Mesh independence of a transient multiphase fluid-solid interaction

I Chirica¹, C M Anghelută², S D Perijoc³, A I Hobjilă⁴ and M Frătiţa⁵

¹, ², ³, ⁴, ⁵ University Dunarea de Jos of Galati, 47, Domneasca Str., Galati, Romania

1, ², ³, ⁴ Ship Design Group, 51, Dogariei Str., Galati, Romania

Corresponding e-mail: marian.angheluta@ugal.ro, cristian.angheluta@shipdesigngroup.eu

Abstract. To prepare for a transient analysis of a damaged ship, a mesh independence test was carried out on an open channel free surface simulation with fluid-solid interaction. The numerical methods applied, the mesh sizing and method, as well as several other parameters which might influence the results were adjusted to obtain comparisons and establish the proper solution for the upcoming simulation. Compared data includes the drag force, dynamic and total pressure applied to the solid, as well as the maximum free surface elevation at the solid domain.

1. The paradigm of modern meshes for fluid flow simulations

A quality mesh is a first step in ensuring that the computational fluid dynamics solver produces a proper solution, while keeping the processing resources as low as possible. Nowadays programming advancements have reached a point where solvers are robust enough that even a “bad mesh” can lead to an accurate result, however, that is undesirable in the field of ship design where the industry strives to reach over-night solutions, no longer limited to only potential flows as the case was until recently.

Take for example Eric Thornhill’s “CFD Simulations of a Ferry in Head Seas” paper for the Defence Research and Development department of Canada. The calculation involved a mesh containing almost eighteen million tetrahedra elements, with a minimum edge size of 5 mm. The 60s simulation took 7 days on a 40 cores workstation. That is a far stretch from the overnight solution one would desire. However, when defining the quality of a mesh, the first part is simple: if a mesh does not converge and doesn’t lead to a result, it is a bad mesh. But following that, things get more complicated, as there are good meshes and even better meshes. Perhaps the most important aspect is the accuracy at which the CFD solution reflects the real-life phenomena. This ties the mesh quality to the mathematics of the solver and the physical modeling of the flow.

NASA’s Stephen Alter underlines the importance of a priori indicators of mesh quality, which, already leading into the scope of this paper, are available in ANSYS’s Fluent module. Orthogonality of the elements is an all-round good indicator of a proper discretization, however, not sufficient to properly evaluate the quality of a mesh. Dannenhoffer however proposes different factors when assessing the mesh: the first one is the validity of it, e.g. are there any negative cells or faces that intersect in the mesh? As far as the knowledge of the authors of this paper goes, negative volume cells will terminate a solver’s mathematical process. The second factor is the geometrical validity, or rather that the mesh accurately
reflects the topology of the model, while the third factor is whether there are shared topological features between the geometry and the mesh, as in cell faces, edges of cell faces etc.

Code developers in the flow solvers field also have their own understanding of mesh quality. In particular, cell skewness and cell size variation seem to be of particular interest. Konstantine Kourbatski of ANSYS pointed out that beyond orthogonality and skewness, the aspect ratios of cells is also important, which we can also link to structural FEA methods.

But on the matter on hand, although many mesh metrics can be established, the fact is that many of them don’t have a direct connection to the mathematical model. It then falls back to the actual simulation characteristics, problem at hand and geometry, as well as obtaining specific domain expertise.

2. Problem description and approach
The study presented in this paper lays the foundation for an ampler research on the global motions and local damaged area effects of an incapacitated vessel. The ultimate goal is to establish a tried and tested method for close-to overnight solutions in appreciating the behavior of a ship in waves, in both intact to begin with, and then damaged state. Current evaluation is based on class empirical and probability-based methods, which are, to some extent, limited when it comes to specific applications.

The study is based on a simulation of an open channel flow, with two fluid phases, water and air, the interface of which is handled using the Eulerian method Volume of fluid. The conservation equations for describing the motion of the fluid are solved separately. In this particular case, the fluid domain is disrupted by a stationary solid cylinder. The problem setup is described below in figure 1 and table 1.

| Model     | Multiphase |
|-----------|------------|
| Viscosity | Laminar    |
| Inlet velocity | 0.01 m/s |
| Initial free surface | 0.25 m |
| Wave height | 0.1 m |
| Wave length | 0.5 m |

![Figure 1. Case geometry](image)

![Figure 2: Water phase volume fraction visual representation](image)
### Table 2: Mesh characteristics for each case studied

|                | DP1       | DP2       | DP3       | DP4       | DP5       | DP6       |
|----------------|-----------|-----------|-----------|-----------|-----------|-----------|
| Element size   | 10 mm     | 10 mm     | 15 mm     | 7.5 mm    | 5.0 mm    | 15 mm     |
| Inflation      | None      | Program controlled | Program controlled | Program controlled | Program controlled | None |
| Assembly meshing| None      | None      | None      | None      | None      | CutCell   |
| Number of nodes| 264472    | 82887     | 31357     | 160938    | 402746    | 177298    |
| Number of elements | 251022 | 397199     | 143971    | 791291    | 2021584   | 157904    |
| Number of tetrahedra | 0      | 378989     | 136001    | 759381    | 1949784   | 0         |
| Number of wedges | 204      | 18210      | 7970      | 31910     | 71800     | 3306      |
| Number of hexahedra | 250818 | 0          | 0         | 0         | 0         | 154208    |
| Minimum aspect ratio | 1.02     | 1.16      | 1.17      | 1.16      | 1.16      | 1.01      |
| Maximum aspect ratio | 2.19     | 12.56     | 12.08     | 17.77     | 14.26     | 3.78      |
| Average aspect ratio | 1.03     | 2.05      | 2.12      | 2.01      | 1.99      | 1.07      |
| Minimum skewness | 0.00      | 0.00      | 0.00      | 0.00      | 0.00      | 0.00      |
| Maximum skewness | 0.46      | 0.83      | 0.84      | 0.85      | 0.85      | 0.58      |
| Average skewness | 0.01      | 0.22      | 0.23      | 0.22      | 0.22      | 0.02      |
| Minimum orthogonal quality | 0.75  | 0.17      | 0.16      | 0.15      | 0.15      | 0.66      |
| Maximum orthogonal quality | 1.00  | 1.00      | 0.99      | 1.00      | 1.00      | 1.00      |
| Average orthogonal quality | 1.00  | 0.77      | 0.76      | 0.78      | 0.78      | 0.97      |

3. Preliminary results and observations

The results for each design point are presented as variation of the corresponding measured value related to flow time.

![Figure 3: Dynamic pressure variation](image)

We immediately notice the significantly increased dynamic pressure values in DP5. This is probably due to the fact that the increased number of cells leads to a more accurate representation of the velocity field, and it also seems to transfer towards the total pressure output on the cylinder wall.
Figure 4: Total pressure variation

The X-direction force is accurately represented in all analysed scenarios.

Figure 5: Total force in X direction

Figure 6: Maximum free surface elevation (calm water at 0.25 m)
The maximum elevation graph however seems to be lacking in data consistency. DP3 scenario has the least qualitative mesh and seems to have a large deviation from the other analyzed scenarios. As far as these analyzed scenarios go, the most accurate mesh seems to be comprised of a solution involving hexahedra elements to reduce the overall number of mesh elements. The CutCell assembly meshing method generated a high-quality mesh, but further studies are needed to evaluate on whether or not it is suitable for complex topologies with more than two phases and interfaces. Safest route is to force a hexahedra method on the mesher algorithm, with inflation on the areas of interest.

Acknowledgement
The paper has been supported by the Doctoral School of Mechanical and Industrial Engineering of “Dunarea de Jos” University, Galati, Romania.

References
[1] Alter S 2004 A structured-grid quality measure for simulated hypersonic flows 42nd AIAA Aerospace Sciences Meeting and Exhibit AIAA 42 2004-612
[2] Thornburg H 2012 Overview of the PETTT workshop on mesh quality/resolution, practice, current research, and future directions 50th AIAA Aerospace Sciences Meeting including the New Horizons forum and aerospace exposition AIAA 50 2012-0606
[3] Stimpson C J 2007 The verdict geometric quality library SANDIA SAND2007-1751
[4] Mavriplis D J 2009 Grid quality and resolution issues from the drag prediction workshop series Journal of aircraft 46 pp. 935-950
[5] Roache P J 1997 Quantification of uncertainty in computational fluid dynamics Annual review of fluid mechanics 29 pp. 123-160
[6] Knupp P M 2007 Remarks on mesh quality 45th AIAA Aerospace Sciences Meeting and Exhibit AIAA 45
[7] ANSYS 18.2.2 Documentation and Fluent Theory guide courtesy of https://www.sharcnet.ca and https://www.ansys.com
[8] Ali B, Somoig P and Ali L, Scientific Bulletin of Naval Academy, Vol. XXI 2018, pg. 531-534.
[9] Dobref V, Tarabuta O, Scientific Bulletin of Naval Academy, Vol. XI-2008, pg. 16-21