

Normal Modes analysis of a ship's substructure - From FORAN CAD data to results using ANSA, Nastran and META

ANSA / META, the pre- and post-processing suite of BETA CAE Systems, is being increasingly deployed in the Maritime Industry. Beyond the suite's functionality that is already established in numerous domains, enhancements are under development to address the special challenges and requirements of the Maritime Industry. Within this framework, BETA CAE Systems cooperates with other system vendors who offer solutions in this domain. Among them is SENER Ingeniería y Sistemas SA, the developer of FORAN System, a worldwide reference program for computer design and marine construction. BETA CAE Systems and SENER, are closely co-operating to offer smooth transition of data from CAD to CAE level. ANSA reads numerous formats of CAD files that carry the geometric data and meta-data to the CAE pre-processing stage, and captures high quality geometric representations and topology.

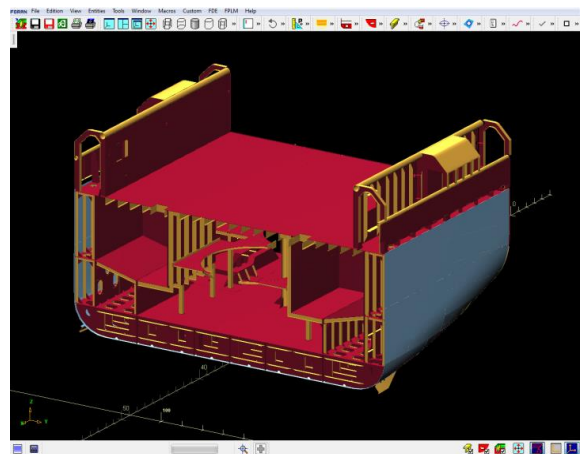
The FORAN to ANSA interface brings significant benefits and accelerates product design for the maritime industry.

Summary

This paper describes the complete process of setting up a normal modes analysis of a ship's substructure, from CAD data created in FORAN to a ready-to-run solver file created by ANSA. Nastran is used for the solution while the results are presented in META post-processor. The process begins with importing the FORAN CAD data to ANSA, cleaning-up the geometry, creating the mesh, and setting up the solver file before outputting it as a ready to run Nastran file.

Geometry

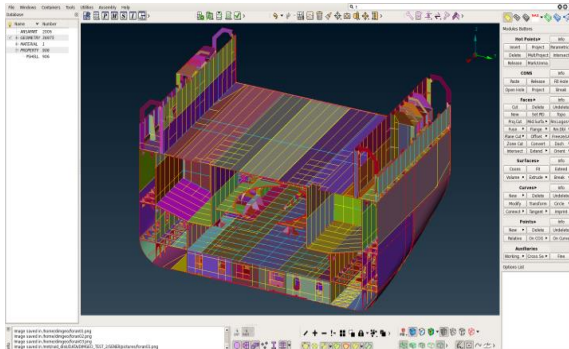
FORAN produces a STEP file that contains the geometry and an xml file that contains all the additional information related to this geometry.



Some geometry clean-up and idealizations are applied in FORAN to facilitate the meshing process. The STEP file, then, is imported to ANSA. A special ANSA script parses the xml file and applies the necessary information to the model. The plate parts of the FORAN file are



translated as ANSA parts while properties with the correct thickness are created and assigned to the geometrical faces. Additionally, the materials are created and assigned to the corresponding properties.



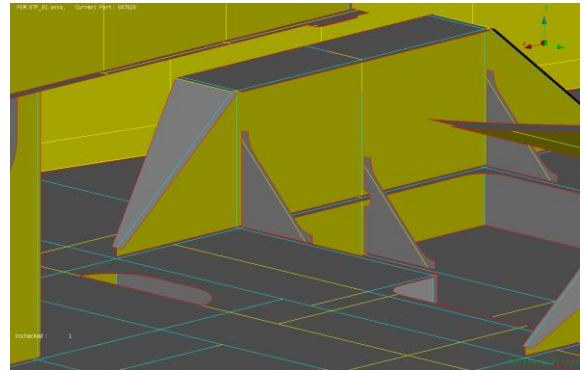
The profile parts are also translated as ANSA parts. In this approach, the additional information of the profile parts (scantling, section area, etc.) is attached to the ANSA parts as comments. Both the plate and profile parts carry the information of the block and ship that they belong to, as part attributes

As the model is composed of shell plate parts there is no need for middle surface extraction.

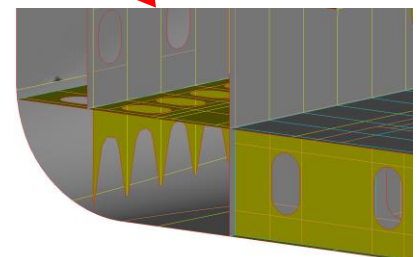
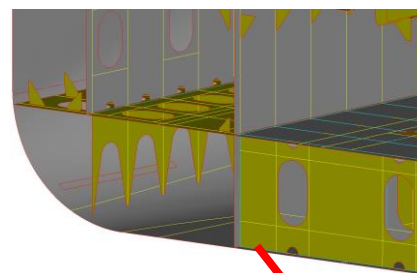
For models composed of plates with solid description, ANSA offers the functionality for the middle surface extraction and the corresponding topology identification.

The embedded geometrical checks of ANSA can be employed for the identification of any problematic areas and, whenever possible, for their automatic correction. The remaining issues are interactively resolved using a whole variety of software functionality. Existing geometrical faces can be

connected, trimmed or extended, while new geometry can be easily created.



Among the additional numerous capabilities in handling geometrical features, holes and welding cuts can be automatically filled and cracks can be closed.



The model is almost 100% symmetrical with differences only in some openings in various plates, but since they do not play an important role in the current analysis, they can be neglected. Thus, half of the

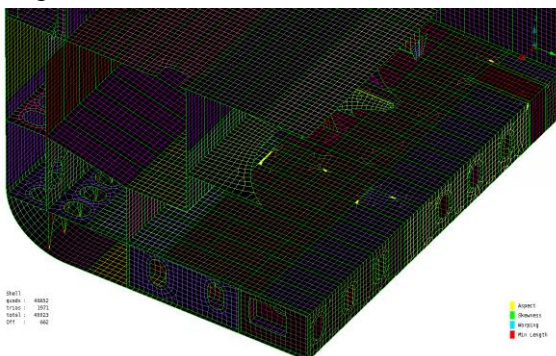


model can be deleted. This greatly reduces the pre-processing time since all the tasks are applied to half model. When the model is ready the other half is created with linked (children) symmetric entities.

Meshing

The Batch Meshing tool of ANSA is used to create the shell mesh. Batch meshing offers significant advantages over the interactive meshing. The user can define the meshing parameters and the quality criteria that should be respected and the mesh is generated on the whole model according to these settings. The de-featuring of the geometry can be automatically performed and special treatment can be implemented on features such as, holes and flanges. Improvement algorithms are automatically applied to the mesh, achieving a result of very high quality.

Any region that might still contain a few elements that violate the quality criteria can be treated further using the numerous available options for local improvement or re-generation of the mesh.

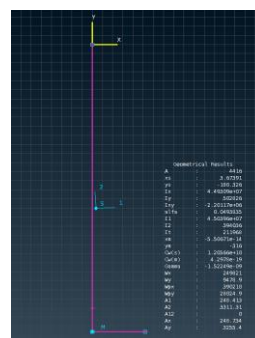


The profile parts can be exported by FORAN as beams or surfaces. In this case,

they were exported as beams and they appear in ANSA as 3d-curves. The aim is to model the profile parts using beam elements connected to the underlying shell mesh. For this reason, the position of these curves must be retained during meshing, since the beam elements pass from these positions. This constraint can be respected by the Batch Meshing tool, minimizing any user interaction.

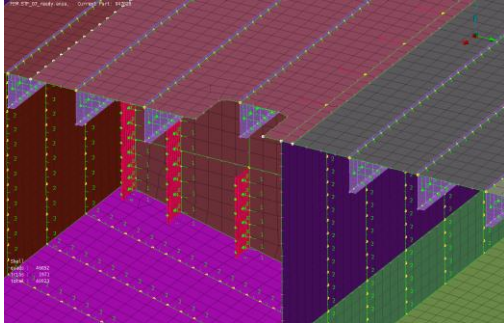
Moreover, ANSA offers a Cross Section tool for the creation of user-defined cross sections and a dedicated embedded solver for the calculation of cross-section properties. Such cross sections can then be used for the definition of the Beam elements.

The model contains stiffeners of various types, so a user defined section is created for each stiffener type. The beam elements can be generated using this database of cross sections and a special tool for stiffeners creation.

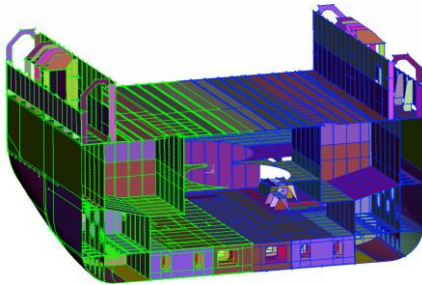


When the user wants to create a stiffener of a specific cross section, he can select this cross section from the database. Then, the edge of the model

that the stiffener will pass from and the beam elements, are automatically created. They are connected to the underlying shell mesh and they are oriented accordingly.



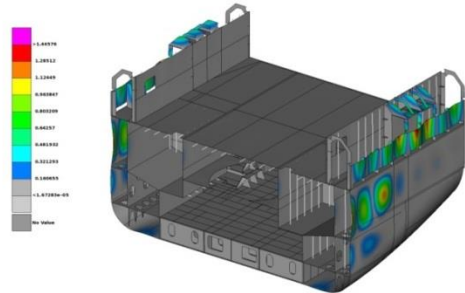
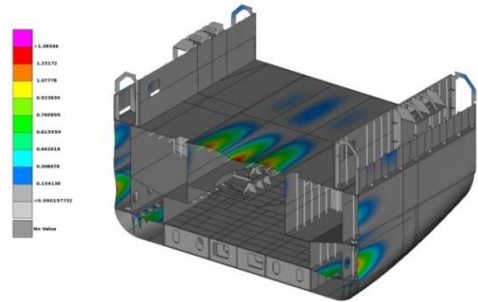
After the creation of the stiffeners, follows the generation of the symmetrical half. As the model is completed the Nastran Header is set-up and the resulting Output is the ready-to-run Nastran file.



Results

The aim of such type of analyses is the avoidance of resonance between fundamental vibration modes and main excitation frequencies. For simplicity reasons, the vibration behavior of the substructure has been regarded as independent of the behavior of the surrounding structure, but nonetheless some conclusions can be made. There are total 128 natural vibration modes in the range up to 20 Hz. Indicatively, we can see

two of the natural modes presented with a deformation scale of 10^2 .



The results are presented in META post-processor, which is capable of handling results from various solvers and for many disciplines, and is famous for its performance on big models and its automation capabilities.

For more information contact
BETA CAE Systems
Email: ansa@beta-cae.com