FINITE ELEMENT ANALYSIS OF COMPOSITE BOATS

Don Campbell¹, don@matrix.co.nz
Brian Jones², brianj@highmodulus.co.nz

Abstract. Advances in computer technology plus the potential for materials saving and boat performance improvement has meant that Finite Element Analysis (FEA) has increasingly become a standard tool for boating designers. However, recent spectacular boat failures from designers employing this methodology have highlighted the need for a systematic and rigorous approach to analysis work. This paper addresses the main issues involved in incorporating FEA into the boat design and production process. Issues include geometry cleanup, element selection, meshing, material and layup definition, load and support specification, analysis, optimisation and results interpretation and presentation. Examples ranging from luxury motor yachts through to high performance racing yachts are discussed.

1. INTRODUCTION

With advances in computer technology there are certain classes of boats that have now become candidates for routine finite element analysis (FEA). These include luxury motor yachts where the analysis budget may be a small portion of the overall budget and racing yachts where performance is the ultimate criterion. Another related class is boats that need to be certified by a regulating authority such as ABS [1], DNV[2] Germanischer Lloyds [3] or Lloyds London [4] that make allowance for FEA, although their underlying criteria appear to be based on basic beam analyses. Price still appears to be a barrier to routine analysis work, but automation procedures, high speed computing and specialist services have enabled FEA to become an increasingly acceptable design tool in the marine industry. However, in order to gain benefits from the process there needs to be specialists in each phase of the process ranging from the initial designer, the composite engineers through to the FEA analysts. A lack of experience and knowledge in any of these steps can lead to design disasters as seen by recent well-documented structural failures where FEA analysis was extensively utilised. However these failures may have been due to other contributing factors such as poor quality control during construction.

High Modulus NZ Ltd and Matrix Applied Computing Ltd have formed a strong strategic alliance to offer the boating industry a service in a narrow and specialized field of the design and build process; namely the structural composite design and accompanying analysis. This part of the whole process is depicted in Figure 1 and details of the various processes are discussed in the following sections.

1 Senior Consultant, Matrix Applied Computing Ltd
2 Director, High Modulus NZ Ltd
around the boat are likely to have a significant margin of error when compared to the model.

There is a danger of spending too much time optimising one aspect for high accuracy when it would be better to run more load cases. A good approach is to bring in more people for a “brain storming” session to determine the relevant load cases to ensure nothing is missed.

Geometry and meshing for local loadings and stress areas such as chain plates and rudder penetrations can be refined and run separately if desired.

One of the great strengths of FEA is also as a comparative or relative tool. In the case of high-end race yachts particularly, one can refine the laminates to improve the stiffness for performance in a number of load scenarios. The absolute results may not be relevant, but by comparing different design scenarios, an optimum can be reached.

2.2 Geometry

Often the geometry has come from the initial boat designer where performance and visual appearance have been the main criteria. This often leads to a mixture of surface and solid geometry with gaps in the hull surfaces, bad surface shapes and mismatched intersections between hull and bulkheads etc. For steel and aluminium boats, some of these issues are easily corrected with modern meshing technology that can ignore the mismatches in geometry and produce a high quality mesh. For a composite boat, there are usually several different regions that will be given a different layup (as specified by the composite designer) and it is more efficient to retain these different areas as separate and connected surfaces as shown in Figure 2.

The initial geometry file may also contain cosmetic geometry and the other important task is to eliminate redundant structure to end up with only structural elements. Care must be taken not to overly simplify the structure by removing significant structural components. Experience has shown it is always better to retain most structural components as it eliminates a question often asked “how does removal of that component affect the results?”

At the end of the geometry phase one should end up with a surface model that can be cleanly meshed with well-formed elements. The geometry should also be constructed in such a way as to anticipate future structural changes. Once a fully developed model has been created it often proves costly to incorporate unanticipated structural changes back into the original model.

If the structure is symmetric, then it is efficient at this stage to only clean up the geometry for the half model.

2.3 Finite Element Meshing

Given the typical length to thickness ratios of most structural boat components (hull, deck, bulkheads etc), the most popular FEA approach is a thick shell analysis where membrane, bending and coupling stiffness terms of the thick shell elements are derived from the orthotropic material properties. Not all finite elements are equal and the suitability of a particular element may be verified using benchmark problems from NAFEMS [5]. This shell approach precludes the extraction of strains in details such as taped joints etc. and every connection of components (e.g. bulkhead to hull) is considered as a fully “taped” or “welded” joint.

With modern meshers such as MSC/Patran [6] meshing should be relatively straightforward and the only real requirement is preservation of element shape (often governed by surface quality) and retaining sufficient elements to adequately represent the structural deformation whilst at the same time allowing solutions in
a short time period. Typical models today will be of the order of 100,000 to 200,000 thick shell elements with each element capable of representing a linear variation of strain throughout the element. Models much finer than this present post-processing difficulties that will become apparent later on. Typical global FEA meshes are shown in Figures 3 and 4.

Once the mesh has been created the model needs to be checked for unwanted gaps in the mesh and element normals should point in a consistent direction. This ensures that results are extracted from consistent surfaces. For example if adjacent areas have reversed normals the outer skin and inner skin may become confused leading to misinterpretation of results. Again, if the structure is symmetric, the mesh should only be developed on this half section and if possible the mesh and geometry should remain associative.

![Image of 20m race yacht Finite Element mesh.](image1)

**Figure 3.** 20m race yacht Finite Element mesh.

![Image of 42m high speed Luxury Motoryacht Finite Element mesh.](image2)

**Figure 4.** 42m high speed Luxury Motoryacht Finite Element mesh.

### 2.4 Material Specification

Based on well-documented engineering design procedures [1,2,3,4,7] the composite designer must take the “structure” and generate an initial layup or material specification. This is developed using their proven design procedures consisting of both structural calculations and practical design experience. Although a given region may consist of many plies, it is often expedient to simplify the layup to a skin-core-skin where the equivalent properties for the “skins” are calculated by the composite engineer from the material properties of each laminate at the specified fibre content. If areas are found to be under-designed, reinforcing strips can then be superimposed on these base “3 ply” layups. The base fibre and laminate properties are usually obtained from handbooks, [8] or material supplier’s data sheets and High Modulus does a significant amount of material testing to derive some of its own laminate properties.

Test problems using this technique and comparing it with a full layup specification have shown good agreement in both displacement and outer surface strain levels. On a typical racing yacht which was analysed using a full layup and “3 ply” layup, the results for both displacement and outer surface strain agreed to within less than 1%. However the simplified modelling approach precludes the use of more sophisticated ply-by-ply failure tools and relies on failure criteria as applied to total surface strains.

For some of the high performance yacht work, more plies are specified (typically 4 plies per skin plus a core on the hull) and in some critical design work, the complete layups with a multitude of plies are often specified. This results in more pre and post-processing effort but does allow more insight into the laminate behaviour.

A drawing showing the different layup regions, as well as an accompanying spreadsheet is produced by the composite engineer and passed to the analyst who is then responsible for transferring that information into the finite element model. A typical layup specification is shown in Figure 5 with accompanying spreadsheet in Figure 6. The specification breaks the yacht up into discrete sections where different combinations of plies can be applied. This enables the designer to place material where it is needed. Consequently, each region labelled in Figure 5 will have a corresponding layup definition in Figure 6. As can be seen in Figure 6, Area B has the same base laminate as Area A plus an additional outside and inside skin.

![Figure 5. Typical layup specification](image3)
<table>
<thead>
<tr>
<th>Laminate</th>
<th>Thickness (mm)</th>
<th>Exx (GPa)</th>
<th>Eyy (GPa)</th>
<th>Gxy (GPa)</th>
<th>Gzx=Gzy (GPa)</th>
<th>v</th>
<th>ρ (kg/m³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>AreaA_OS</td>
<td>2.0</td>
<td>13.57</td>
<td>17.80</td>
<td>5.10</td>
<td>3.00</td>
<td>0.17</td>
<td>1760.0</td>
</tr>
<tr>
<td>CORE</td>
<td>15.0</td>
<td>0.05</td>
<td>0.06</td>
<td>0.03</td>
<td>0.03</td>
<td>0.30</td>
<td>100.0</td>
</tr>
<tr>
<td>AreaA_IS</td>
<td>2.0</td>
<td>12.10</td>
<td>17.66</td>
<td>5.60</td>
<td>3.00</td>
<td>0.19</td>
<td>1760.0</td>
</tr>
<tr>
<td>AreaB_Layer2_OS</td>
<td>1.0</td>
<td>14.80</td>
<td>14.80</td>
<td>31.60</td>
<td>3.00</td>
<td>0.8</td>
<td>1520.0</td>
</tr>
<tr>
<td>AreaB_Layer1_OS</td>
<td>2.0</td>
<td>13.57</td>
<td>17.80</td>
<td>5.10</td>
<td>3.00</td>
<td>0.2</td>
<td>1760.0</td>
</tr>
<tr>
<td>CORE</td>
<td>15.0</td>
<td>0.05</td>
<td>0.06</td>
<td>0.03</td>
<td>0.03</td>
<td>0.30</td>
<td>100.0</td>
</tr>
<tr>
<td>AreaB_Layer1_IS</td>
<td>2.0</td>
<td>12.10</td>
<td>17.66</td>
<td>5.60</td>
<td>3.00</td>
<td>0.19</td>
<td>1760.0</td>
</tr>
<tr>
<td>AreaB_Layer2_IS</td>
<td>1.0</td>
<td>14.80</td>
<td>14.80</td>
<td>31.60</td>
<td>3.00</td>
<td>0.8</td>
<td>1520.0</td>
</tr>
</tbody>
</table>

Figure 6. Typical layup definition in spreadsheet format.

2.5 Specification of the finite element layup

Often layups are specified as regions of different “thick shell composite” properties derived from knowledge of how plies overlay each other. With a modern laminates modeller such as in MSC.Patran Laminates Modeller [9], the process is much more logical and meaningful. Base plies can be draped over surfaces or groups of surfaces and reinforcing plies draped on top of these base plies. This draping procedure requires the specification of a starting point, an application direction and reference direction and will then take account of how the directions of the fibres will change over a curved surface. It will even produce the flat pattern shape of the material needed to drape to a particular curved surface area and indicate if a surface cannot be successfully draped. Figure 7 shows a typical fibre orientation from draping onto a curved surface.

The layup technique employed at Matrix involves the following steps.

a) Creation of a ‘dummy’ layup. This is used to link the layup data stored in the supplied spreadsheet to the finite element model by defining ply regions, reference and application directions. Only a single ply is applied to each region, which speeds up the layup definition process. This procedure produces a “session” file for MSC/Patran that may be edited and replayed to create the true layup.

b) Import the layup definitions into customised software to take the material model spreadsheet supplied by High Modulus (Figure 6), interface it to the session file (produced in step (a) above) and automatically produce a new session or replay file that automatically generates the desired layup. This procedure is suitable for any number of plies used in the layup.

The process outlined above was developed for increased efficiency and reliability, eliminating any error in translating the supplied material properties and makes subsequent runs with different materials a simple procedure. The layup can be checked through the use of graphics that will depict individual ply angles, stacking sequence, total ply thickness etc. A typical stacking scenario is shown in Figure 8.

Figure 7. Fibre orientation of a draped ply on a curved surface.

Figure 8. Typical stacking scenario, ply thickness (mm)

Reflection of composites is not an automatic process. Finite element software generally seeks to retain consistent element normals on reflection and as a consequence the “element material” directions are rotated. This means that a ply oriented at say 45 degrees is not reflected correctly and needs a 90-degree rotation for correct alignment. There is an automated layup reflection option that has been recently implemented (at the request of Matrix) within the MSC.Laminates Modeller. However, this option removes element groupings from the reflected model. These groupings are important and enable the analyst to view different sections of the model separately. For example the deck, hull, bulkheads, longitudinals etc. can all be viewed independently of each other and this greatly assists in results review and presentation.

Matrix has developed its own software to perform the reflecting procedure that allows more control of the reflection process and retains element groupings.

The draping process has been approximated in the past by specification of a global material orientation angle that does not take into account fibre realignment on the curved surfaces. Test problems completed on real structures show the simplified method tends to underestimate stiffness (by 5% on a recent test hull),
which may be an important limitation when seeking very stiff structures for "top end" race yachts.

From the layup specification, the element properties are generated and a curved surface consisting of say 10 different layup regions may end up with over a 100 different element properties as the software assigns different properties based on the material ply directions derived from the draping process. The simplified procedure mentioned in the previous paragraph would only return one property per layup region.

The properties are generated for a thick shell element which include a membrane and bending stiffness as well as the coupling between bending and membrane. That is, an unbalanced composite element in pure tension will deflect out-of-plane.

2.6 Load and Constraint Specification

The loads from the various bureaus of standards tend to be based on simplified beam models that do not readily translate into a general shell finite element model. For example the sagging load case may be specified as a bending moment profile that can be represented as opposing shear forces applied at the specified locations along the hull to reflect the relevant bureau’s desired load distribution.

When applying these loads to the shell model, care must be taken that induced loads do not result in very high strain concentrations and the loads should be spread in some consistent manner around the hull to avoid this. It may also be necessary to move the load slightly to a bulkhead location to avoid gross local bending of the hull under the applied line loads. Similarly rig loads are spread into the surrounding area using special load interpolation elements rather than concentrating the load at a single point producing unrealistic strains.

For luxury yachts, discrete mass from items such as the engine and generator can also be beamed onto support points using the interpolation elements. Distributed mass from other fittings is often spread to the structure by using a dummy ply with low stiffness and appropriate density. This does not affect the structural performance but allows a better representation of the mass distribution within the boat.

When not designing to a scantling rule, the loads are a little more difficult to specify, although the designers and composite engineers have the freedom to develop a number of different loading scenarios and observe the effect on the structure. Generally the load cases will be designed to induce worst bending and torsion scenarios as well as bow slam acceleration loads. Any hull wave profile can be specified and the resulting hydrostatic pressure distribution generated. High Modulus tends to start with the simplest loading of a boat at dock and progresses through several load cases to a boat plunging down a monstrous wave in the southern ocean. Figure 9 is a plot of pressure contours on the hull caused by a specified wave profile. The wave profiles are generally generated from knowledge of the boat length and a typical and worst sea state. The hydrostatic pressure is then calculated for each element based on its depth below the wave surface.

Matrix uses MSC.Nastran [10] for its marine analysis work along with a technique used in the aerospace industry for unsupported structures. This technique, called inertia relief, eliminates the need for artificial supports to constrain the boat on the water in a particular orientation. The software takes the resulting load applied to the boat and then determines the accelerations in all directions that need to be applied to the boat to balance the loads. For example a boat at dock with hydrostatic pressure below the design water line should return an inertial acceleration of 1g if the full mass of the boat is represented in the model. However, the designers also have a responsibility to produce consistent loads that are balanced in some sense or artificially high balancing inertial loads will be generated.

3. RESULTS PROCESSING

One of the most important immediate results is the overall hull bending which gives an indication of the hull stiffness. If the boat is artificially constrained at a single point location, there may be some artificial rigid body rotation that obscures the overall deflected shape. The use of inertia relief places the displaced shaped in the optimum position so the bow and stern deflect almost equally about the central axis. Figure 10 shows a typical balanced deflected shape of a racing yacht.

As well as overall deflection, different components can also be checked for interference. For example, the doors...
should still open when the boat is under load. Often this interference is very difficult to detect from a displaced shape plot and a useful technique is to place a series of low stiffness rods across the structural opening. Knowing the length of the rods and the strain in the rods, the relative deflection can be readily calculated.

Once satisfied with displacements, the individual components (hull, bulkheads, decks etc) are contoured for strain. In general the maximum and minimum surface principal strains are contoured. Normally these strains are extracted at the outer extremities of the inside and outside skin, though they can be shown at any individual ply for a multi-ply layup. At High Modulus, the general design criteria is to limit the maximum principal strain in the region of 0.2% to 0.3% for carbon laminates resulting in a margin of safety of approximately 3 to 4.

The use of strain tensor plots provides better insight into the strain flow and gives an indication of the direction of extra plies needed to reinforce areas of concern. It also greatly assists with clarification of what areas are in tension, compression or shear. The tensors are difficult to view and if the mesh is too fine, the tensor plot becomes too cluttered to be of much use. Figure 11 is a tensor plot of part of a yacht bulkhead.

Figure 11. Principal Strain Tensor plot

Another useful post-processing technique is to isolate different components and to plot the free body forces on these components. This assists with designing the taping used for example to tape the bulkheads to the hull etc. Figure 12 shows the force vectors at part of a hull to longitudinal boundary.

Figure 12. Force Vector plot at hull to longitudinal boundary.

4. REFINED ANALYSIS

If there is an area of concern, a refined analysis may be carried out on part of the structure. One of the most convenient methods is to use a sub-structuring technique where the boundary displacements from a “coarse” analysis are used as boundary conditions for the refined area. The refined area may use a completely different mesh as long as it ties in with the boundary nodes. Within MSC.Patran the boundary displacements from the coarse analysis can be extracted by a simple keystroke and this eliminates any translation errors in going from the coarse to the refined model. This technique has been successfully used to examine the potential for buckling in isolated areas of the structure where a full buckling analysis would be impracticable.

5. OPTIMISATION

A boat project usually involves several design loops where different layups are considered, weak areas reinforced, excess material removed etc. This process may be generally thought of as optimising the design. This process can be automated to some extent through the use of computer optimisation. For example the objective may be to minimize the weight of the hull whilst constraining the peak bending deflection to be less than “x mm” and the maximum principal strains in the hull to be less that 0.3%.

The specifications of the layup (ply thickness and orientation) are set as the design variables, the loads applied and optimisation run to arrive at a minimum weight that satisfies the design constraints.

It is extremely difficult to optimise on ply angles as design variables (response varies as the cube of the angle) so a viable technique is to use a 0, +45, 90 skin-core-skin layup and have the 8 skin ply thicknesses as the design variables. Any other starting array of angles may also be used. The total ply thickness may be constrained to remain constant. The optimised solution then gives the designers the relative percentage of plies in the specified directions to achieve the optimum (say maximum stiffness for a given weight). This technique has worked particularly well for race boats but has not
been applied to any luxury yachts. As with all optimisation problems, the optimised solution must be able to be manufactured.

6. CONCLUSIONS

If properly specified, FEA can be of great assistance to the design process. As well as giving more confidence in a design, it can enhance understanding of the structural behaviour and assist in arriving at a more cost effective or better performing design.

The essential steps can be summarised as follows:

1. Define what is required from the analysis and what load cases need to be considered.
2. Clean up the geometry suitable for composite meshing.
3. Mesh with a suitable element size to capture the strain variation under the applied loads.
4. Check for mesh continuity, consistent normals, etc.
5. Define the layup and check material angles and thicknesses.
6. Define the load cases, restraints and analyse.
7. Review the results to check for consistency and agreement with expected results.
8. Repeat steps 5 to 7 to arrive at a suitable design.

Arriving at a suitable design does, of course, rely on realistic load estimates and the structural performance of the boat will ultimately depend on how well the boat is manufactured.

References

1. American Bureau of Shipping, USA
2. Det Norske Veritas, Norway
3. Germanischer Lloyd, Germany
4. Lloyds Register of Shipping, England
5. Composite Benchmarks, NAFEMS Ref:-R0031, Issue:2, England
6. MSC.Patran Version 2000, MSC.Software, USA
9. Using MSC.Patran Laminates Modeller, MSC.Software, USA
10. MSC.Nastran 2001, Quick Reference Guide, MSC.Software, USA