Hydrodynamic Drag Force Predictions for Amphibious Military Vehicles

İlker Kurtoğlu

FNSS Savunma Sistemleri A.Ş. Oğulbey Mah. Kumludere Cad. No: 11 Gölbaşı Ankara TURKEY

Abstract

Amphibious military vehicles are very important in the battle field due to their flexible operating environment. In general, amphibious military vehicles are designed for water operations in still lakes and rivers with low current speed. There are also some examples of amphibious military vehicles operating in open water environment with harsh sea state conditions. Having this operational flexibility makes the amphibious military vehicles strategically important for armies. However, making a military vehicle amphibious brings challenging problems especially in the design stage. Drag force prediction is obligatory for the thrust requirement determination. High drag forces mean high thrust requirements which may also affect the selected thrust system. Due to high thrust demand, the system may be bigger, and hence heavier, and this weight increase may affect the amphibious performance of the vehicle. Therefore, estimating the drag forces for defined amphibious operation conditions is very important. Also, optimizing geometrical form of the vehicle according to the amphibious capability of the vehicle becomes more important. In this study, the drag forces on an amphibious military vehicle at different operation velocities are predicted using Incompressible Computational Fluid Dynamics (ICFD) solver implemented in LS-DYNA®. In the simulations, the vehicle model is scaled according to the Froude number used in the verification tests. The test scale is limited by the pool dimensions. The drag tests are performed in Istanbul Technical University Naval Architecture and Design Faculty laboratories. Drag force predicted from the simulations for a predefined speed is compared with the one obtained from the pool tests. The accuracy of the drag force estimated from simulations is reviewed for different solution parameters of LS-DYNA® ICFD solver.

1 General Introduction

Drag force predictions are very important for amphibious military vehicles. Thrust requirements can be estimated by reliable scaled model tests of the vehicle. The scaled model tests are used for verifying the simulation models. However, making the design iteration with the tests can be very expensive. Due to this, optimization of the vehicle geometry can be done by CFD simulations and the final results can be verified with the pool tests using the final geometry.

The model scale used in this study is determined by Istanbul Technical University Naval Architecture and Design Faculty. The main constraint on the model scale is the pool width and depth of the drag test laboratory. The scale is determined in such a way that the depth and side effects are negligible on the drag forces of vehicle.

2 Simulation Model

The free surface flow model composed of two different materials, namely air and water. Since the levelset method is utilized for the solution of free surface flow, the air is modeled as vacuum, hence no density and viscosity are defined. The general view of the model is shown in figure below.

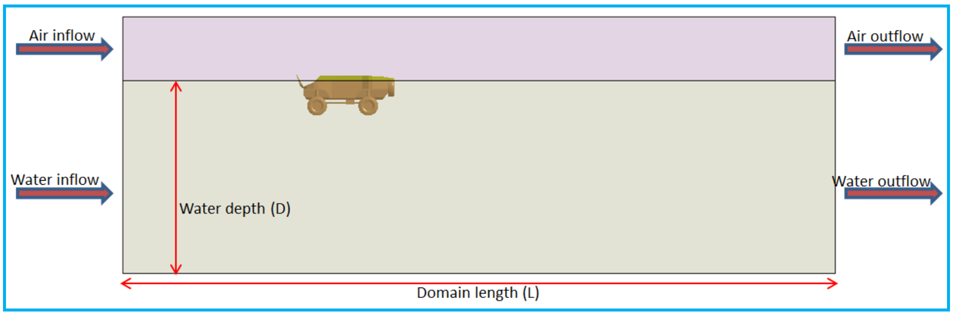

Fig.1: Side view of the simulation domain.

In the simulations, a constant speed inflow and outflow is assumed making the vehicle fixed in the domain. No fluid structure interaction is defined; hence a pure CFD problem is solved. No boundary layer is defined since the viscous forces are assumed to be negligible for s military vehicle geometry. Only pressure drag forces are examined. The turbulence effects are also ignored in this study and will be investigated later.

The inlet and outlet velocities are defined with **`*ICFD_BOUNDARY_PRESCRIBED_VEL`** keyword. The side walls presenting the air domain has a free slip boundary condition defined with the keyword **`*ICFD_BOUNDARY_FREESLIP`**. The surface of the vehicle has no-slip boundary condition and is defined with **`*ICFD_BOUNDARY_NONSLIP`** keyword. The drag forces are extracted from the vehicle surfaces using the keyword **`*ICFD_DATABASE_DRAG`**.

Different MGSF parameters of the **`*ICFD_CONTROL_MESH`** are tried in order to see the difference in the quality of solid tetrahedral mesh generated by LS-DYNA®. Triangular elements are used in the first model for vehicle surface meshing and domain meshing, as well. A sample view for the mesh of the domain (section view) is shown below.

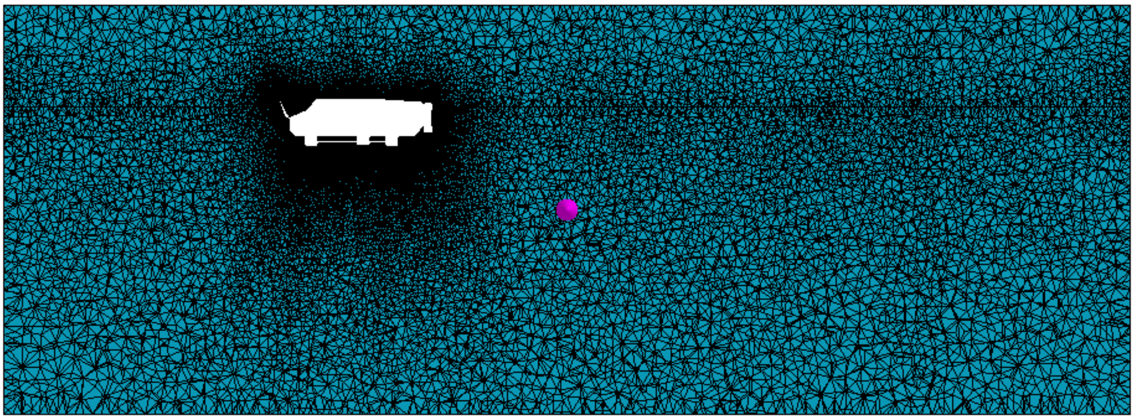

Fig.2: Fluid domain mesh (MGSF=1.41).

3 Results

The first simulation case is performed for 3 different values of MGSF in the **`*ICFD_CONTROL_MESH`** keyword, namely 1.41 (the default value), 1.323 and 1.118. The number of generated nodes and elements are shown in the table below.

	MGSF=1.41	**MGSF=1.323**	**MGSF=1.118**
# of Nodes	787927	915452	1394514
# of Tets	4604772	5424635	8492632

Table 1: Generated domain by LS-DYNA®.

The quality of the elements is important as well as the number. Since the vehicle geometry is very complicated, there are some small and distorted elements in the surface mesh. Hence, the solid mesh has also some distortions but when compared to the total number, the ratio is negligible. The simulations are performed for 15 seconds and several outputs of the simulations are compared that are listed below:

- Required Volume vs. Current Volume
- CFL Time Step vs. Run Time Step
- Instantaneous and Average Drag Force

The velocity of the inflow and outflow is 0.9262 m/s according to the model scale. The time step is taken as 0.005s for the first simulation cases. The required volume and current volume comparison is shown below.

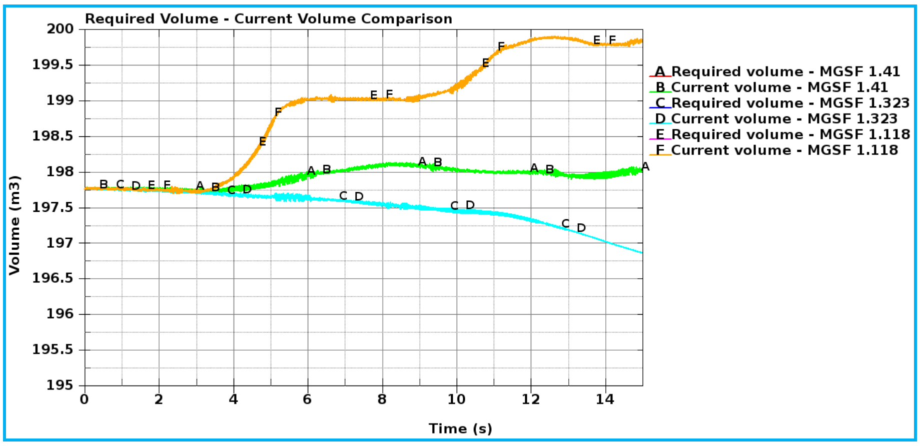

Fig.3: Required volume vs. Current volume.

The time step comparison is also performed for the simulations. The results show that the selected time step is well above the calculated CFL time step calculated by LS-DYNA® for the model which may not be desirable for a free surface problem. The results are shown in figure below.

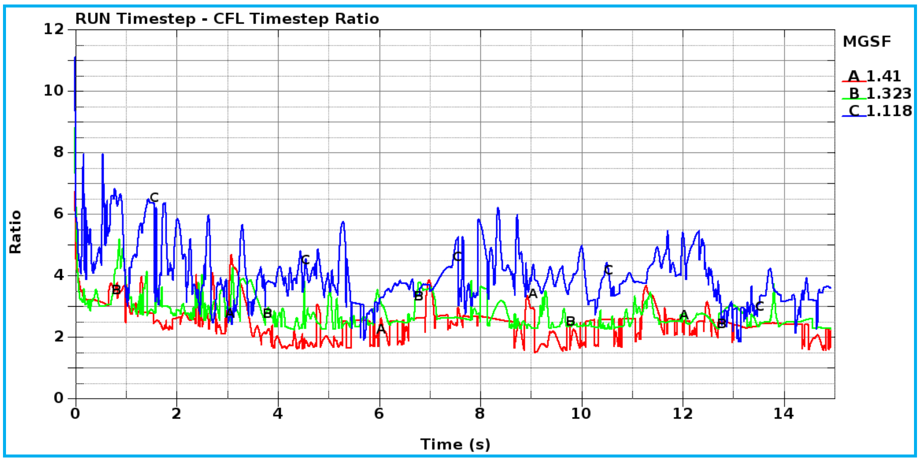

Fig.4: Run time step and CFL time step ratio.

When instantaneous and average drag forces are investigated, it is observed that the simulation time should be longer to have a more stable convergent behavior. However, the results still can be used for the comparison with test results since they also have an oscillating behavior.

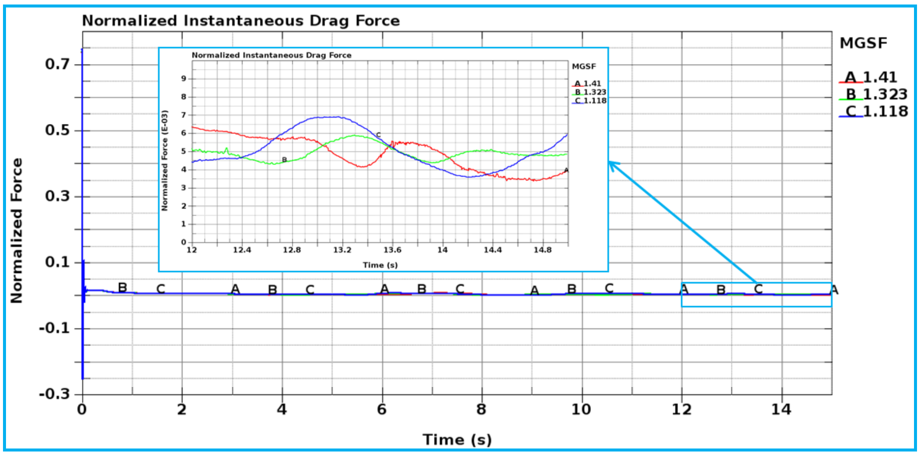

Fig.5: Instantaneous drag force comparison.

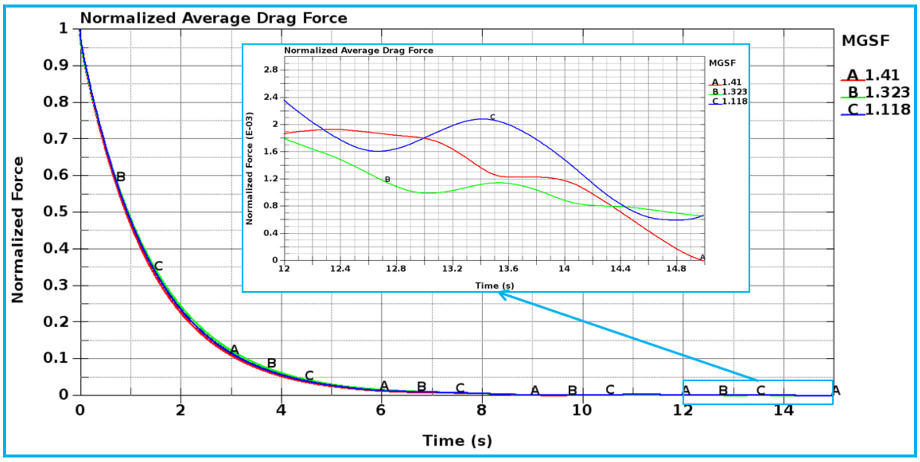

Fig.6: Average drag force comparison

The X velocity distribution on the free surface is shown in the figures below. The results are shown for the 14th second of the simulations. By looking at these results, one can conclude that the domain should be larger in the front and rear of the vehicle in order the wake to be diminished.

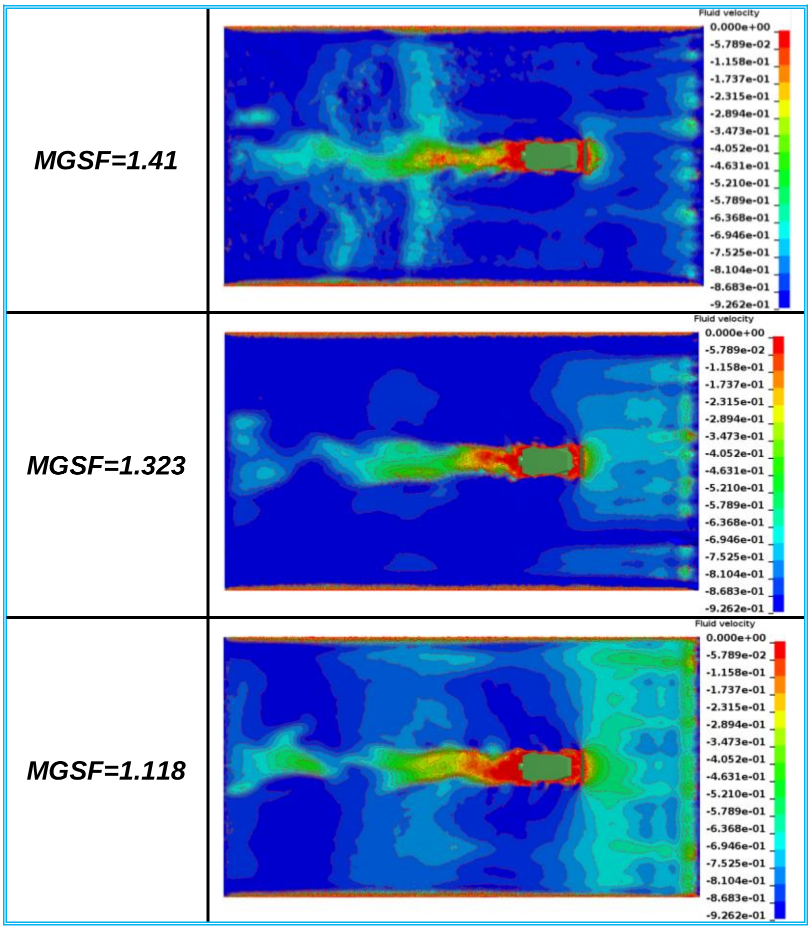

Fig.7: Free surface X velocity distribution.

The model with MGSF=1.41 is also solved with a variable time step size using an assumption of 1.05*CFL. The comparison of the behaviors is shown in figure below.

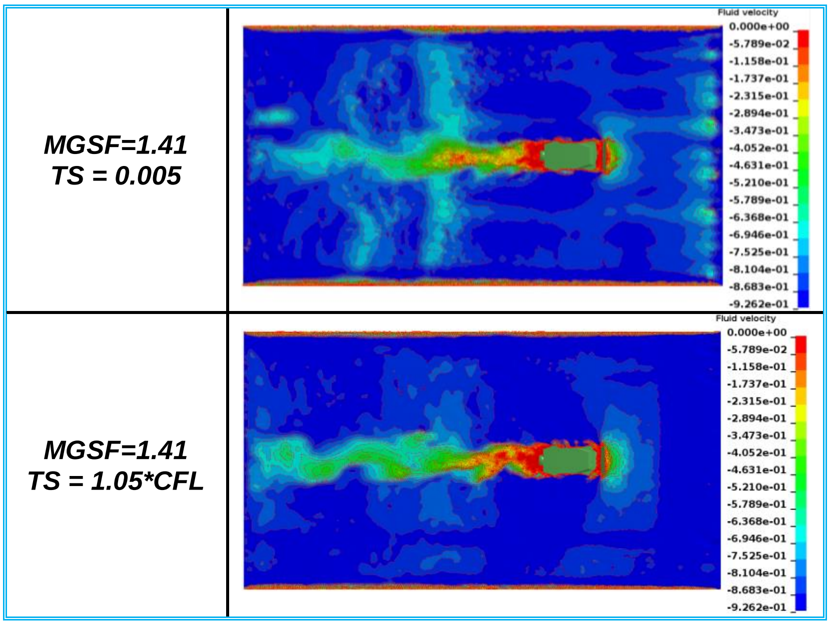

Fig.8: Fixed time step and variable time step comparison of X velocity.

The required volume vs. current volume and instantaneous drag force comparisons are also shown in figures below.

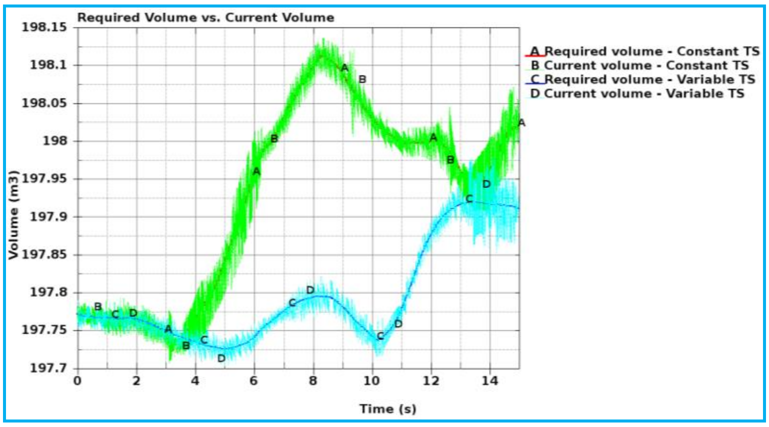

Fig.9: Required volume and current volume comparison.

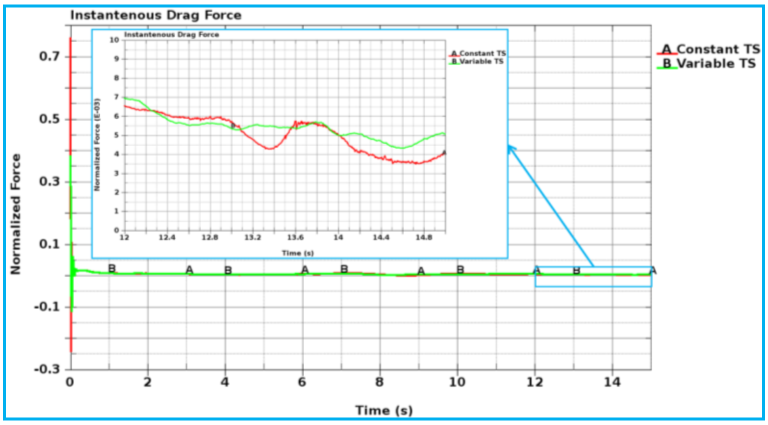

Fig.10: Instantaneous drag force comparison.

Another model is also prepared with refined mesh. The mesh is denser at the free surface region that is close to vehicle. The simulations are performed with default parameters of LS-DYNA®. The top view of the refined mesh is shown in figure below.

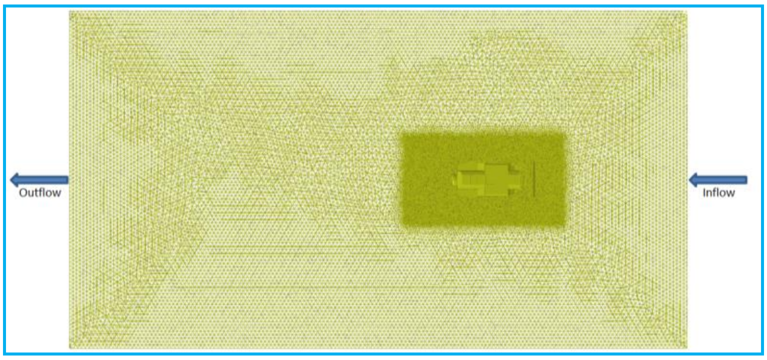

Fig.11: Refined mesh, free surface top view.

In this set of simulations using the new created refined mesh, various trials are performed to see the effect of ***ICFD_CONTROL_PARTITION** keyword. Same model is run with default parameters and also with the PTECH=4 parameter. When this parameter is used, the CFL time step is calculated as 0.013177 and therefore the simulation time step is taken as 0.015. In the case where ***ICFD_CONTROL_PARTITION** keyword is not used, the CFL time step is changing during the simulation but 0.005 time step is taken as the reference. The results are shown in figures below.

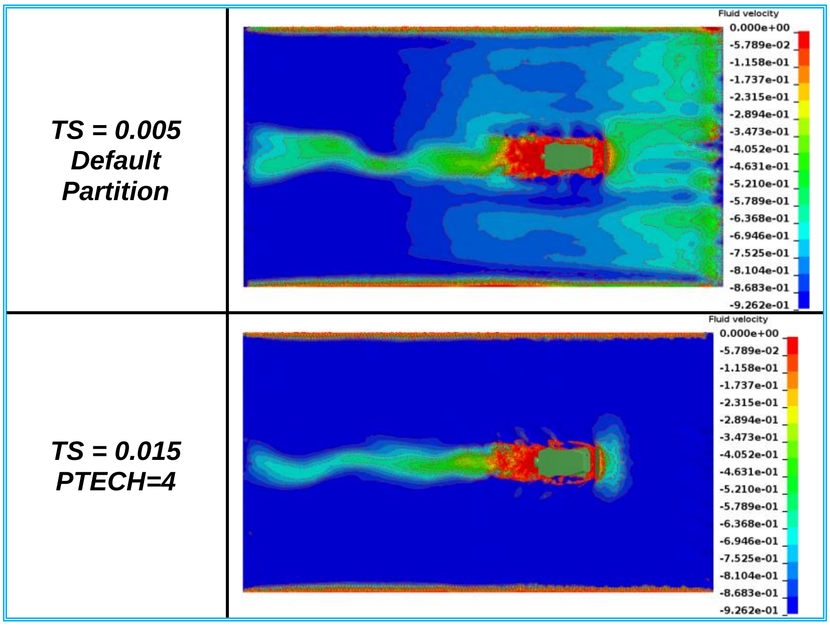

Fig.12: Free surface velocity distribution.

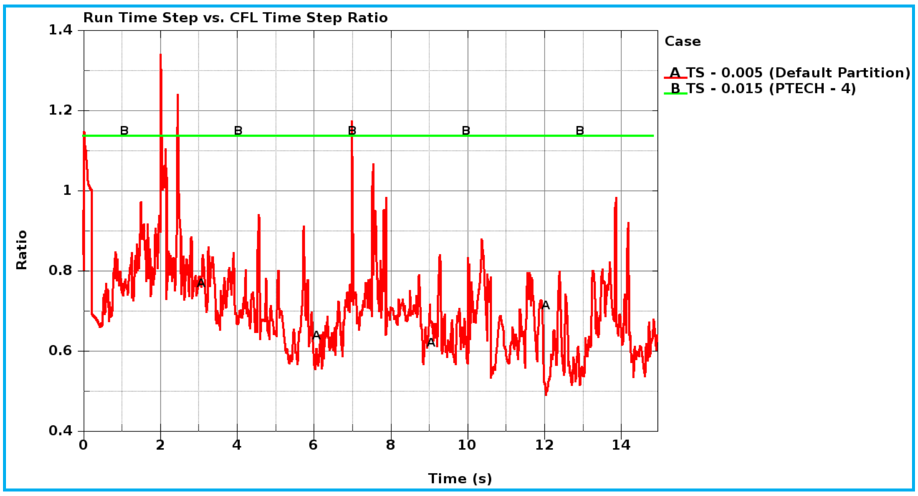

Fig.13: Run time step vs. CFL time step ratio.

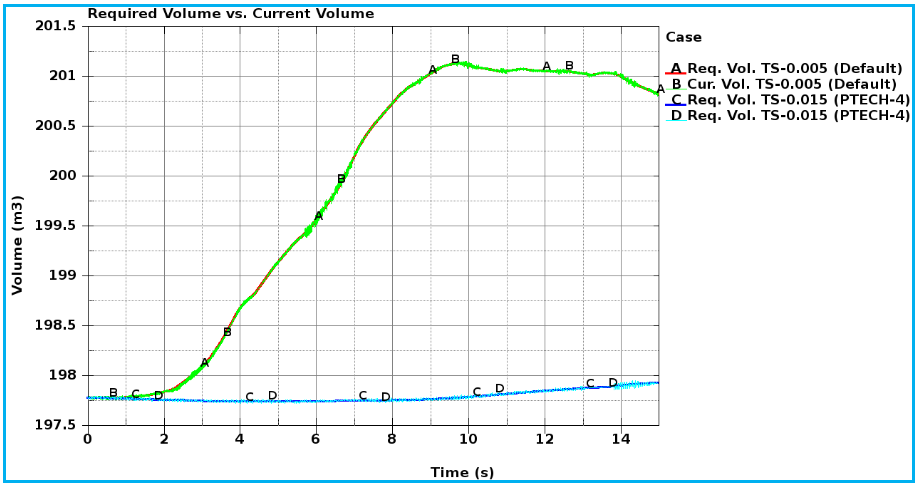

Fig.14: Required volume vs. current volume comparison.

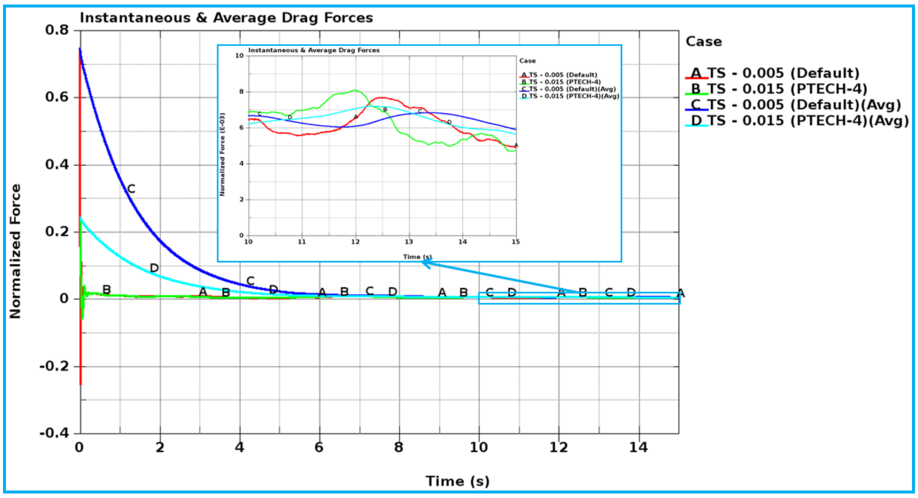

Fig.15: Instantaneous and average drag force comparison.

4 Test Verification

As it is stated before, the model scale tests are performed in Istanbul Technical University Naval Architecture and Design Faculty. When the test results are compared with the simulation results for the predefined velocity, the simulation results seem to be 10% higher at most. That is an acceptable accuracy being in the conservative side for the design.

5 Summary and Future Work

In this work, the hydrodynamic drag force prediction is made for 4x4 amphibious military vehicles with ICFD solver capability of LS-DYNA®. Various parameters and mesh strategies are tried and results are compared with each other.

The PTECH=4 parameter set in the **`*ICFD_CONTROL_PARTITITON`** keyword changed the behavior of the free surface. Moreover, the time step selection has also an effect of the simulation results especially in the velocity field of the free surface. Depending on the number of cores used in the simulations, sometimes the simulation gets stuck in the decomposition stage when the keyword **`*ICFD_CONTROL_PARTITION`** is used with PTECH=4.

In general, the average drag force converges to similar values in the simulations however the validity should also be checked with the behavior of free surface, inlet and outlet. A good and acceptable correlation is achieved with the model scale test results when oscillation of volume is lesser.

For the cases where the cruise speed of the vehicle is high, dynamic trim effects will become dominant in the drag forces and the simulations cannot be performed with pure CFD solution. At this time, fluid structure interaction should also be used in order to take the dynamic trimming of the vehicle into account. Further studies will be made for higher water cruise speeds.

As it is stated in the beginning, the turbulence effects are ignored but will become important with increasing cruise speed. Various turbulence models will be tried and the results will be verified with the model scale tests. The full scale vehicle model simulations will also be performed without simplifications the geometry and the differences with the scaled model will be determined.

6 Acknowledgements

I hereby thank to Prof. Dr. Metin Taylan and Prof. Dr. Ali Can Takinacı from İstanbul Technical University for their continuous support in amphibious mobility calculations and pool tests.

7 Literature:

[1] LS-DYNA® Keyword User's Manual Volume 3, R9, Livermore Software Technology Corporation, 2016.

[2] LS-DYNA® ICFD Theory Manual, Livermore Software Technology Corporation, 2014.