International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-8 Issue-5, June 2019

Parametric CFD Model Setup for Stability Investigation of SLV during Amphibious Mode

Prashant Rane, T Micha Premkumar

Abstract*: Submerged land vehicle (SLV), water operations are limited to amphibian movement. In this paper an existing SLV has been examined in terms of amphibious capabilities. The external flow CFD analysis is commonly used to study the behavior of objects submerged in continuous non static fluids. It aims to determine how efficiently a vehicle can move through the medium and how fluid flow and vehicle motion affect each other in the process. In the present scenario when the SLV is moving under water, the fluid is not confined between wall type conditions such as pipes or confined flow cases. The fluid is free to move around the object and interact only with its external "wet" layer. This means that it is the flow involving the body shape that must be analyzed. By evaluating the pressure and velocity distribution around the SLV, generated by the resistances of the fluid, we can determine the various forces acting on the SLV in different planes and analyze the flow-dynamic stability of the SLV when it is undergoing deep fording.

Index Terms*: Submerged Land Vehicle, Deep Fording, CFD, Stability, Dynamic Forces.

I. INTRODUCTION

The Submerged Land Vehicle should be able to ford across a water obstacle without getting bogged. The fording depth is usually limited by the height of the air intake of the engine, and to a lesser extent the driver's position. The typical fording depth for a SLV is 90 to 120 cm. (3-4 Feet.). However, with preparation some SLV are able to ford considerably deeper waters. The effect of the water current becomes a key performance factor when the water obstacle is wide. Therefore the hydrodynamic analysis of the Submerged Land Vehicle to meet the mission requirement becomes an important design feature. Several problems involving the flow of fluid around submerged objects are encountered in the various engineering fields. Such problems may have either a fluid flowing around a stationary submerged object or an object moving through a large mass of a stationary fluid or both the object or the fluid being in motion. Some of the examples which may be quoted are the motion of very small objects such as fine and particles in air or water, very large objects such as airplanes, submarines, automobiles, ships etc., moving through air or water and the structures such as buildings, bridges etc., which are submerged in air or water. Similar is the case of Submerged Land Vehicle carrying out deep fording through a water obstacle. In the analysis and design of such objects the knowledge of the forces exerted on them by the fluid is of significant importance [1].

Rane and Premkumar [2] have proposed a methodology for the analysis of forces and stability criteria evaluation of SLV's during deep fording operations. Figure 1 presents their methodology for dynamic stability evaluation. The sample evaluation was carried out on a common SLV. Vehicle propelling thrust was calculated using the engine torque and power transmission box efficiency obtained from SLV design. Rolling resistance, gradient resistance and damage resistance were added to calculated an estimated pull with certain factor of safety. In the dynamic stability methodology, these forces were compared with hydrodynamic forces such as drag, lift, buoyancy and lateral body force obtained from classical fluid mechanics correlations. It was observed that with use of correlations and the SLV's basic geometry data, a dynamic stability check can be established. However, the fluid forces are strongly dependent on actual SLV body design, local flow conditions and vehicle relative speed. Hence Rane and Premkumar have proposed to use a more accurate evaluation of fluid dynamic forces acting on the SLV using 3D CFD models and introduce these model results into the dynamic stability programme (Figure 1) to get a design fording operation limit or estimate an impact of change in the SLV body.

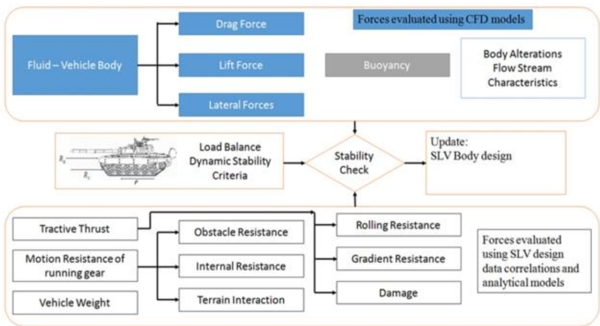

Fig.1 Force analysis for dynamic stability of SLV during deep fording operation

II. METHODOLOGY

Figure 2 shows the Methodology applied for the analysis of external flow around the SLV body. The main stages of the whole process have been listed here. These consist of creating a base CAD model first. The CAD model is real scale geometry but a representative configuration and not all specifications are necessary.

Manuscript published on 30 June 2019.
* Correspondence Author (s)
Prashant Rane, Centre for Defence Technology Studies, Hindustan Institute of Technology & Science, Chennai, India.
T Micha Premkumar, Centre for Defence Technology Studies, Hindustan Institute of Technology & Science, Chennai, India.

Once the CAD of the physical system is prepared it needs to be transferred to an analysis system. ANSYS software, academic version was used in the reported study. The ANSYS Geometry module is used to import the CAD model and is used to convert the physical SLV representation into an under-water movement simulation or a virtual test system. ANSYS Meshing has been used for creating the computational mesh and the solver used for flow calculations is CFX. The final stage of the analysis is post-processing where all variables of interest such as pressure and velocity field are visualized and studied. All quantitative data important for evaluating the dynamic stability of the SLV are also evaluated in the post-processor.

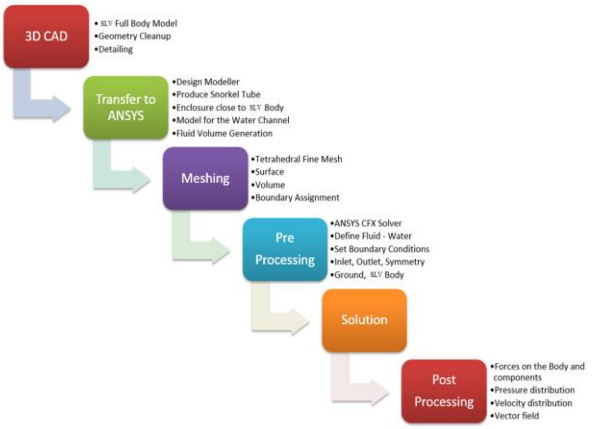

Fig.2 Methodology of the analysis

2.1 3D CAD Model

3D CAD software Solid works 2014 version was used to create the geometry of standard SLV body and the main components forming the exterior of the body were assembled together. Some details on the surface of the SLV body have been retained as these might create boundary flow separation effects. Figure 3 shows the resultant CAD model of the vehicle and Figure 4 shows the SLV in front, top and side views.

Fig.3 3D CAD model of the SLV used in the analysis

2.2 Parametric Model of the CFD Analysis

In order to get complete data on the forces acting on the SLV body under different vehicle speed or under flow and no flow conditions or under various orientation positions of the SLV relative to the flow, it is necessary to place the boundaries and also the SLV body at right positions in the virtual channel geometry.

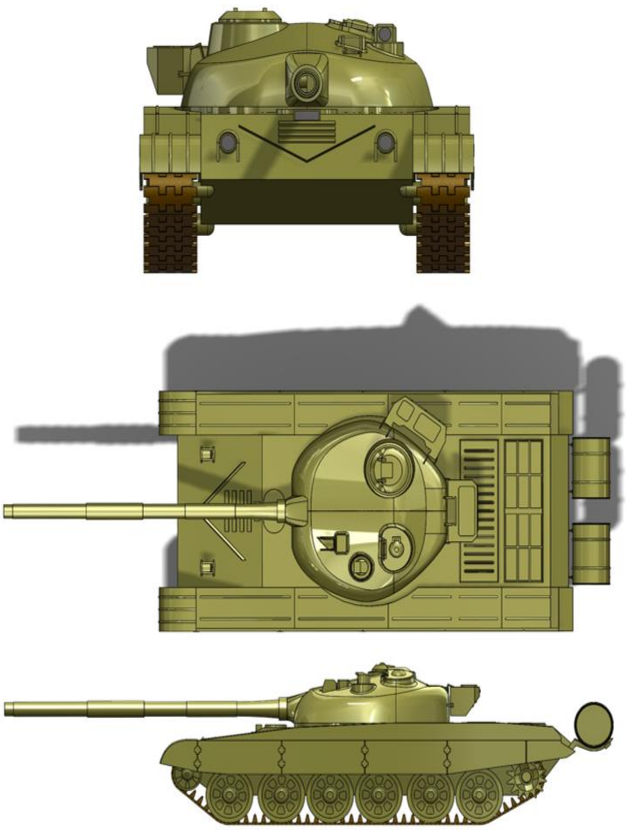

Fig.4 3D CAD model of SLV, Front, and Top and Side views

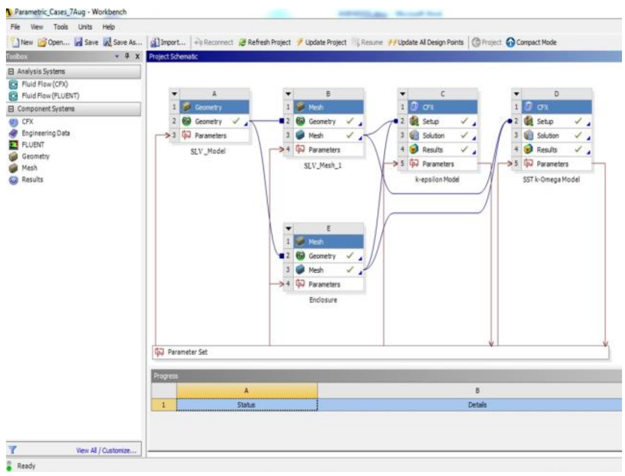

Fig.5 Parametric setup of the analysis system – Layout

To automate the process of setup, a parametric system was built in ANSYS workbench. The angle of alignment of the SLV body with respect to the channel, the velocity of flow and the velocity of SLV were defined as input parameters. In addition, the turbulence model of k-epsilon and SST k-Omega were setup for each of the conditions analysed. Figure 5 shows the layout of the parametric system and Figure 6 shows the table of parameters. Quantities data for forces was also set as output parameter from the system. For each set of parameters, a design point was created for calculation by CFX solver.

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication (BEIESP)

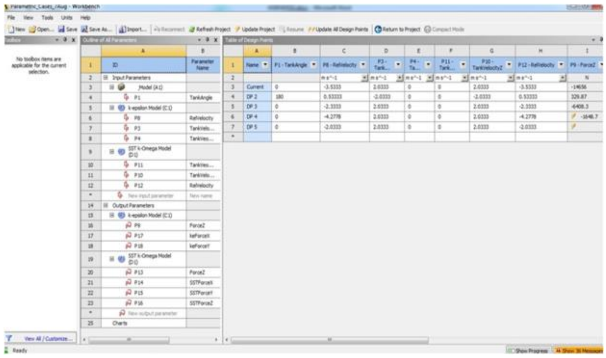

Fig.6 Parametric setup of the analysis system –Parameter Table

Table 1 presents the various cases that were analyzed in the study. Two main turbulence model used commonly in industrial external fluid dynamics flow computations are the k-epsilon and SST k-Omega turbulence models. Both of these models were considered at the set operating conditions. At a flow velocity of 1.5 m/sec three SLV alignment positions – Opposed Flow (0 degree), Cross Flow (90 degree) and Aligned Flow (180 degree) with flow were considered. These positions have an effect on the relative velocity between the SLV and flow and also on pressure distribution. In opposed flow condition, three vehicle velocities were analyzed and also a condition of water obstruction was analyzed.

Table.1 CFD cases analysed for various SLV Body Alignments

Case	Turbulence Model	SLV Velocity	SLV Alignment	Water Velocity	Relative Velocity
		m/s	Degree	m/s	m/s
1	k-epsilon	2.03333	0	1.5	3.53333
2		2.03333	90	1.5	1.5
3		2.03333	90	3.5	1.5
4		2.03333	90	8	1.5
5		2.03333	90	10	1.5
6		2.03333	180	1.5	-0.53333
7		0.83333	0	1.5	2.33333
8		2.7777	0	1.5	4.2777
9		2.03333	0	0	2.03333
1	SST k-Omega	2.03333	0	1.5	3.53333
2		2.03333	90	1.5	1.5
3		2.03333	180	1.5	-0.53333
4		0.83333	0	1.5	2.33333
5		2.7777	0	1.5	4.2777
6		2.03333	0	0	2.03333

2.3 Flow Geometry Generation

The geometry constructed in CAD system is a representation of the vehicle body and for the analysis of flow around the body a geometry representing the fluid domain needs to be generated. This is achieved by first importing the whole SLV assembly into ANSYS Geometry tool and performing surface clean-up operations to avoid any holes and unwanted features to be present on the model. The whole body is than enclosed in an exterior volume representing the fluid part and by performing Boolean operations, the SLV body is removed from the exterior volume. The resultant is that the exterior volume is left with a pocket that is bounded by the surface of the SLV's body. This surface can be treated as the SLV body in flow simulations.

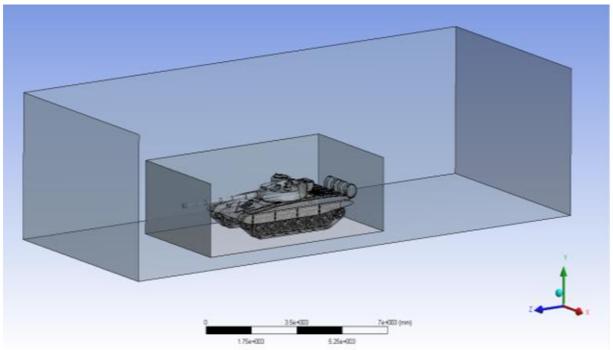

Fig.7 SLV body positioned in the enclosure

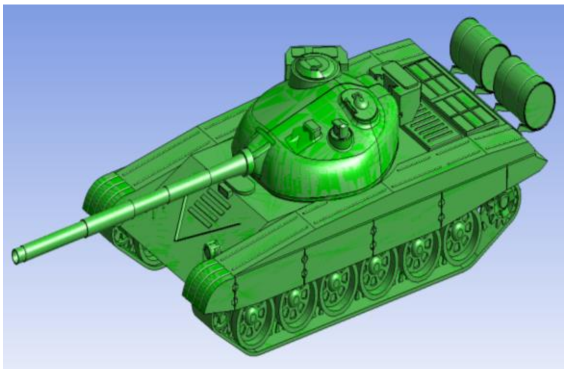

Fig.8 Wetting surfaces for the SLV body in bounding volume of the flow

In order to capture flow gradients accurately around the vehicle body, a particularly fine mesh is desired surrounding the vehicle. But if this fine mesh is used throughout the flow domain than a considerable large mesh count is resulted and computations are not possible with available hardware resources. Hence the mesh is given a growth rate of about 2 from the SLV body to the external bounding flow volume.

On the other hand in order to facilitate creation of a simple and parametric rotation of the SLV body with respect to the flow, the SLV body is first enclosed in a relatively smaller rectangular domain and this domain with the SLV body is than positioned inside a large full flow domain as shown in Figure 7 and Figure 8. The boundary condition for flow calculation are required to be specified only on the external faces of this large flow domain whereas the inner rectangular volume is directly connected to each other by fluid to fluid interfaces.

2.4 Computational Mesh generation

Figure 9shows the fluid geometry being brought into the meshing tool. A complete tetrahedral mesh is specified in the whole domain except for the exterior boundaries where an inflation layer with 5 layers has been applied in order to capture near wall effects. Figure 10 shows the tetrahedral mesh in the interior volume of the bounding block.

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication (BEIESP)

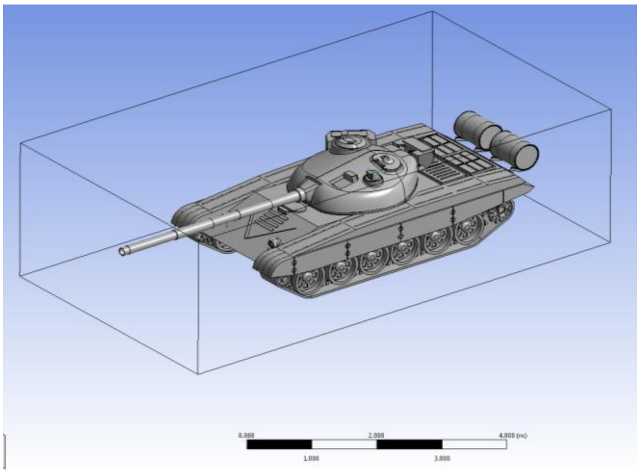

Fig.9 The SLV body with enclosure in the meshing tool

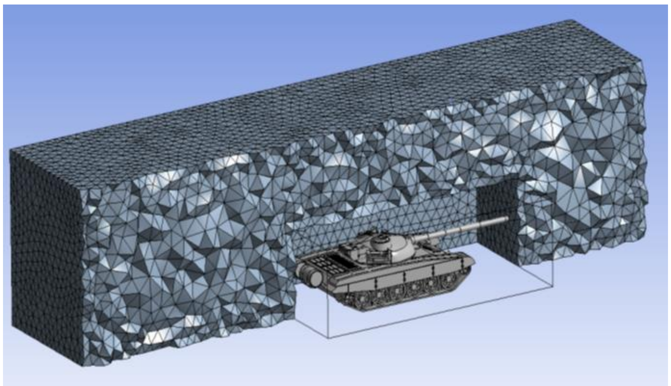

Fig.10 Tetrahedral mesh in the interior of the bounding volume

Figure 11 and 12 show the mesh on the surface of the SLV and the ground boundary in close views.

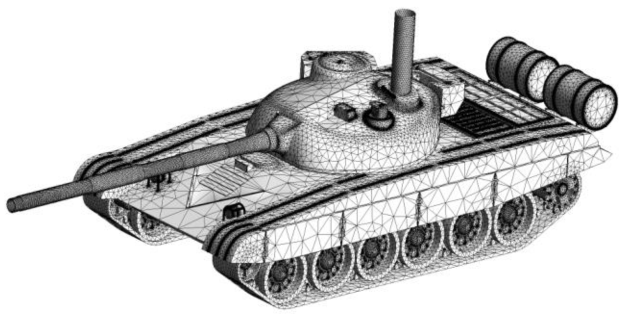

Fig.11 Mesh on the SLV body

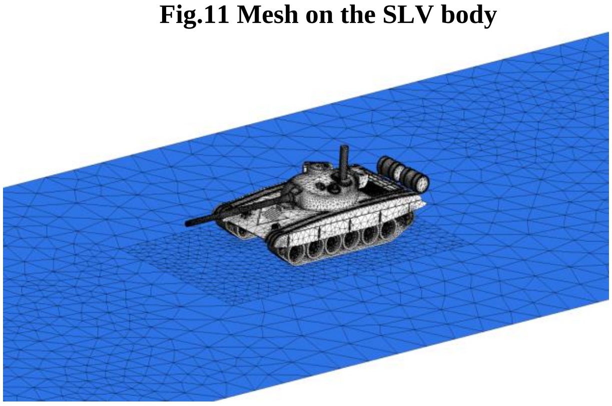

Fig.12 Mesh on the ground surface and on the SLV body

III. CFD MODEL DESCRIPTION

Figure 13 shows the CFD model in one of the design conditions. The SLV is in opposed flow position. The main boundaries of the model have been highlighted in the figure. The length of channel from rear end of the SLV to the outlet of the domain is considerably long in order capture the trailing eddies and back pressure conditions accurately.

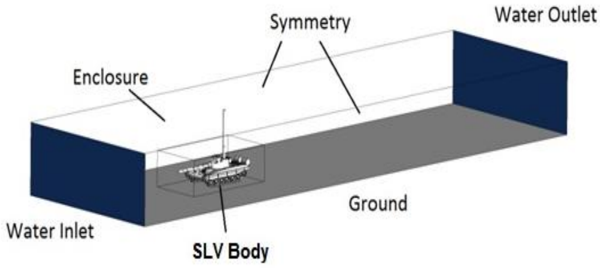

Fig.13 CFD model in the pre-processor with boundaries

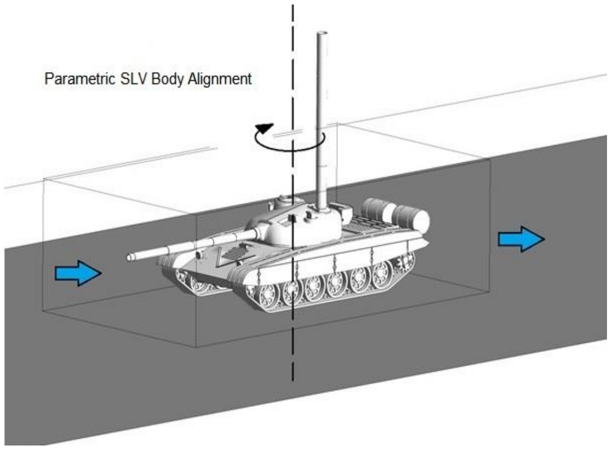

Fig.14 Parametric positioning of the SLV body in the flow geometry and axis of orientation

Figure 14 shows the SLV body enclosed in the rectangular domain that is parametrically positioned relative to the main flow direction. The axis of rotation has been highlighted. Figure 15 highlights the Water Inlet boundary. It is defined as an Inlet condition with Cartesian velocity components specified. The x and y components were set as zero while the z component was defined as a parameter which changed magnitude and direction according to the design point under consideration.

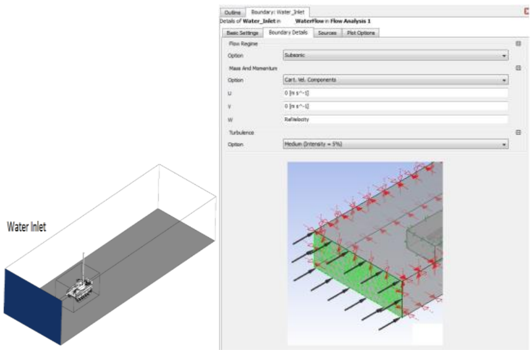

Fig.15 Inlet boundary condition at upstream of the channel

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication (BEIESP)

Figure 16 highlights the Water Outlet boundary condition. This is defined as an opening type of boundary which allows the flow to enter and leave the domain. A static pressure of 0 bar has been specified and hydrostatic pressure gradient has been ignored as its effect on drag force is negligible.

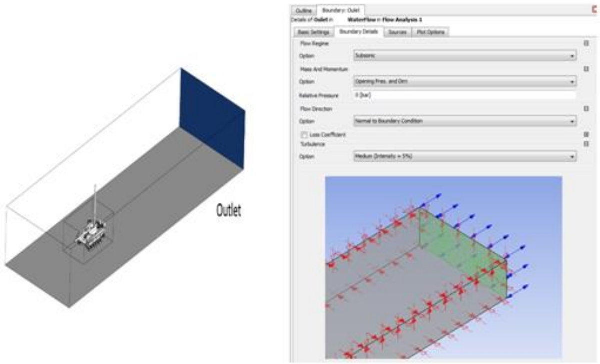

Fig.16 Outlet boundary condition at downstream end of channel

Figure 17 shows the boundaries on the left and right extremes of the domain. They have been set as Symmetry in order for the flow to be aligned with the channel length.

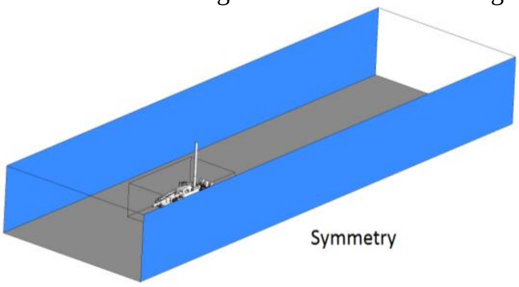

Fig.17 Symmetry boundary condition on the left and right side of the SLV body

3.1 Solver Settings and Convergence

CFX solver was set with a High resolution advection scheme for continuity and momentum equations. For turbulence equations a First Order discretisation scheme has been selected as these is the default setting of the solver. The solver was set in steady state mode with maximum iterations set as 100. The convergence criterion was set as $1x10^{-4}$ for r.m.s residuals.

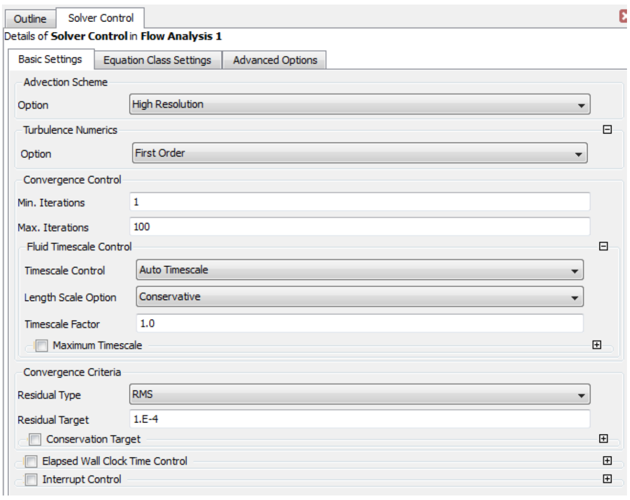

Fig.18a CFD solver settings

Figure 18a shows the main solver settings. Figure 18b presents an example of the residual levels reached by each of the equations during calculations. Similarly, Figure 18c represents the convergence of the continuity, u, v and w momentum equations. They show that the target set for r.m.s residuals has been achieved by all the equations. Figure 18d represents the convergence of turbulence equations. Again the target set for r.m.s residuals has been reached by both turbulence quantities.

```
======================================================================
OUTER LOOP ITERATION =    98                    CPU SECONDS = 1.996E+04
----------------------------------------------------------------------
|       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
+----------------------+------+---------+---------+------------------+
| U-Mom                | 1.02 | 8.3E-05 | 2.5E-02 |       1.2E-03  OK|
| V-Mom                | 0.98 | 8.1E-05 | 2.0E-02 |       5.1E-03  OK|
| W-Mom                | 1.02 | 1.1E-04 | 3.6E-02 |       9.2E-03  OK|
| P-Mass               | 0.97 | 8.5E-06 | 9.1E-04 |  8.9  6.2E-02  OK|
+----------------------+------+---------+---------+------------------+
| K-TurbKE             | 0.98 | 4.8E-05 | 7.1E-03 |  5.7  2.1E-02  OK|
| O-TurbFreq           | 1.01 | 3.8E-06 | 2.3E-04 | 12.6  1.6E-03  OK|
+----------------------+------+---------+---------+------------------+
```

Fig.18b Solver report of residual level for each equation

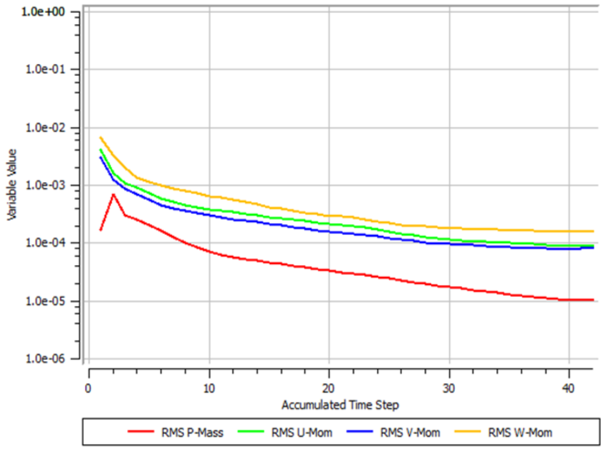

Fig.18c Plot showing the convergence of continuity, u, v and w momentum equations

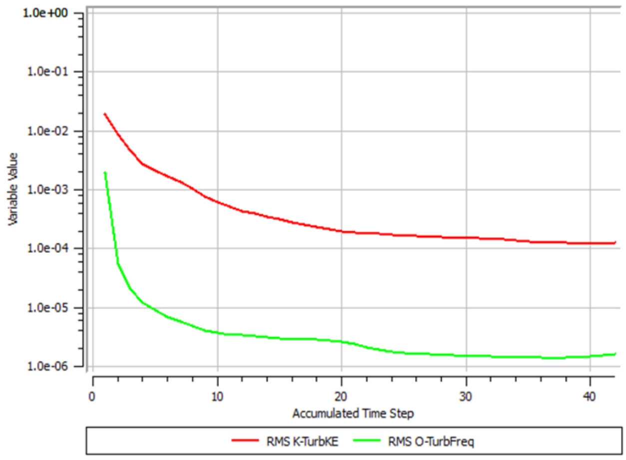

Fig.18d Plot showing the convergence of k-Omega equations

Initially the full analysis work flow from geometry, meshing, solver setup and solution is tested on the default parameter values. This helps in correcting any issues that may arise due to mesh sizing and geometry cleanup commands used in the process.

*Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication (BEIESP)*

Also little iteration to improve the quality of the computational mesh are required to be performed in order to use it for a automated design study. Once this is established and from the solver monitor plots it is found that the convergence is reproducible and solution is stable, the setup is ready for a parametric evaluation. Since turbulence models play an important role in the flow solution, in this study two commonly used turbulence of k-epsilon and SST k-Omega were set as one of the CFD model parameters in addition to the geometry and flow boundary conditions. From the solution, the post processing variables are then set as output parameters. Essentially these are the drag, lift and lateral forces acting on the SLV body as defined by the Cartesian coordinate system. With SLV alignment with the flow direction being an input parameter, the drag and lateral force output parameters can be defined using the same direction vectors. Thus a fully parametric CFD model for the dynamic stability analysis is setup and provides inputs to the procedure presented in Figure 1.

IV. CONCLUSION

In this research work an existing SLV design has been examined in terms of amphibious capabilities. The analysis are being planned with the consideration of fixed underwater ground conditions; however there can be situations the underwater ground conditions are loose making traction force availability low. Under these conditions, the SLV can become dynamically unstable. In future it is necessary to have a standardized procedure to evaluate on field underwater ground conditions. A set of flow conditions can be tabulated for expected drag, lift and lateral forces and should be made available to the engineers. These data will help in taking on ground decisions. CFD modeling was performed using best practices suggested for such external body flow analysis. From the forces obtained using CFD models and the traction forces obtained using empirical correlations, the dynamic stability of the SLV under various fording conditions can be analyzed. Detailed results and discussions will be presented in next article.

REFERENCES

1. Helvacioglu, S., Helvacioglu, I.H. and Tuncer, B. “Improving the river crossing capability of an amphibious vehicle”. In: *Ocean Engineering*, Volume 38, Issues 17–18, 2201-2207, 2011.
2. Rane, P., Premkumar, T. M., Analysis of Forces and Stability in Submerged Land Vehicle During Deep Fording Operations. International Journal of Recent Technology and Engineering, ISSN: 2277-3878, Volume-8, Issue-1, 2019.
3. Headquarters, U.S. Army Material Command “Engineering Design Hand book (Wheeled Amphibians)” AMCP, 706-350, 1971.
4. M. G. Bekker. “Mechanics of Off-the-Road Locomotion”SAGE Journals, Volume: 16 issue: 1, page(s): 25-44, 1962.
5. More, R.R., Adhav, P., Senthilkumar, K., Trikande, M.W. “Stability and Drag Analysis of Wheeled Amphibious Vehicle using CFD and Model Testing Techniques”. In: *Applied Mechanics and Materials*, Volume 592-594, 1210-1219, 2014.
6. Modi, P.N. and Seth, S.M. “Hydraulics and Fluid Mechanics”. Twelfth edition, 1998.
7. Wong, J.Y. “Performance of Off-Road Vehicles”. In: *Terramechanics and Off Road Vehicle Engineering*, Second Edition, 129-154, 2010.