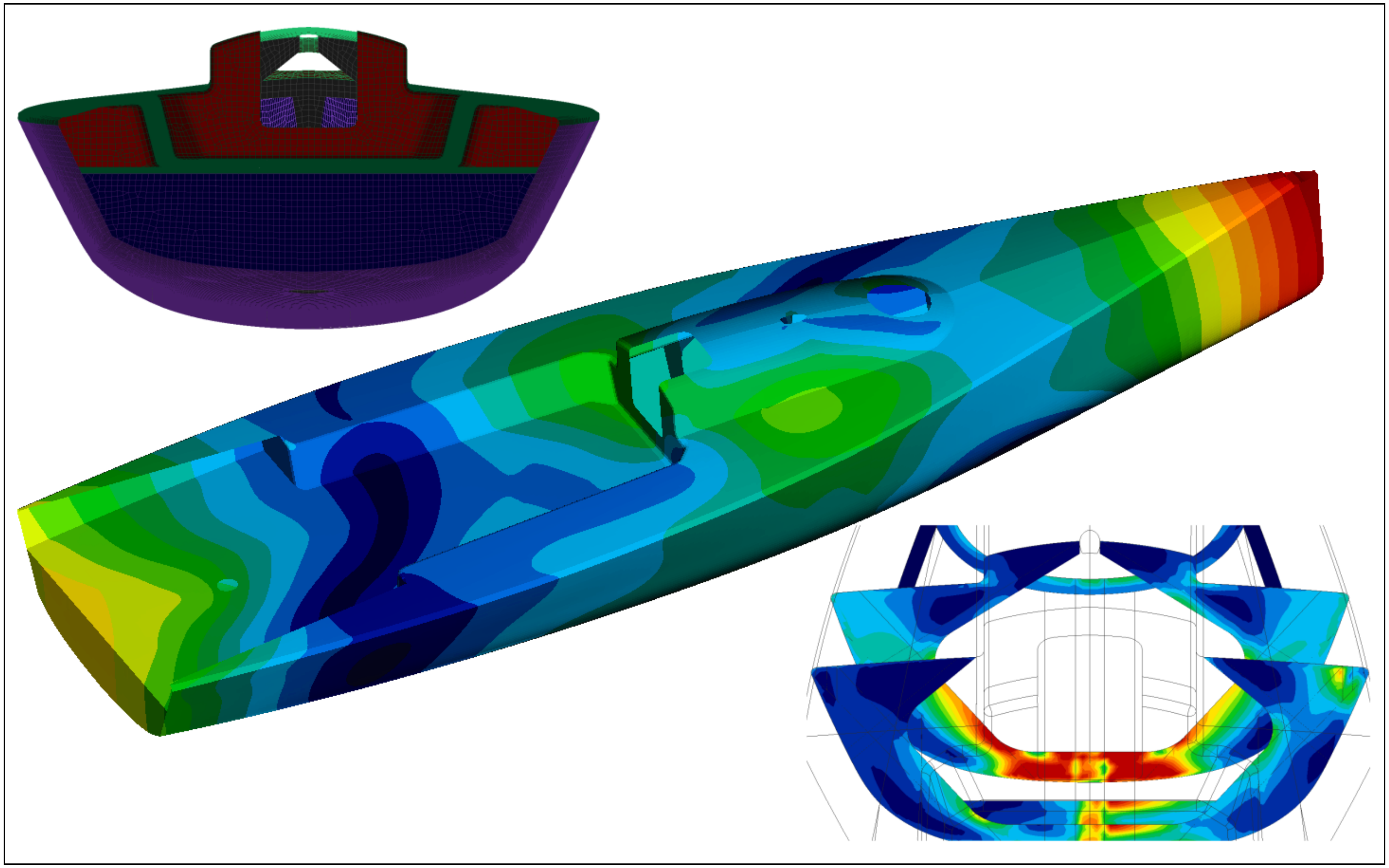
ARISTON TECHNOLOGIES

IBEX 2010 - SESSION 403
FEA FOR STRUCTURAL COMPOSITES OPTIMIZATION



PRESENTED BY:
DAVE FORNARO
PRINCIPAL ENGINEER
ARISTON TECHNOLOGIES, LLC

54 Julia Ct., Kingston, RI 02881 • 401-575-8144 • www.aristontech.com • info@aristontech.com

OUTLINE

1. BASICS OF FINITE ELEMENT ANALYSIS (FEA)
2. FEA COMPARED TO FIRST PRINCIPLES
3. FEA COMPARED TO SCANTLINGS
4. ANATOMY OF A FINITE ELEMENT ANALYSIS
5. TIPS & RECOMMENDATIONS

BASICS OF FINITE ELEMENT ANALYSIS (FEA)

- Finite Element Analysis is an approximate method for studying continuous physical systems, by which the system is broken down into discrete geometric elements interconnected at node points. These finite elements and their interrelationships, along with applied loads and boundary conditions, are represented by a set of simultaneous partial differential equations which are solved using matrix algebra.
- FEA is most useful for systems that are geometrically too complicated for closed-form (textbook) solutions, but it is often benchmarked using systems with known solutions (e.g. beam in bending, column in compression, pressure vessel, etc…)
- FEA can be used to solve problems involving displacement, strain, stress, buckling, vibration, temperature, fluid flow, electro-magnetism and more…
- Solutions can be linear, non-linear, static, dynamic, etc…
- Materials can be isotropic, orthotropic, elastic, plastic, hyperelastic, etc…
- Mesh density highly influences accuracy of results; A quality mesh is essential

BASICS OF FINITE ELEMENT ANALYSIS (FEA), ctd…

- Linear Analysis:
  - Materials have a linear stress-strain relationship (e.g. metals below yield)
  - Deflections are small and vary linearly with applied loads
- Non-Linear Analysis:
  - Materials may have a non-linear stress-strain relationship (e.g. metals beyond yield)
  - Deflections may be large and not always be proportional to the applied loads
- Static Analysis:
  - Loads and boundary conditions do not vary with time
  - Can also be a “snapshot” – i.e. quasi-static approximation of a dynamic system
- Dynamic Analysis:
  - Loads and boundary conditions may vary with time
  - FE solution is an iterative (step-by-step) approximation of a continuous event

BASICS OF FINITE ELEMENT ANALYSIS (FEA), ctd...

- Isotropic Materials:
  - Properties are the same in all directions
  - Defined by two elastic constants – Young's Modulus (E) and Poisson's Ratio (ν)

- Anisotropic Materials:
  - Properties may vary in all directions
  - Defined by 21 elastic constants

- 3D Orthotropic Materials:
  - Two orthogonal planes of symmetry with different properties in each
  - Defined by 9 elastic constants (E1, E2, E3, G12, G13, G23, ν12, ν13, ν23)
  - Applicable to solid composite materials

- 2D Orthotropic Materials:
  - Orthotropic in one plane only, with constant properties normal to that plane
  - Defined by 6 elastic constants (E1, E2, G12, G13, G23, ν12)
  - Applicable to planar composite materials

BASICS OF FINITE ELEMENT ANALYSIS (FEA), ctd…

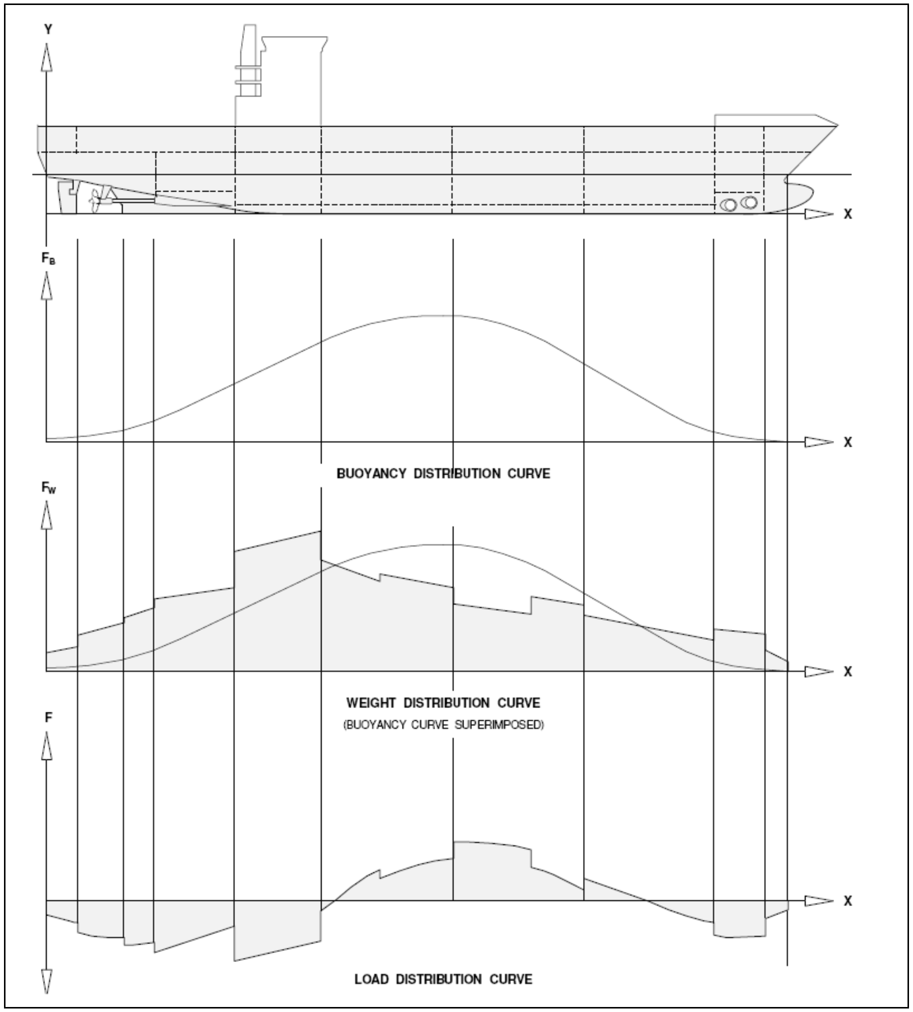
- Majority of yacht design-related composites FEA applications will be linear static analyses using 2D orthotropic properties for laminates (shell elements)
- Non-linear and dynamic analyses are useful for specialized studies (e.g. slamming), although linear "quasi-static" simplifications are often made
- Isotropic properties are generally used for foam core (shell or solid elements)
- 3D orthotropic properties can be used when through-thickness stresses of orthotropic materials are required (e.g. honeycomb core modeled as solid elements)

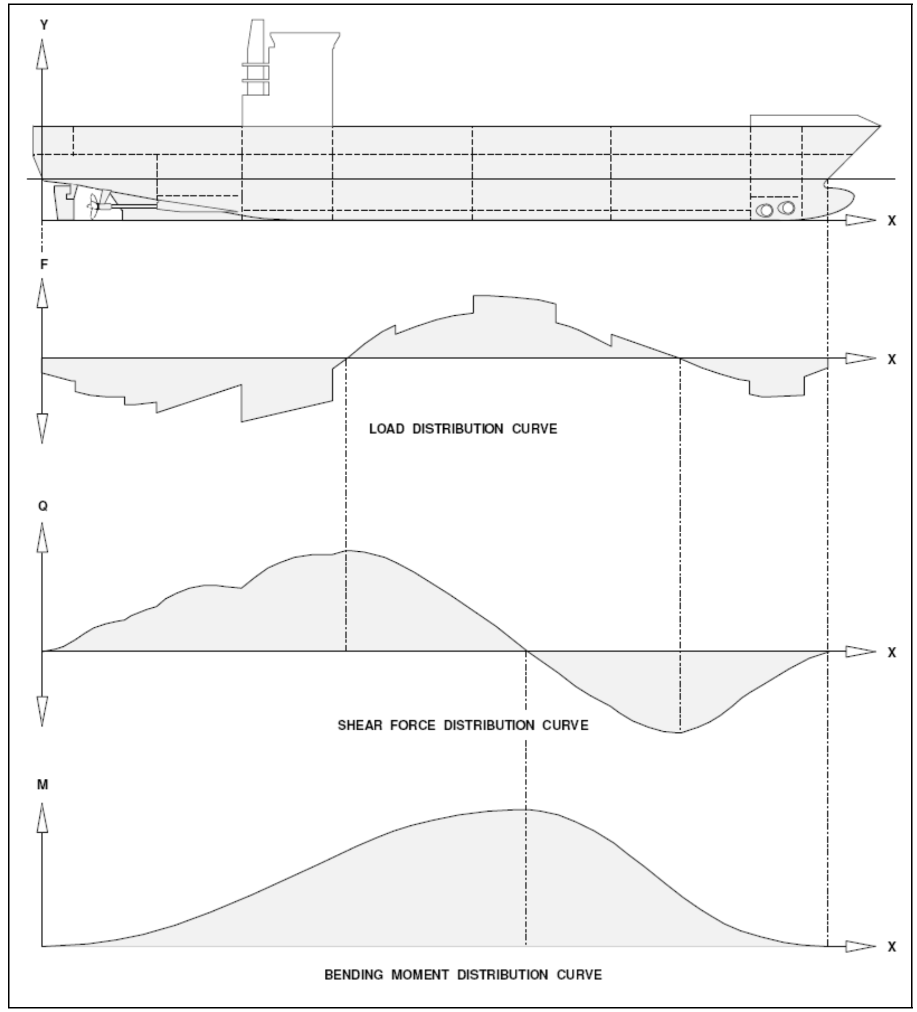
FEA COMPARED TO FIRST PRINCIPLES

- First principles (textbook) analysis methods for simple systems can often be used to estimate the response of more complicated systems (e.g. hull in bending modeled as a beam with variable cross-section and inertia properties)
- First principles methods are commonly used to assess hull scantlings for cargo ships subject to self-weight, cargo weight and buoyancy forces. Calculations are made for shear forces and bending moments, which are used as inputs to scantling rules
- Buoyancy force per unit length, $F_B = \rho g a$
- Weight force per unit length, $F_W = mg$
- Net vertical applied force, $F = F_B - F_W = \rho g a - mg$
- Shear Force, $Q = \int F dx = \int(\rho g a - mg)dx$
- Bending Moment, $M = \int Q = \iint F dx = \iint(\rho g a - mg)dxdx$

IBEX 2010 - SESSION 403
FEA FOR STRUCTURAL COMPOSITES OPTIMIZATION

FEA COMPARED TO FIRST PRINCIPLES, ctd...





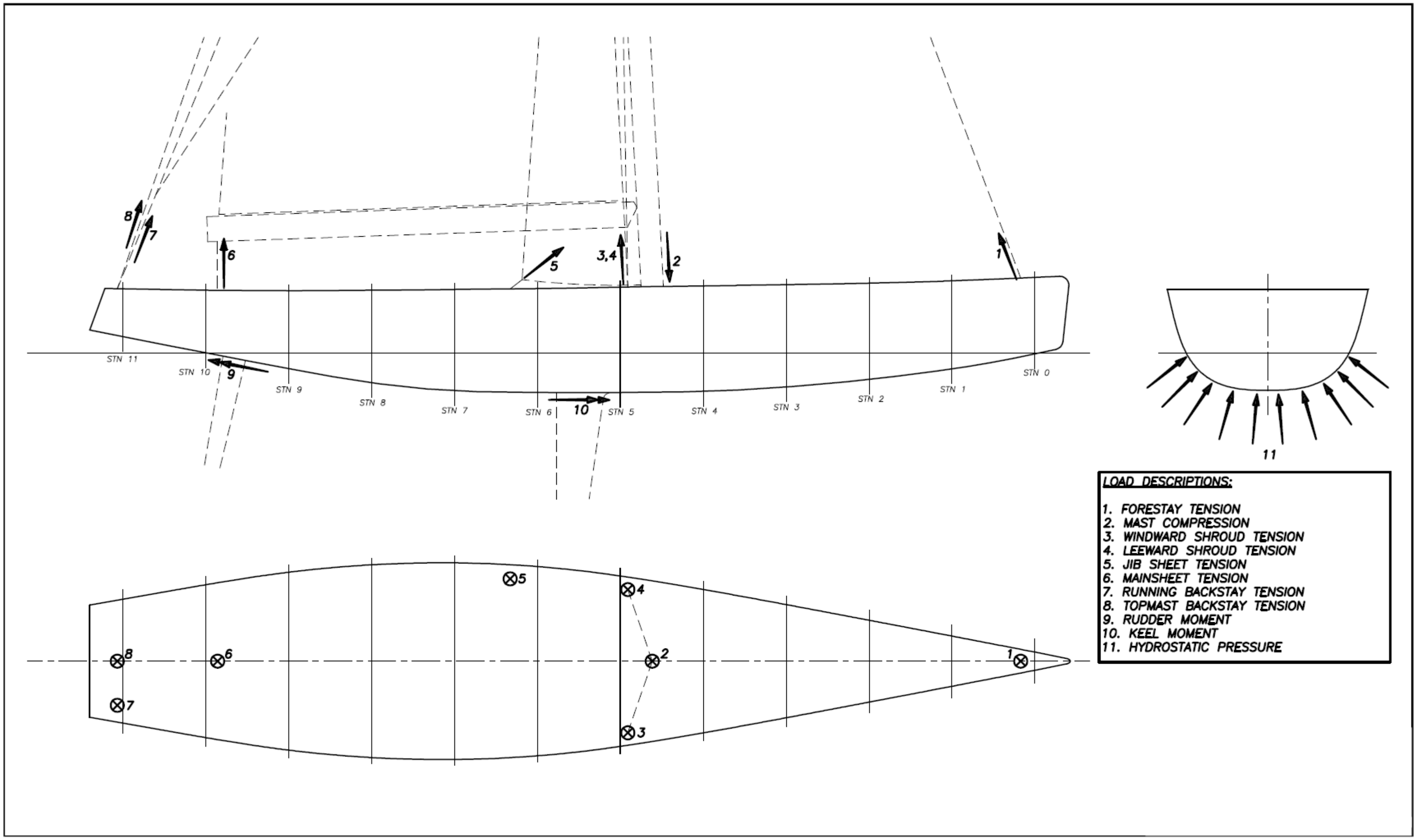
Simplified first-principles global hull strength assessment – load distributions (left), shear force and bending moment curves (right)

FEA COMPARED TO FIRST PRINCIPLES, ctd…

- Note that while this approach is first principles-based, it is not a simple hand calc! While it could be programmed in Excel, this type of analysis is usually carried out using dedicated software such as GHS, DELFTship or HST
- Given the bending moment and shear force, a nominal bending stress at the extreme fiber distance can be calculated according to $\sigma = Mc/I$ and a nominal shear stress calculated according to $\tau = Q/A$
- This would at best be a very rudimentary estimate of global bending stress, and will not address local loads, sectional shape changes, stress concentrations, etc…
- This can be useful for simple cases, but what if you had this…

IBEX 2010 - SESSION 403
FEA FOR STRUCTURAL COMPOSITES OPTIMIZATION

FEA COMPARED TO FIRST PRINCIPLES, ctd...



Load case for upwind sailing – global stress and deflection indeterminate, can't be accurately assessed by first principles

FEA COMPARED TO FIRST PRINCIPLES, ctd...

- First principles calculations can be useful for providing basic information regarding overall sectional shape and strength requirements for simple geometries and loading conditions, either by direct (though approximate) assessment of stress and deflection, or as required input to a scantling rule

- First principles calculations can also be very useful for analyzing local strength of panels, beams, fittings, attachments, etc... although assumptions must be made regarding loading and boundary conditions that compromise accuracy

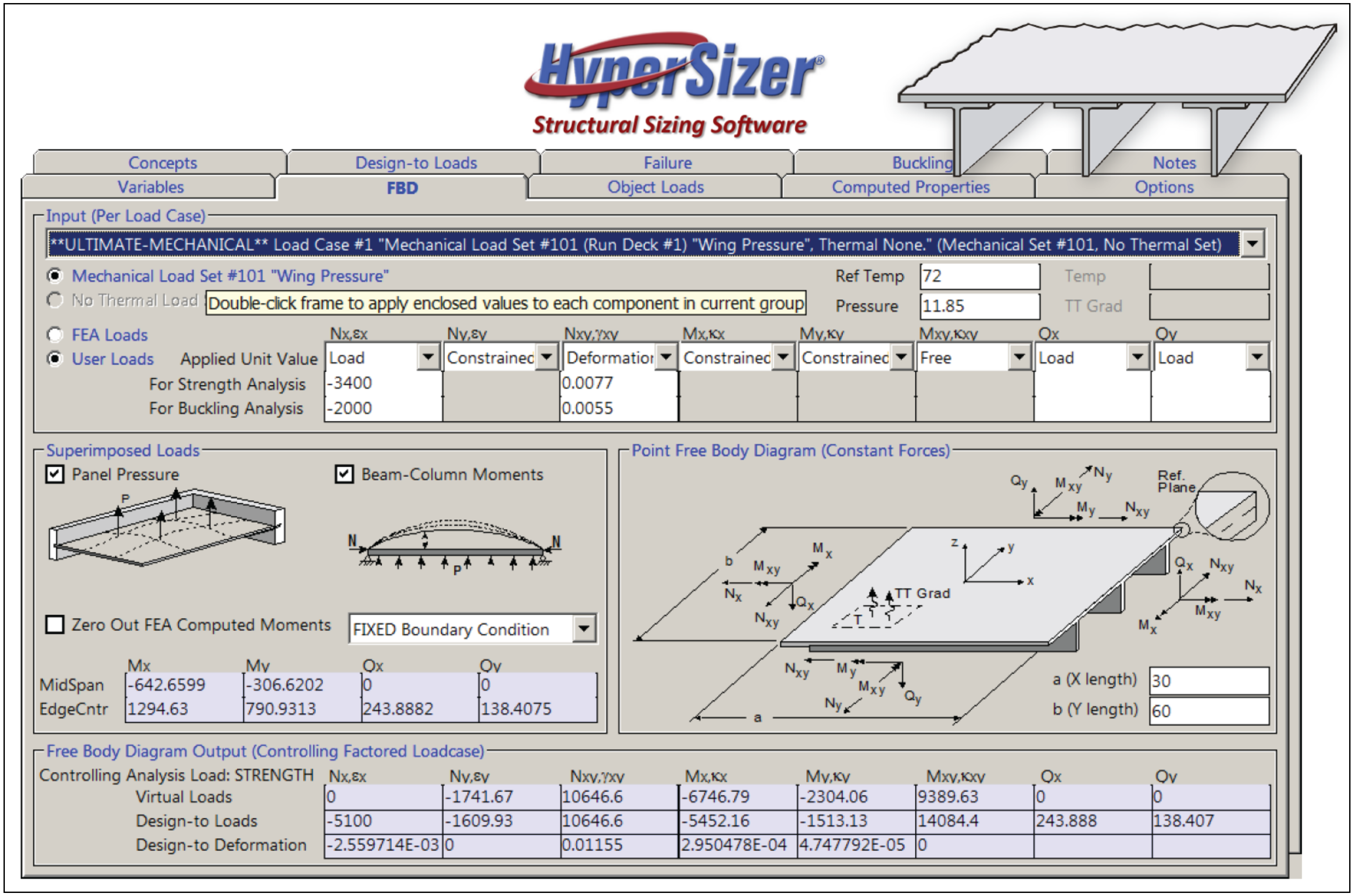
- Many useful first principles-based composites laminate, panel and beam analysis software are available including CoDA, ESAComp, HyperSizer, CompositePro, etc...

- FEA by comparison has the potential to provide close to exact solutions for global deflection and stress given a quality model

- FEA can also provide detailed deflection and stress results for local features such as bulkheads, frames, stringers, panels, fittings and attachments... but only if these features are modeled in sufficient detail

IBEX 2010 - SESSION 403
FEA FOR STRUCTURAL COMPOSITES OPTIMIZATION

FEA COMPARED TO FIRST PRINCIPLES, ctd...



Typical first-principles / classical laminate theory-based software for ply, laminate, beam and panel analysis

FEA COMPARED TO SCANTLINGS

- Scantling rules are generally a combination of first principles calculations and databases of empirical knowledge which have been used to develop acceptable design practices for yachts falling into various categories – e.g. sailing yachts, power yachts, high-speed craft, naval craft, cargo ships, barges, etc…

- As discussed above, first principles calculations for global shear forces and bending moments from hydrostatics programs are often used as inputs to scantling rules. This is particularly true for rules governing cargo ships, less true for those governing recreational craft

- Scantling rules typically include first principles/empirical calculations for:
  - Hull panel pressures based on location & other factors (e.g. dynamics)
  - Bulkhead pressures based on location & other factors (e.g. collision, watertight)
  - Required skin thicknesses, section modulii and moments of inertia based on panel pressure and stiffener spacing, material properties, core thickness, etc…
  - Required core thickness based on type and shear strength requirements
  - Required stiffener spacing, depth, section modulus & moment of inertia

FEA COMPARED TO SCANTLINGS, ctd...

- Scantling rules generally do not provide detailed global or local stresses, although the calculations for section modulus and moment of inertia are often based on approximated first principles calculations for stress and stiffness
- Scantling rule calculations for panels and beams include similar simplifications and assumptions regarding loading and boundary conditions as for first principles-based software
- A good scantling rule will get you in the ballpark, and may provide the means for some degree of basic optimization (e.g. materials, stiffener spacing, core thickness, etc...) but is not an effective tool for detailed optimization of strength or stiffness
- FEA by comparison offers much more detailed understanding of the response of the structure to the applied loads, and far more complete capabilities for studying changes to materials and layout in order to optimize the design

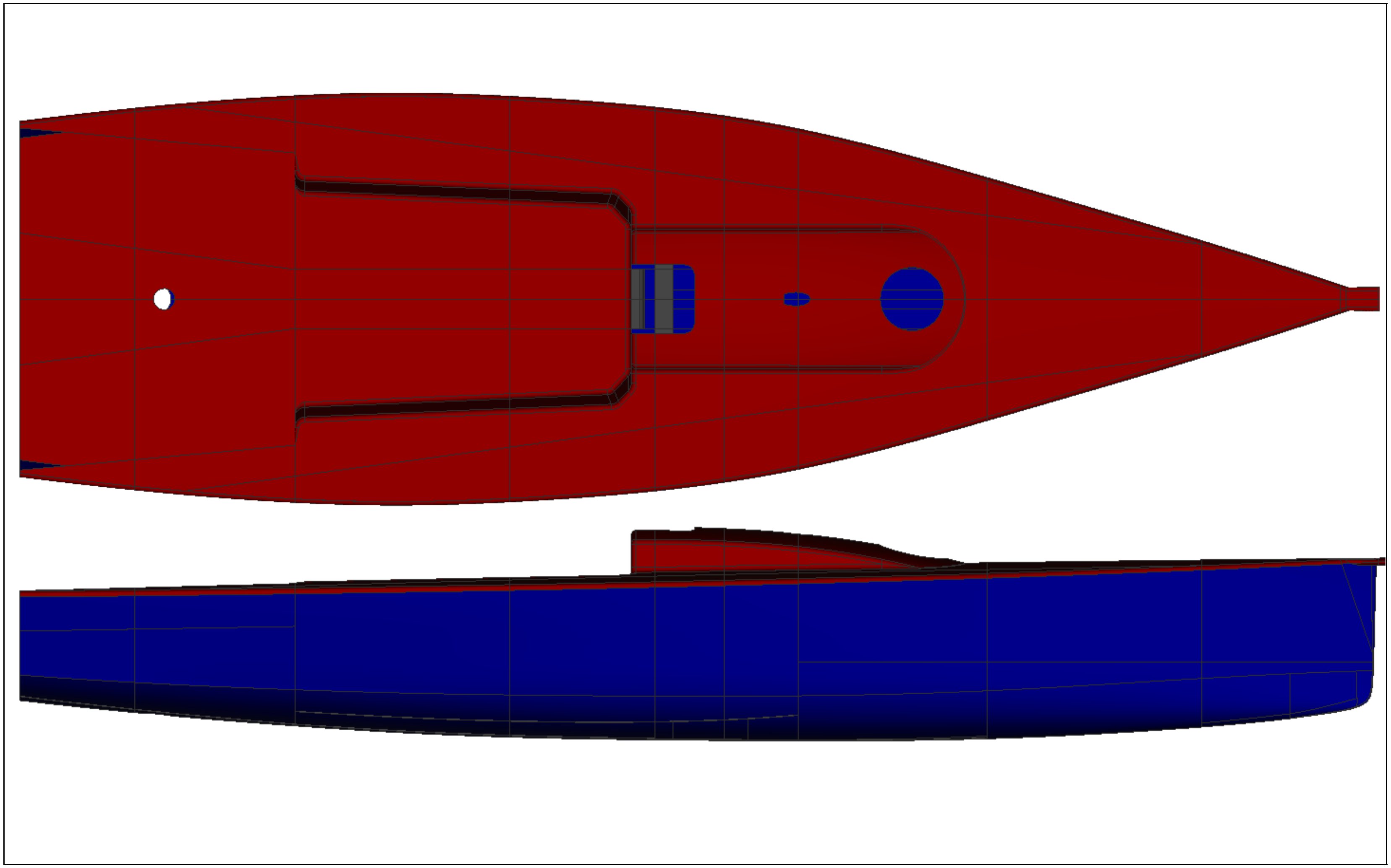
ANATOMY OF A FINITE ELEMENT ANALYSIS

- FEA process illustrated using a model of a 42-foot racing yacht
- Design by Mick Price at Weaver-Price Design & Engineering; Structural engineering & optimization by Dave Fornaro at Ariston Technologies
- Hull modeled in FASTSHIP; Deck & internal structure modeled in PRO/ENGINEER; Pre- & Post-Processing done with FEMAP; Solutions run using NEi/NASTRAN
- Major steps in the process detailed on the following slides:
  - GEOMETRY
  - MESH
  - MATERIAL PROPERTIES
  - LOADS & CONSTRAINTS
  - SOLUTION & RESULTS
  - OPTIMIZATION

ANATOMY OF A FINITE ELEMENT ANALYSIS – GEOMETRY

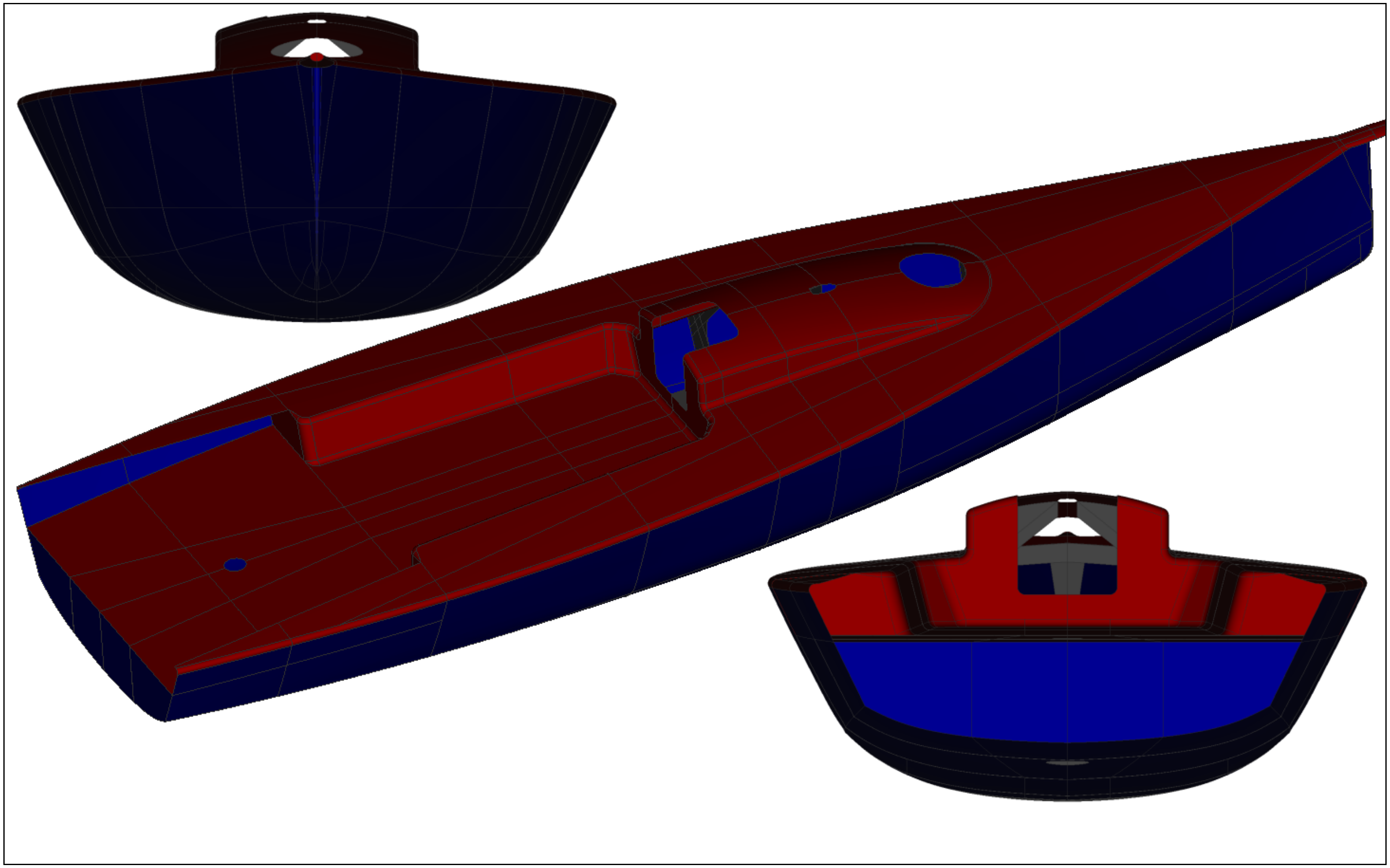
- 3D model of geometry to be analyzed is required
- Surface models preferred over solid models
- Quality surface models avoid lengthy editing or re-work prior to meshing
- Hull & deck modeled at molded (usually outer) surfaces
- Internal frames and stringers modeled at centerline surfaces
- All surfaces split at intersections with other surfaces and at laminate zone extents
- All surfaces must have consistent normal vectors for defining laminate thickness direction (normal vectors ideally defined on surfaces prior to meshing and inherited by elements during the meshing process)

ANATOMY OF A FINITE ELEMENT ANALYSIS – GEOMETRY



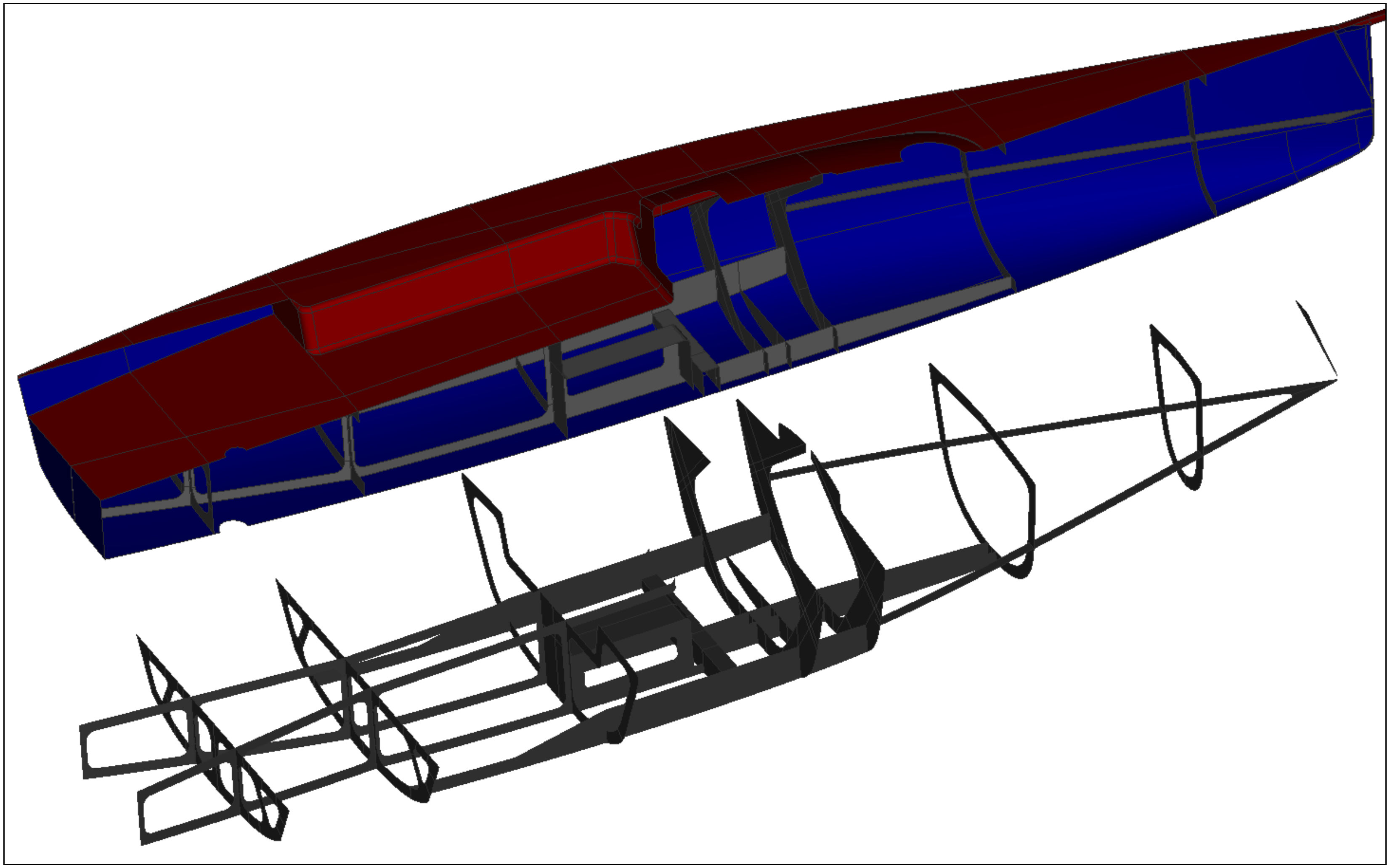
Geometry (surface) model – hull and deck shell

ANATOMY OF A FINITE ELEMENT ANALYSIS – GEOMETRY



Geometry (surface) model – hull and deck shell

ANATOMY OF A FINITE ELEMENT ANALYSIS – GEOMETRY

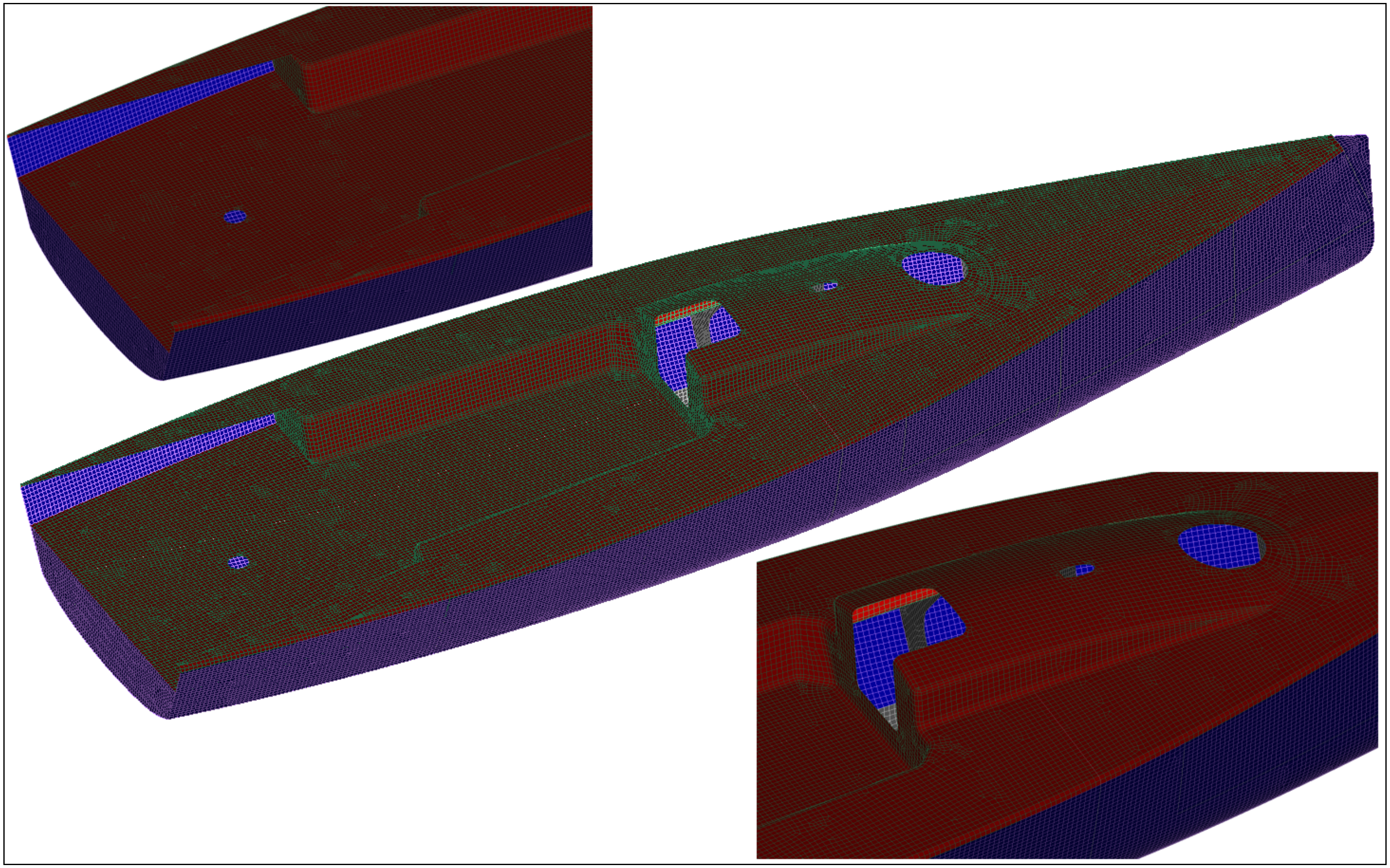


Geometry (surface) model – internal structure

ANATOMY OF A FINITE ELEMENT ANALYSIS – MESH

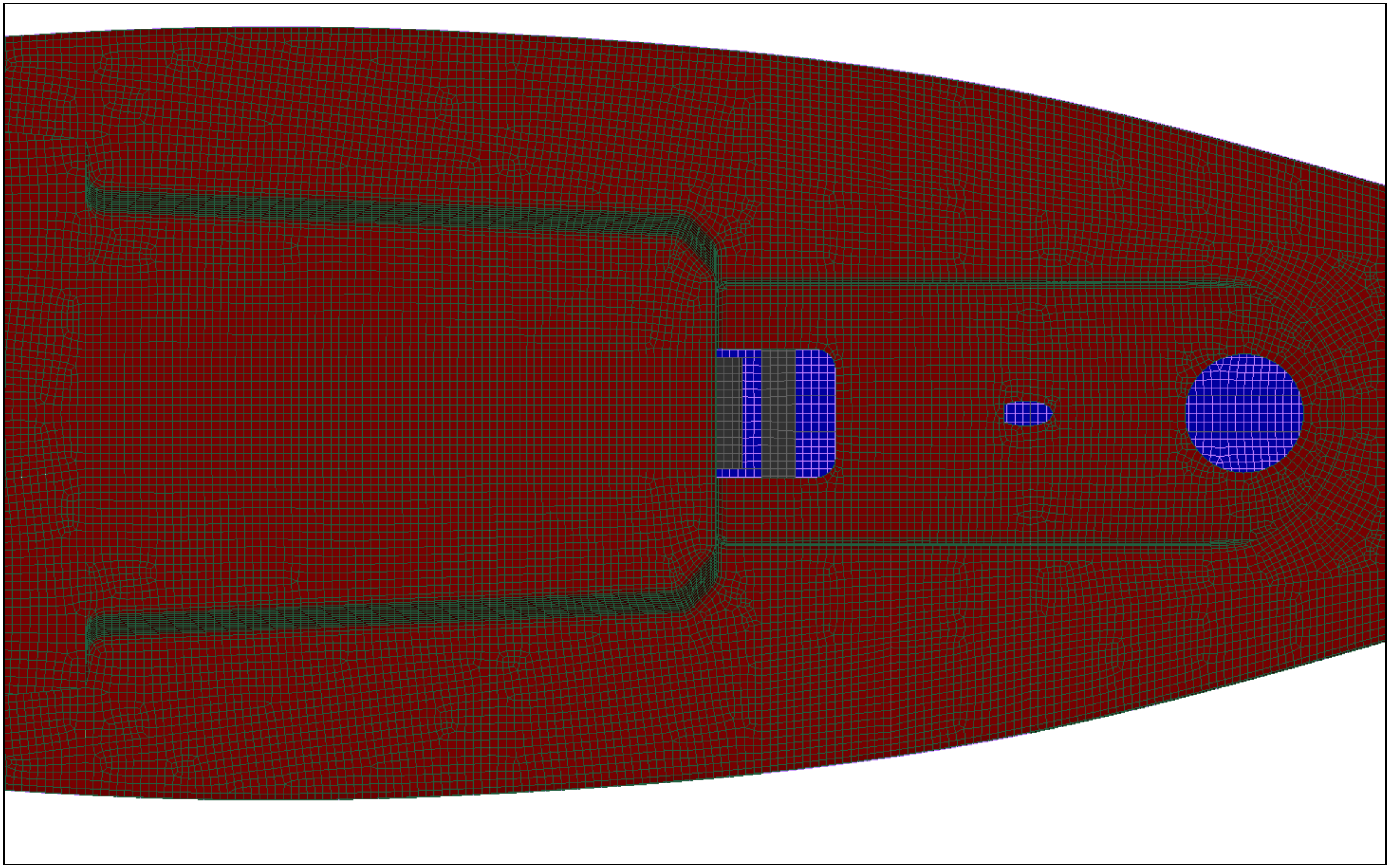
- Auto-meshing pre-processor typically utilized
- All curves and surfaces seeded with mesh controls to produce a quality mesh
- Quadrilateral (4-sided) shells preferred over triangular (3-sided) shells for accuracy
- Mesh should be quad-dominant, but a few tris are unavoidable in transition areas
- Mesh density should be sufficient to replicate geometric shapes with high fidelity
- Mesh should be ordered and structured (look pretty)
- Pre-processor mesh quality controls should be utilized to search for connectivity problems(free edges) as well as warped, skewed or otherwise badly formed elements and any errors should be corrected

ANATOMY OF A FINITE ELEMENT ANALYSIS – MESH



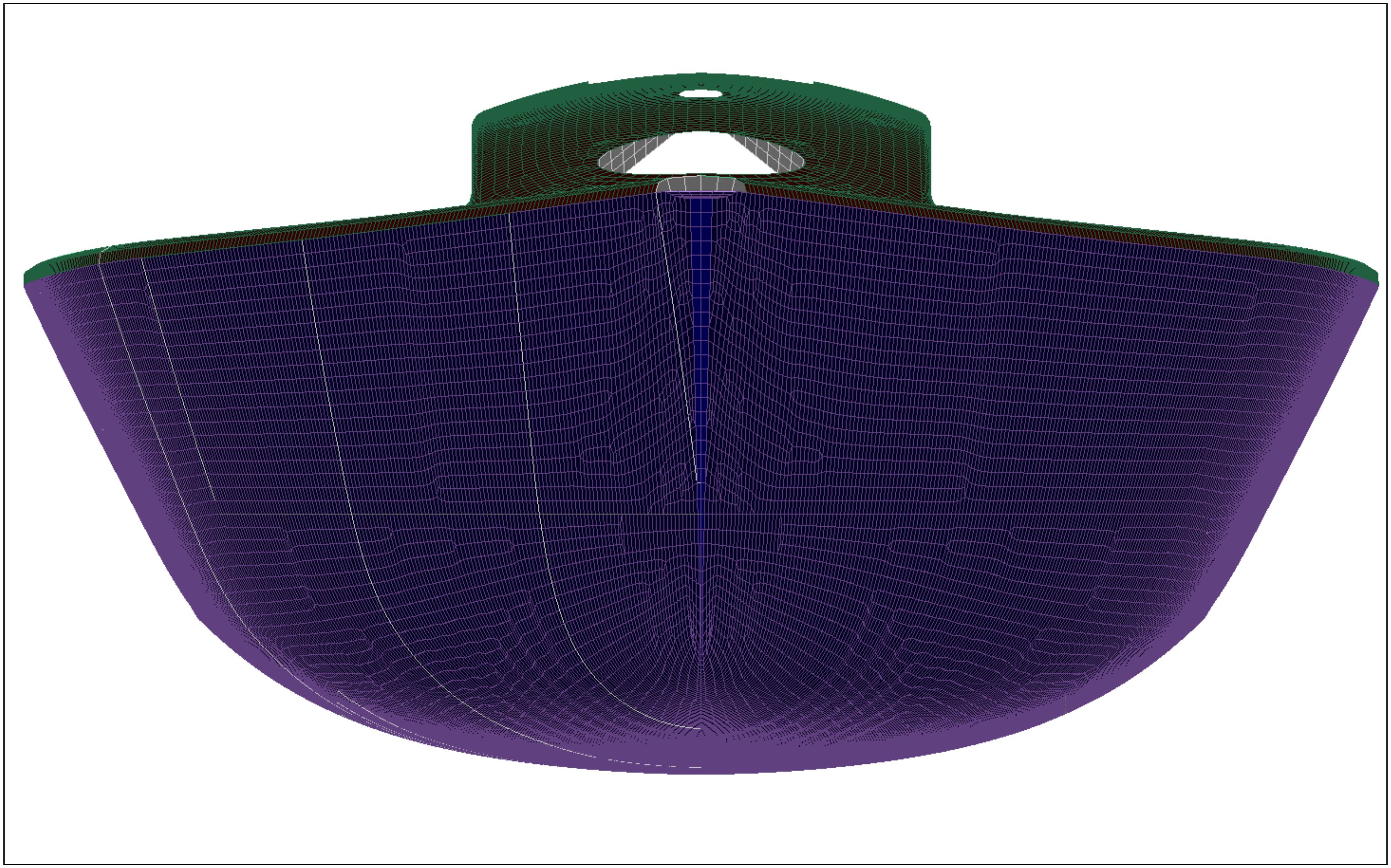
Finite Element Mesh – hull and deck shell

ANATOMY OF A FINITE ELEMENT ANALYSIS – MESH



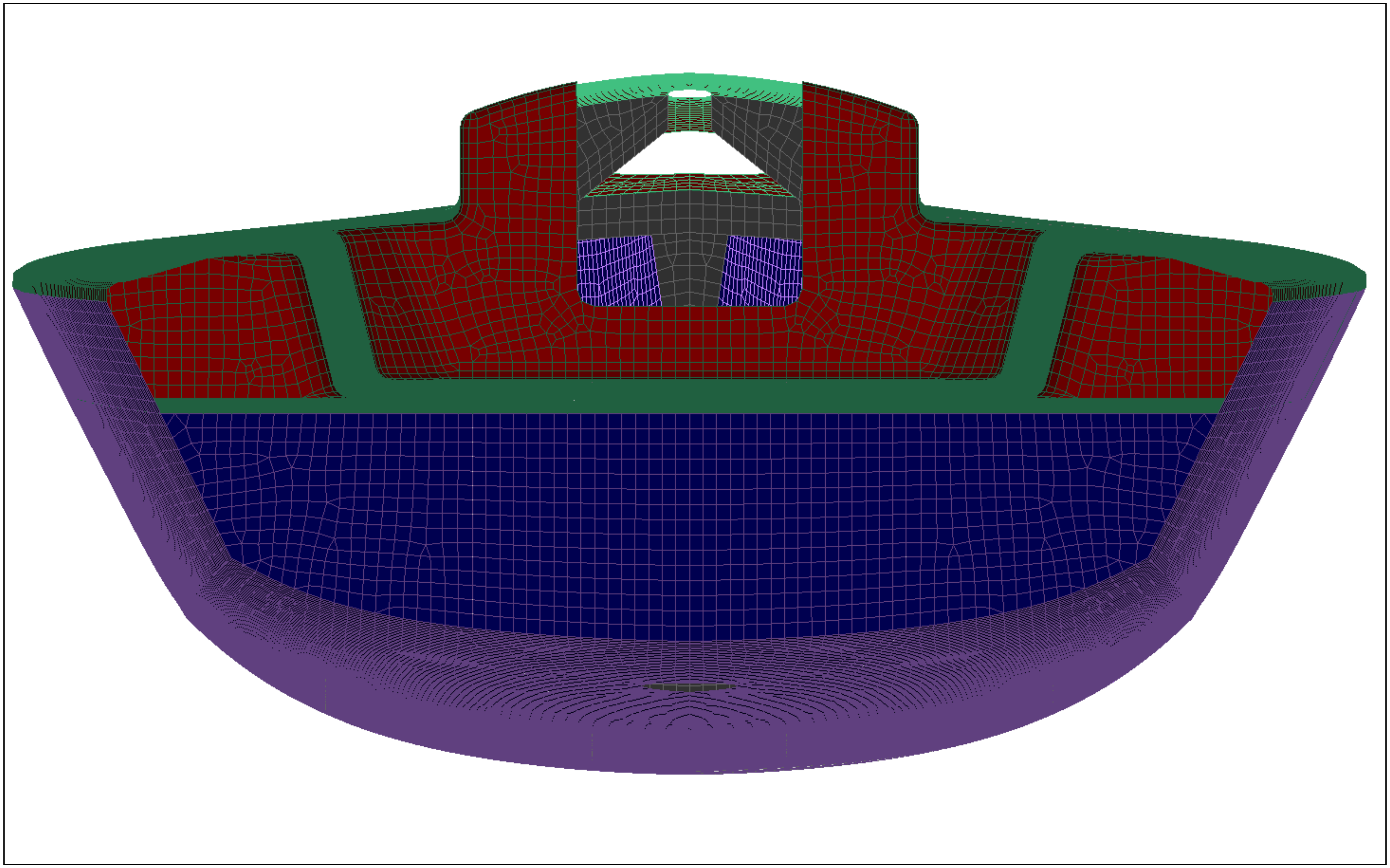
Finite Element Mesh – hull and deck shell

ANATOMY OF A FINITE ELEMENT ANALYSIS – MESH



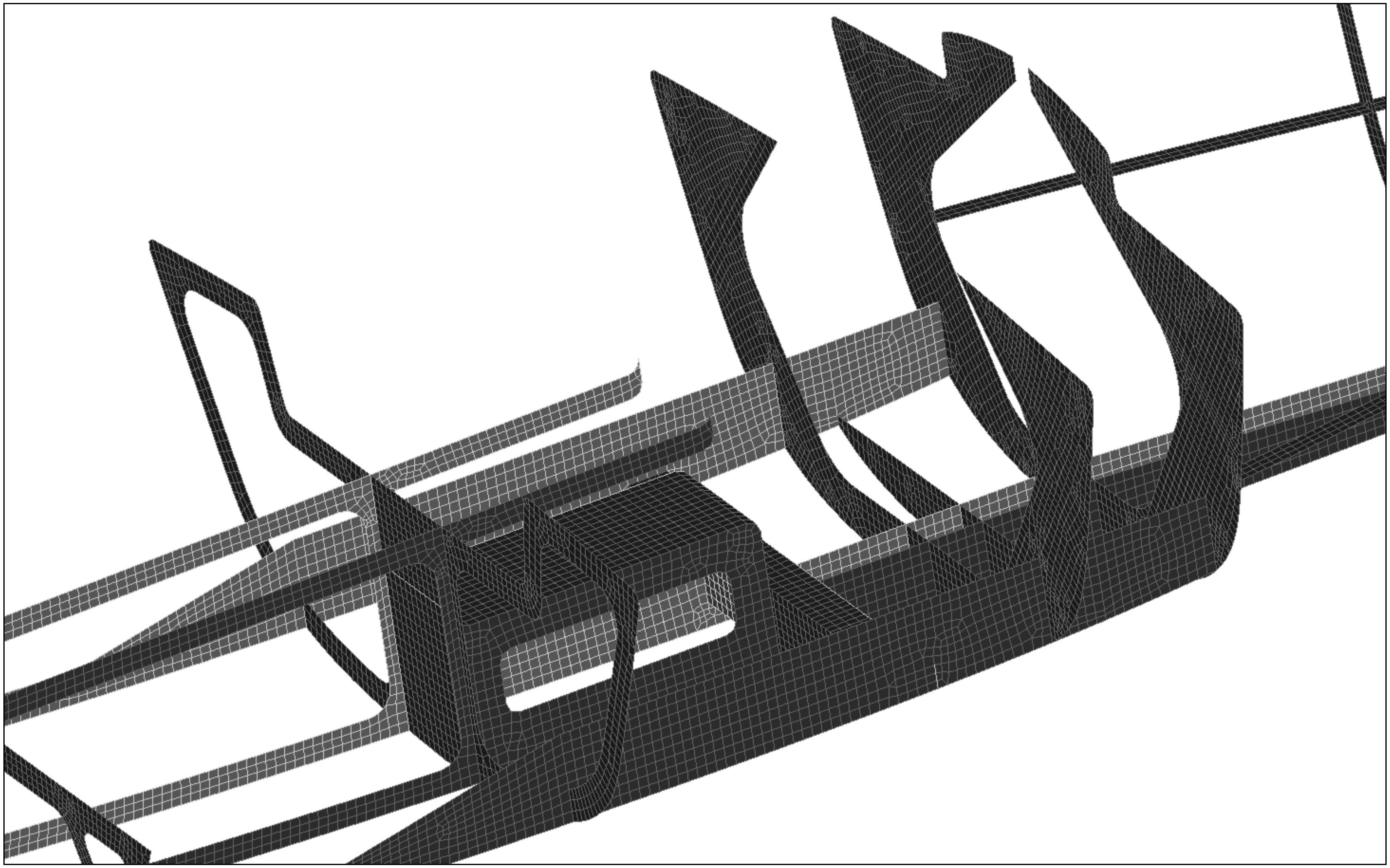
Finite Element Mesh – hull and deck shell

ANATOMY OF A FINITE ELEMENT ANALYSIS – MESH



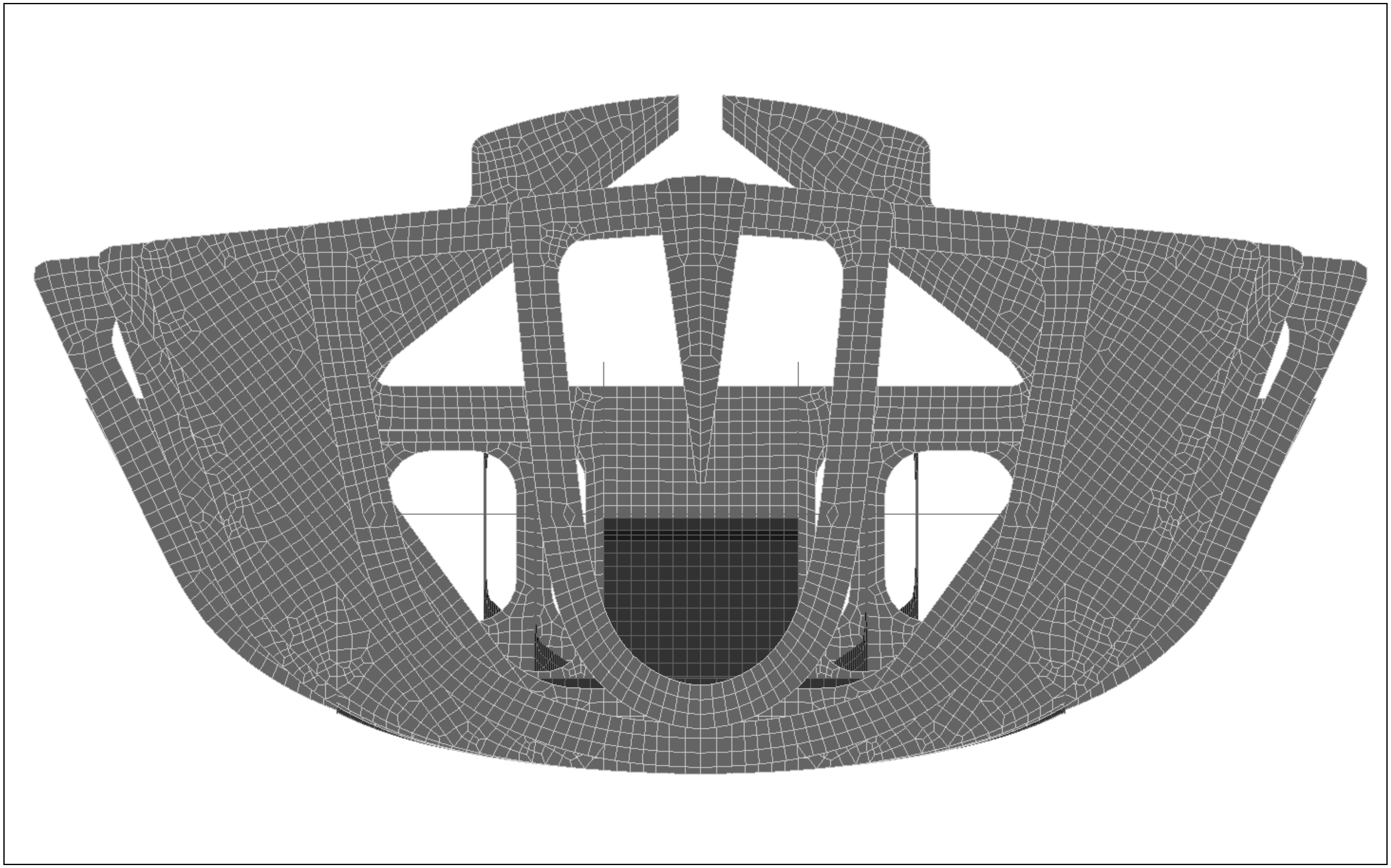
Finite Element Mesh – hull and deck shell

ANATOMY OF A FINITE ELEMENT ANALYSIS – MESH



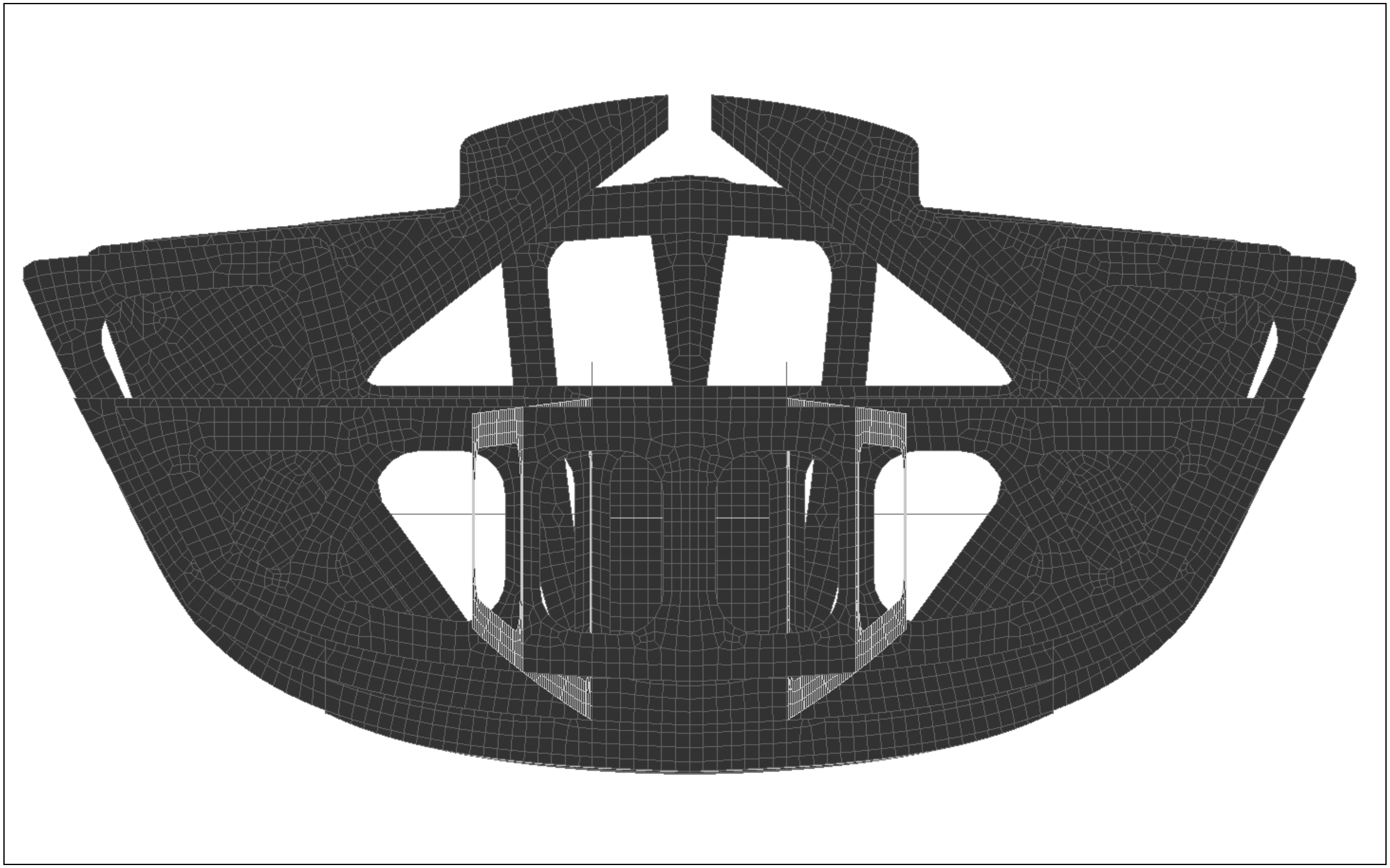
Finite Element Mesh – *internal structure*

ANATOMY OF A FINITE ELEMENT ANALYSIS – MESH



Finite Element Mesh – internal structure

ANATOMY OF A FINITE ELEMENT ANALYSIS – MESH



Finite Element Mesh – internal structure

ANATOMY OF A FINITE ELEMENT ANALYSIS – MATERIAL PROPERTIES

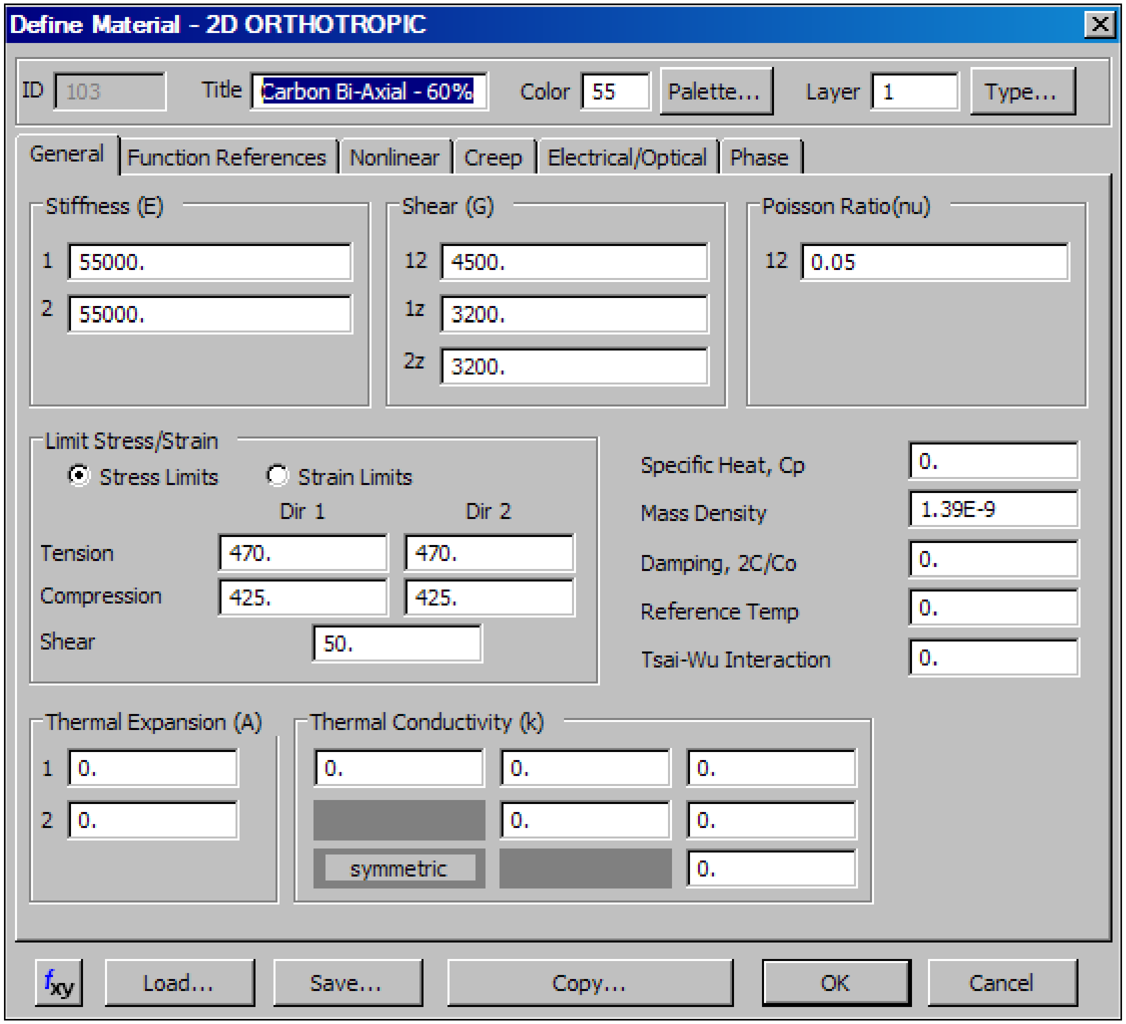
- Individual plies modeled as 2D orthotropic materials defined by 6 elastic constants required to run an analysis:
  - E1 = Elastic Modulus in 1-fiber (parallel) direction
  - E2= Elastic Modulus in 2-fiber (transverse) direction
  - G12 = Shear modulus in 12-plane
  - G13 = Through thickness modulus in 13-plane (often taken as resin modulus)
  - G23 = Through thickness modulus in 23-plane (often taken as resin modulus)
  - ν12 = Poisson's Ratio in 12-plane

- Strength properties required only for failure index calculation:
  - σ1t = Tensile Strength in 1-fiber (parallel) direction
  - σ1c = Compressive Strength in 1-fiber (parallel) direction
  - σ2t = Tensile Strength in 2-fiber (transverse) direction
  - σ2c = Compressive Strength in 2-fiber (transverse) direction
  - τ12 = Shear strength in 12-plane

ANATOMY OF A FINITE ELEMENT ANALYSIS – MATERIAL PROPERTIES

- Ply data can be taken from vendor testing, in-house testing, standards; Can be difficult to get good data (esp. for shear) – estimates sometimes have to be made
- All elements must have normal direction defined, relative to which laminate stack thickness is developed
- All elements must have material angle defined, relative to which fiber directions are oriented (methods include vector projection, draping)
- Pre-processor quality control tools such as vector displays and backface shading should be used to confirm that all normal vectors and material angles are correct; If these are wrong, the analysis will be meaningless
- Laminates are built-up using pre-processor functionality to assemble plies at correct angles relative to material angle for each element; Large models (up to 150 laminates) require good housekeeping to create, manage and update laminates; Can integrate with Excel or other proprietary interfaces for managing plies and laminates

ANATOMY OF A FINITE ELEMENT ANALYSIS – MATERIAL PROPERTIES

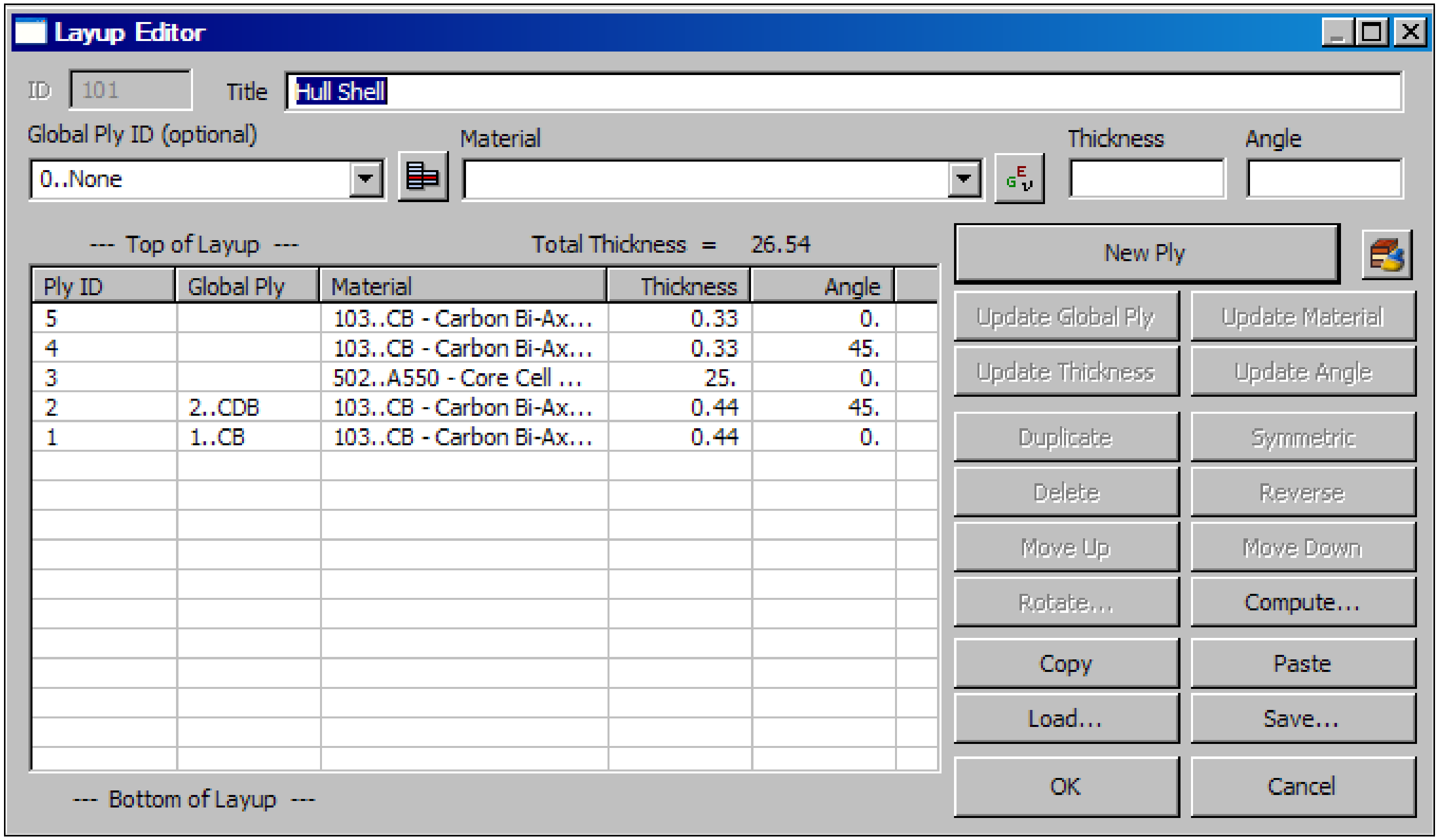


Typical ply properties input – Biaxial carbon fiber at 60% fiber weight fraction

ARISTON TECHNOLOGIES

IBEX 2010 - SESSION 403
FEA FOR STRUCTURAL COMPOSITES OPTIMIZATION

ANATOMY OF A FINITE ELEMENT ANALYSIS – MATERIAL PROPERTIES



Typical hull shell laminate stack build-up – Biaxial carbon fiber skins alternating 0/90 and +/-45;
Dry fiber weights – Outer skin 800gm/m2 (24oz/yd2); Inner skin 600gm/m2 (18oz/yd2);
Corecell A550 foam core, 25mm thickness

Note: Similar laminates defined for deck & internal structure; Highly detailed models with many reinforced areas designed to optimize strength and stiffness can have up to 150 laminate zones

54 Julia Ct., Kingston, RI 02881 • 401-575-8144 • www.aristontech.com • info@aristontech.com

ANATOMY OF A FINITE ELEMENT ANALYSIS – MATERIAL PROPERTIES

HEXCEL — TYPICAL MECHANICAL VALUES ON EPOXY PREPREG LAMINATES

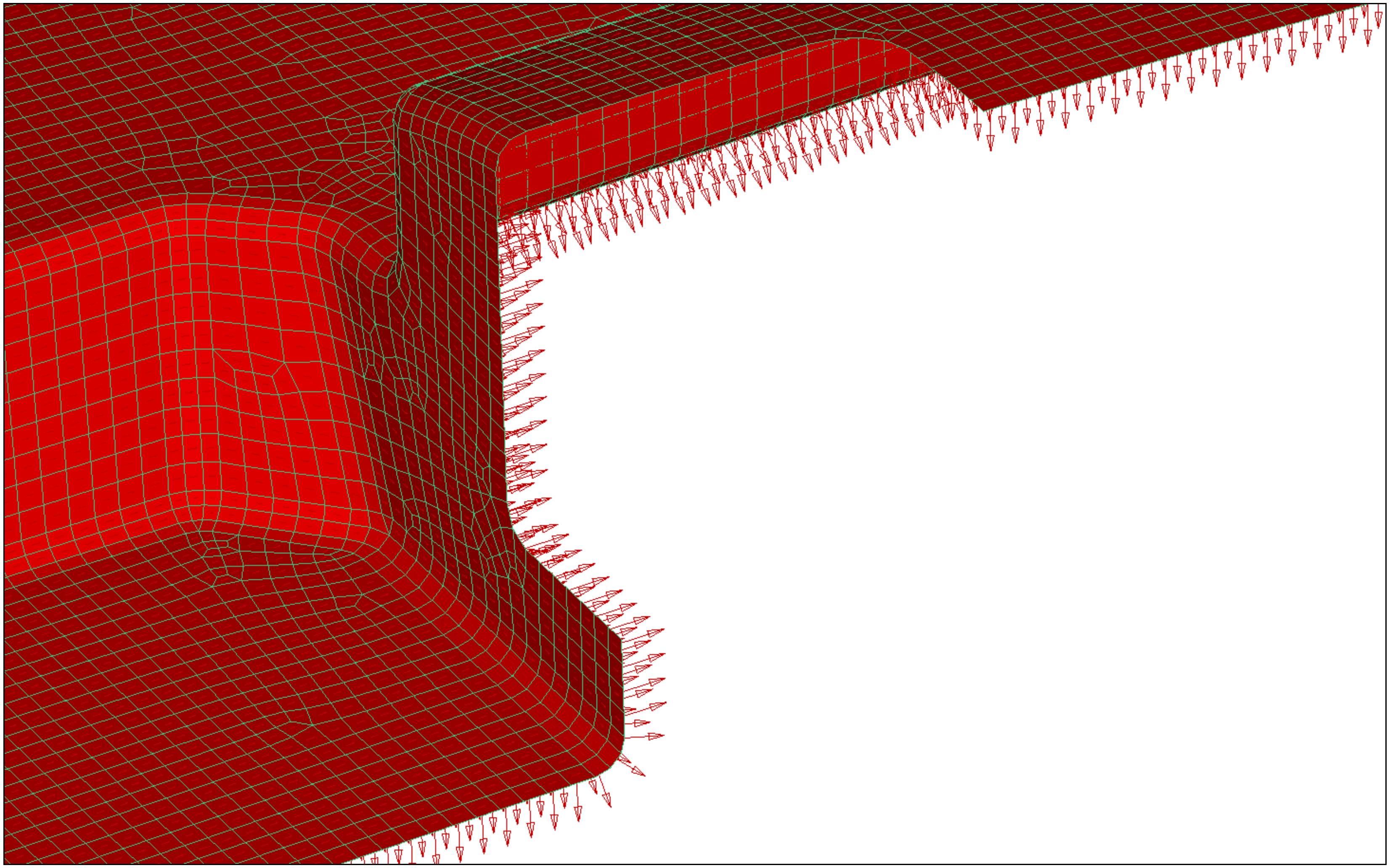
t 90°, 0°

Volume content of fibres :
≈ 60 % (Carbon)
≈ 50 % (E-glass - Aramid)

			UNITS	E-GLASS UD	E-GLASS Fabric	ARAMID UD	ARAMID Fabric	HIGH STRENGTH CARBON UD	HIGH STRENGTH CARBON Fabric	INTERMEDIATE MODULUS CARBON UD	INTERMEDIATE MODULUS CARBON Fabric
Tensile	σ /	III	MPa	1100	600	1100	500	2000	800	2400	900
	σt	≡	MPa	35	550	35	450	80	750	80	850
	E /	III	GPa	43	20	60	30	130	70	170	90
	Et	≡	GPa	8	19	8	30	9	65	9	90
	Poisson's ratio υ			0.28	0.13	0.34	0.2	0.25	0.05	0.27	0.05
Compression	σ /	III	MPa	900	550	250	150	1300	700	1600	800
	σt	≡	MPa	150	500	150	150	250	650	250	750
	E /	III	GPa	42	17	75	31	115	60	150	80
	Et	≡	GPa	10	16	5.5	30	10	55	11	75
Flexure	σ /	III	MPa	1200	700	550	400	1800	1000	1400	1200
	E /	III	GPa	42	20	40	25	120	65	140	75
In-plane shear	σ		MPa	60	55	45	40	95	80	95	80
	G		GPa	4	4.2	2.1	4	4.4	5.5	4.4	5
Interlaminar shear	σ	III	MPa	75	50	60	50	80	70	80	70

Generic ply properties for e-glass, Kevlar, carbon – use only if test/mfr data unavailable

ANATOMY OF A FINITE ELEMENT ANALYSIS – MATERIAL PROPERTIES

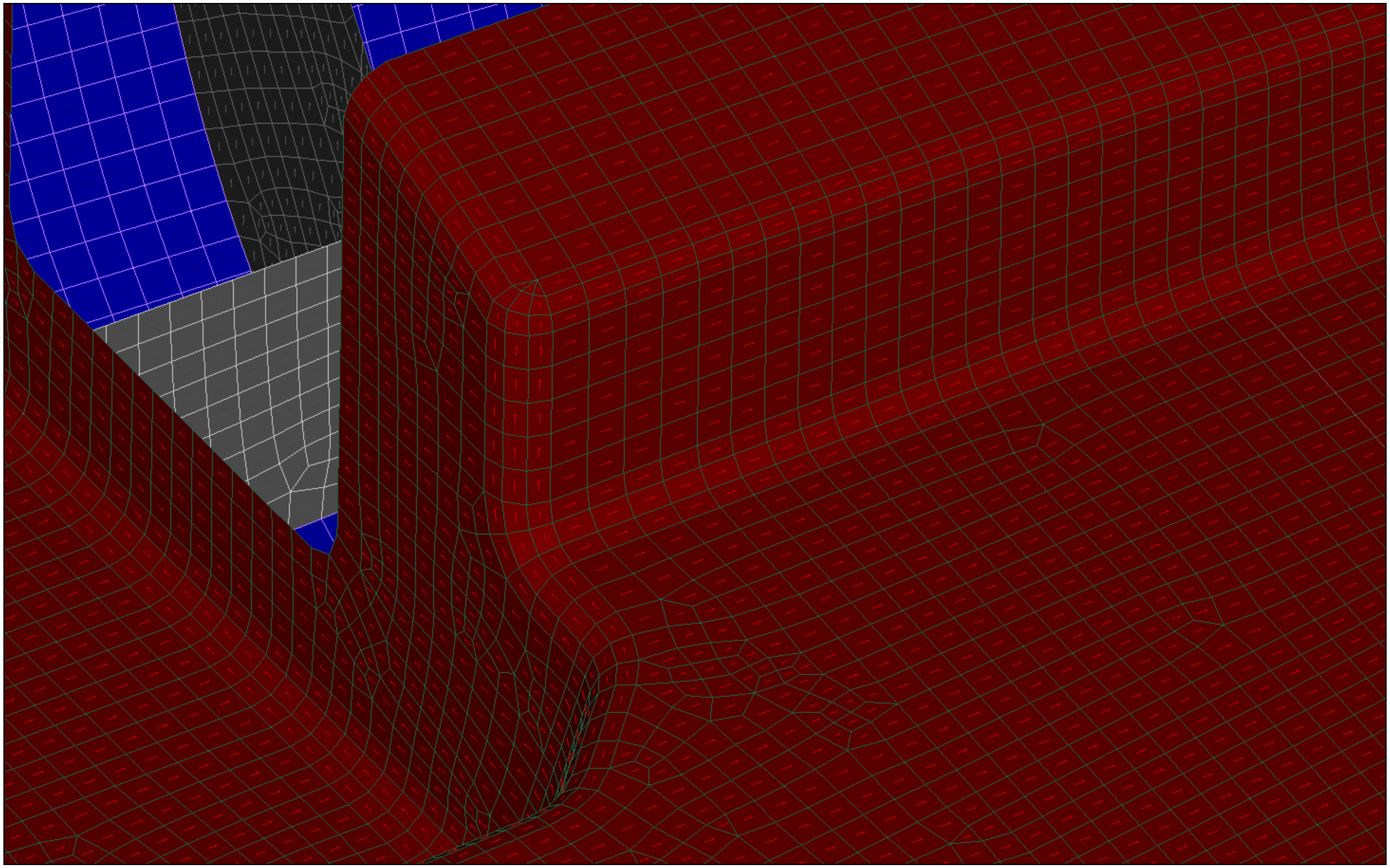


Vectors indicating shell element normal directions (must be consistent to yield correct thickness offset)

ARISTON TECHNOLOGIES

IBEX 2010 - SESSION 403
FEA FOR STRUCTURAL COMPOSITES OPTIMIZATION

ANATOMY OF A FINITE ELEMENT ANALYSIS – MATERIAL PROPERTIES



Vectors indicating shell element material orientation (must be carefully defined relative to ply/laminate properties)

ANATOMY OF A FINITE ELEMENT ANALYSIS – LOADS & CONSTRAINTS

- Accurately modeling realistic load cases is critical to successfully utilizing FEA for structural composites optimization

- Typical load cases can be operating loads or limit loads; Required safety factors and other acceptance criteria should be developed prior to beginning the analysis

- Global load cases representing steady-state operation should be as close to fully force and moment balanced as possible; This can often require input from various sources (CFD, VPP, Hand-calcs, rule-of-thumb); Whatever the source(s) of the load data, the complete load picture should be balanced (or as close as possible)

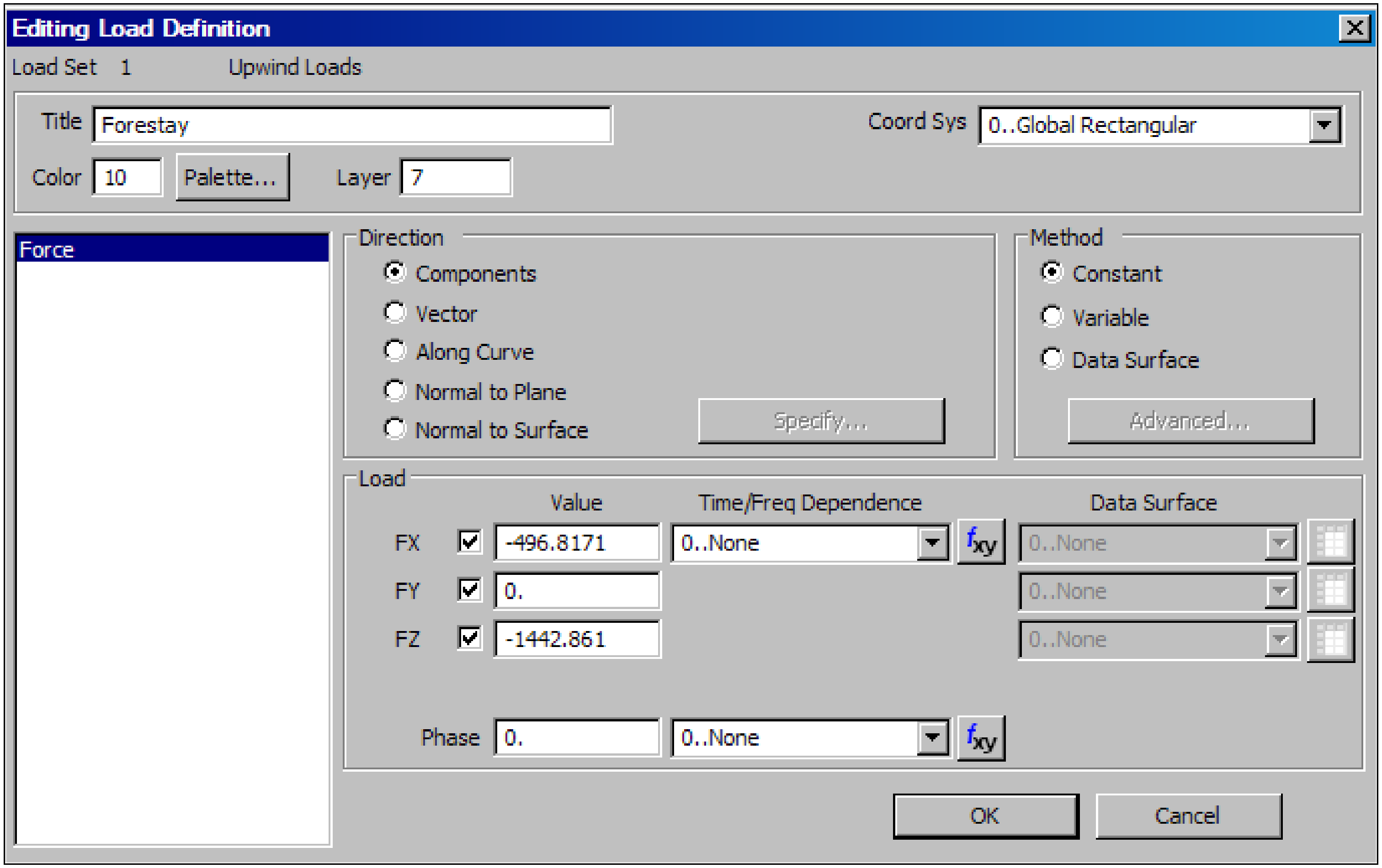
- Artificial constraints should be avoided if possible; Floating structures are inherently unconstrained and should be modeled as such for best results; Inertia relief can handle small residuals but should not be a crutch for poor load case development

- Details can be studied by extracting local models from the global model complete with internal loads and boundary displacements; These models can then be refined to study details more closely as long as the overall stiffness is not changed substantially

ANATOMY OF A FINITE ELEMENT ANALYSIS – LOADS & CONSTRAINTS

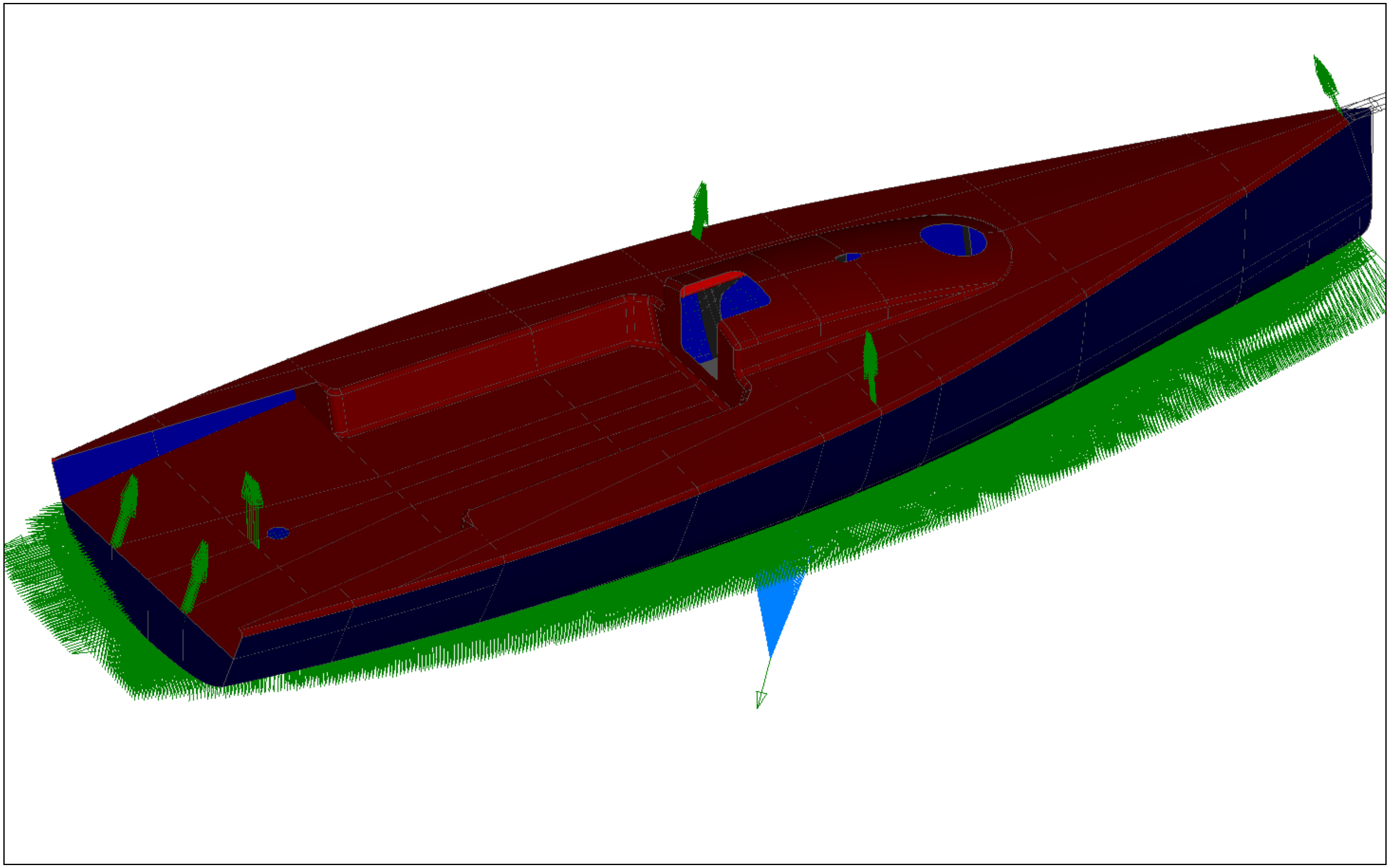
- Steady-state, upwind sailing, 20 degrees heel, typical operating condition
- Loads include:
  - Forestay
  - Backstay
  - Windward V1/D1 shrouds
  - Leeward V1/D1 shrouds
  - Mast compression
  - Mainsheet
  - Jib sheet
  - Keel
  - Rudder
  - Hydrostatic pressure

ANATOMY OF A FINITE ELEMENT ANALYSIS – LOADS & CONSTRAINTS



Typical load input (per node) – forestay load components in x- (aft) and z- (up) directions

ANATOMY OF A FINITE ELEMENT ANALYSIS – LOADS & CONSTRAINTS



Vectors indicating various loads applied throughout the model

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS

- Results parameters include various metrics for deflection, stress, strain and failure index; In particular for composites there are many components of stress/strain that can be studied for both the laminate as a whole as well as for each ply

- In cases where certain principal stresses dominate, then direct comparison to test results for the equivalent strength values can be made; Studying principal strain vectors can give good clues as to the flow of strain in the laminate which can help with determining the orientation of reinforcements

- More often, the state of stress is multi-axial and too complicated to be simply compared to one or more principal strength values; In this case it is more appropriate to utilize one of several failure indices

- Failure Indices are mathematical models that predict failure based on the combination of maximum principal, minimum principal and maximum shear stresses in each ply, relative to the respective strength values for the ply

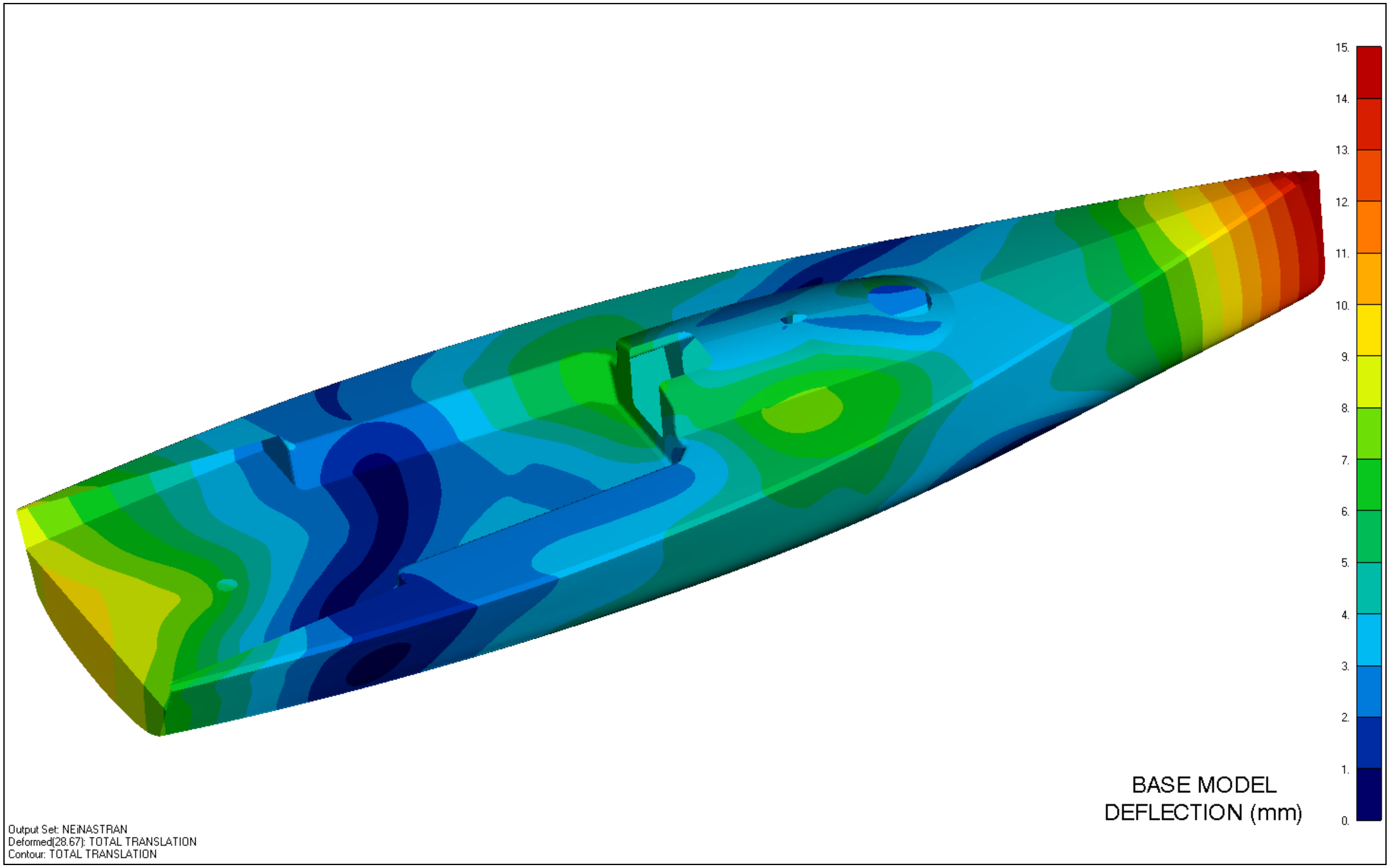
- Different failure indices are appropriate for different types of laminates; Need to be sure to choose the appropriate failure index for the application being studied

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS

Theory	Failure Index	Remarks
Hill	$\frac{\sigma_1^2}{x^2} - \frac{\sigma_1\sigma_2}{x^2} + \frac{\sigma_2^2}{y^2} + \frac{\tau_{12}^2}{s^2} = \text{F.I.}$	Orthotropic materials with equal strengths in tension and compression.
Hoffman	$\left(\frac{1}{x_t} - \frac{1}{x_c}\right)\sigma_1 + \left(\frac{1}{y_t} - \frac{1}{y_c}\right)\sigma_2 + \frac{\sigma_1^2}{x_t x_c} + \frac{\sigma_2^2}{y_t y_c} + \frac{\tau_{12}^2}{s^2} - \frac{\sigma_1\sigma_2}{x_t x_c} = \text{F.I.}$	Orthotropic materials under a general state of plane stress with unequal tensile and compressive strengths.
Tsai-Wu	$\left(\frac{1}{x_t} - \frac{1}{x_c}\right)\sigma_1 + \left(\frac{1}{y_t} - \frac{1}{y_c}\right)\sigma_2 + \frac{\sigma_1^2}{x_t x_c} + \frac{\sigma_2^2}{y_t y_c} + \frac{\tau_{12}^2}{s^2} + 2F_{12}\sigma_1\sigma_2 = \text{F.I.}$	Orthotropic materials under a general state of plane stress with unequal tensile and compressive strengths.
LaRC02	See the NEiNastran User's Manual, Reference 5.	Orthotropic materials comprised of unidirectional plies under a general state of plane stress.
Puck	See the NEiNastran User's Manual, References 12 and 13.	Orthotropic materials comprised of unidirectional plies under a general state of plane stress.
Max Stress	$\text{Max}\left[\left(\frac{\sigma_1}{X_t}\right), \left(\frac{\sigma_2}{Y_t}\right), \left(\frac{\lvert\tau_{12}\rvert}{S}\right)\right]$	None
Max Strain	$\text{Max}\left[\left(\frac{\varepsilon_1}{X_t}\right), \left(\frac{\varepsilon_2}{Y_t}\right), \left(\frac{\lvert\gamma_{12}\rvert}{S}\right)\right]$	None

Mathematical definitions for various orthotropic material failure theories (NEi/NASTRAN Reference Manual)

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS

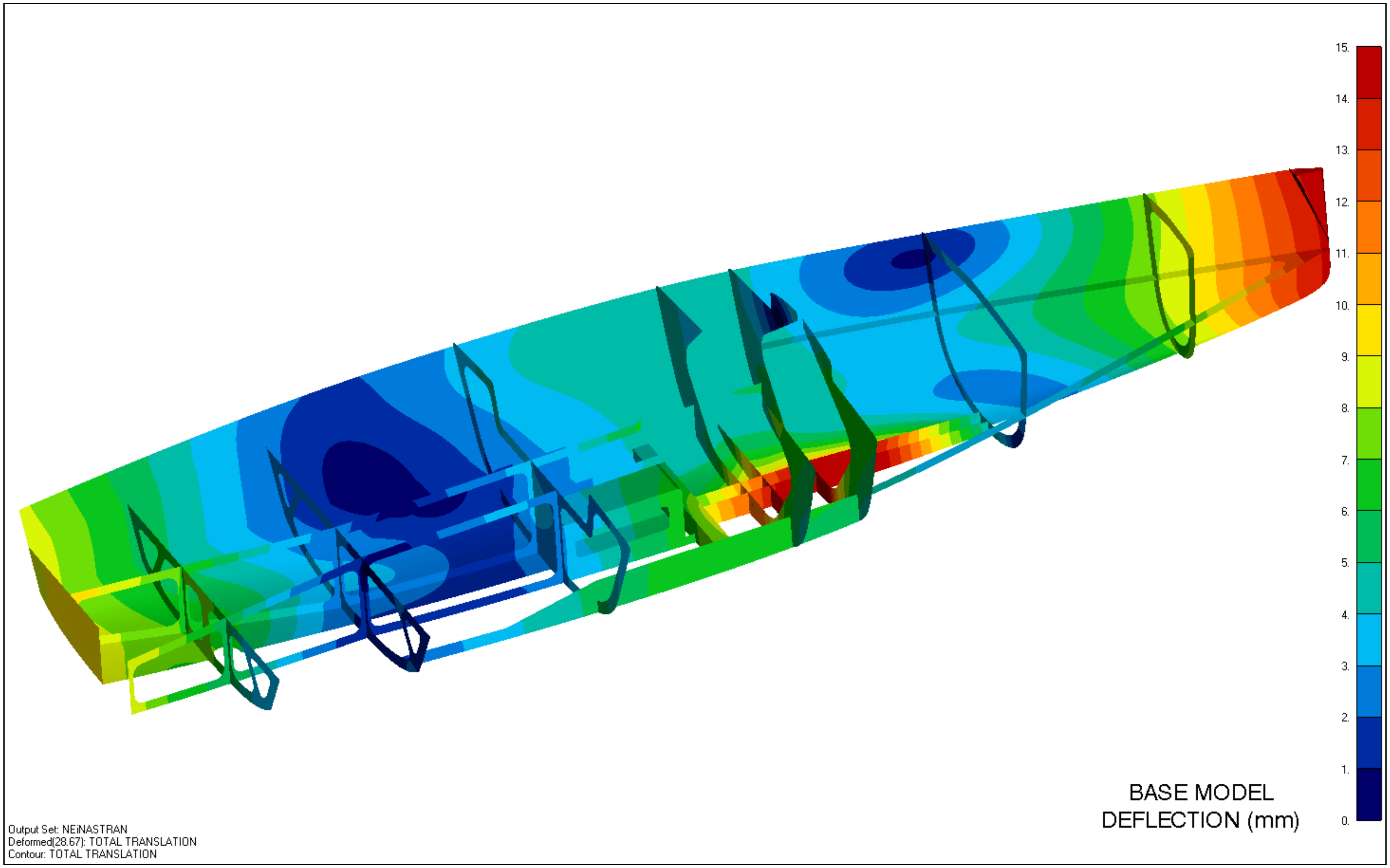


Base Laminate Model – Global deflections

ARISTON TECHNOLOGIES

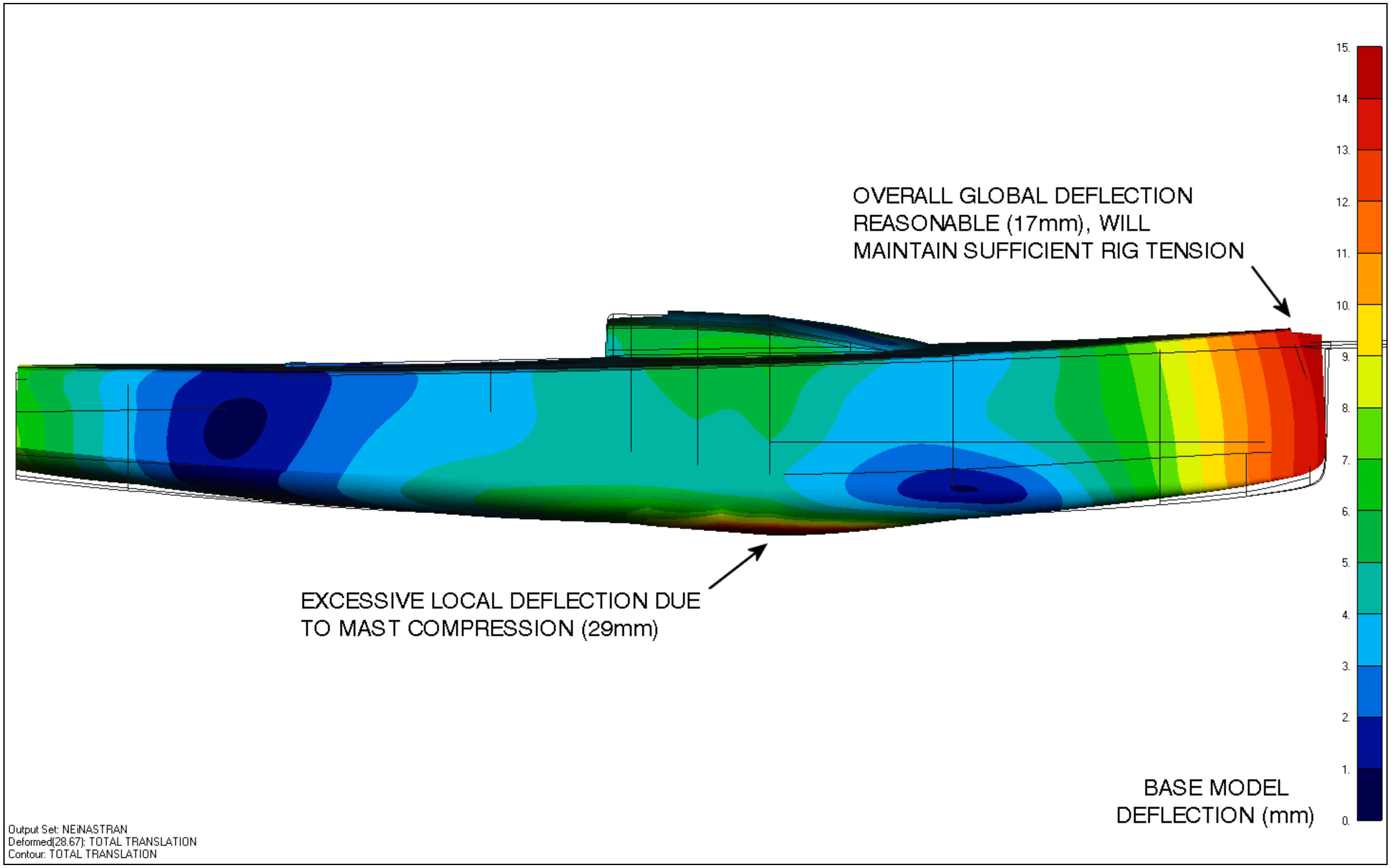
IBEX 2010 - SESSION 403
FEA FOR STRUCTURAL COMPOSITES OPTIMIZATION

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



Base Laminate Model – Global deflections

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



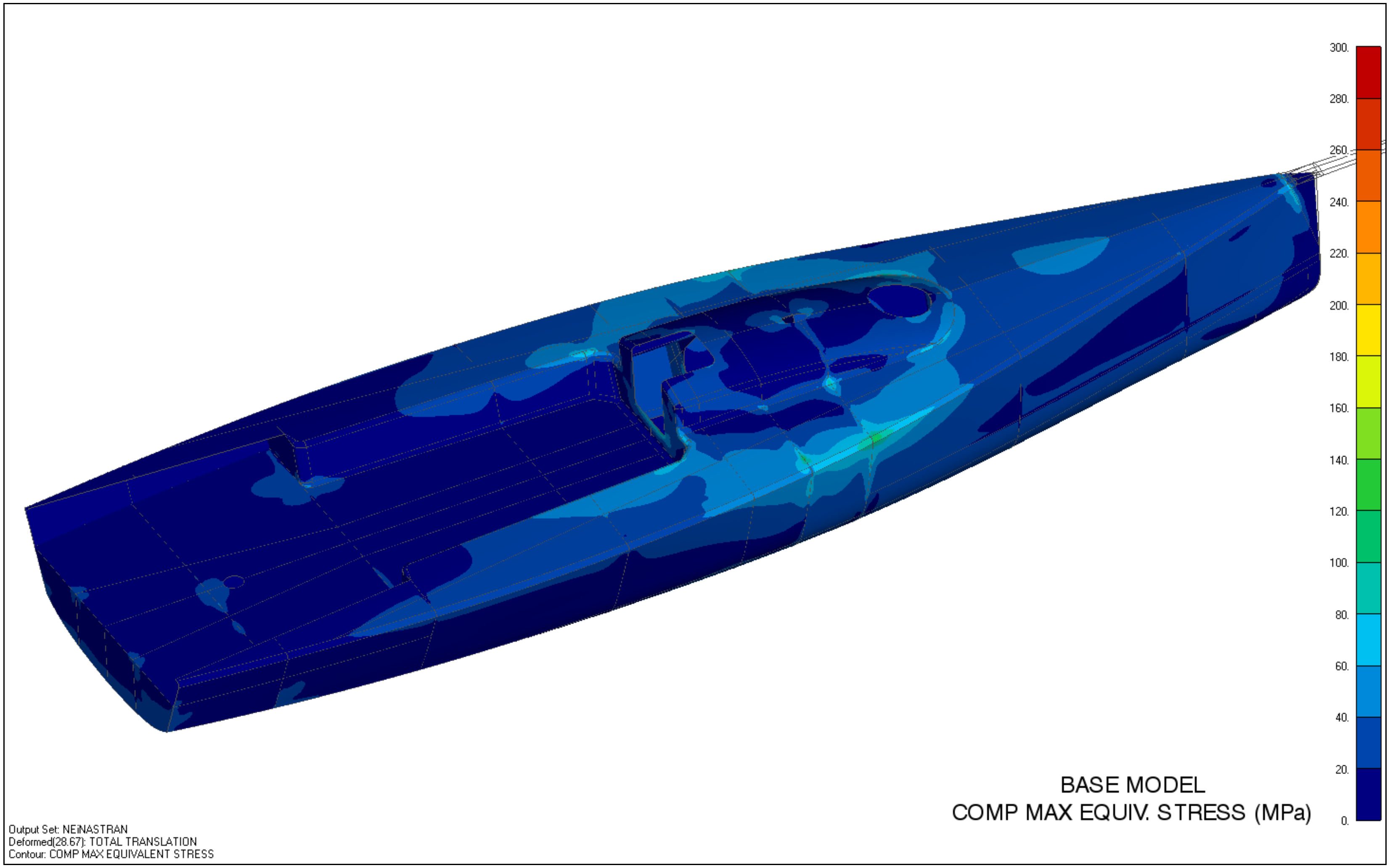
Base Laminate Model – Global deflections

ARISTON TECHNOLOGIES

IBEX 2010 - SESSION 403
FEA FOR STRUCTURAL COMPOSITES OPTIMIZATION

THE INTERNATIONAL BOATBUILDERS' EXHIBITION & CONFERENCE

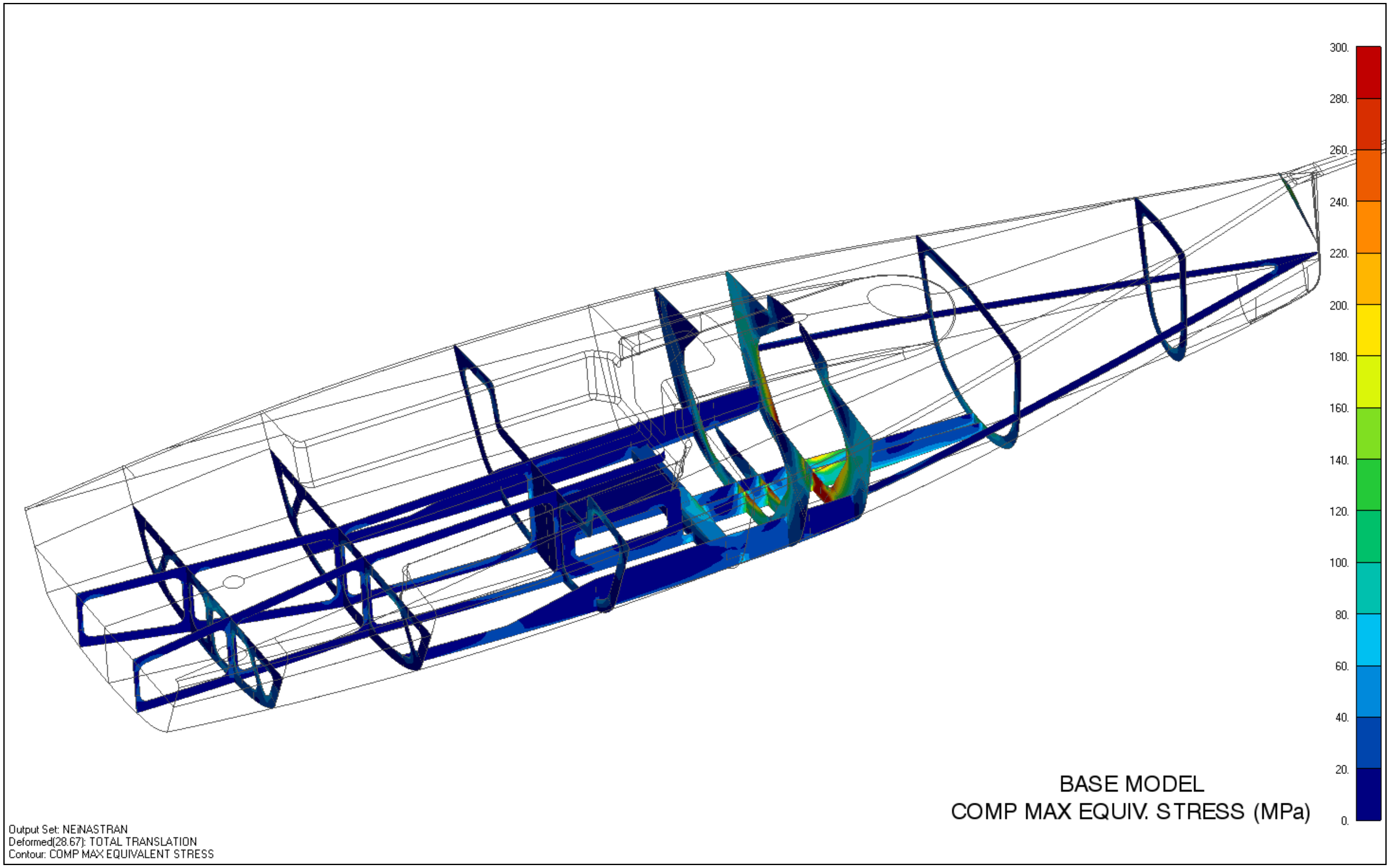
ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



Base Laminate Model – Composite Maximum Equivalent Stress (Max. of all plies)

54 Julia Ct., Kingston, RI 02881 • 401-575-8144 • www.aristontech.com • info@aristontech.com

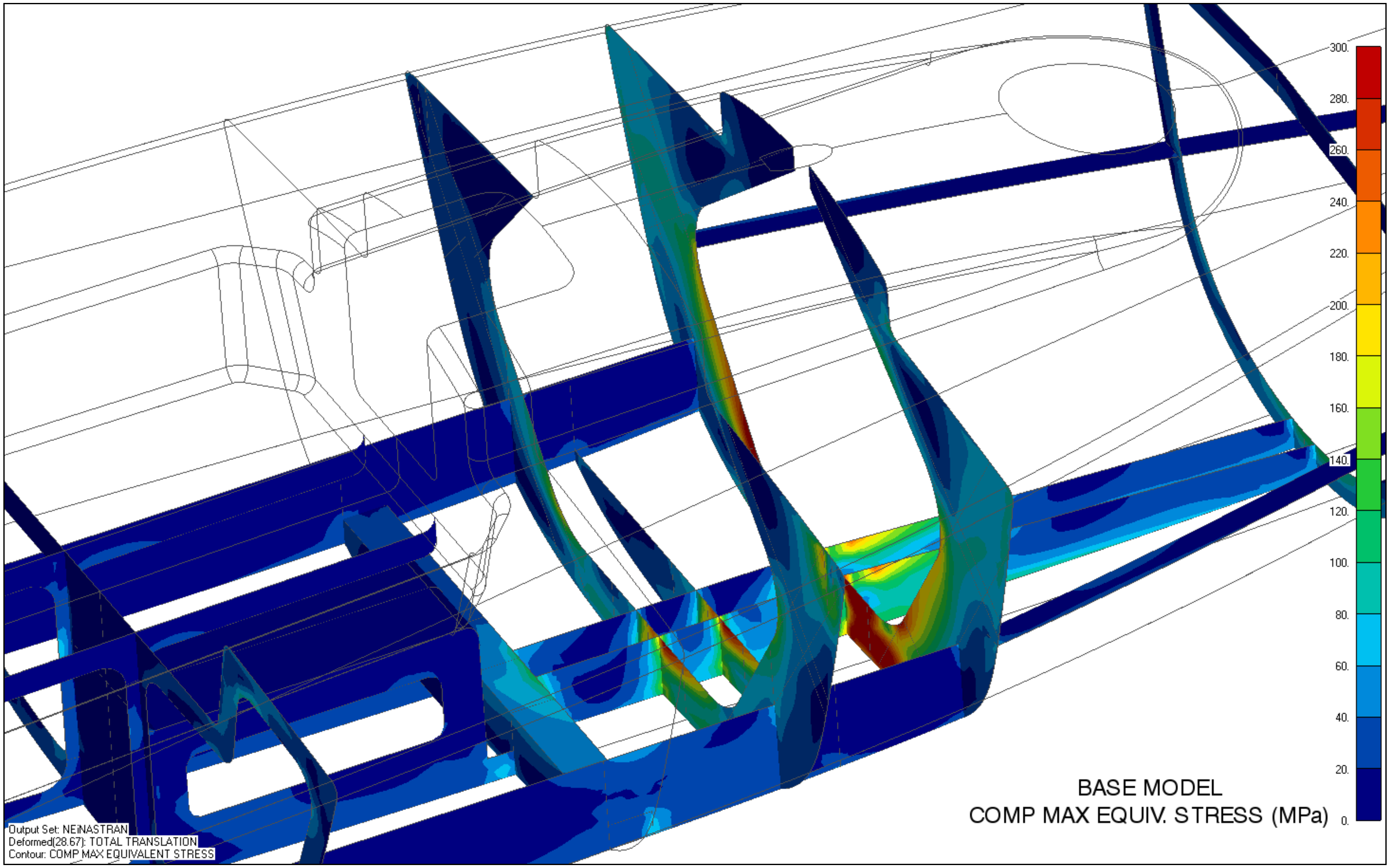
ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



Base Laminate Model – Composite Maximum Equivalent Stress (Max. of all plies)

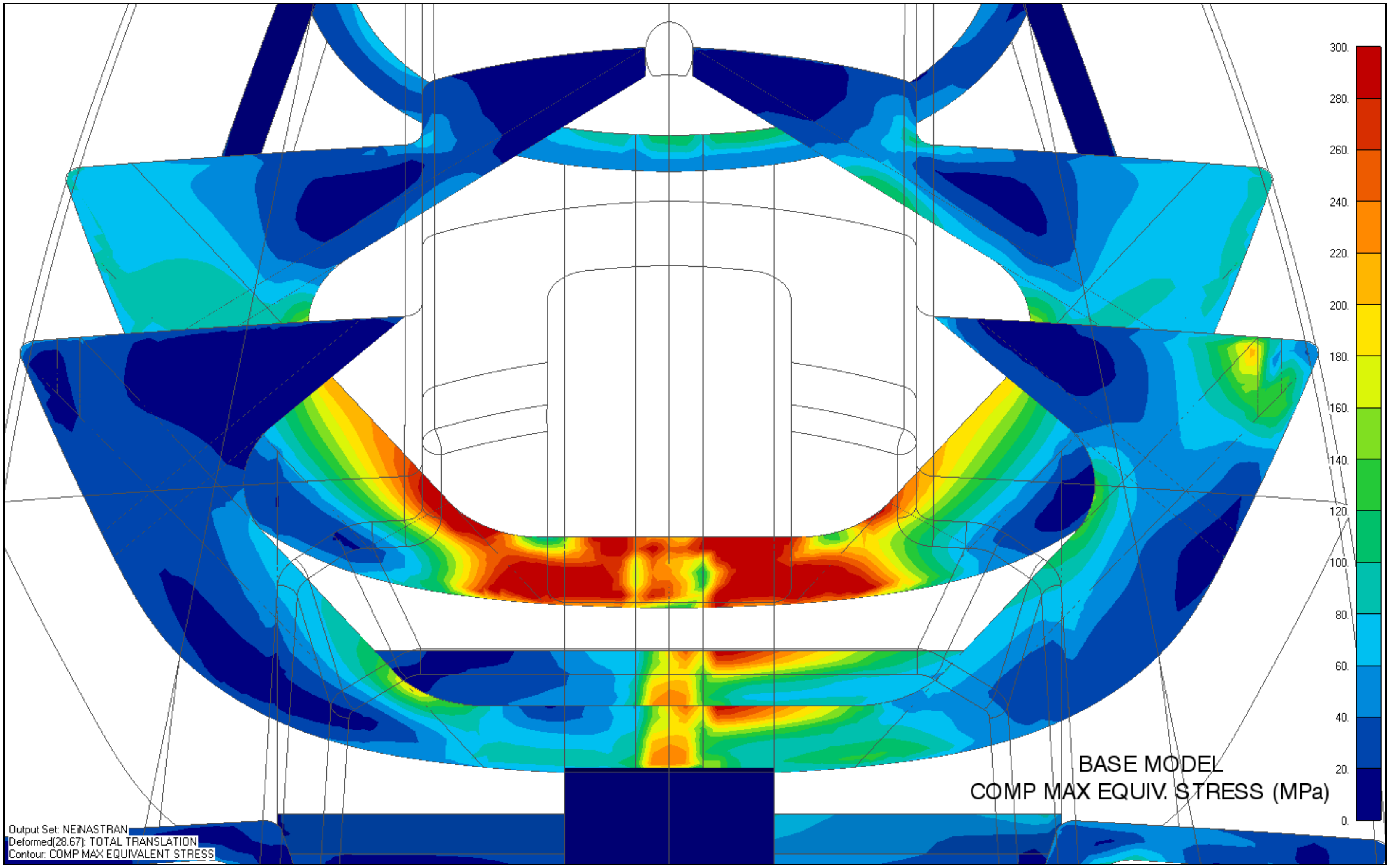
IBEX 2010 - SESSION 403
FEA FOR STRUCTURAL COMPOSITES OPTIMIZATION

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



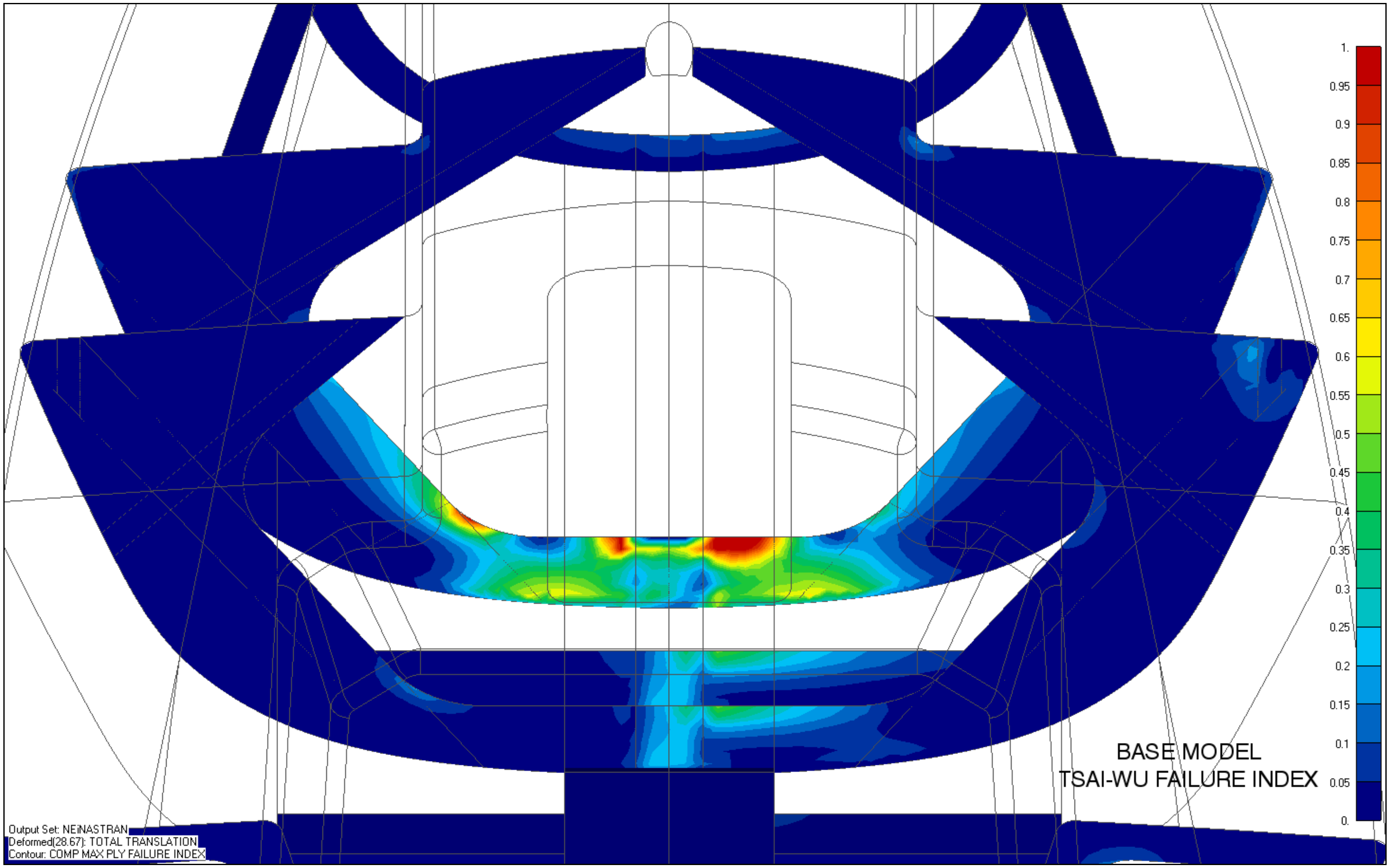
Base Laminate Model – Composite Maximum Equivalent Stress (Max. of all plies)

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



Base Laminate Model – Composite Maximum Equivalent Stress (Max. of all plies)

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



Base Laminate Model – Tsai-Wu Failure Index (Max. of all plies)

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS

- Images shown for Comp. Max. Equivalent Stress and Tsai-Wu failure index are maximums of all plies; Many more images could be shown for ply-by-ply assessment, but those shown above are sufficient to illustrate the concept

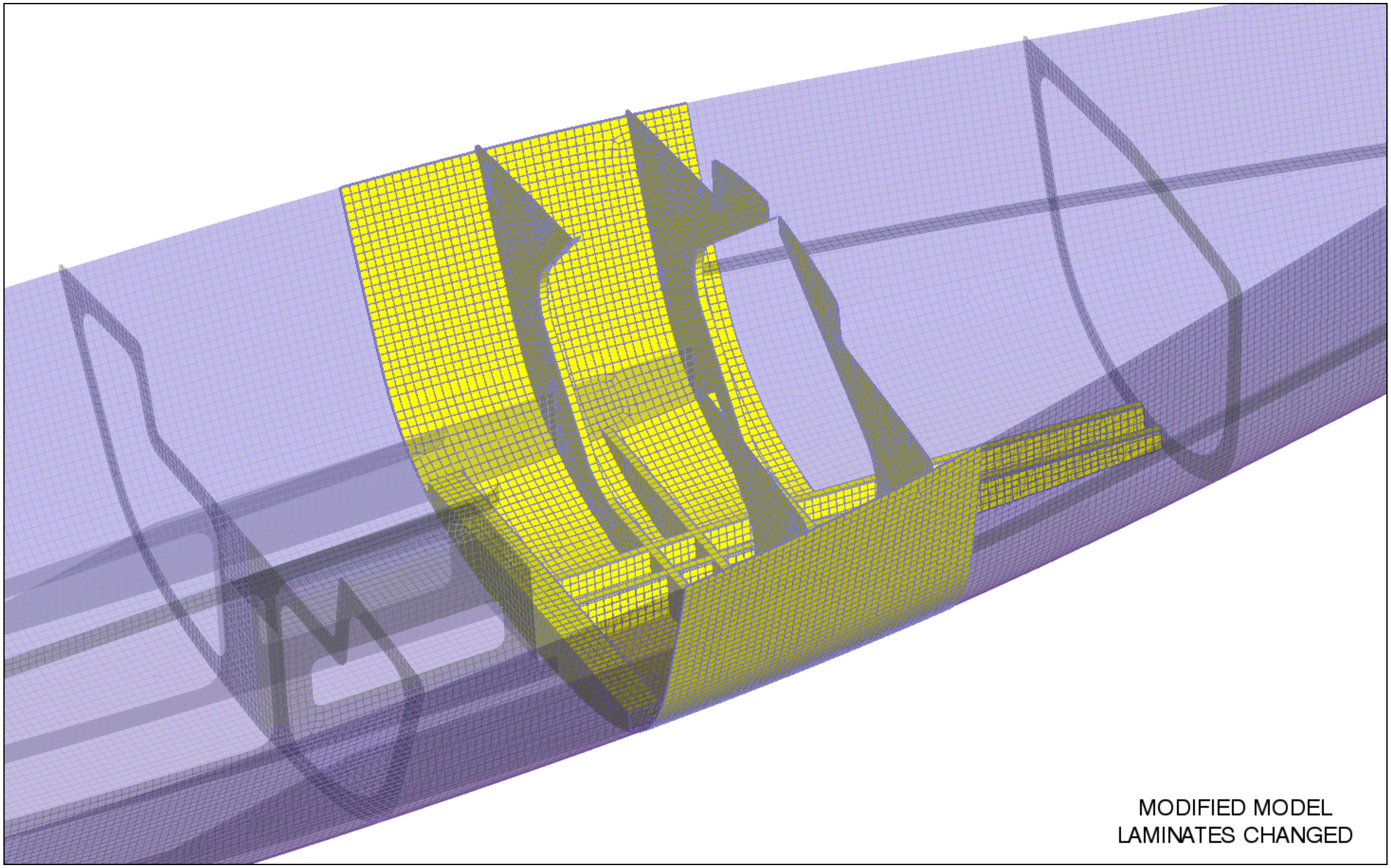
- A full assessment of results would consider maximum principal, minimum principal and maximum shear stresses; principal strain vector orientations; and failure indices for each ply throughout the model

- Model updates to resolve deflection and stress issues at mast base:

  - Hull shell laminate increased in thickness locally in way of mast base and surrounding frames and longitudinals
  - Mast and chainplate frame laminate thicknesses increased
  - Longitudinal web laminate thickness increased
  - Return flanges added around mast and chainplate frame inner cutouts
  - Flanges added to longitudinals

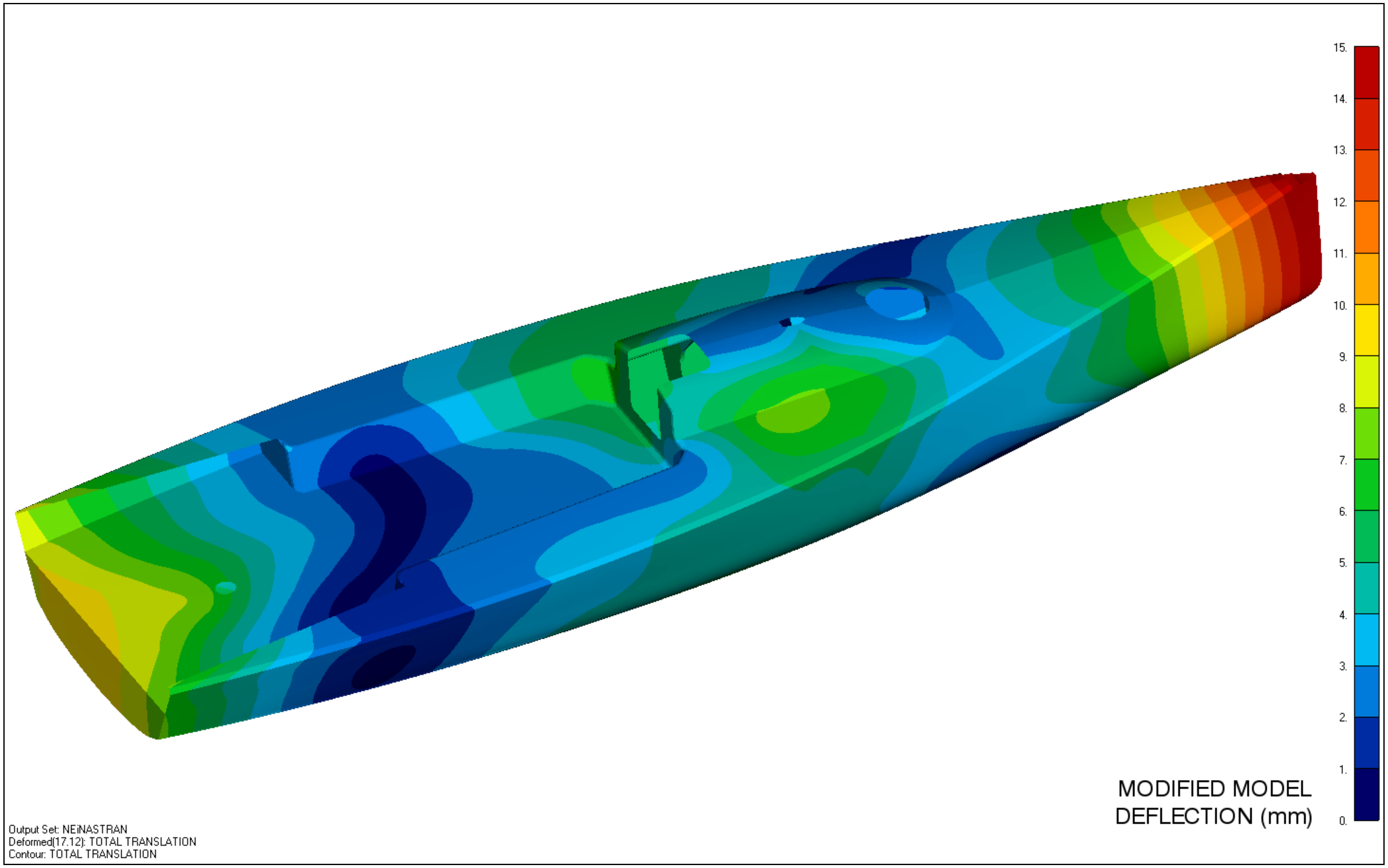
- Images showing updated results follow…

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



Optimized Laminate Model – Areas changed from base model

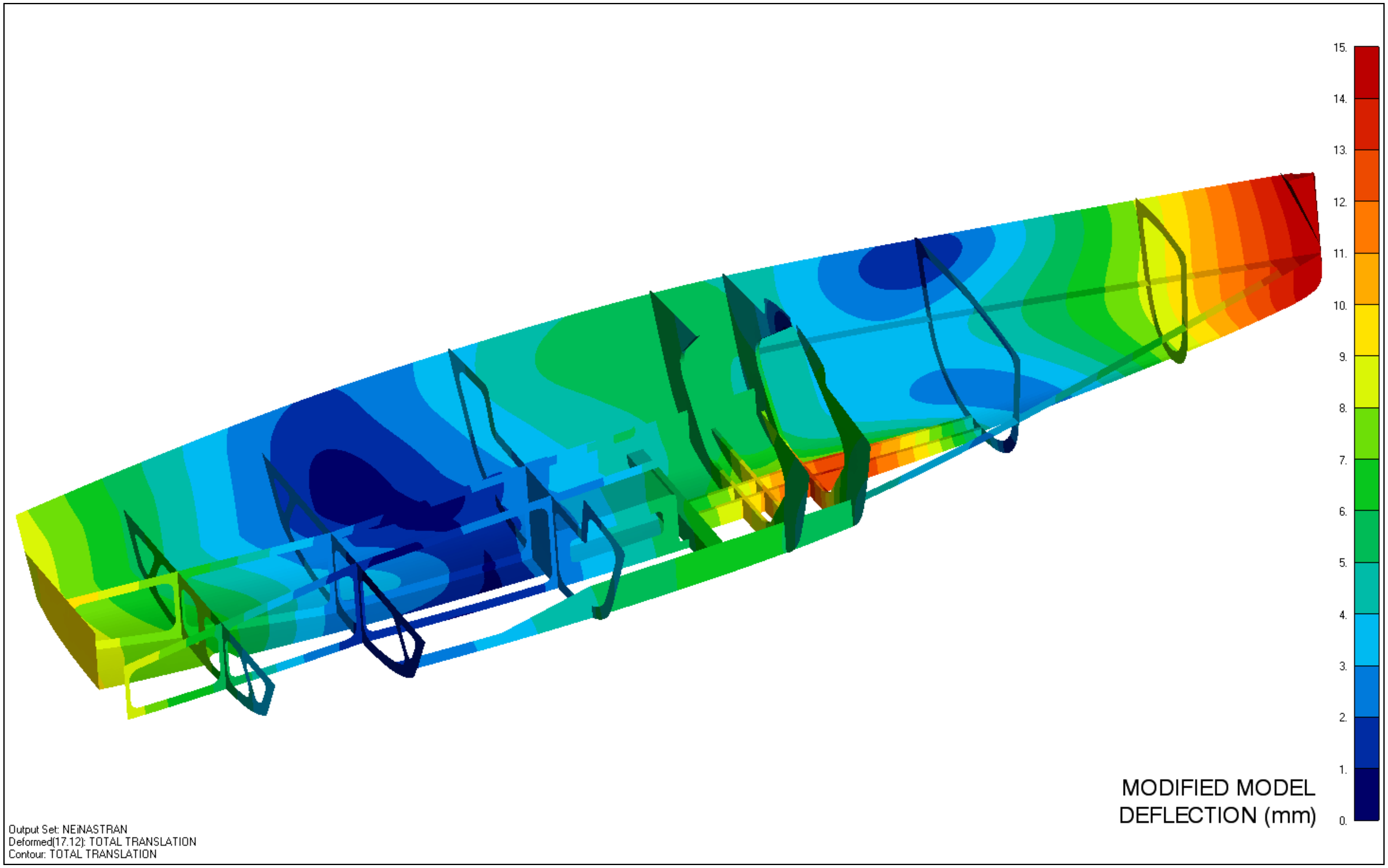
ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



Optimized Laminate Model – Global deflections

IBEX 2010 - SESSION 403
FEA FOR STRUCTURAL COMPOSITES OPTIMIZATION

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS

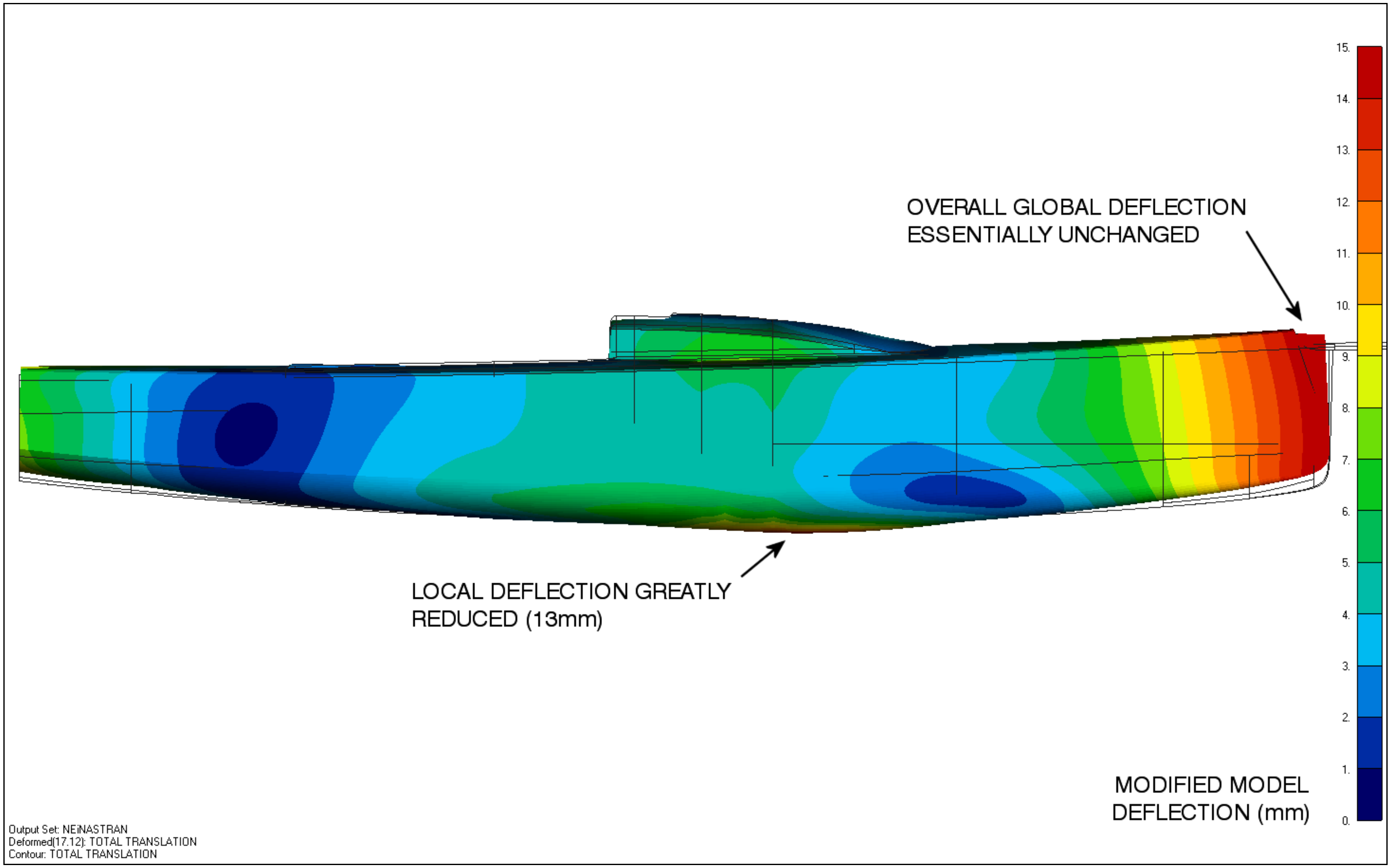


Optimized Laminate Model – Global deflections

ARISTON TECHNOLOGIES

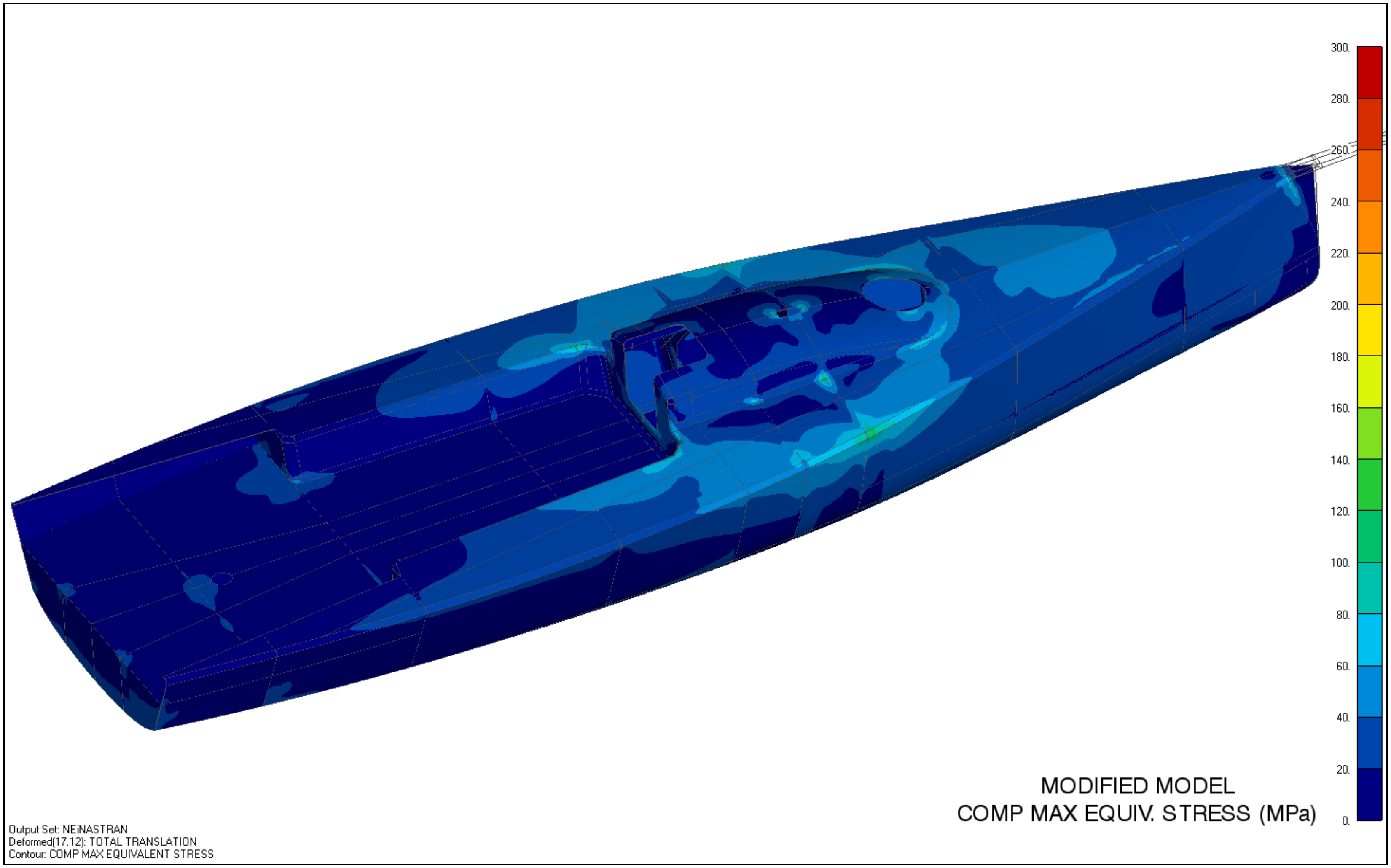
IBEX 2010 - SESSION 403
FEA FOR STRUCTURAL COMPOSITES OPTIMIZATION

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



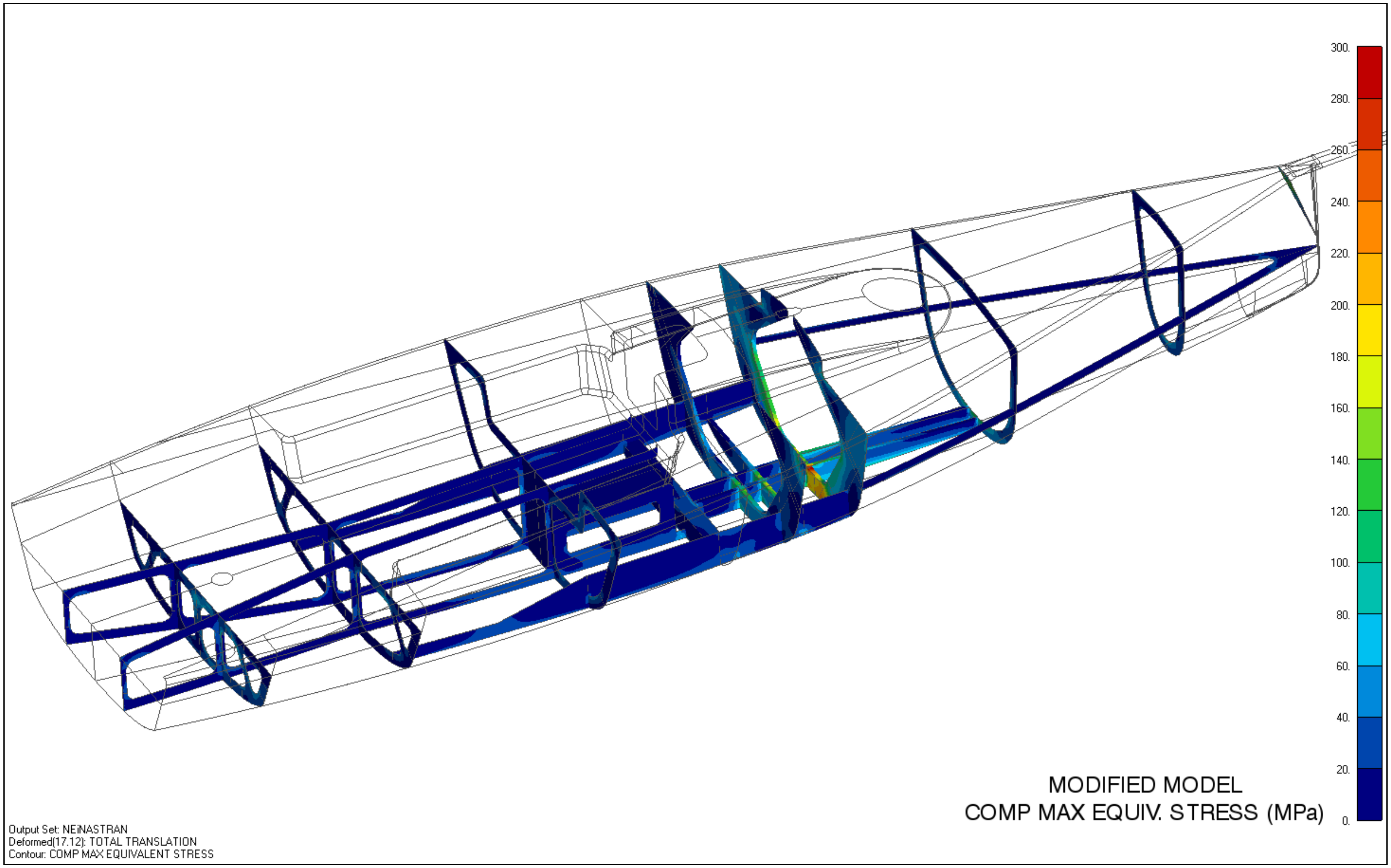
Optimized Laminate Model – Global deflections

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



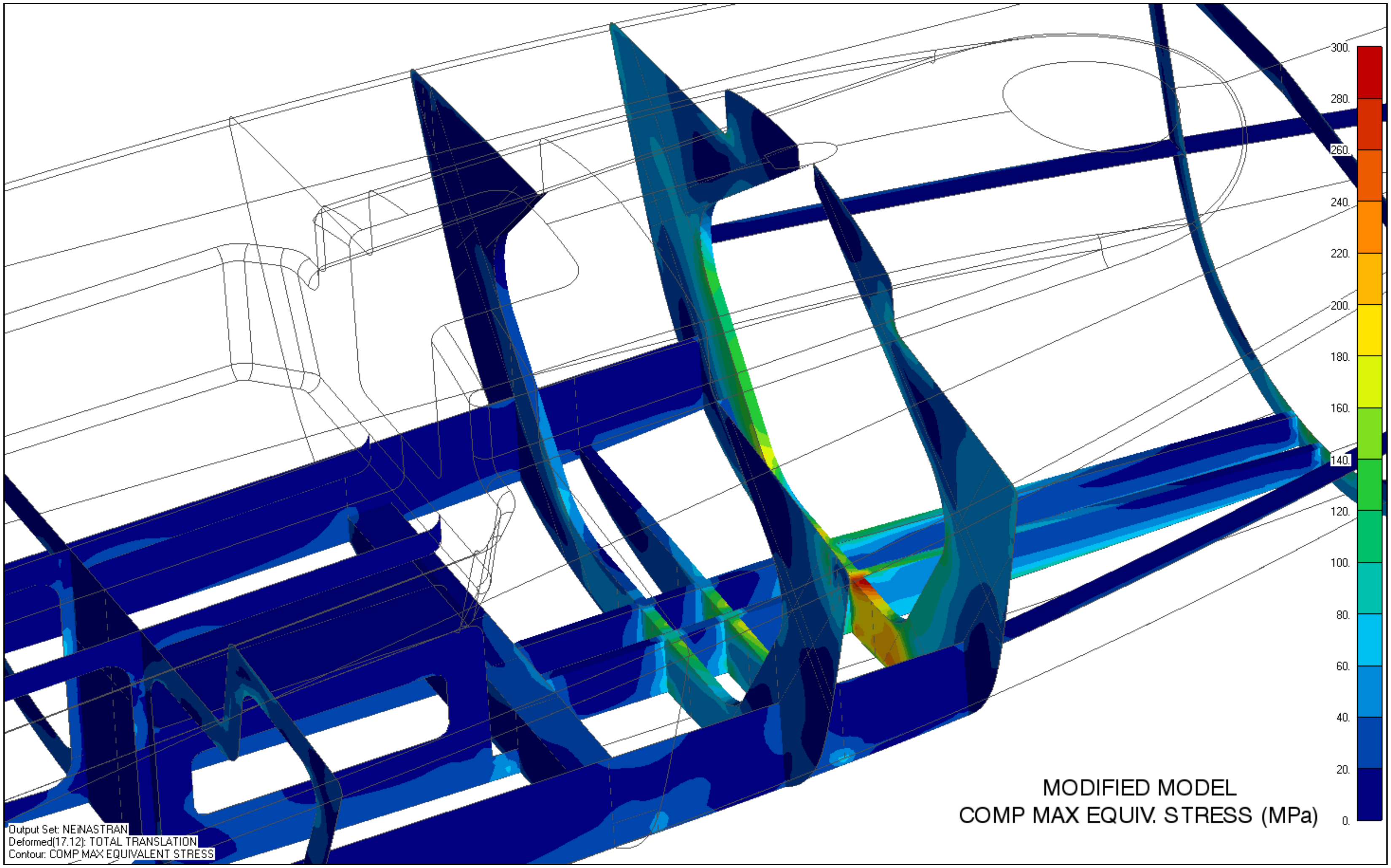
Optimized Laminate Model – Composite Maximum Equivalent Stress (Max. of all plies)

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



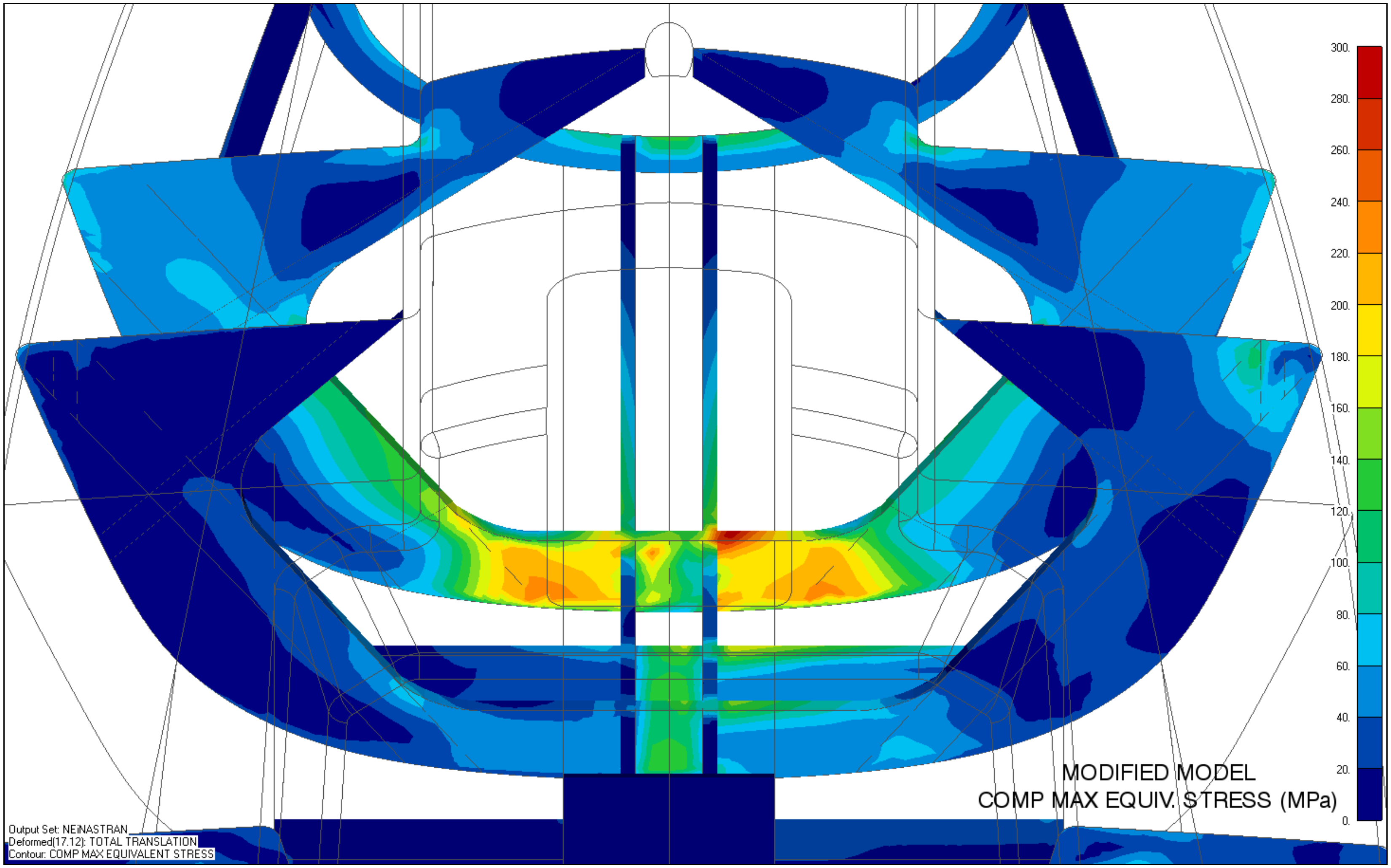
Optimized Laminate Model – Composite Maximum Equivalent Stress (Max. of all plies)

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



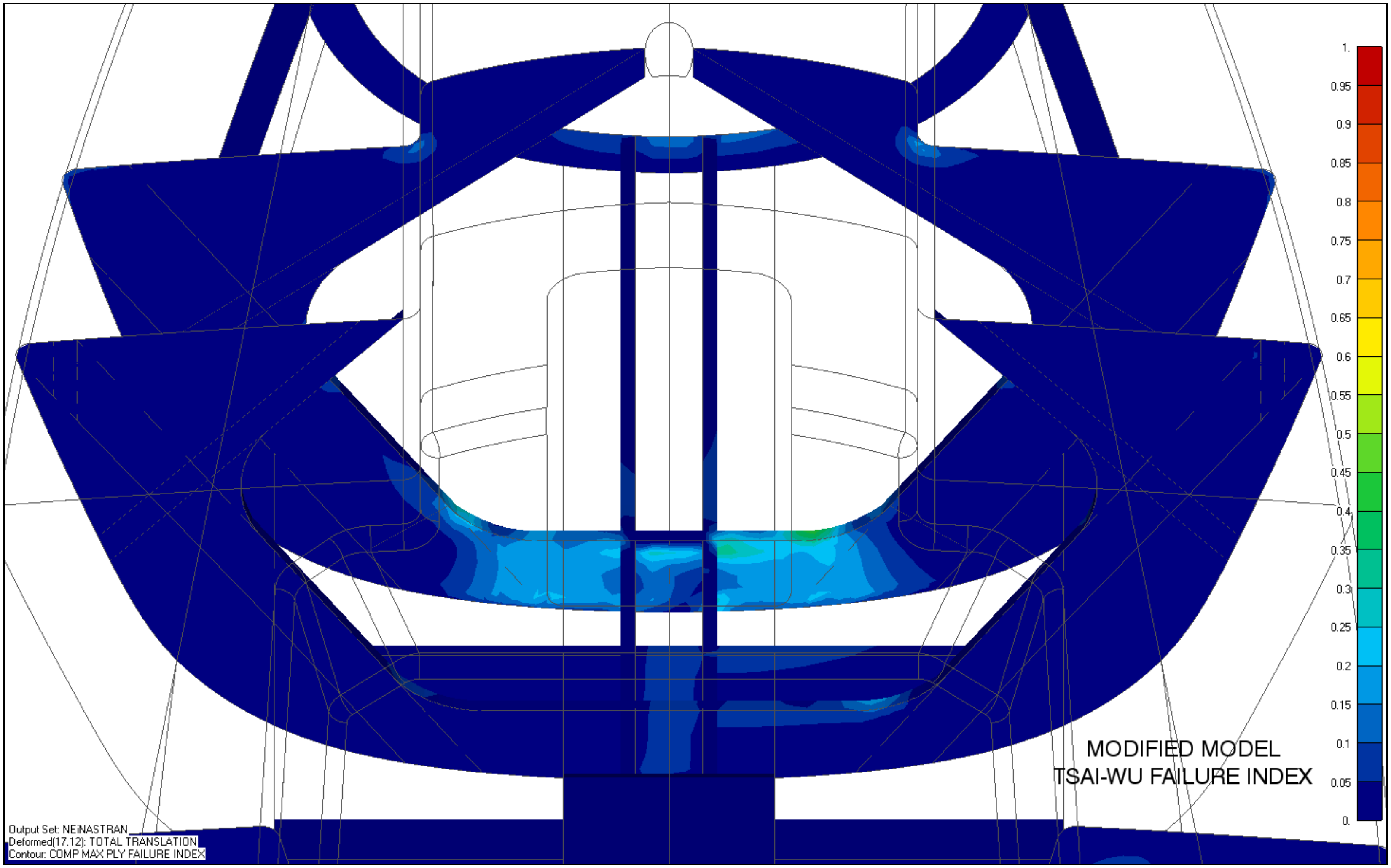
Optimized Laminate Model – Composite Maximum Equivalent Stress (Max. of all plies)

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



Optimized Laminate Model – Composite Maximum Equivalent Stress (Max. of all plies)

ANATOMY OF A FINITE ELEMENT ANALYSIS – SOLUTION & RESULTS



Optimized Laminate Model – Tsai-Wu Failure Index (Max. of all plies)

IBEX 2010 - SESSION 403
FEA FOR STRUCTURAL COMPOSITES OPTIMIZATION

TIPS & RECOMMENDATIONS

- FEA is not a replacement for sound engineering best-practices
- FEA should be used as a supplement to fundamental structural design approaches based on first principles and/or scantlings
- FEA can provide a more detailed understanding of both the global and local structural response to applied loads
- FEA can be used for optimization of global deflection and strength as well as for localized studies such as panel deflection & strength, beam/stiffener/girder sizing, assessment of critical fittings & attachments, etc…
- FEA is only useful if done correctly and accurately; “Quick & Dirty” FEA is an oxymoron – an accurate solution to an incorrectly or incompletely posed problem will be of no benefit but rather could be erroneous and misleading

TIPS & RECOMMENDATIONS

- FEA should only be performed by someone competent and experienced enough to fully understand the process and recognize the potential pitfalls
- Composites FEA is an order of magnitude more complicated than isotropic materials, in terms of both material characterization and results interpretation
- Be critical! Common sense should rule the day when assessing what FEA is telling you; If it doesn't make sense, then it's probably not right; Search the model for errors in meshing, materials, loads, constraints, etc…
- Careful documentation is critical, as the database of information generated can quickly become overwhelming
- When done properly, FEA is unmatched as a tool for composites optimization