Review in Numerical Simulation in Naval Application

Cristian-Milică Niță¹, Ionuț-Cristian Scurtu², Sorin Ovidiu Cupsa³
¹ Maritime University of Constanta, 104 Mircea cel Batran, RO900663, Constanta, Romania ²
² Mircea cel Batran Naval Academy, 1 Fulgerului, RO900218, Constanta, Romania
³ Ceronav, Constanta, Romania

Abstract. Numerical Simulation (CFD/FEA is a powerful engineering investigation software for fluid flow motions. CFD has been widely used to simulate the flow pattern around ships and propellers. This paper discusses the fundamentals in developing naval CFD/FEA models for ship shaped model. The simulation model is provided to enable the design of efficient and low cost naval projects CFD. A review of the current CFD used in naval domain is provided. Some limitations of the ANSYS software are also discussed.

Keywords: computational fluid dynamics; CFD; naval; computer simulation

1. Introduction
ANSYS is a modeling package for general purpose numerical finite elements, solving a wide range of mechanical calculation issues. These issues include dynamic analysis, structural (both linear and nonlinear), heat and fluid transfer as well as electromagnetic issues. In general, a finite element solution can be divided into three steps.
Step 1. Preprocessing: -Definition of the problem.
- Major steps in preprocessing (i).
- Define key start points / start lines / start areas / start volumes, defining element type and material / geometry properties (ii).
- Place lines / areas / volumes as needed (iii).
The amount of detail required will depend on the dimensionality of the analysis, ie 1D, 2D, asymmetric and 3D.
Step 2. The solution. Assigning tasks, constraints and solving
It is necessary here to specify tasks (point or pressure), constraints (translational and rotational) and finally solve the set of equations.
Step 3. Post-processing: further processing and viewing results.
At this stage you would like to see (i) lists of nodal displacements, (ii) force element and moments, (iii) deformation curves, and (iv) stress contour diagrams or temperature maps.
There are two ways to use ANSYS for ship shaped model:
- The first is through the graphical user interface or workbench GUI. The Ansys method follows the popular Windows conventions.
- The second is through command files in C++ programming language. The command file approach has a stricter learning curve for many but has the advantage that the entire analysis can be described in a small text file, usually in less than 50 command lines.
It allows easy model changes and minimal file space requirements; the presented model of ship shape is below.

![Ship shape model subjected to momentum in numerical simulation](image)

**Fig. 1. Ship shape model subjected to momentum in numerical simulation**

Utility menu: Utility menu [A] contains functions that are available on the whole screen in the ANSYS session, such as file controls, selections, graphical commands and parameters.

(2) Input line: Input line [B] displays program messages and allows direct input of orders.

(3) Toolbar: Toolbar [C] contains buttons that frequently execute ANSYS command usage. Several buttons are available.

(4) Main Menu: The Main Menu [D] contains the primary ANSYS functions, organized by preprocessor, solution, general postprocessor and design optimizer. From this menu, the vast majority of modeling commands are issued.

(5) Graphical windows: The Graphic Window [E] is the place where a graphical choice is displayed and made. Here is the model in its various stages, the construction and the results from the analysis can be seen.

2. Building a naval model in numerical simulation

The ANSYS workbench platform is python based and has capabilities to analyze finite elements, from a static, linear and / or simple analysis to a dynamic, transitive, non-linear and / or complex analysis. Building a finite element model requires more time than any other part of the analysis. First, the name of a job task and an analysis title should be specified. The preprocessor is used to define element types, material properties, and geometric patterns. Except for magnetic field analyzes, any system of units can be used as long as it ensures that the units are compatible for all input data. Units can not be set directly from the GUI.

To establish units as an International Unit System (SI) in ANSYS: Material Library → Select Units.

The various phenomena treated in science and engineering are often described in terms of differential equations, formulated using the continuous mechanics models. The exact solutions for differential equations, however, are generally difficult to obtain. Solving differential equations under different conditions, such as limit or initial conditions, leads to the understanding of the phenomena and can predict the future of the phenomena (determinism). Numerical methods in various complex problems are adopted to obtain numerical approximate solutions of requested differential equations.

3. The Governing Equations

Choosing the next weighting function determines the subdomain method or the finite volume method.

\[ w_i(x) = \begin{cases} 1 & (x \in D) \\ 0 & (x \notin D) \end{cases} \]

We consider a limit value problem described by a differential dimensional equation:
\[
\begin{align*}
    \frac{d^2u}{dx^2} - u &= 0 \quad (0 \leq x \leq 1) \\
    BC: u(0) &= 0 \quad u(1) = 1
\end{align*}
\]

The linear operator \( L[\cdot] \) and the function \( f(x) \) in equation (2.6) are defined as follows:
\[
L[\cdot] = \frac{d^2(\cdot)}{dx^2} \quad f(x) \equiv u(x)
\]

For simplicity, use \( x \) as a test function \( \phi_i \);
\[
\tilde{u}(x) = \sum_{i=0}^{n+1} c_i x^i
\]

To meet the necessary limit conditions,
\[
c_0 = 0, \quad \sum_{i=1}^{n+1} c_i = 1
\]
results:
\[
\tilde{u}(x) = x + \sum_{i=1}^{n} A_i (x^{i+1} - x)
\]

If the second term on the right of equation (2.10) is chosen as the first solution for an approximation \( \tilde{u}_1(x) = x + A_1 (x^2 - x) \)

the error is
\[
R = \frac{d^2 \tilde{u}}{dx^2} - \tilde{u} = -A_1 x^2 + (A_1 - 1)x + 2A_1 \neq 0
\]
\[
\int_0^1 w_i R dx = \int_0^1 [1 - (-A_1 x^2 + (A_1 - 1)x + 2A_1)] dx = \frac{13}{6} A_1 - \frac{1}{2} = 0
\]

Therefore, the approximate order of first order is obtained as
\[
\tilde{u}_1(x) = x + \frac{3}{13} x(x - 1)
\]
who admits the exact solution
\[
u(x) = \frac{e^x - e^{-x}}{e - e^{-1}}
\]
as shown by the dotted and solid lines in Figure 2.

![Fig. 2. Comparison of results obtained by different types of discrete analysis.](image-url)
When the weighting function \( w_i \) in equation (2.4) is chosen equal to the test function \( \varphi_i \), it is called the Galerkin method,

\[
w_i(x) = \varphi_i(x) \quad (i = 1, 2, \ldots, n)
\]

and thus Equation (2.4) changes to:

\[
\int_D \varphi_i R dv = 0
\]

This method determines the coefficients \( a_i \) by directly using the equation (2.6) or by integration by parts.

4. The constitution of the CFD/FEA model . Numerical analysis with 3D-FEM for ship shaped model

In the fifth step modeling of loading cases is included and structural numerical analysis is performed with 3D-FEM models developed over the entire length of the ship. Thus, distortion and stress distributions are obtained in the analysis of the global and local strength of the ship's body.

Loads operating on the ship's body are of three types:
- gravitational loads, given by the weight of the structural elements of the ship and other components of the displacement, excluding the weight of the cargo;
- Tests in cargo, considered as local hydrostatic pressures on body structure in cargo holds (in the case of containers being considered as concentrated forces at the gripping points);
- Hydrostatic pressures from the quasi-static equivalent wave, in the following cases: \( hw = 0 \) (calm water) and \( hw \neq 0 \) (according to the static values prescribed by the naval registers), using an iterative algorithm for free shipboard balancing (floatability and longitudinal slicing), implemented with its own system of GEO macro-commands within the FEM program.

Fig. 3. Spirals design two different storage approaches of the design data.

The fact that design information is missing in a 3D format is one of the reasons why Finite Element Analysis (FEA) is not usually introduced in the early stages of design - although it behaves as a validation tool in subsequent design stages. FEA is often the first and also the only reason for creating a 3D model in the first phases of structural design. This means that the finite element model (FE) is probably the only place where design information is in 3D. In the naval industry, typically, geometry information and FE model ownership are too idealized to be fully used by other disciplines. Therefore, the FE model cannot be considered as the optimal archive for design information.

FEA is a very time-consuming task, normally performed as little as possible, and serves more as a tool for validating the design. Therefore, the first EAA is performed late in the design process (Figure 4).
The primary purpose of the FEA in early design phases is to ensure that the ship's overall responses are within tolerable limits and to ensure that the overall layout of ship structures is reasonable. As mentioned previously, FE models required for global response often need a long time to create FEAs for general use. In addition, local models can be created to support the design and validation of critical structural details as well as areas of high stress concentration.

5. Applications of numerical simulation
Due to the above-mentioned reasons, new and innovative designs are not pursued because they tend to be labor intensive (ie very expensive). This situation could be changed if the FEA was done in a much shorter time, therefore, making the business more attractive, even if the classification authorities do not impose it.

This would lead to better design if the FEA could be introduced earlier in the design process. The process would be even more effective if the FE model was extracted from the latest design information for each analysis.

Often, the FE model of the gear unit for the new analysis is based on the previous GE gear model and is updated according to the new design information. With a planar results approach this system would be the only reasonable way to work, but if 3D design information is available, a new opportunity opens.

Fig. 5. Location of FEA activities in the typical and ideal vessel design process. [10]

The full FEA process can be broken down into key activities and individually examined. These activities are introduced in Figure 6 and discussed in more detail below.

Fig. 6. The time budget for FEA process entities (relative to 2D design approach). [10]
6. Conclusions
The objective of this review is to discuss the current state of numerical modelling in naval domain. The constitution of the CFD/FEA model is done with Ansys. The CFD applications for naval projects are a low cost and efficient way to investigate propulsion ship dimensions and parameters in any real environmental condition. The CFD/FEA applications for the naval industry have some limitations on both numerical algorithms and computational power.

References

[1] Domnișoru, L.; Găvan, E.; Popovici, O., (2005), *Analiza structurilor navale prin metoda elementului finit*. București: Editura Didactică și Pedagogică, p. 11-88.

[2] Domnișoru, L., (2009) *Analiza structurilor navale prin metoda elementului finit. Aplicații numerice*. Galați: Editura Fundației Universitare „Dunărea de Jos”, p. 31-303. Disponibil online: https://www.scribd.com/doc/81811146/Analiza-Structurilor-Navale-Cu-FEMAP-2009, Accesat: 12.01.2017.
[3] Doig, R.; Bohm, M.; Stammer, J.; Hernandez, P.; Grösch, S.; Kohn, D.; Brånhult, J.; Bitterling, B., (2009), Integrating Structural Design and Assessment, 8st Int. Conf. Computer and IT Appl. Maritime Ind. (COMPIT), Budapest, pp.374-389.

[4] Holmberg, T.; Hunter, S.D., (2011), Increasing efficiency in the ship structural design process. In: Proceedings of the 10th International Conference on Computer and IT Applications in the Maritime Industries, COMPIT’11, 2011 May 2–4. Berlin, Germany, Hamburg: Technische Universitat Hamburg-Harburg. p. 536–550.

[5] Hughes, O.F.; Paik, J.K., (2010), Ship Structural Analysis and Design. New Jersey, USA: The Society of Naval Architects and Marine Engineers.

[6] Kurki, T., (2010), Utilization of Integrated Design and Mesh Generation in Ship Design Process, 9th Int. Conf. Computer and IT Appl. Maritime Ind. (COMPIT), Gubbio, pp.311-318.

[7] Oancea, L., (2015), Modelarea fluxului de containere printr-un terminal în analogie cu mecanica fluidelor. Teză de doctorat. Coordonator științific: prof. univ. Dr. Ing. Dumitru Dinu, Universitatea Maritimă Constanța, Facultatea de Electromecanică Navală, p. 40-45.

[8] Paik, J.K.; Thayamballi, A.K., (2003), Ultimate Limit State Design of Steel-Plated Structures. Chichester: John Wiley.

[9] Radeș, M., (2006), Analiza cu elemente finite. Traducere a cursului Finite Element Analysis, Facultatea de Inginerie în Limbi Străine, Filiera Engleză, Universitatea Politehnica București, 1992, pp. 9-47.

[10] *** https://maestromarine.com/wp-content/uploads/2015/09/51_holmberg.pdf

[11] Gyongyosi T, Corbescu B, Puiu D, Valea Ş, Panaitescu V 2017 Pipe ice plugging tests using a ~ 15% exhaust liquid nitrogen vapor restriction for the freezing device mounted on a 200 mm horizontal pipe, in a stationary flow regime, The 10th Annual International Conference on Sustainable Development through Nuclear Research and Education, Nuclear 2017. Nuclear Technology and Materials. Pitești, May 24-26, 2017, Proceedings of Nuclear 2017, part 1/3, pag. 291-298

[12] Gyongyosi T, Valea Ş, Puiu D, et al, I. R. 10220/2014 Influența mărimii orificiului de evacuare în procesul formării dopului de gheață în conducta orizontală cu Dn 200 mm în absența curgerii apei demineralizate, SCN Pitești