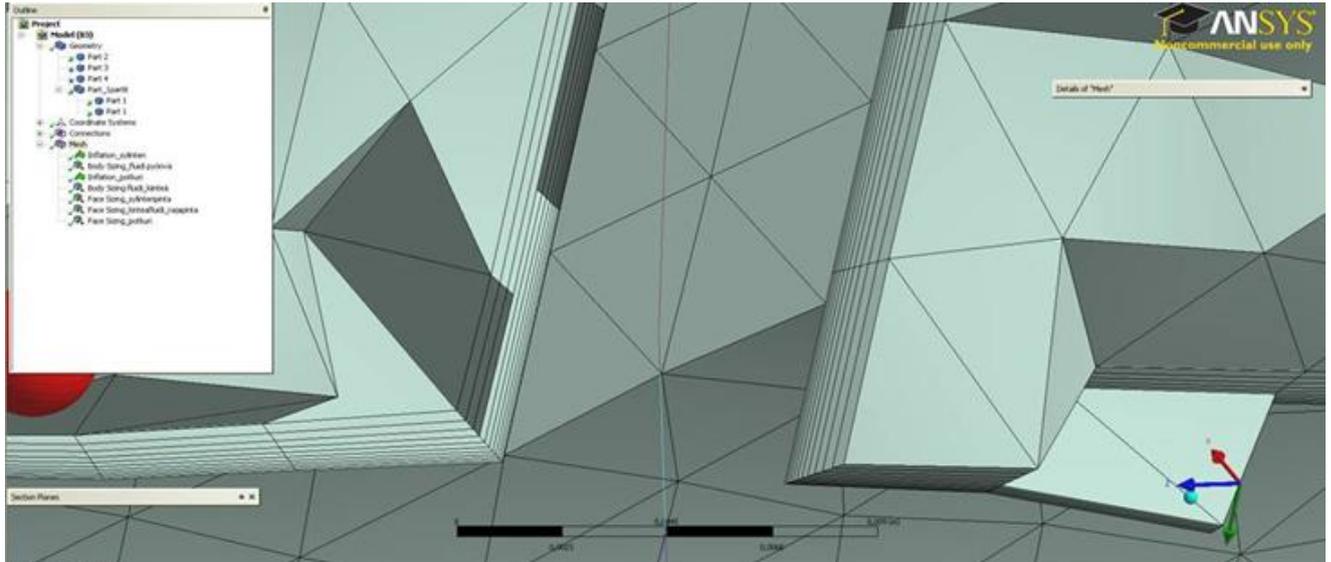
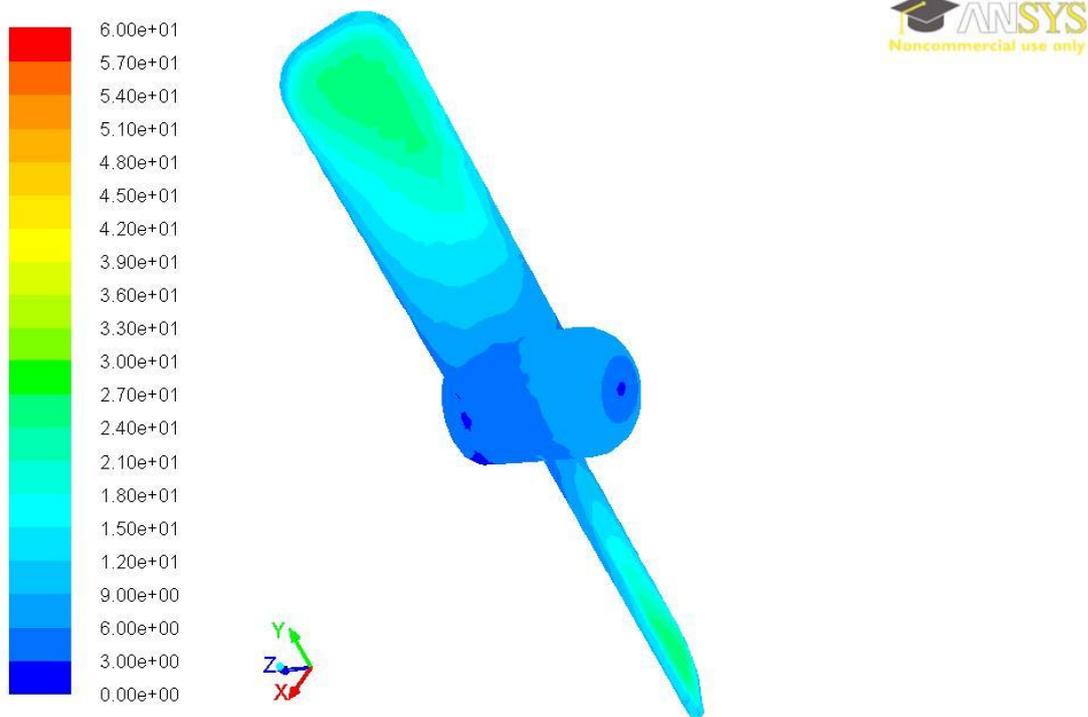


Figure 9 Boundary layers on the propeller surface



Contours of Wall Yplus (Time=6.0000e-01)

Oct 05, 2011
ANSYS FLUENT 13.0 (3d, pbns, rke, transient)

Figure 10 Yplus parameter on the propeller boundary layer

5. Methods

Sequential serial coupling algorithm scheme used in co-simulation is presented in Figure 11. Code A is solid mechanics code (Abaqus) and code B is CFD code (Fluent). The data is transferred between the codes in both directions. Before starting the co-simulation, the CFD-

code sends the pressure data to the FE-code. During iteration step, data is not changed between the codes. Both codes exchange the data before the iteration step. Filled circles in the scheme represent the time points.

It is also possible to use parallel coupling, where iteration steps of both codes are calculated in parallel. However, in this case serial coupling was used.

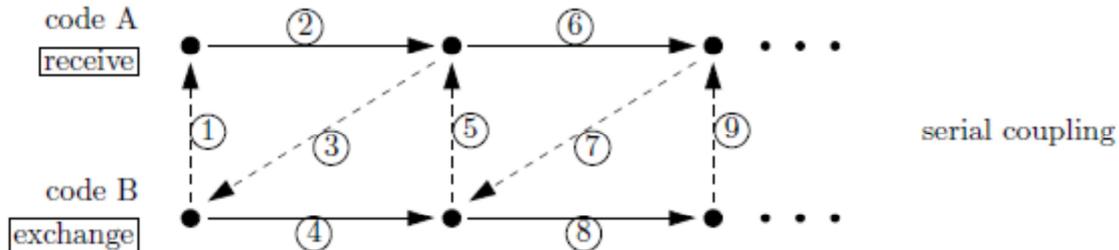


Figure 11. Coupling configuration used in current serial coupling. Code A is solid mechanics code (Abaqus) and code B is CFD code (Fluent). Both codes exchange the data before iteration.

Structural responses were calculated in Abaqus by using implicit time integration method. Large displacement theory was used (geometric non-linearity). Hilbert-Hughes-Taylor time integration method was used. It was needed to introduce moderate numerical damping in time integration (*Dynamic, application=MODERATE DISSIPATION, initial=NO) in order to get stabilized solution.

The meshes of mechanical FE-model and CFD-model were non-matching, and the data exchange between the codes (Nodal position and Pressure) was carried out by interpolation using shape functions. Transient coupling configuration scheme used was "Explicit-Transient", see Figure 11. Time step used in co-simulation was 100 μ s. The coupling step in co-simulation must be small enough in order to get stabilized solution.

For a transient simulation one solver could require smaller time steps for an accurate and/or stable solution process. Forcing the other code to apply the rather small time step might yield an inefficient process. MpCCI allows codes to subcycle, i. e. use a smaller time step size than the coupling time step size. However, in this case subcycling was not used.

Explicit-transient coupling algorithms exchange data once during each time step. In coupled simulations, in which the interaction between the codes is strong, this can cause stability problems. For these cases it is also possible to use so called iterative transient coupling scheme (Jacobi algorithm or Gauss-Seidel algorithm), where the data are exchanged between the codes during iteration. However, it was not necessary to test this scheme in this case.

6. Results

Totally 8000 time steps were simulated (totally 0.8 seconds) which correspond 4 full rotations of the propeller. Parallel processing was used for the Fluent computation, number of parallel processes was 6. Total co-simulation wall-clock time with Windows 7 workstation was about 80 hours. MpCCI monitor representing exchanged variable (pressure field) during co-

simulation on the propeller surface at 0.75s is shown in Figure 12. Due to different mesh densities the local pressure fields are not exactly similar, but overall levels are very similar.

Deformed CFD mesh around the propeller is shown in Figure 13. It can be noted that propeller blade deformation is quite high and the CFD mesh smoothing works well during simulation. The cells in the boundary layer are not deformed during smoothing.

Calculated pressures at rotating speed 300 rpm are shown in figures Figure 14 and Figure 15. Velocity vector fields are presented in figures Figure 17 - Figure 19 and velocity streamlines are shown in Figure 20. The pressure and velocity fields obtained seems to be reasonable.

Axial displacements at tip of the propeller blade (purple) and propeller hub is presented in Figure 21. It can be seen that most dominating component of axial displacements of the propeller is the blade frequency, about 10 Hz. This is caused by cylinder structure located at the wake of the propeller. For transversal and vertical displacements of the hub the dominating component is rotating frequency (5 Hz), see Figure 22. However, orders of magnitude of transversal and vertical displacements of the hub are significantly smaller.

Von Mises stresses of the propeller structure at 0.8 s are shown in Figure 23.

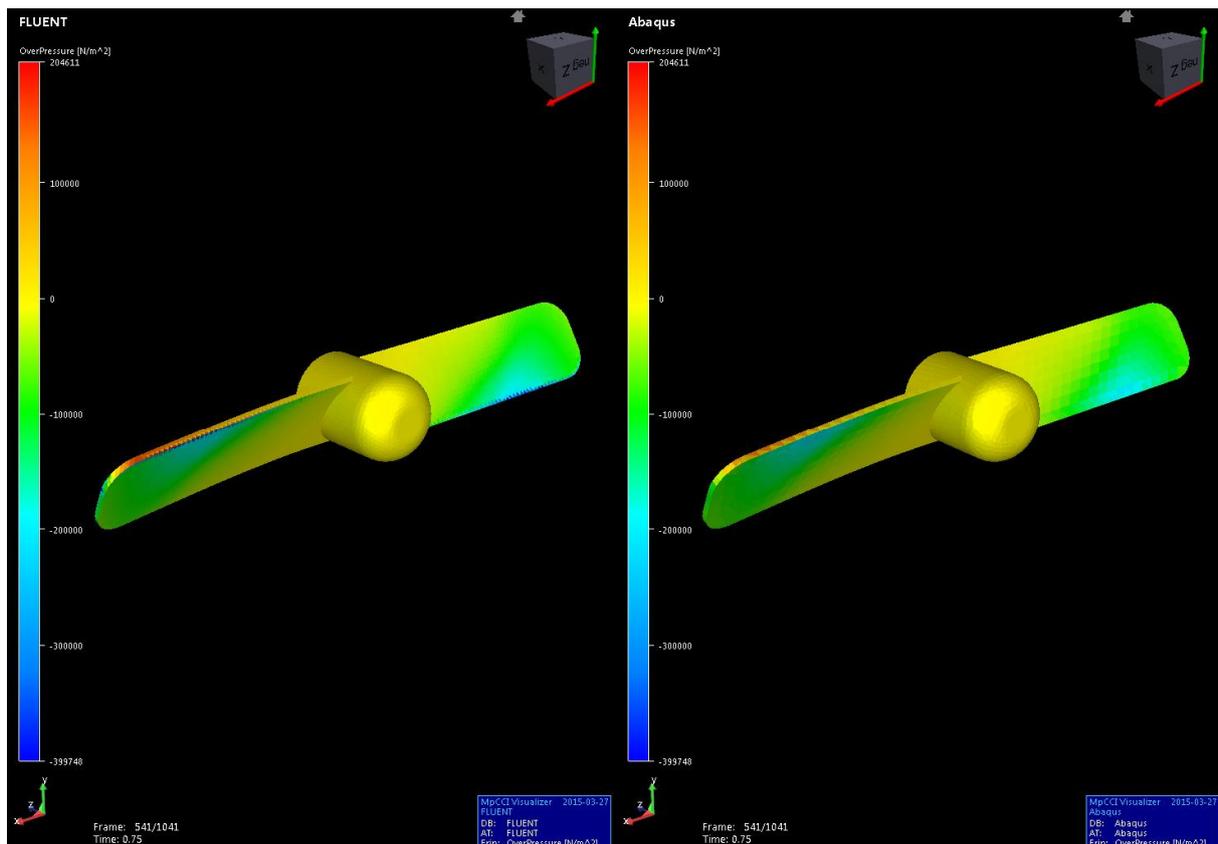


Figure 12 MpCCI monitor representing exchanged variable (pressure field) on the propeller surface at 0.75s

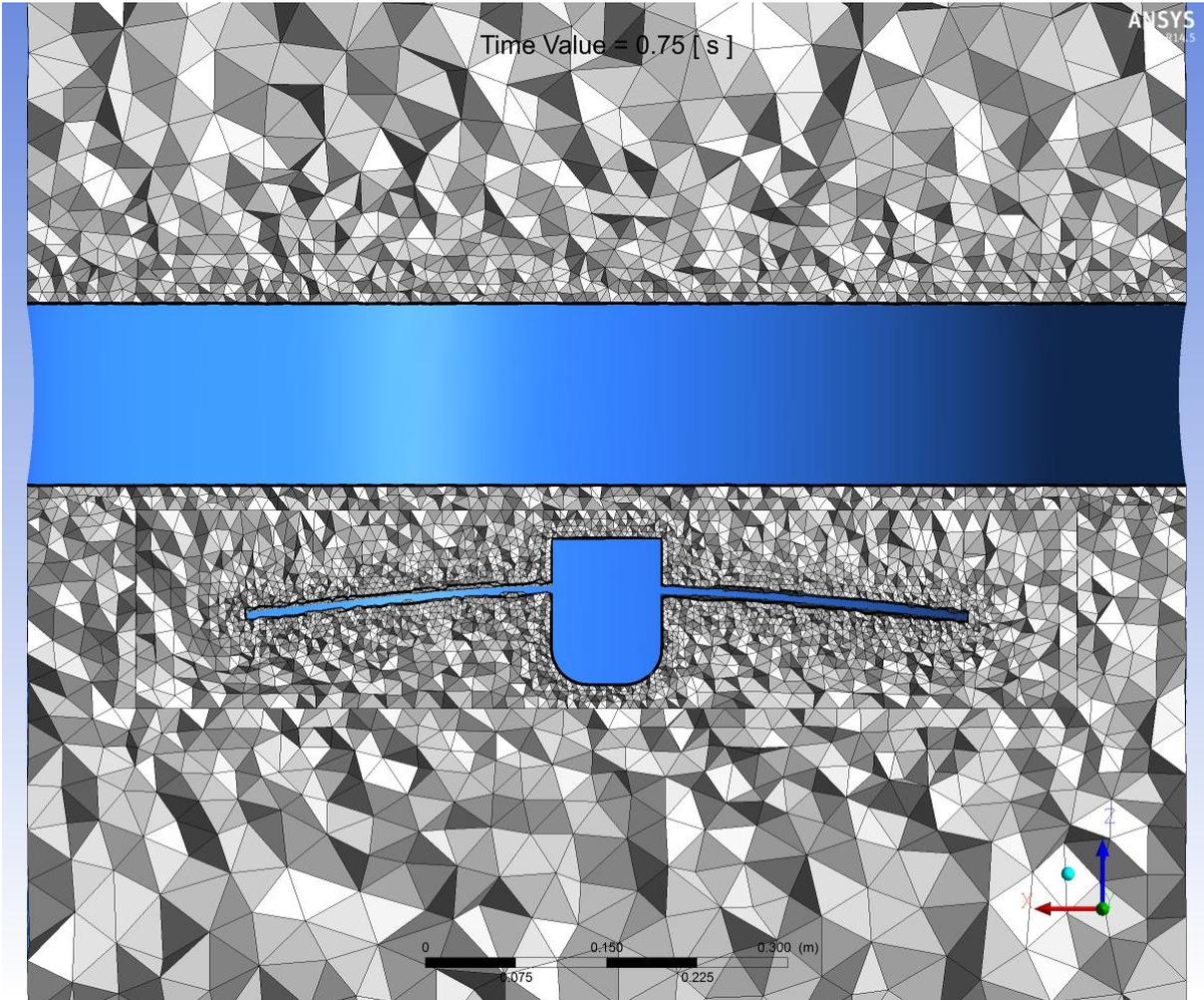


Figure 13 Deformed CDF mesh around the propeller

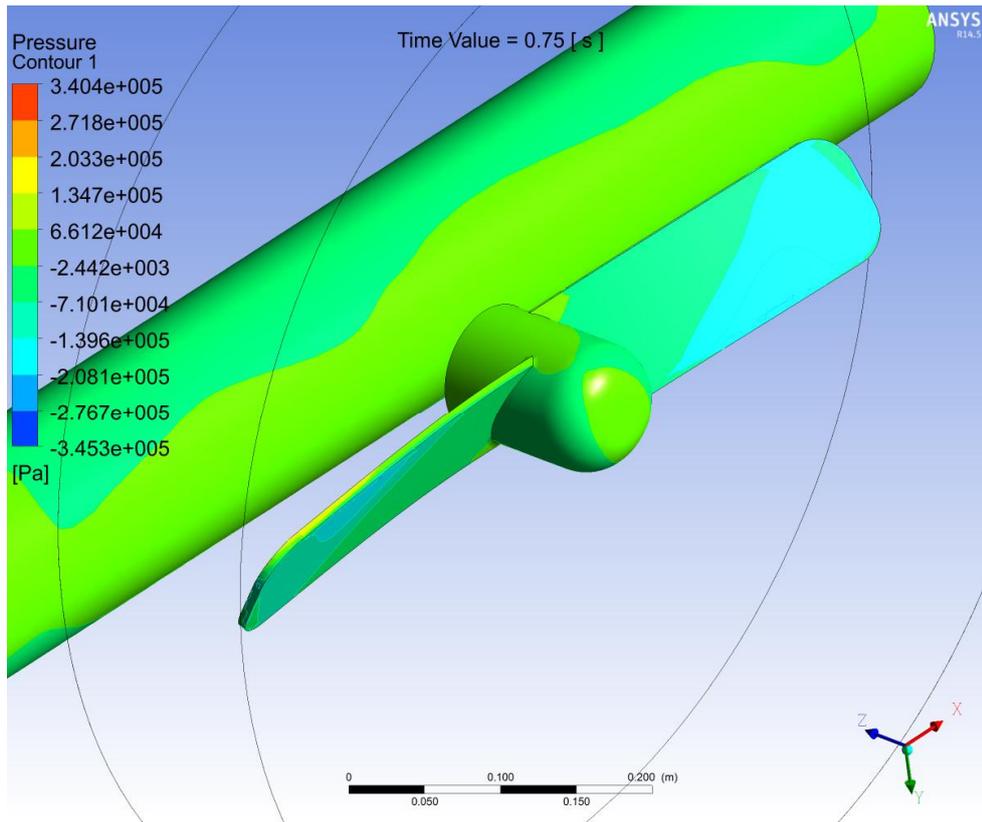


Figure 14 Pressure on the propeller and the cylinder surface at 0.75s

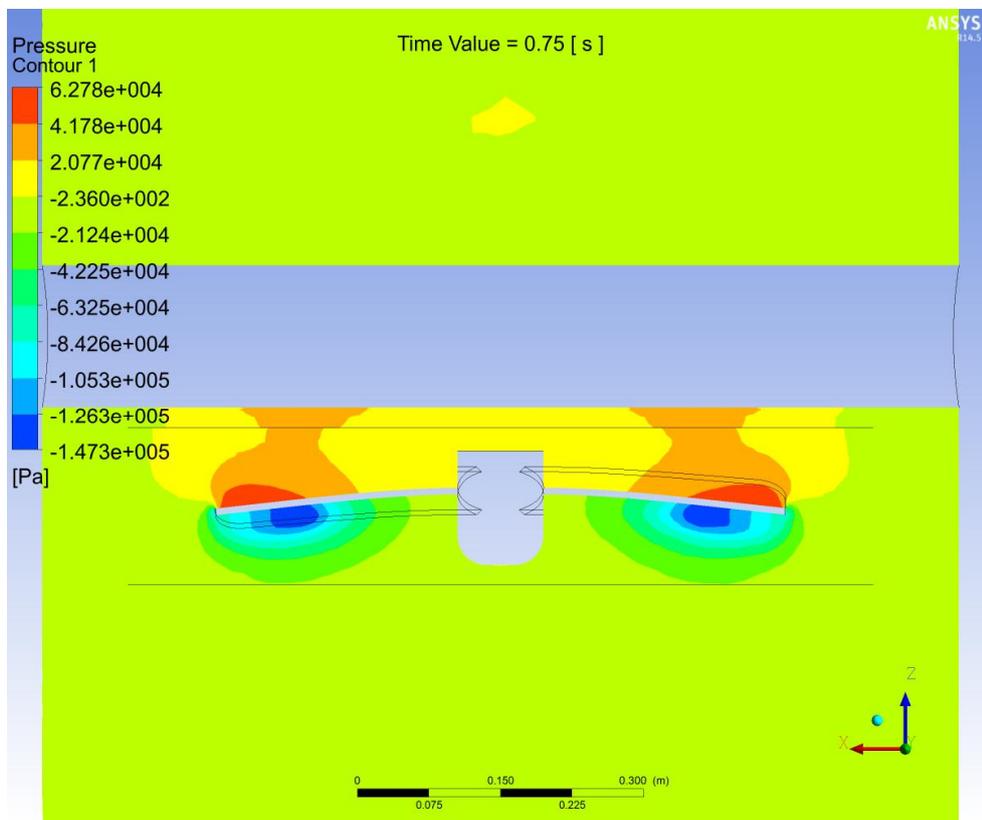


Figure 15 Pressure on the XZ plane at 0.75s

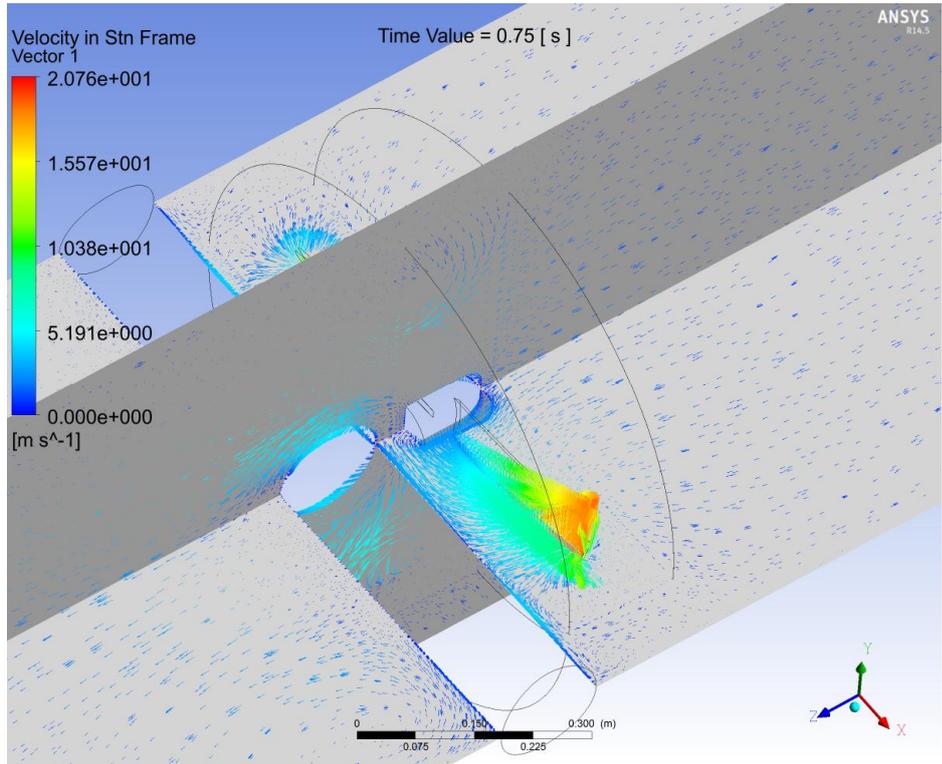


Figure 16 Velocity vector field at 0.75s

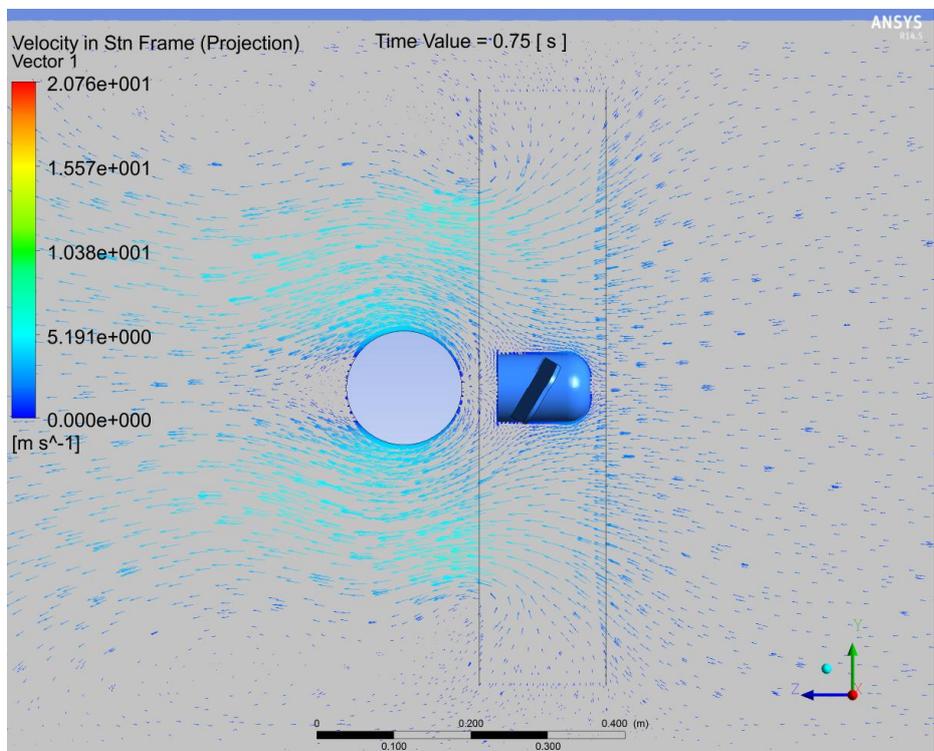


Figure 17 Velocity projection on YZ plane at 0.75s

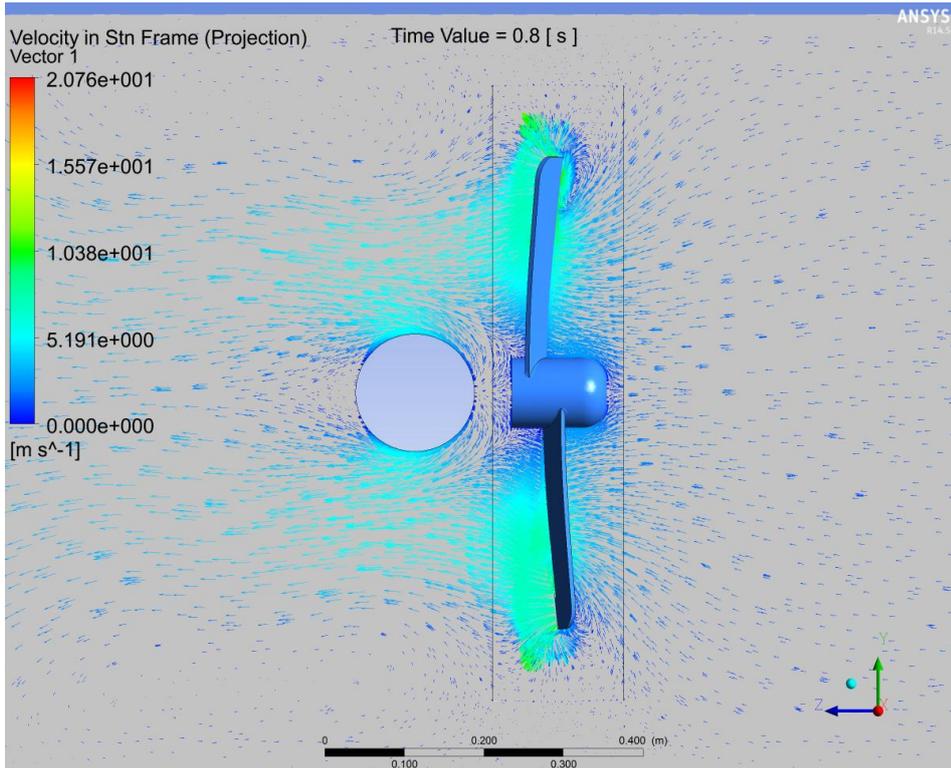


Figure 18 Velocity projection on YZ plane at 0.8s

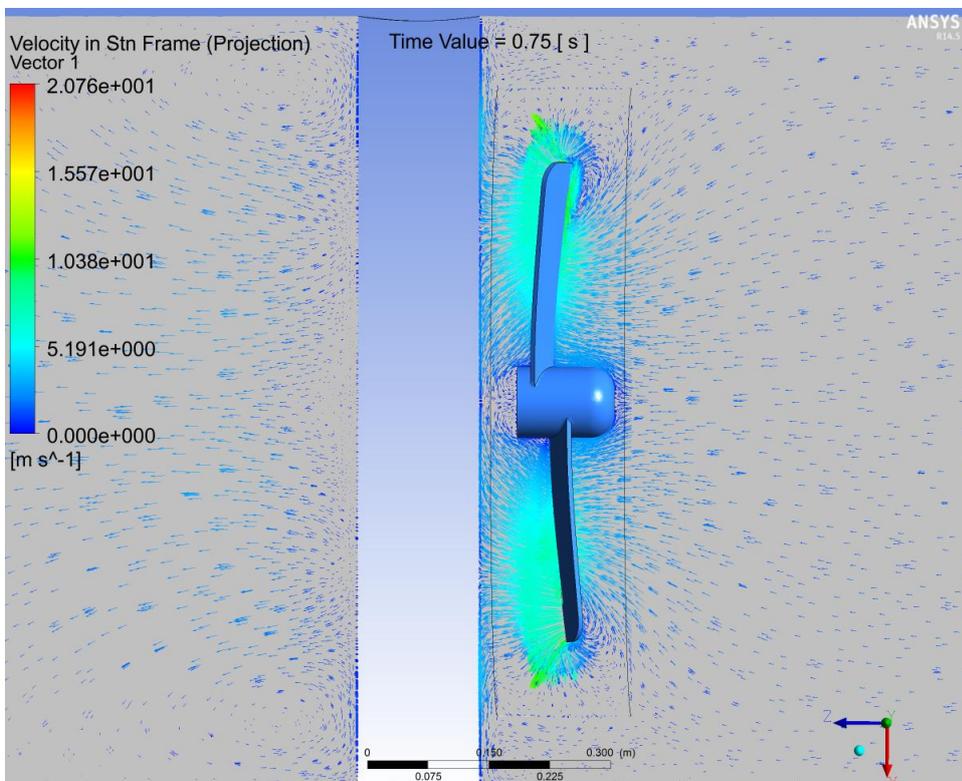


Figure 19 Velocity projection on XZ plane at 0.75s

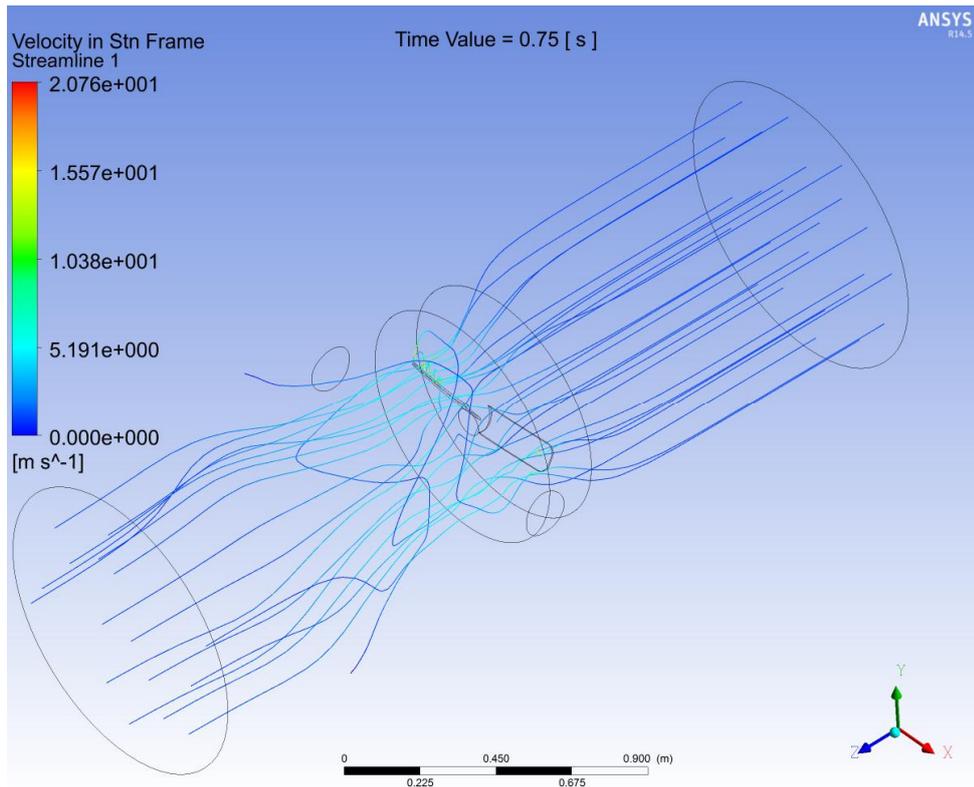


Figure 20 Velocity streamline at 0.75s

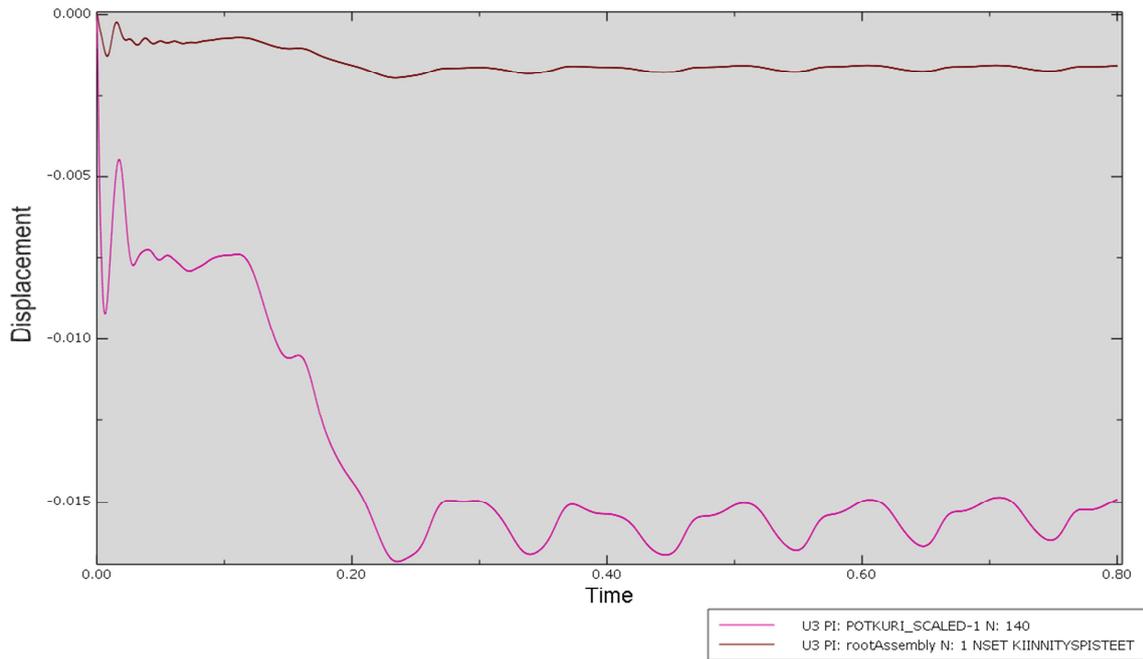


Figure 21 Axial displacements at tip of the propeller blade (purple) and propeller hub (brown)

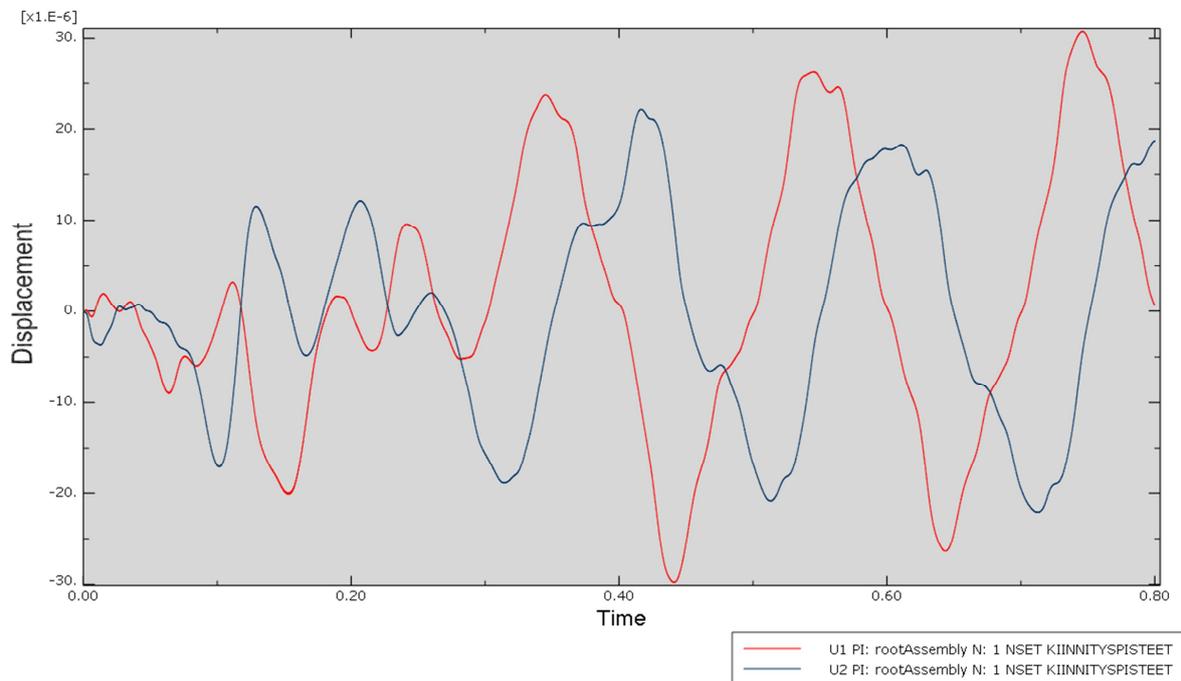


Figure 22 Transversal (red) and vertical (blue) displacement of the propeller hub.

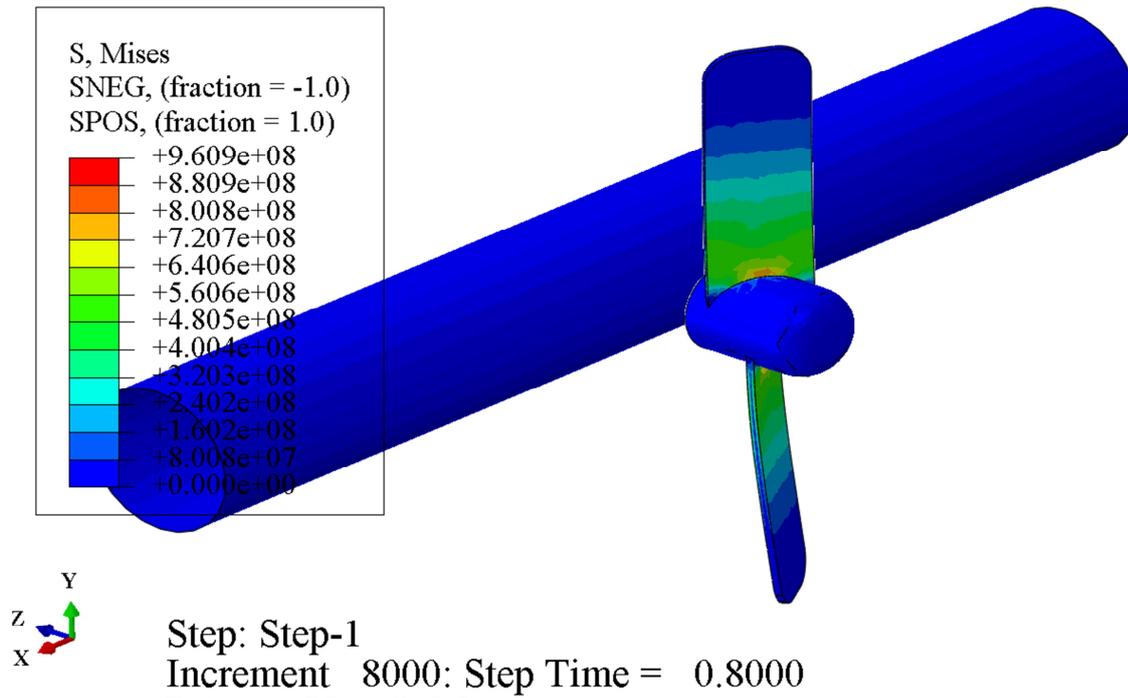


Figure 23 Von Mises stress of the propeller structure at 0.8s.

7. Conclusions and Summary

Two-way FSI co-simulation with Fluent CFD-code and Abaqus structural FE-code utilizing MpCCI was applied successfully. In an FSI simulation, a structure deforms due to forces caused by a fluid flow, while the deformation changes the fluids boundary. So called weak coupling was used, which means that each problem is solved separately and during each time step some variables are exchanged and inserted into the equations of the other problem (pressure and nodal position in this case). Example structure in this case study was a rotating propeller and a non-rotating cylinder located at the wake of the propeller. The cylinder represents for example the body of the azimuthing thruster.

The pressure and velocity fields obtained seemed to be reasonable. It was found that most dominating component of axial displacements of the propeller is blade frequency. This is caused by cylinder structure located at the wake of the propeller. For transversal and vertical displacements of the hub, the dominating component is rotating frequency.

The goal of this study was to demonstrate two-way FSI co-simulation utilising MpCCI, and therefore the CFD-model was not optimized for propeller calculations. In real applications for example k- ω turbulence model with finer boundary layer could be better for the rotating propeller. Also mesh refinement in the propeller wake should be considered.

In the future work the results obtained should be compared with results with one-way co-simulation in order to find out impact of two-way coupling for this type of problems. Also computing times could be compared between these simulations.

References

- [1] Fluent 14.5 Manual
- [2] MpCCI 4.3.1-1 Documentation, Fraunhofer Institute for Algorithms and Scientific Computing SCAI, Germany, 2014