

Improvements in the Commercial Viability of Finite Element Analysis (FEA) for Accurate Engineering of Marine Structures

Paul Bosauder¹, paul.bosauder@matrix.co.nz

Don Campbell², don.campbell@matrix.co.nz

Brian Jones³, brian.jones@highmodulus.co.nz

Abstract. Advances in computer technology plus the potential for materials saving and structural performance improvement has meant that Finite Element Analysis (FEA) has increasingly become an essential tool for composite designers. Although the finite element analysis of composite structures has become relatively common in aerospace, automotive racing and marine applications, little commercial development has taken place to streamline what can often be a tedious and time-consuming model development process. Model complexity, cost and generation lead-time remains a restrictive factor on the full scale adoption and efficiency of finite element composite analysis. In addition recent spectacular composite marine failures have highlighted the need for improved quality assurance measures for both analysis and manufacturing.

This paper presents advancements in finite element pre-processing technologies for composite structural design and analysis. Advanced pre-processing tools specifically customized for composite analysis have been developed and validated. The origin of this technology stems primarily from the analyses of America's Cup Class (IACC) yachts and other high performance marine vessels and has enabled the deployment of finite element analysis with increasing accuracy, speed and versatility. Workflows for a complete design to analysis process are presented with a particular focus on the customized software tools developed and the recent successful MAXIMUS yacht design analysis project.

1. INTRODUCTION

Finite Element Analysis is now being used routinely in the design of complex composite structures in the automotive, aerospace and boating industry. In the automotive industry, composite analysis is mainly confined to racing teams while there has been a recent move to more composite components in the commercial aerospace industry. The most well known recent commercial project is that of the Boeing 777 where 90% of the aircraft's structural weight is derived from composite materials. In the marine industry all high performance yachts such as IACC are composite and there are an increasing number of luxury boats being constructed out of composite materials. With the high cost of composite materials, there is a significant return on investment on the use of FEA to optimise the design both from a materials, strength and stiffness perspective.

Although many FEA programs have composite analysis capability, there are still only a few packages (i.e. MSC.Patran Laminate Modeler) that allow draping of the material and hence account for a change of material fibre direction during the draping process. Also in composites there is the

ability to apply local reinforcing, often leading of hundreds of different material regions. There is a general lack of software that provides a good interface between the composite designer and the FEA analyst and this paper outlines a methodology that has been developed to streamline this process. Using the recent example of the MAXIMUS yacht, this paper outlines typical workflows and demonstrates how these tools can provide control of the layup process and significantly reduce model development and editing time.



Figure 1. MAXIMUS Composite Yacht during Rolex Transatlantic Challenge

1 Senior Consultant, Matrix Applied Computing Ltd

2 Senior Consultant, Matrix Applied Computing Ltd

3 Director, High Modulus NZ Ltd

2. OVERVIEW OF FEA WORKFLOW FOR COMPOSITE STRUCTURES

Over the past decade 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 detailed structural composite design and accompanying FEA.

During this time the finite element modelling process has evolved allowing more detailed analysis, faster model development and improved quality assurance (QA). Figure 2 outlines the finite element modelling process for composite structures while emphasising some of the proprietary developments (shaded) that have taken place.

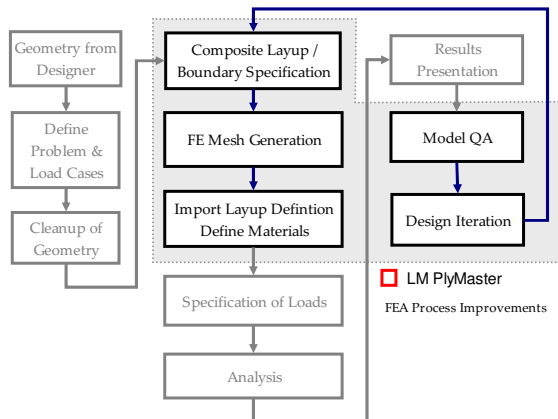


Figure 2. Finite Element Analysis (FEA) Procedure for Composite Structures.

The purpose of FEA remains to ensure adequate structural performance from both a stiffness and safety perspective. Geometry must be imported into a finite element environment, meshed, material properties defined, loads specified and the resulting displacements and strains calculated. At the end of the solution process results must pass QA inspections. Results may then be utilised to optimise the existing design. In most cases this involves some manipulation of the laminate definition.

Figure 3 is a plot of the MAXIMUS finite element model recently developed by Matrix and High Modulus that has been contoured by laminate region. This illustrates regions where composite designers have chosen to specify unique definitions

for laminate properties. The finite element representation of laminate regions are chosen to provide a reasonable numerical approximation of the real structural configuration.

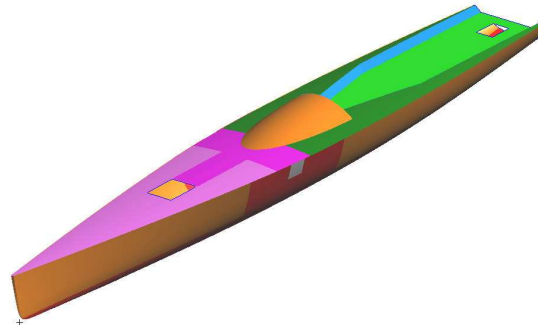


Figure 3. Finite Element Model MAXIMUS Contoured LaminateRegion

The most time consuming aspect of developing a yacht composite FEA model remains in defining the composite layup. This is mainly due to the anisotropic nature of composites, draping of the underlying fabrics and the shear volume of individual plies that define a typical yacht finite element model. Figure 4 illustrates exactly how customised pre-processing software can automate a large percentage of the procedure.

LM PlyMaster is an MS Excel and MSC.Patran customization developed at Matrix during previous America's Cup campaigns. Ultimately LM PlyMaster enables laminate definitions to be imported from a composite engineer's spreadsheet directly into the finite element modelling environment.

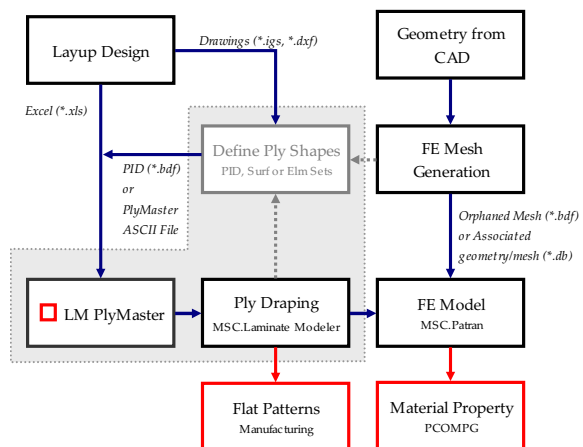


Figure 4. Detailed Workflow for Laminate Property and Flat Pattern Generation.

This software customisation provides a two-way interface from MSC.Patran Laminate Modeler to MS Excel. Thus, 2D-orthotropic materials, draped plies and layup stack definitions are transferred directly without analyst input. In addition, the application contains tool sets to manage reflections of symmetric or partially symmetric geometry, finite elements, draped plies and layup stacks, both within the MSC.Patran Laminate Modeler and MS Excel environments. Advanced modelling features such as boundary splits, thickness offsets, and ply groupings are also supported.

The key advantages of implementing such a method are:

- Reduced modelling effort and time as material definition and reflection processes are automated.
- Improved QA by removing human input of data.
- Provision for editing/defining finite element material definitions with a MS Excel environment.
- The composite engineer can work within a familiar MS Excel environment without detailed knowledge of FEA software.
- Improved visualisation of layup stack.
- Two-way communication between MSC.Laminate Modeler and MS Excel.

3. MODEL DEFINITION & CREATION

Further information will be provided for each of the critical processes defined in Figure 2, moving from the initial geometric design through to the creation of the finite element model and its corresponding material model.

3.1 Geometry

In the marine industry, often the geometry has come from the initial boat designer where performance and visual appearance have been the main criteria [12]. This can lead 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.

3.1.1 Laminate Considerations

For a composite boat, there are usually several different regions that will be given a different layup. This must be given some thought prior to meshing and a decision made how to best treat these laminate regions. Essentially one of either two main approaches is most commonly applied.

1. Geometry used to define ply zones
2. Finite elements used to define ply zones

(i) Geometry used to Define Ply Zones

In this case different ply or laminate zones are defined by separate and connected surfaces as shown in Figure 5. Thus specification of ply shapes or areas can be performed by selecting individual surfaces or sets of surfaces. If additional reinforcement is required then finite element sets may also be selected and mixed finite element and surface ply definitions are supported.

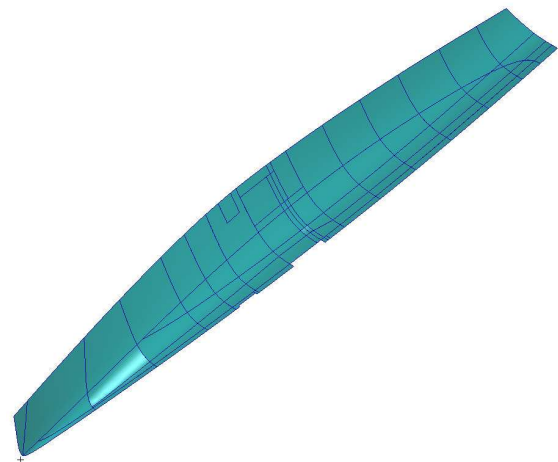


Figure 5. Hull Geometry for MAXIMUS broken at Intersections and Laminate Regions.

(ii) Defining regions by Elements

Alternatively laminate zones may be approximated as element sets instead of surface sets. Thus geometric boundaries at laminate zones will not be required. This approach is more common in automotive applications where third-party meshing tools are implemented. The output from the third-party meshing tool is an orphaned finite element mesh (e.g. *.bdf) which is carried forward into the rest of the modelling workflow.

Unique element sets are often given a common material property ID (PID). These PIDs can be

referenced later to define laminate regions within LM PlyMaster [11].

Ultimately both approaches are valid and supported within the LM PlyMaster modelling frame work and often the ability to mix these methodologies is required. A comparison of geometric and finite element ply-zone definitions is provided in Figure 6.

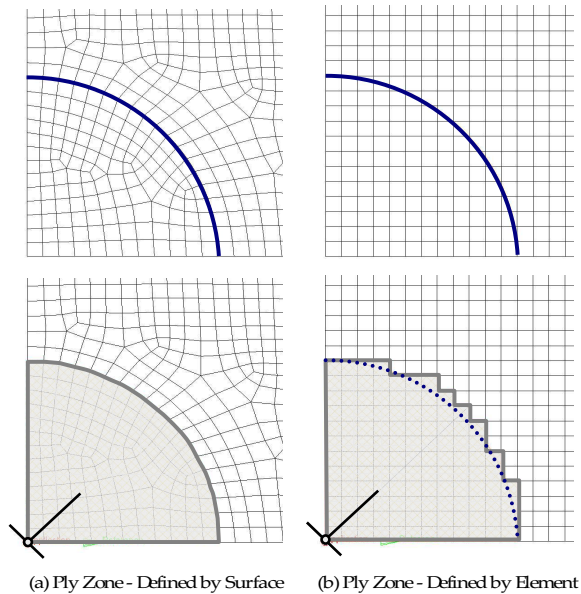


Figure 6. Comparison of Geometric and Finite Element Based Ply-Zone Definitions.

Within the context of marine applications and specifically the MAXIMUS finite element project, 90% of the laminate definitions were performed via surface definition.

3.1.2 Symmetry Considerations

If the majority of the structure is symmetric, which is true of many automotive, aerospace and marine structures, then it is efficient at this stage to only develop a half model and reflect the geometry later [12]. This approach can be applied even for structures that are not completely symmetric in terms of geometry and laminate definitions.

Certain geometric asymmetries do not necessarily preclude the development of a half model during the geometry and meshing stages. By consolidating asymmetric features in the half-model the correct geometry may still be represented. Post reflection of unwanted elements and surfaces are removed

before analysis to define the desired topology. Highly unsymmetrical geometries can also be treated as exceptions and ignored during the reflection process.

The asymmetric forestay bulkhead of the MAXIMUS is a good example of this consolidation technique. Figure 7a shows the original bulkhead “as designed” and Figure 7b the “as modelled” bulkhead. In order to obtain the correct bulkhead topology, the cut-out was meshed and those elements removed from the appropriate side (port in this case) prior to analysis.

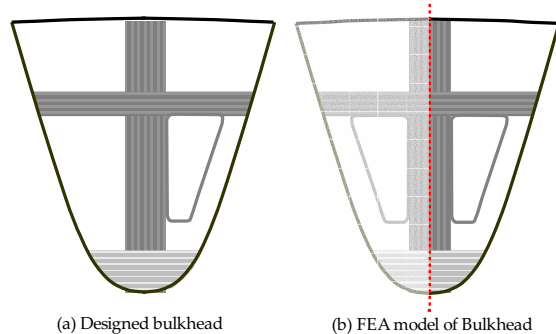


Figure 7. Asymmetric Forestay bulkhead as modelled in the MAXIMUS yacht finite element model.

3.2 Finite Element Mesh Generation

Given the typical length to thickness ratios of most automotive, aerospace and marine structures, 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. This shell approach precludes the extraction of strains in details such as taped joints etc. and every connection of components is considered as a fully “taped” or “welded” joint.

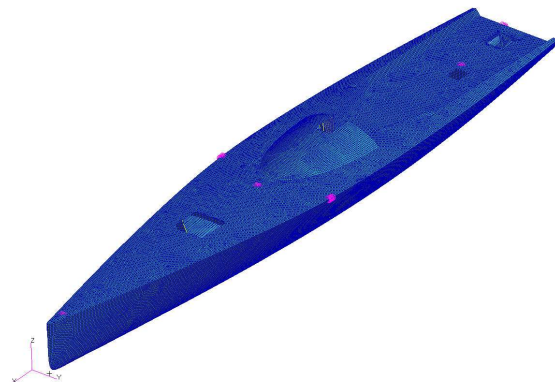


Figure 8. Global Finite Element model of MAXIMUS.

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 200,000 to 400,000 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, particularly with strain tensor plots [12]. The global finite element model developed for MAXIMUS is shown in Figures 8 and 9.

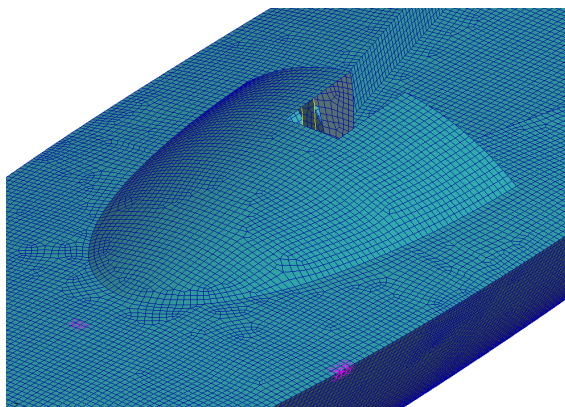


Figure 9. Global Finite Element Model of MAXIMUS

Again, if the structure is symmetric, the mesh should only be developed on this half section and, depending on the modelling approach, mesh and geometry should remain associative.

3.3 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. Once the layup design has been completed it must be transferred into the finite element model for analysis. The workflow for this process has already been illustrated in Figure 4.

Drawings 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. This 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.

Figure 10 is plot of the forward hull section of the MAXIMUS yacht contoured by laminate region. A typical composite yacht finite element model may include as many as 100 laminate or ply regions and more than 2,000 individual plies in the laminate stack for the entire model.

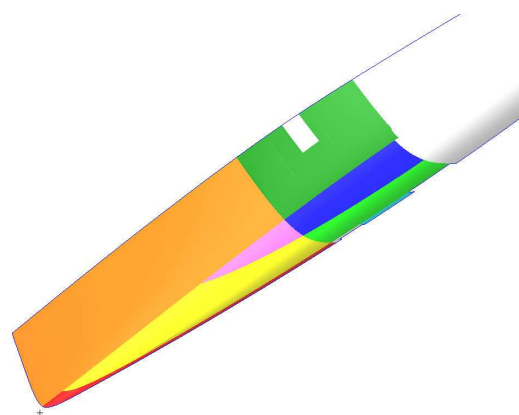


Figure 10. Laminate Region Definitions for MAXIMUS Forward Hull

There are essentially two fundamental approaches to how material properties are derived for the finite element model:

1. Equivalent skin-core-skin Layup
2. Ply-by-Ply Layup

These will be discussed in more detail but the methodology chosen is dependant on many factors including required accuracy, budget and project time-frame requirements.

3.3.1 Equivalent Skin-Core-Skin Layup

Although a given region may consist of many plies, the equivalent skin-core-skin method allows simplification of the layup to an equivalent 3-ply skin-core-skin representation. 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 most of its own material ply 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 [12]. 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.

3.3.2 Ply-by-Ply Layup

For a highly detailed analysis of the individual ply strains or optimisation of the ply layup, a more detailed investigation into the performance of the laminate is required. Thus the complete layup or some representation thereof is specified. This requires more pre- and post-processing effort, but allows more insight into the laminate behaviour. The Ply-by-Ply approach is required for investigation of more complex failure modes such first-ply failure.

Typically very little difference will be seen between the two methods regarding stiffness for marine sandwich structures but Ply-by-Ply would be required for accurate analysis of solid laminates with respect to bending stiffness and torsional effects.

A comparison of Equivalent and Ply-by-Ply inputs is shown in Figure 11. This indicates that far fewer individual plies are required for the equivalent method but more ply material definitions.

Laminate	Angle (deg)	t (mm)	Exx (GPa)	Eyy (GPa)	Gxy (GPa)	Gzx=Gz y (GPa)	v	ρ (kg/m ³)
Equivalent skin-core-skin layup								
Area_A_OS	0	2	21.8	15.1	6.0	3.0	0.29	1760
Area_A_Core	0	15	0.08	0.08	0.03	0.03	0.30	100
Area_A_IS	0	1.6	15.1	21.8	6.0	3.0	0.20	1760
Ply-by-Ply layup								
Area_A_OS4	0	0.5	35.3	7.6	3.7	3	0.30	1770
Area_A_OS3	-30	0.5	"	"	"	"	"	"
Area_A_OS2	30	0.5	"	"	"	"	"	"
Area_A_OS1	90	0.5	"	"	"	"	"	"
Area_A_Core	0	15	0.08	0.08	0.03	0.03	0.30	80
Area_A_IS1	90	0.4	35.3	7.6	3.7	3	0.30	1770
Area_A_IS2	-60	0.4	"	"	"	"	"	"
Area_A_IS3	60	0.4	"	"	"	"	"	"
Area_A_IS4	0	0.4	"	"	"	"	"	"

Figure 11. Comparison of Equivalent and Ply-by-Ply Material Properties

3.3.3 Draping of Composite Materials

Early finite element models of composite material employed a projected ply technique where material orientations were specified by a global material orientation angle. However, this technique does not take into account fibre realignment on the curved surfaces during draping of the plies. Test problems completed on a typical IACC hull has shown that this projected ply method tends to underestimate stiffness in the order of 5%-10%. Also, this level of error increases quickly with increased surface curvature and is an important limitation when seeking very stiff structures for high performance race yachts.

Modern laminate modelling tools such MSC.Laminate Modeler enable the draping process to be modelled and accurate local fibre orientations computed. 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.

The flat patterns computed by MSC.Laminate Modeler are highly accurate and have successfully been used to develop manufacturing drawings in recent yacht projects.

Figure 12 shows typical fibre orientations and flat pattern computed for a unidirectional draped at 45° onto the forward section of the MAXIMUS hull.

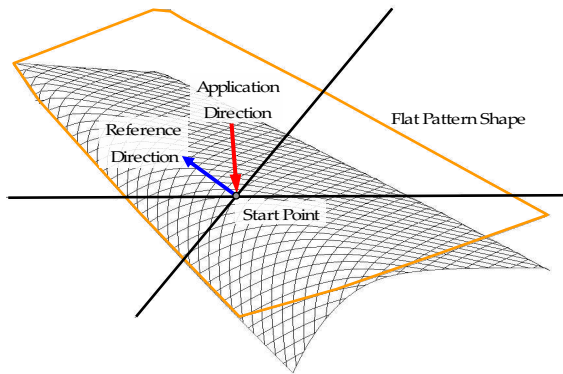


Figure 12. Draped 45° Ply and Resulting Flat Pattern on Hull Shell of MAXIMUS Yacht Computed via MSC.Laminate Modeler.

3.4 Specification of the finite element layup

Often layups are specified as regions of different thick shell composites where properties are derived from a knowledge of how the plies overlay each other. With a modern laminates modeller such as in MSC.Laminate Modeler [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.

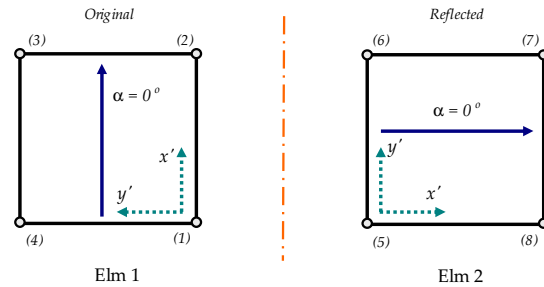
From the layup specification, the element properties are generated automatically and a curved surface consisting of say 10 different layup regions may end up with over a 100 different element property definitions (PCOMPG) as the software assigns different properties to groups of elements based on the material ply directions derived from the draping process. The projected ply procedure mentioned in the Section 3.3.3 would only return one property per layup region.

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

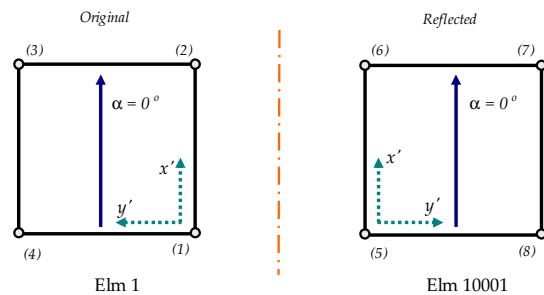
3.5 Reflection of Composite Finite Element Models

Reflection of composites is not an automatic process. Finite element software generally seeks to retain consistent element normal definitions on reflection and as a consequence, the element internal axes and hence material directions are

rotated. This means that an oriented orthotropic ply is not reflected correctly and needs a 90-degree rotation for correct alignment. This is illustrated in Figure 13 which compares general purpose and LM PlyMaster element reflection procedures.



(a) General Finite Element Reflection of 2D Orthotropic Material



(b) LM PlyMaster Reflection of 2D Orthotropic Material

Figure 13. Comparison of General Purpose and LM PlyMaster Finite Element Mirroring Procedures.

Automated layup reflection has been implemented within the MSC.Laminate Modeler environment. 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 and longitudinals can all be viewed independently of each other and this greatly assists in results reviewing and presentation.

The LM PlyMaster reflection toolset provides more control of the reflection process and retains surface/element groupings. It is possible to build symmetric and partially symmetric models reflecting each component of the model individually or collectively. In addition, this tool ensures fixed ID offsets for surfaces and elements that have been reflected. This has several advantages that will be discussed further in subsequent sections.

4 LM PLYMASTER DEVELOPMENT PROJECT

The role of LM PlyMaster in the analysis of composite structures is simply to enable the composite design to be transferred with minimal effort and error into the finite element model.

The composite design can be effectively defined as:

- 1) The shape or geometry of ply zones. (defined as drawings *.igs, *.dxf)
- 2) The material and ply definitions to be applied to each zone or region. (defined in MS Excel spreadsheets containing material, ply and layup data)
- 3) Draping Information (start points, reference directions, offsets...etc)

The LM PlyMaster development project has typically focused on these three areas while providing tools to speed general composite model development (i.e. support for model reflection, laminate offset, two-way MSC.Patran to MS Excel data transfer...etc). Visual Basic routines implemented within a dynamic link library (dll) are referenced into MS Excel and are used to drive the MS Excel component of LM PlyMaster while MSC.Patran PCL routines drive actions performed within MSC.Patran and MSC.Laminate Modeler. Figures 14 and 15 are screenshots from LM PlyMaster MS Excel component.

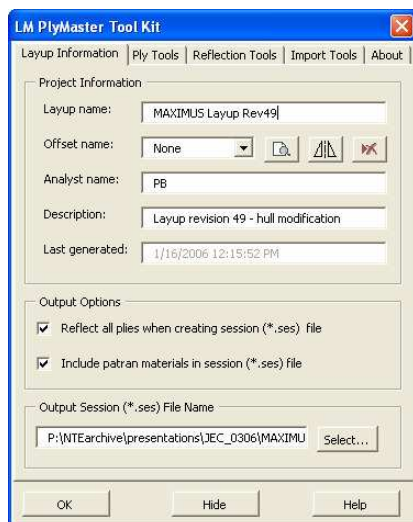


Figure 14. Screenshot of LM PlyMaster Main Form

Ply Id	Ply Name	Area	Material	Thickness (Unit - L)	Angle (Deg)	Group	Symmetric
1000	Deck_1_skin_OS_1	Deck_1	Deck1_skin_OS	6.3	0	→	1
1002	Deck_1_skin_OS_2	Deck_1	Deck1_skin_OS	6.3	0	→	1
1003	Deck_1_skin_OS_3	Deck_1	Deck1_skin_OS	6.3	-45	↘	1
1004	Deck_1_core_1	Deck_1	Deck1_core	20	45	↗	1
1005	Deck_1_core_2	Deck_1	Deck1_core	20	65	↗	1
1006	Deck_1_core_3	Deck_1	Deck1_core	20	65	↗	1
1007	Deck_1_skin_IS_1	Deck_1	Deck1_skin_IS	4.4	60	↗	1
1008	Deck_1_skin_IS_2	Deck_1	Deck1_skin_IS	4.4	40	↗	1
1009	Deck_1_skin_IS_3	Deck_1	Deck1_skin_IS	4.4	20	→	1
1010	Deck_2_skin_OS	Deck_2	Deck2_skin_OS	4.5	5	→	1
1011	Deck_2_core	Deck_2	Deck2_core	20	0	→	1
1012	Deck_2_IS	Deck_2	Deck2_IS	5.5	10	→	1
1013	Deck_3_skin_OS	Deck_3	Deck3_skin_OS	5.6	10	→	1
1014	Deck_3_core	Deck_3	Deck3_core	20	15	→	1
1015	Deck_3_skin_IS	Deck_3	Deck3_skin_IS	7.7	10	→	1
1016	Deck_4_skin_OS	Deck_4	Deck4_skin_OS	7.8	10	→	1
1017	Deck_4_core	Deck_4	Deck4_core	20	0	→	1
1018	Deck_4_skin_IS	Deck_4	Deck4_skin_IS	7.8	-10	→	1
1019	Hull_skin_OS	Hull	Hull_skin_OS	5.5	10	→	1
1020	Hull_core	Hull	Hull_core	30	0	→	1
1021	Hull_skin_IS	Hull	Hull_skin_IS	7.5	-10	→	1
1022	Hull_patch_OS	Hull	Hull_patch_OS	4	25	↗	1
1023	Bulkhead_1	Bulkhead_1	Bulkhead_1	7	45	↗	1
1024	Bulkhead_2	Bulkhead_2	Bulkhead_2	7	45	↗	1
1025	Bulkhead_3	Bulkhead_3	Bulkhead_3	7	45	↗	1
1026	Bulkhead_4	Bulkhead_4	Bulkhead_4	7	45	↗	1

Figure 15. Screenshot of LM PlyMaster Ply Sheet.

4.1 Definition of Ply Zones

Traditional methods of defining ply zones typically have involved the analyst importing geometry for laminate regions and either breaking geometry to define discrete regions or using sets of elements to specify where plies will be created within MSC.Laminate Modeler. This ultimately requires the user to select regions within the pre-processor and if the model is re-meshed and/or surfaces modified then there is no quick method of regenerating the layup information.

LM PlyMaster enables ply zones to be defined via either the creation of a 'template' layup or specification of property identifier (PID) sets. The 'template' layup is created by defining a single template ply on each unique region. These ply definitions are then exported to LM PlyMaster and stored in MS Excel for later use. This data need only be defined once per region and is then linked to the layup data stored in the supplied spreadsheets. The critical information that is defined by these template plies includes surface/element numbers (to define the area), start points, application directions, reference directions, boundary splits, offsets...etc.

Modifications to the layup, ply or material definitions are quickly re-transferred back into MSC.Laminate Modeler and ultimately into the MSC.Patran database once draping calculations have been performed.

4.2 Import of Material and Ply Data

Material and ply data specifying the number of plies applied in each region, angles, and thicknesses are imported from the composite engineer's spreadsheet. 2D Orthotropic material data referenced by each ply is also imported at this stage. Figure 16 is a sample spreadsheet containing both ply and material data for the MAXIMUS yacht. The format of this spreadsheet is flexible enough to accommodate both equivalent ply and ply-by-ply definition methodologies.

1	A	B	C	D	E	F	G	H
1	Material Name	Exx	Eyy	Gxy	Gxz	Gyz	v	Density
2	CC	62000	62000	4200	3000	3000	0.1	1.519E-06
3	SM	130000	8700	6400	3000	3000	0.3	1.519E-06
4	HM	215000	9300	4200	3000	3000	0.3	1.519E-06
5								
6	Ply Name	Material	Thickness	Angle				
7	Mast_Area_A_ply_1	CC	0.190	0				
8	Mast_Area_A_ply_2	HM	0.291	0				
9	Mast_Area_A_ply_3	SM	0.291	90				
10	Mast_Area_A_ply_4	HM	0.291	0				
11	Mast_Area_A_ply_5	SM	0.291	45				
12	Mast_Area_A_ply_6	HM	0.291	0				
13	Mast_Area_A_ply_7	HM	0.291	0				
14	Mast_Area_A_ply_8	SM	0.291	-45				
15	Mast_Area_A_ply_9	HM	0.291	0				
16	Mast_Area_A_ply_10	HM	0.291	0				
17	Mast_Area_A_ply_11	SM	0.291	45				
18	Mast_Area_A_ply_12	HM	0.291	0				
19	Mast_Area_A_ply_13	HM	0.291	0				

Figure 16. Example Material and Ply Input Spreadsheet for LM PlyMaster

4.3 Linking Ply Layup and Zone Definitions

The analyst must decide which plies should be applied to each region. LM PlyMaster automates this procedure providing there is some consistency in the names of the 'template' plies and the ply data imported from the composite engineer's spreadsheet. The algorithm is able to match individual plies to their respective zones based on a string type search. The following table provides an example of when this process is likely to be successful.

Dummy Ply Name	Imported Ply Name	Matched
Hull_A_template	Hull_A_OS2	OK
Hull_B	Hull_B_core15mm	OK
PlyTemplate1	Hull_A_OS4	NO

Figure 17 shows an example LM PlyMaster spreadsheet with recently imported ply zone, material and ply layup data. The automated routine has been used to match up the imported ply layup to template plies created within MSC.Laminate Modeler. This process of matching plies may also be performed manually by selecting the desired area from within a drop-down box in LM PlyMaster (as shown in Figure 15).

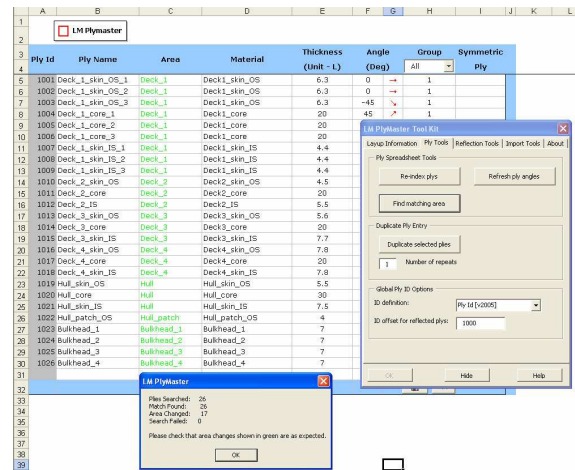


Figure 17. LM PlyMaster Spreadsheet with automated Ply Zone assignment.

Once complete, a valid laminate definition is defined and it may be exported into MSC.Laminate Modeler for draping, material model development and ultimately the property definitions for each individual finite element.

4.4 Export Layup data to MSC.Patran and MSC.Laminate Modeler

The process outlined above was developed for increased efficiency and reliability, eliminating any error in translating the supplied material properties. In addition performing subsequent runs with different materials becomes 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.

5 RESULTS PROCESSING

In many composite analyses, stresses are a little meaningless and strains in different plies are reported. Traditional methods for post-processing composite finite element models have focused on maximum values reported on the inside and outside surfaces of a composite.

However, finite element codes typically convert the laminate definition into layers for post-processing and, as a consequence of ply drop-off, it becomes impossible to select a single layer that defines an equivalent inside or outside skin.

This is demonstrated in Figure 18a, which defines a typical composite stack consisting of a base skin-core-skin laminate with local symmetric patching. In this example the finite element software splits each ply into a global series of layers as shown in Figure 18b. The result is that each individual layer contains mixed information so results for individual plies cannot be readily tracked.

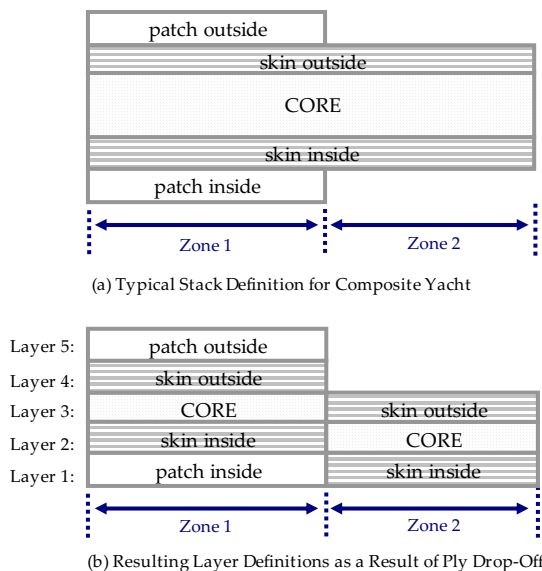


Figure 18. Example of Layer and Ply Tracking Issues in Post-Processing Composite Finite Element Models.

MSC.Nastran provides a solution to the inside/outside skin problem by computing inside and outside skin strains and reporting them in separate layers referred to as z1 and z2. The definition of inside and outside is a function of the element normal and stacking sequence for the ply region. This technology has been available in MSC.Nastran for many years.

The ability to track strain results in individual plies has only been recently resolved. The MSC.Nastran solver from version 2005 forwards supports the PCOMPG property definition. This contains an ID reference for each ply defined within the model that is carried through the solver and into the output

results database. Thus finite element results can be reported for each ply based on this unique ply ID.

This concept is also supported within both MSC.Laminate Modeler and LM PlyMaster. The PLY ID defined in the LM PlyMaster spreadsheet will be exported to MSC.Laminate Modeler and used to define the PCOMPG definition. As a result, information for a single ply is tracked all the way from the composite design spreadsheet, through draping, finite element solution and into results post-processing. The analyst performing the analysis work can view a ply in the design spreadsheet and monitor predicted strain levels and failure quantities with confidence.

6 DESIGN MODIFICATION

Once the designer has set up a spreadsheet, modification to the layup and alteration of the FEA model is a relatively automated feature. For example the thickness of several plies can be increased, the material of several plies altered and additional reinforcing plies added to selected regions. This can all take place at the spreadsheet level and once the designer is satisfied, this is passed from LM PlyMaster through to MSC.Patran and the model automatically updated. Thus the composite engineer can work within a familiar environment with minimum effort for the analyst to modify the FEA model.

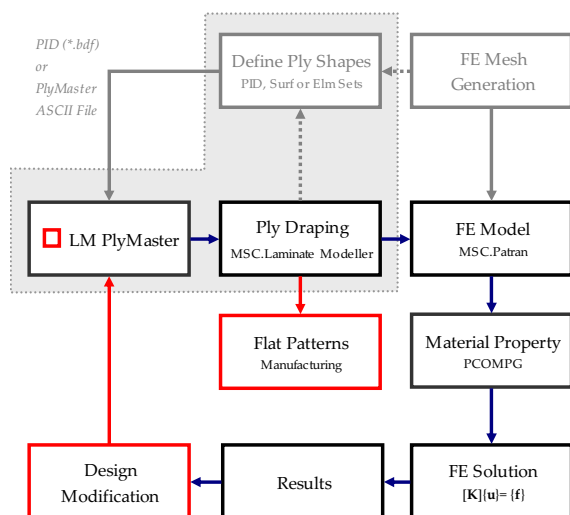


Figure 19. Extended Workflow for Design Modifications

Figure 19 provides an example workflow illustrating how laminate design modifications are automated within the LM PlyMaster environment.

Design modifications of a topological nature such as the addition of a bulkhead or stiffener require some additional effort to define draping parameters. However once draping parameters are defined either by the creation of dummy plies or by manual input then re-generation of the composite material model is completed automatically driven by LM PlyMaster.

7 CONCLUSIONS

Using the above techniques it is estimated there is a productivity gain factor of 3 to 4 in initial model generation and development and up to factors of 10 for each model iteration. In additional developed models inherit a higher standard of QA with fewer opportunities for analyst error. The LM PlyMaster tool has been continuously developed during real-life analyses of advanced composite structures such as MAXIMUS and IACC yachts. In recent campaigns this has enabled engineers to consider upwards of 100 different layup and geometric configurations, whereas in previous campaigns, these had been limited to many fewer iterations.

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 2005, MSC.Software, USA
7. Raymond J. Roark, Warren C. Young, "Formulas for Stress and Strain", McGraw Hill Inc. USA
8. Stephen W Tsai , "Theory of Composites Design", 1992. Think Composites, Dayton Ohio USA.
9. Using MSC.Patran Laminate Modeler, MSC.Software, USA
10. MSC.Nastran 2005, Quick Reference Guide, MSC.Software, USA
11. LM PlyMaster, Quick Reference Guide, Matrix Applied Computing Ltd, NZ
12. Finite Element Analysis of Composite Boats, High Performance Yacht Design Conference 2002, Auckland, NZ