WEST-EAST HIGH SPEED FLOW FIELD CONFERENCE
19-22, November 2007
Moscow, Russia

# MODELING FLUID STRUCTURE INTERACTION FOR AEROSPACE APPLICATIONS

**Andrey A. Aksenov, Kirill A. Iliine and Vladimir V. Shmelev**

**Abstract.** An approach for solving Fluid Structure Interaction in aerospace application is presented in this paper. The proposed approach is based on the two-way coupling between CFD code FlowVision and FEA code ABAQUS. The codes are coupled directly without using any 3rd party software or intermediate structure. A direct link offers a full control over the load transfer and interpolation error free data exchange between the codes. The direct link is implemented using special meshing techniques (submerged meshes in FlowVision). FE mesh is subtracted from the Cartesian CFD mesh; all links between CFD mesh cell and outside faces of the finite elements are preserved. Node displacements are transferred directly between FlowVision and Abaqus without any interpolations. The above approach is illustrated with simulation of helicopter emergency landing on the water surface (helicopter equipped with flexible landing ballonets). The simulation objective is to estimate maximum loads on the helicopter hull caused by splashdown. The ballonets should absorb some of the impact and decrease acceleration on the helicopter crew. Results of two simulations are compared: helicopter lands on a rigid surface (ground) and on the still water surface (splashdown).

## 1. INTRODUCTION

Pneumatic ballonets have wide application in different technical areas. Small overall sizes, low weight and high specific load capacity made pneumatic ballonets a universal design element in many engineering applications. Ballonet light structures are used in the motor-car construction, aviation, and shipbuilding as e.g. emergency rescue elements:

- automobile airbags;
- inflatable emergency chutes in the passenger aircraft;
- splashdown devices;
- life rafts, etc.

Helicopters accomplishing long flights above large water areas must obligatory have possibility for save emergency landing on the water surface (Figure 1). The ballonets are inflated by crew in case of splashdown. Ballonet damps the landing forces and is used to secure the helicopter buoyancy.

Figure 1. Helicopter with inflated ballonet

During the splashdown the ballonets are deformed and absorb the energy reducing the forces and making the emergency landing safe for the crew and helicopter.

Inflation of ballonet and its deformation during impact of helicopter with rigid surface (land) is fast unsteady-state process. This problem is solved successfully in different finite-element codes and is rather standard task for Abaqus. However in case of impact with water surface the problem cannot be solved in existent commercial FEA codes. Accurate and adequate solving of this problem requires simultaneous simulation of water motion, ballonet deformation and impact of the helicopter hull with water; thus it requires modeling strong Fluid-Structure Interaction (FSI).

In this paper we present approach for solving such problems using Abaqus FEA code and FlowVision CFD code. Helicopter hull, suspension of ballonets and ballonets themselves are defined in Abaqus as flexible bodies represented by FE mesh. The FE mesh is imported to FlowVision as one of the boundaries of the computational domain. The FE representation can freely move and deform in the computational domain.

Split method is used to couple the solutions generated by both codes. Abaqus simulates kinematics and deformation of the ballonets. After completing each simulation time step the new node coordinates are passed to FlowVision. Based on the updated node coordinates FlowVision generates new CFD mesh and calculates new loading case. The loads (pressure) are then transfer to Abaqus to calculate new deformation.

The described approach for linking FlowVision and Abaqus uses direct coupling and doesn't require any additional software. The advantage of this approach is a simple and controllable interface between the codes. There is no interpolation involved in the data exchange process, which results in fully controllable and interpolation error free data transfer between the FEA and CFD.

The splashdown of helicopter with inflated ballonets was simulated using this approach. Simulation of splashdown was compared with impact of the helicopter with land. Maximum loadings on the helicopter hull and acceleration forces are compared between those two cases.

## 2. NUMERICAL METHOD

### 2.1. FEA mesh

Helicopter and ballonets with suspension are fully defined in Abaqus and described by finite-element mesh. The used FE element type is arbitrary and does not have any effect on the coupling between Abaqus and FlowVision.

## 2.2. CFD mesh

Borders of the fluid computational domain in FlowVision are described as surface mesh (Figure 2). This surface mesh comes from a CAD describing the helicopter geometry (VRML or STL formats), or it is imported from FEA volume mesh as outside faces of the elements or like FEA surface mesh. Helicopter hull and suspension is defined by volume FE mesh, ballonets are defined via shell elements. Splashdown of helicopter is so-called “outer problem” in hydrodynamics. It means the outside border of the computational domain must be defined. This border serves to approximate boundary conditions “in infinity”. In this case the outside border is a surface mesh got from CAD system. CFD mesh is generated between these two surfaces. Cartesian mesh is created initially in the fluid computational domain. This mesh is dynamically adapted to the boundaries or to a solution. Adaptation means subdivision of the original cell into 8 smaller cells, or coursing the mesh by merging 8 neighboring cells in one bigger (Figure 3). To accurately approximate curvilinear boundary conditions a Sub-Grid geometry resolution method is used [3,4]. This method is based on a Boolean subtraction between Cartesian volume mesh with local adaptation and curvilinear boundary of the computational domain. If subtracting boundary is a FE mesh (in our case it is surface of the helicopter ballonets) a direct link between CFD mesh and FE mesh is specified; see details of this method in paper [2].

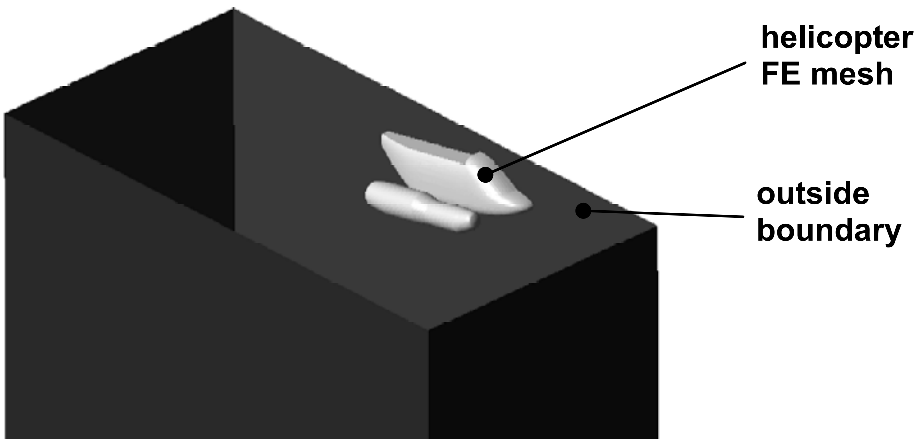



Figure 2 Boundaries of fluid computational domain.

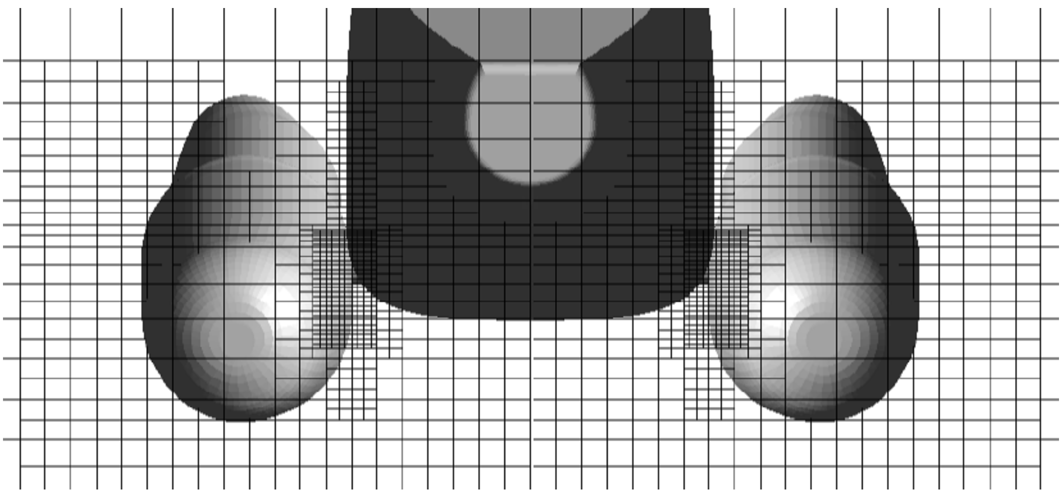


Figure 3 CFD mesh with Local Adaptation and Sub-Grid geometry resolution

## 2.3. Governing Equations

Modeling structure is performed by Abaqus/Explicit to take into account deformation of the helicopter, suspension, ballonets and its kinematics behavior. Governing equations for deformed structure in terms of discrete finite-element model are the following:

$$\mathbf{M}\frac{d^2\mathbf{u}}{dt^2} = \mathbf{P} + \mathbf{P}_f - \mathbf{I} \tag{1}$$

where **M** is the mass matrix of the finite element system, **u** – displacement of the nodes. **P** is non-hydrodynamic force acting on the structure, **I** is the internal element force. $\mathbf{P}_f$ is hydrodynamic force equals

$$\mathbf{P}_f = P \cdot \mathbf{s},$$

where **s** is vector-area of external face of the element, $P$ is a fluid pressure, calculated from Navier-Stokes equations. Navier-Stokes equations in integral form applied to calculation grid of fluid flow domain are:

$$\frac{d}{dt}\iint_{\Omega,\tau} \mathbf{V}d\Omega dt + \oint_S \mathbf{V}(\mathbf{V}-\mathbf{W})d\mathbf{s} = -\oint_S \frac{P}{\rho}d\mathbf{s} + \oint_S D d\mathbf{s}\,. \tag{2}$$

Integral form of continuity equation is

$$\oint_S (\mathbf{V}-\mathbf{W})d\mathbf{s} = 0,$$

where **V** is fluid velocity, $\mu$ - viscosity, $\rho$ - density, $\mathbf{W} = \dot{\mathbf{u}}$ - velocity of the structure surface, $\Omega$ and S is a volume and a surface of the cell of fluid flow computational domain, $\tau$ is time increment for fluid flow simulation.

## 2.4. Coupling Abaqus and FlowVision

Split numerical method is used for 2-way coupling deformation equations and fluid motion equations. Exchange of information between Abaqus and FlowVision takes place at specified by the user time intervals $\Theta^{n+1}$ (FSI time step), $\Theta^{n+1} = T^{n+1} - T^n$, where $T^{n+1}$ and $T^n$ are time moments of synchronization between both solutions. Inside each FSI time step both codes can do several (or one) time increments. In present implementation an explicit splitting algorithm is used. The disadvantage of explicit method can be use of small FSI time step. But this disadvantage is compensated by fast calculation speed as not internal iterations are necessary. The coupling splitting method is the following:

- Initially Equation 1 is calculated by ABAQUS to obtain displacement of the nodes $\mathbf{u}^{n+1}$ corresponding to time step $\Theta^{n+1}$. Fluid pressure $P^n$ is obtained from previous time step $\Theta^n$ and assumed constant during the time step $\Theta^{n+1}$.
- Displacement of the nodes $\mathbf{u}^{n+1}$ is transferred to FlowVision; velocity of the deformed surface **W** is calculated.
- Equation 2 are calculated by FlowVision to obtain fluid loading on the structure.
- Pressure $P^{n+1}$ is transferred to ABAQUS at the end of all FlowVision time increments at moment $T^{n+1}$.

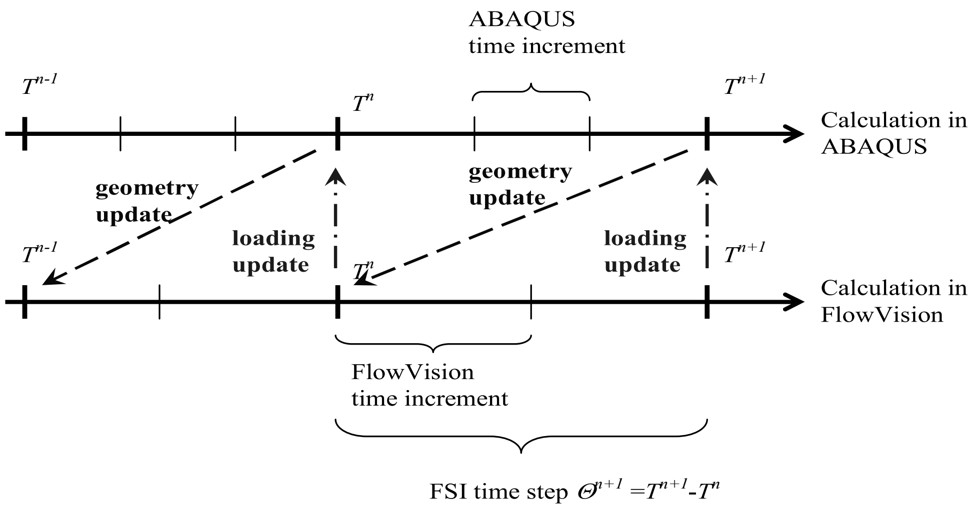



Figure 4 Explicit coupling procedure

## 2.5. Numerical method for solving equations of fluid dynamics

FlowVision uses an Euler approach to solve fluid dynamics equations in the computational domain with moving boundaries. Short description of the method used can be found in our previous papers [1,2]. Navier-Stokes equations are solved by split method for physical variables described in [5].

## 2.6. Multi-Physics Manager (MPManager)

MPManager controls Abaqus and FlowVision during their coupled simulation and transfers loadings from FlowVision to Abaqus and in the counter direction - node displacements from Abaqus to FlowVision. To set up a FSI simulation is very easy for users who are familiar with Abaqus and FlowVision. User creates Abaqus project for simulating splashdown of helicopter almost in the same way as for simulating impact with the rigid surface. In Abaqus' input file, the user must define the outside surface of the helicopter, suspension and ballonets as surfaces for which Abaqus User Subroutines will be invoked. This Abaqus input file is imported into FlowVision project to define the deformable body. Link between two meshes is built automatically and is hidden for the user. In the MPManager user defines only the path to Abaqus and FlowVision codes and defines the FSI time step. During simulation the user can view the results on-line via FlowVision post processor.

# 3. RESULTS

3D simulation of helicopter splashdown and impact with rigid surface was performed for half of the model because of symmetric helicopter hull. The ABAQUS and FlowVision models and FSI results are described below

## 3.1. Problem Statement for structure

Model consists of 3 parts – helicopter hull, ballonets and ballonet suspension. The helicopter hull is defined as absolutely rigid shell with 2 mm thickness. Ballonet is modeled as elastic shell of 5 mm thickness. Material of ballonet is reinforced rubber.

Finite-element model of the helicopter is shown in Figure 5. Shell finite element is used for hull and ballonets. Following loadings, boundary conditions and initial conditions are specified:

- ballonet inflation is 1 bar;
- all degree of freedom except vertical was fixed;
- gravity force is applied;
- initial speed of helicopter model is 3 м/с and directed vertically downward.

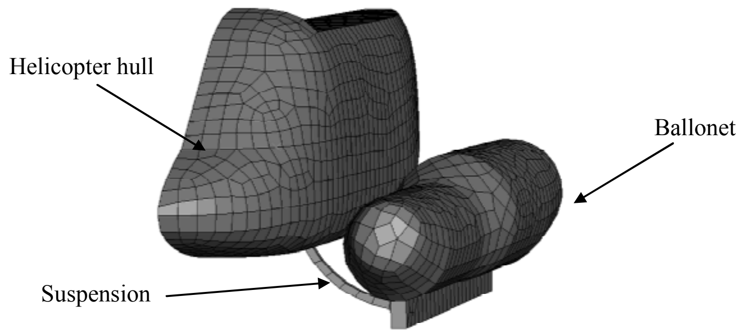



Figure 5 Helicopter finite-element model (one-half)

### 3.2. Problem statement for fluid flow

Fluid flow around the helicopter is simulated for one half of the helicopter model. A CFD model is shown in figure 6. At initial time moment helicopter was just above the water surface. Vertical speed of helicopter and its acceleration under gravity force is defined by Abaqus. Boundary conditions of fluid computational domain are shown in Figure 6.

A finite-volume mesh with local adaptation is shown in Figure 7. The mesh is refined between the ballonet and helicopter hull to accurately resolve water jet formed during helicopter splashdown.

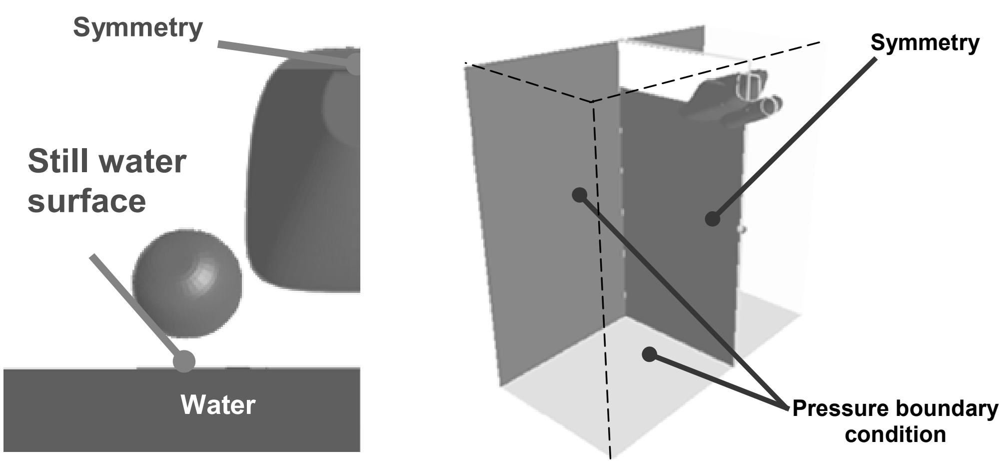



Figure 6. CFD model of helicopter splashdown.

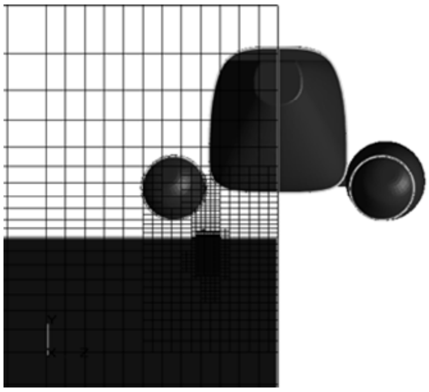


Figure 7. Finite-volume mesh with local adaptation near hull and ballonets.

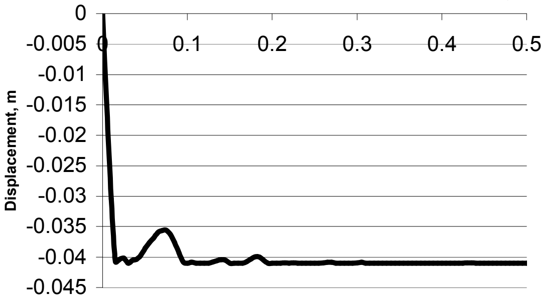
## 3.3. FSI simulation

One of the important characteristic of the landing or splashdown is speed of center mass and its acceleration. Using these values force acting on crew during impact is defined. Vertical displacement of center mass is shown in Figure 8 for both cases – impact with land (Figure 8,a) and splashdown (Figure 8,b). Acceleration of center mass is shown in Figure 9. As one can see the action of acceleration is shorter at landing, but more intensive on amplitude. Splashdown is softer, but action of the acceleration is longer.

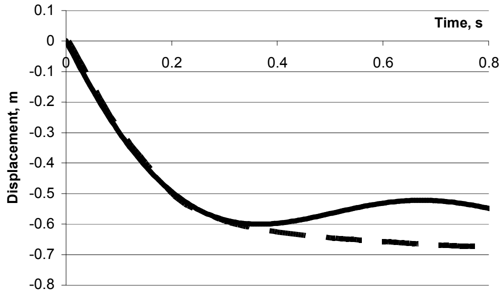
The influence of the ballonet stiffness on the behavior of the helicopter during splashdown is investigated. Center mass displacement and acceleration are shown in Figures 8,b and 9,b by solid and dashed lines. Simulation of helicopter with rigid ballonets is provided without coupling with Abaqus, only FlowVision is used for this simulation. One can see that ballonet deformation lead to deeper submersion of the helicopter in water. Acceleration as expected is larger in case of rigid ballonets.

Deformation of ballonets with suspension is shown in Figure 10 at different time moments. When ballonets are submerged into the water, its underwater part is affected by hydrostatic pressure and slightly squeezed. This results in to bigger submerging of helicopter in comparison with rigid ballonets.

Helicopter generates waves during impact with the water surface. These waves are shown in Figure 11. Wave generation results in oscillating vertical speed of helicopter with decreasing amplitude after impact. One can see that helicopter with soft ballonets generates waves more intensively. It results in better absorption of kinematical energy and lower accelerations during splashdown.

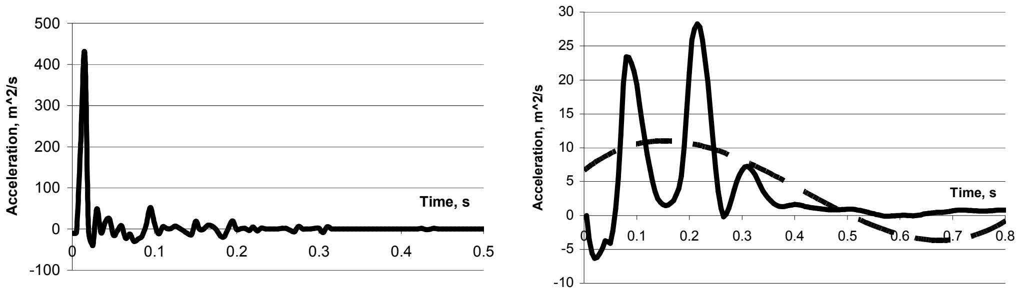



a)



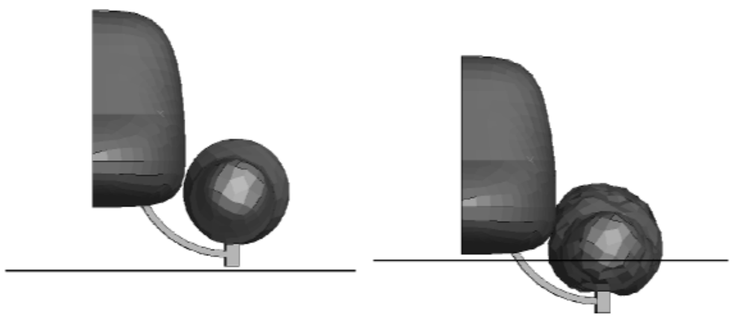

b)

Figure 8. Vertical displacement of helicopter center mass.
a) Landing. b) Splashdown; solid line – rigid ballonets; dashed line – deformed ballonets



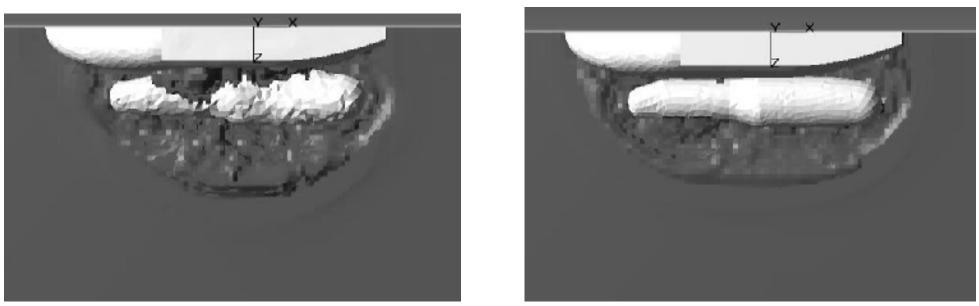

a) b)

Figure 9. Vertical acceleration of helicopter center mass.
a) Landing. b) Splashdown; solid line – rigid ballonets; dashed line – deformed ballonets



a) b)

Figure 10. Deformation of ballonets at times a) t=0 and b) t=0.825 sec after splashdown of helicopter.



a) b)

Figure 11. Comparison of water distribution over the helicopter for a) soft and b) rigid ballonets, time is 0.825 s after splashdown.

## 4. CONCLUSION

FSI simulation based on 2-way coupling between FlowVision and Abaqus is successively used for modeling emergency splashdown of helicopter with pneumatic ballonets. The FSI approach provides designers with more accurate and detailed information of the processes during helicopter splashdown. In this way acceleration on the crew, loadings on helicopter hull and suspension of ballonets, maximum loads on suspension, reserve buoyancy can be adequately predicted. Designer can choose stiffness of ballonets and their construction to fulfill the requirements for recomended acceleration loads on the crew and ballonets submerging in water.

## 5. REFERENCES

[1] A. Aksenov, K. Iliine, T. Luniewski, T. McArthy, F. Popielas, R. Ramkumar "Oil Leakage Through a Valve Stem Seal", Proc. Abaqus User Conference, 2006, Boston, USA, 2006

[2] A. Aksenov, A. Dyadkin, T. Luniewski, V. Pokhilko "Fluid Structure Interaction analysis using Abaqus and FlowVision", Proc. Abaqus User Conference, 2004, Boston, USA, 2004

[3] Aksenov A, Dyadkin A, Pokhilko V. "Overcoming of Barrier between CAD and CFD by Modified Finite Volume Method", Proc. 1998 ASME Pressure Vessels and Piping Division Conference, San Diego, ASME PVP-Vol. 377-1., 1998

[4] Aksenov A., Dyadkin A., Gudzovsky A., 1996, "Numerical Simulation of Car Tire Aquaplaning". Computational Fluid Dynamics '96, J.-A. Desideri, C.Hirsch, P.Le Tallec, M.Pandolfi, J.Periaux edts, John Wiley&Sons, pp. 815-820.

[5] Belotserkovsky, 1994 "Numerical Methods in Continuum Mechanics", Moscow, Fizmatlit, 2nd edition, p.441