

The Development of TruPly, an Efficient Composite Optimization Tool for Simulia Abaqus

Martin Gambling¹, Shanshan Wang²

¹ Managing Director, GRM Consulting Ltd, Warwickshire, UK, martin@grm-consulting.co.uk

² Software Developer, GRM Consulting Ltd, Warwickshire, UK, shanshan.wang@grm-consulting.co.uk

1. Abstract

Over recent years the demand for efficient optimization tools for composite structures has seen the development of several commercial solutions, which have been adopted by industries such as Formula 1, Aerospace and Wind Energy. One such tool is VR&D Genesis, which has provided an effective gradient-based composite optimization process since 1996. More recently, Genesis has been adopted as the industry standard optimization tool within Formula 1 since the introduction of GRM OptiAssist, a dedicated composite optimization environment, developed to link effectively with the laminating tools used within Formula 1.

A limiting factor in the adoption of optimization for composites has been the complexity of efficiently managing the material, element and laminate set-up of composite optimization models, including ply draping effects through the optimization process. Additionally, the requirement to transfer composite data from a user's preferred FE solver data structure to another for optimization has often discouraged its regular application.

In industries such as Formula 1, companies have been adopting Simulia Abaqus as their preferred analysis solver for composites due to its advanced analysis capabilities. The solver does, however, not support an efficient optimization toolset such as the capabilities available in VR&D Genesis.

This paper details the development of an integrated composite optimization tool for Abaqus, developed through support of major Formula 1 teams. With the objective of utilizing Genesis as the core optimizer, the development of the composite optimization tool, TruPly, has presented a number of technical challenges. Most prominent of these has been the fully automated conversion and management of data from the native Abaqus solver structure to Genesis' data structure.

With the objective of TruPly being to provide an efficient composite optimization solution supporting ply sizing, Topometry and the ability to mix the methods, the paper will discuss the development challenges faced in achieving the objectives. Through a discussion of several case study applications a review of the capabilities of TruPly will be made confirming the current capabilities of the tool and the future plans for its further development.

2. Keywords: Composites, Optimization, Abaqus, TruPly, grm-consulting

3. Introduction

The following paper overviews the development of a new composite laminate optimization tool, TruPly, as part of an overall new software package called TruProducts Design. TruProducts is a suite of structural design optimization tools available for Abaqus users, embedded as a plugin environment to Abaqus/CAE. The tools include:

- TruForm – Topology optimization
- TruPly – Composite Laminate optimization
- TruShape – Topography and Free-shape optimization
- TruSize – Shell Sizing and Topometry optimization

TruForm provides Topology optimization for Abaqus users and has been available commercially since its launch in 2010. It provides users with the ability to optimize for maximum stiffness or to minimize mass whilst limiting stress or plastic strain. Working with Abaqus/Standard solutions, TruForm typically converges on optimal topology designs in between 5 and 10 Abaqus solvers, working for both linear and nonlinear models.

Following the success of TruForm a more ambitious development task has been undertaken to create a complete suite of structural optimization tools for Abaqus, which has culminated in TruProducts Design, comprising TruPly, TruShape and TruSize. TruProducts Design, therefore provides shell sizing, Topometry, topography and composite laminate optimization as an integrated solution within the Abaqus/CAE environment.

Most challenging, and also interesting, has been the development of the composite laminate optimization tool, TruPly, which is discussed in more detail in this paper. The capabilities of TruPly, the technical challenges faced during its implementation and examples are discussed. Conclusions are then drawn regarding the current capabilities of the tool and the plans for its ongoing development.

4. Optimization Capabilities

4.1 Optimization Methods

When optimizing composite laminates there are two key methods that can be considered; ply pattern optimization (Topometry) and detailed sizing of existing plies. Topometry optimization works by the user defining candidate plies over a structure of the desired materials and ply orientations. The thickness of each ply of each element is independently designed to meet the defined optimization objective and constraints. Additionally, coarse Topometry may be defined, allowing elements to be efficiently grouped in patches. The coarsening method allows often more manufacturable laminates to be developed directly from the Topometry stage, negating the requirement for a second, sizing optimization step.

For both Topometry and Sizing methods ply angles and discrete optimization are available. TruPly supports many response types, which include stiffness, failure index, laminate thickness, layup variation, mass and volume. Equation responses may also be defined. Figure 1 demonstrates the available optimization methods applied to a simple box section under bending load.

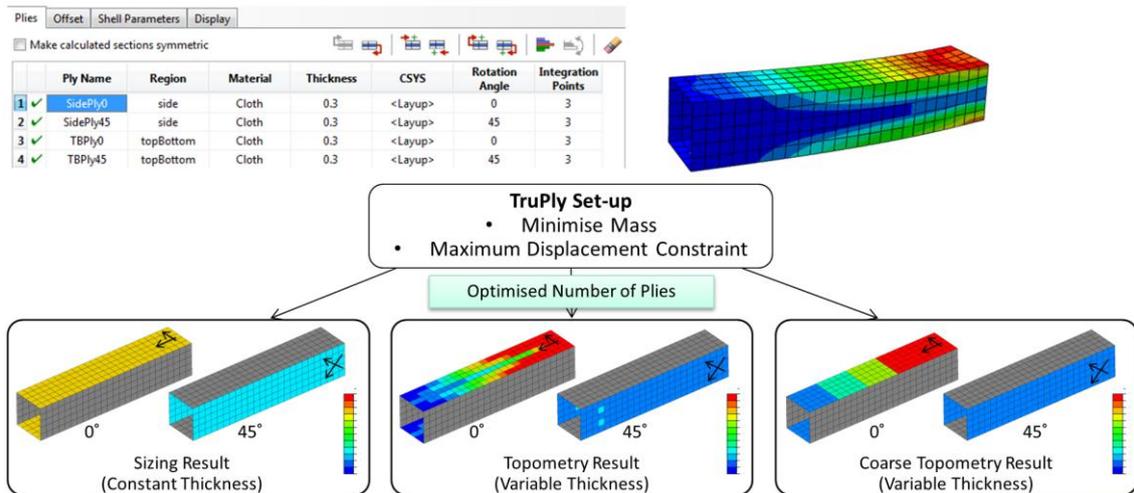


Figure 1: TruPly Optimization Methods

Traditionally, the optimization of composite laminates has considered both methods sequentially, creating a 2 stage optimization process. TruPly has been developed to allow users to perform both Topometry and Sizing optimization methods and also allows a combination of both methods.

4.2 User Interface

Using the python development interface within Abaqus/CAE, TruProducts Design has been developed as a fully integrated optimization plugin. The TruPly interface has been developed to work with composite laminates defined using Abaqus' composite layup properties. Figure 2 shows the TruPly design set-up page, which allows users to define the allowable changes to the laminate. The page allows users to selectively assign thickness and orientation design variables to plies and, additionally, define Topometry splitting of plies.

Once an optimization problem has been defined it may be submitted either directly on the user's workstation or exported for submission on a remote solver system.

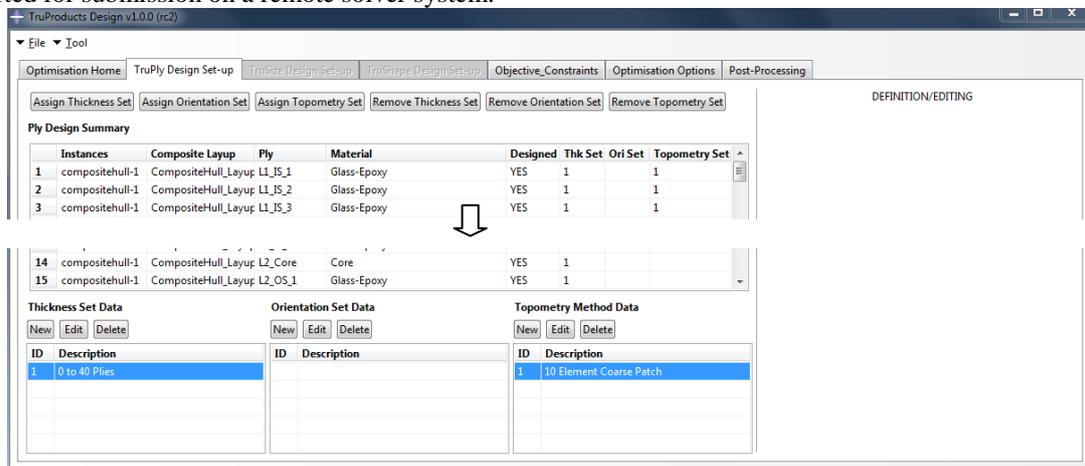


Figure 2: TruPly Design Set-up in

4.3 Optimization Workflow

Using TruPly, users have two optimization study ways to review and analyse optimization data. TruPly allows users to submit optimization studies directly or export them to run on a remote solver. After opening Abaqus/CAE model database, TruPly GUI panel can be loaded from Abaqus/CAE plug-ins list. Plies in the model will be listed in the ply summary table, detailing the parent instance and composite layup and the referenced material. Users are allowed to specify which plies are to be designed and how they are allowed to change from composite layups designed in the model. Once optimization data set, users can choose submitting optimization data or exporting data to do analysis. About optimization methods, TruPly supports ply sizing, Topometry and mixed sizing and Topometry methods. The sizing optimization process allows users to optimize an existing laminate construction, defining allowable changes to the number of plies and angle of each designed ply. The Topometry optimization uses the element- by- element method to sizing optimization. And the mixing of two methods allows some plies to be sizing optimized (assuming a fixed ply pattern) whilst the ply pattern of other plies may be varied through Topometry. Once optimized, users can automatically update the composite layup within Abaqus/CAE. Meanwhile, users can review ODB analysis and optimization results. The basic optimization processes of TruPly is illustrated as the below diagram (Figure 3).

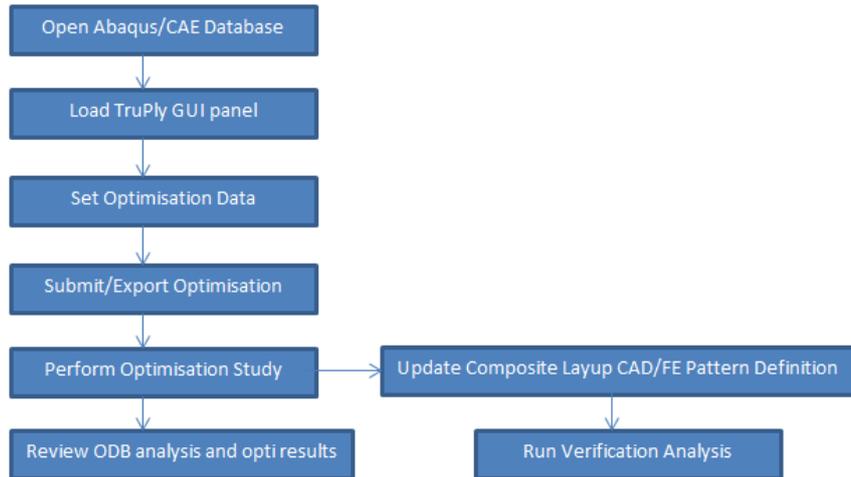


Figure 3: TruPly Optimization Workflow

5. Development

With the objective of utilising Genesis as the core optimizer, a major task involved converting data from the native Abaqus solver structure to Genesis in a fully automated process. Further to this a program workflow needed to be developed, which would allow sizing, Topometry or mixed optimization methods to be applied, automatically updating composite layup definitions defined by either CAD or FE pattern definitions. Honoring the native CAD based ply patterns was a key feature to allow users to apply TruPly within the Dassault Systems CATIA to Simulia workflow. Creating responses data efficiently also presented several technical challenges. In the following sections, the paper will take examples to discuss these three main technical challenges.

5.1 Convert Abaqus Analysis Data

Converting bulk data is the most significant challenge in the development of TruPly. Data should be converted from the native Abaqus solver structure to Genesis structure. The following two examples illustrate the challenges of converting bulk data.

5.1.1 Calculating Element Material Angle

Take a quadrilateral plate/shell element (QUAD4) of the structural model as an example. In Genesis, CQUAD4 card is created to define a quadrilateral plate/shell element of the structural model. THETA in CQUAD4 card is used to specify the material property orientation angle in degrees. As illustrated in the figure 4, the value of THETA is the value of angle θ . How to get the value of angle θ is the challenge in data converting due to a fundamental difference in how Abaqus and Genesis input data define this angle. The following three steps are calculating the value of THETA.

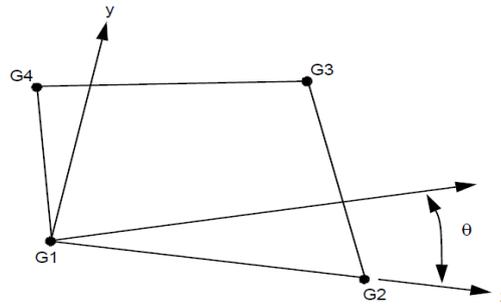


Figure 4: plate element with inplane rotational stiffness

- In order to calculate the element angle, normal vector should be created. Set $V2 = \overrightarrow{G1G2}$ and $V3 = \overrightarrow{G1G3}$ ($V2$ and $V3$ are two vectors), then create normal vector $n = \text{cross}(V2, V3)$.
- Project vector to the plane. The value of projected vector should be based on the normal vector created in the first step. Then using the equation $V_{prj} = \text{cross}(n, \text{cross}(V1, n))$ to get the value of projected vector. $V1$ is calculated based on element data in *Distribution from Abaqus file. The equation choosing is the hard part in these three steps. After several times testing and trying, the final equation is decided.
- Calculate angle between V_{prj} and $V2$. Using the value of projected vector and $V2$, element angle can be calculated easily.

5.1.2 Create Tied/Glue Contact Data

In Abaqus input file, *Tie is used to impose tie constraint between pairs of surfaces. However, in Genesis, using CGLUE card to define a pair of contact surfaces to be glued upon contact. In order to utilise Genesis to be a core optimiser, *Tie data should be converted to CGLUE card. Before converting data, analysing data is a significant step in the processes of data management. In *Tie data card in Abaqus input file, each data line defines a pair of surfaces, slave surface and master surface, that will be tied together. Each *Tie should be created to a CGLUE card in Genesis input file. Meanwhile, slave surface and master surface should be converted to BSURFE cards. The processes of creating BSURFE card is prominent challenge in creating glue contact data. First of all, surface type should be specified from Abaqus input file. Then based on surface type define the values of GA_i and GB_i . For solid elements, GA_i is the identification number of a GRID connected to a corner of element face. And GB_i is the identification number of the GRID connected to the corner diagonally opposite to GA_i on the face of CHEXA or CPENTA element. The process of specifying the values of GA_i and GB_i seems like guessing puzzles. There are no any useful information and references about getting the values of GA_i and GB_i without guessing and testing. During several times guessing and testing, the values of GA_i and GB_i are finally defined based on different surface type. At last, BSURFE card has been created.

5.2 Update of Composite Layup Definition based upon Optimization Results

As mentioned before, both sizing and Topometry methods are available within TruPly and, additionally, the two processes may be mixed. For both optimization steps the results are used in 3 ways:

- Direct update and solving of Abaqus input file
- Addition of optimization results to the odb results data file
- Update of Composite layup within Abaqus/CAE

5.2.1 Sizing Optimization Process – Number of plies and plies orientation.

The sizing optimization process allows users to optimize an existing laminate construction, defining allowable changes to the number of plies and angle of each designed ply. Once optimized the user may automatically update the composite layup definition within Abaqus/CAE. The challenges of sizing optimization process is the process of updating Abaqus input file based on changed number of plies and angle of each designed ply and the process of modifying composite layup definitions within Abaqus/CAE. During the optimization process TruPly will automatically update the *shell section data and perform a verification Abaqus solve. To enable efficient laminate data handling within Abaqus/CAE, the user interface allows the update of the composite layup pattern definitions, maintaining the ply pattern definitions in their native geometry/FE format.

5.2.2 Topometry Optimization Process – Ply placement optimization.

Using an element-by-element or patch-by-patch approach to ply sizing optimization the optimal ply patterns may be efficiently determined. Candidate plies are defined in an initial composite layup and the numbers of plies of each type that may be used are defined. The results will highlight the optimal ply patterns required to achieve the

defined objective function whilst meeting the chosen constraints. TruPly provides the results as contours appended to the ODB file and allows the user to automatically update the Composite Layup definition within Abaqus/CAE. Based on element-by-element approach to modify Abaqus input file is a difficult step in Topometry optimization. Using an initial composite layup and the numbers of plies of each type creates candidate plies. The development of TruPly improves the optimization performance and reduces development time and costs. As a complete suite of efficient design optimization tools within the Abaqus/CAE environment, TruPly do a useful support of Formula 1 teams.

6. Example Applications

The following section details two example applications of the composite optimization capability of TruProducts (TruPly). The examples demonstrate the methods available for ply placement optimization (Topometry) and the sizing optimization of laminates, maintaining the native geometry association.

6.1 Yacht Hull Sizing Optimization

The hull of a yacht (figure 5) presents an excellent opportunity to achieve significant mass savings through application of composite optimization techniques. However, in the case of such an application, the consideration of the manufacturing process must be carefully considered. In this example, clearly zoned ply patterns have been defined, which are efficient to manufacture. Sizing optimization is therefore applied, maintaining these existing ply patterns, which have been defined by geometry regions. Utilising this approach also allows the direct and efficient interfacing to composite laminate definitions within CATIA; the process allowing users to automatically update laminate definitions in CATIA, based upon TruPly optimization results.

An example hull ply definition is shown in Figure 6. The baseline laminate has been developed

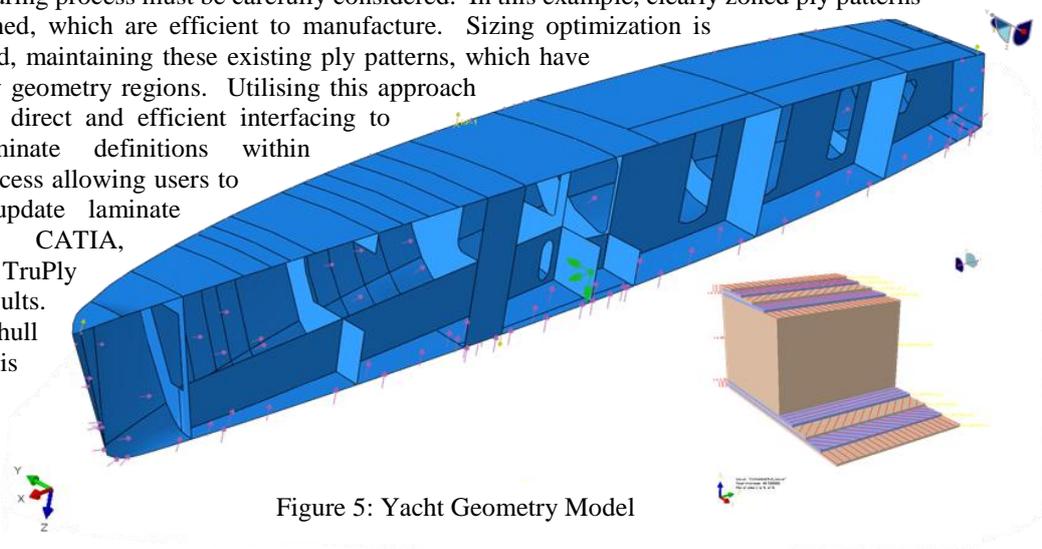


Figure 5: Yacht Geometry Model

through the traditional iterative processes, which meets the yacht's strength requirements. The hull laminate was constructed from glass epoxy uni-directional plies and was constructed as a balanced laminate with a foam core. The total mass of the baseline hull was 4.04 tonnes and the performance under the abuse load case is shown in figure 7. The performance target for the hull is to not exceed a failure index limit of 1.0 under the abuse load case. As can be noted on figure 7, the majority of the hull is within the failure index (FI) limit, whilst some localized areas exceed the limit by a small margin, with a maximum recorded FI of 1.3. These are considered acceptable.

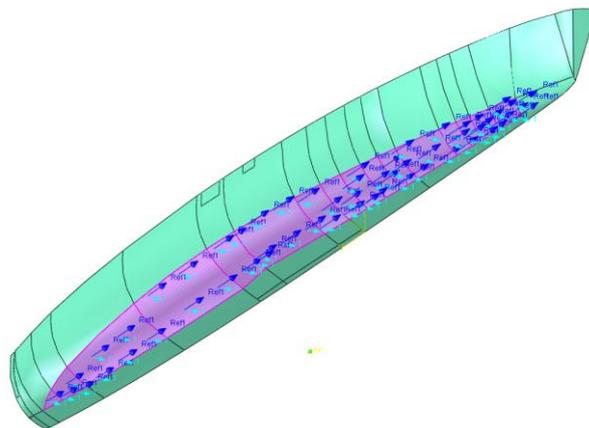


Figure 6: Example Hull Ply Pattern Definition

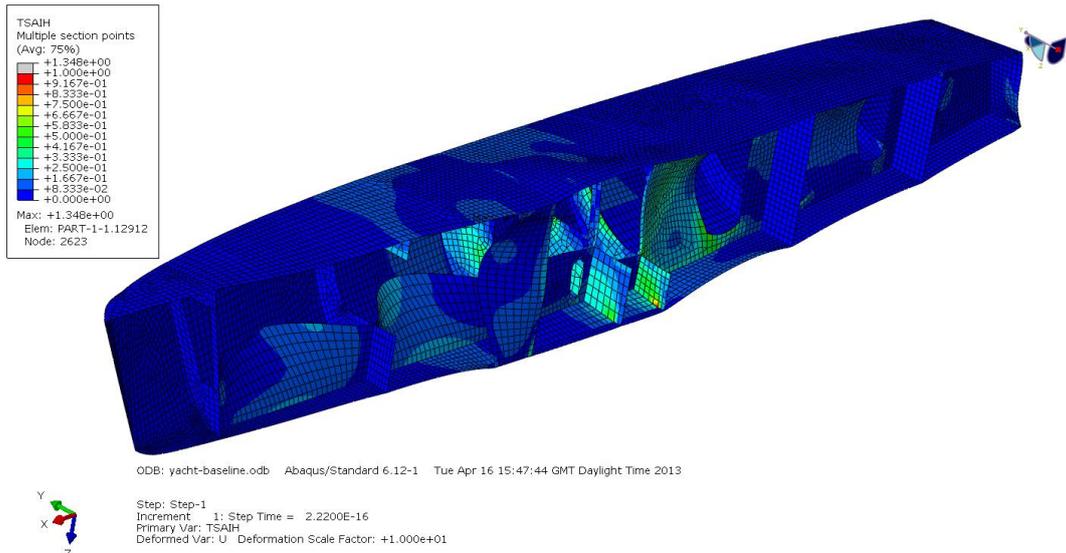


Figure 7: Baseline Hull Laminate Failure Index under Abuse Loading

6.1.1 Design Optimization Studies

Using the baseline laminate as a starting point for sizing optimization, TruPly was used to allow each ply to be varied in thickness from 0 to 40 times its initial ply thickness. The objective of the optimization was to minimize the mass of the glass-epoxy plies which not exceeding a failure index of 1.0. Additionally, a constraint was defined to limit the laminate thickness to a minimum of 1mm on all surfaces. Two design optimization studies were performed; firstly assuming a balanced laminate is maintained and, secondly, allowing the laminate to vary freely each side of the core material.

The optimization studies were solved directly within Abaqus/CAE on a local workstation. Both optimization studies took 35 design iterations to converge, followed by an Abaqus verification analysis solve. Once the optimization was complete the TruPly post-processing tools were used to automatically update the composite layup definitions of each model to reflect the optimization results. Table 1 below provides a summary of the two optimization results in comparison to the baseline design.

Design	Number of Plies	Mass (tonnes)
Baseline	1318	4.04
Balanced Laminate Optimization	1104	2.7
Free Laminate Optimization	983	2.26

Table 1: Summary of Yacht Sizing Optimization Results

6.1.2 Analysis Verification

As is shown in Figure 7, both the balanced and freely optimized hull laminates meet the failure index constraint of 1.0. Review of the baseline, balanced optimized and freely optimized failure index results does show an increasing average failure index over the structure. As the constraint on the design is 1.0, this demonstrates how the optimizer is developing more efficient structures, eliminating redundant material whilst remaining within the design constraints.

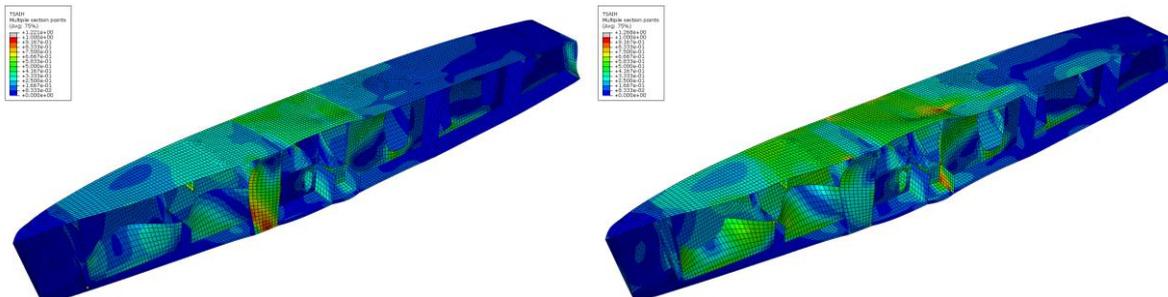


Figure 7: Optimized Hull Laminate Failure Index under Abuse Loading

The optimization of the yacht hull provides a useful demonstration of the typical performance savings that may be achieved through the application of TruPly’s optimization methods. In this example, the use of sizing optimization only has enabled the ply patterns to remain defined by the native CAD surfaces of the hull model. This has the significant benefit of allowing the optimized hull laminate results to be updated directly to CATIA, maintaining the laminate definition at the designer level.

6.2 Ply Placement Optimization of a Formula 1 Front Wing

VR&D Genesis has been established as the industry standard composite laminate optimizer in the Formula 1 industry for almost ten years. More recently, however, the primary analysis solver used by several teams has changed from Nastran to Abaqus, which uses a significantly different input data structure. This has made the direct application of Genesis as an optimizer a more complicated process due to the introduction of a data conversion step. TruPly has therefore already been adopted by Formula 1 teams using Genesis who have migrated their primary analysis solver to Abaqus.

To demonstrate the application of TruPly to a typical Formula 1 application, a 2010 style front wing (Figure 8) main plane has been optimized. The example demonstrates the application of Topometry optimization to determine the optimal ply patterns required to meeting the loading.



Figure 8: 2010 Style Formula 1 Front Wing

In this example, the wing is subjected to two load cases, which are the aerodynamic pressure loads and a regulatory bending load defined by the Formula 1 governing body (FIA).

6.2.1 Design Optimization Set-up

For the optimization of the front wing, the design objective was to maximize the displacement of the rear, outer edge of the wing under the aerodynamic loading. The design constraints were to ensure no failure in the main plane occurred during the aerodynamic loading and that the end of the wing does not exceed a displacement of 16mm under the FIA bending load case. Additionally, the laminate thickness was constrained to not exceed a maximum thickness of 5.0mm.

To perform the ply placement optimization, 4 initial plies were defined in the composite layup. The first ply was defined as non-designable and was a cloth outer ply at an angle of 0 degrees. The three remaining plies were defined as uni-directional fibres oriented at 0, +45 and -45. Each designable ply was allowed to vary in thickness from 0 to 25 times its original ply thickness. The initial layup construction is detailed in figure 9.

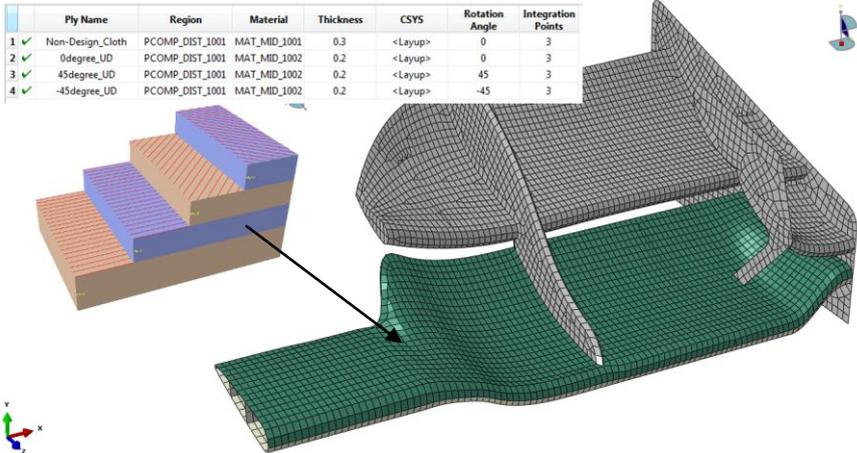


Figure 9: Initial Candidate Layup Construction

The optimisation study was performed using two very differing coarsening levels for the ply pattern development. The first used a small patch size of 25mm x 25mm in order to develop the most optimal solution. The second assumed a patch size of 150mm x 150mm in order to develop ply patterns that were easy to interpret into a manufacturing solution.

6.2.2 Optimization Results

Both optimization studies converged on an optimal solution after 20 design iterations, followed by a single Abaqus confirmation analysis. As can be noted in table 2, the objective of maximizing the aerodynamic displacement is greater for the lower coarseness level. This is to be expected as the optimizer has more freedom to vary the element/ply thicknesses independently, however, the resulting design will be more complicated to manufacture. Figure 10 shows the total laminate thickness of both optimization results, and shows clearly the difference in complexity of the two optimization studies.

Optimization	Design Objective (Aero Load Displacement - mm)
High Coarseness (150mm x 150mm)	97.1
Low Coarseness (25mm x 25mm)	111.2

Table 2: Summary Front Wing Optimization Results

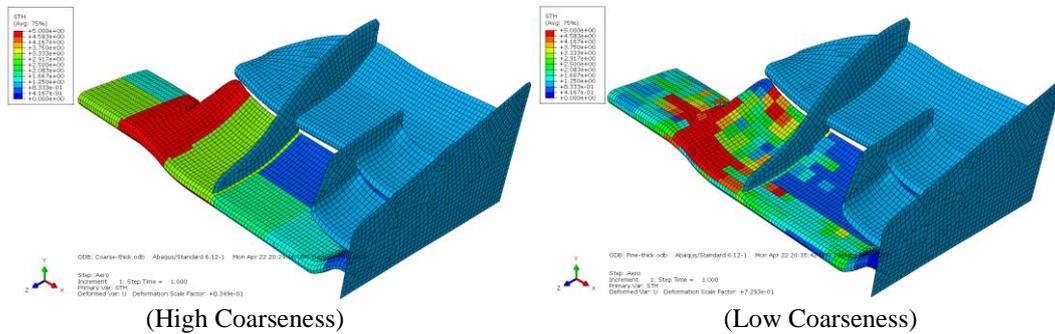


Figure 10: Front Wing Optimized Laminate Thickness

6.2.3 Analysis Verification

Following optimization of the front wing, TruPly's 'update composite layup' option was used to automatically convert the Topometry results into an updated laminate for the front wing main spar, considering the high coarseness results. The update process automatically converted the initial 3 candidate plies into a total of 54 plies; 28 on the top of the main wing section and 26 on the bottom. Example plies are shown in figure 11. Once updated, the performance of the newly laminated wing section was assessed and the performance detailed in table 3. The results demonstrate that the wing meets the displacement and strength requirements whilst achieving a displacement under aerodynamic loading of 96mm.

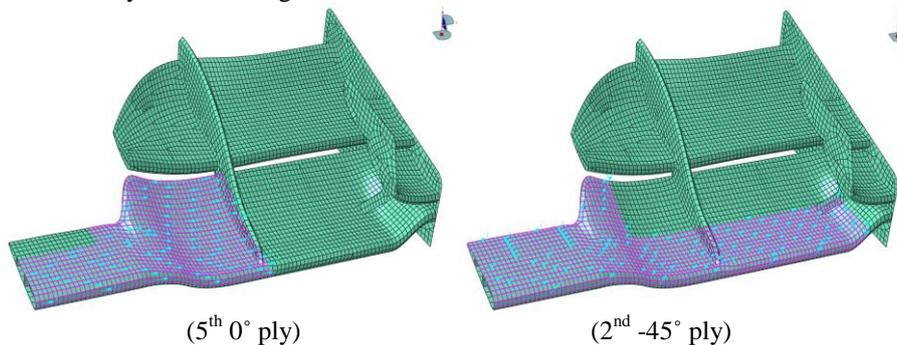


Figure 10: Front Wing Optimized Laminate Thickness

Performance Measure	Target	Performance
FIA Displacement (mm)	<16mm	15.9mm
Aerodynamic Load Failure Index	1.0	0.98
Aerodynamic Load Rear Displacement (mm)	Maximize	98mm

Table 2: Summary Front Wing Optimization Results

7. Summary and Conclusions

The demand for ever lighter and more cost efficient structures continues to increase the commercial and technical gains that businesses can achieve through the application of design optimization methods. Key to the successful adoption of optimization tools is their complete integration into a business's primary structural analysis solver environment. TruProducts Design, incorporating TruPly, has therefore been developed as a fully embedded optimization toolset within Abaqus/CAE with the aim of making efficient optimization available to FE analysts. The challenges of creating TruPly, which are detailed within this paper, have been quite notable. Most significant of these has been to fully automated conversion of the native Abaqus/Standard input data format to the format required for internal optimization solver VR&D Genesis.

The results of the development are a new breed of optimizer for Abaqus users, providing many unique capabilities including strength based ply-pattern optimization, layup variation constraints and, most notably, a two-way interface between the optimizer and CATIA laminate definitions.

8. References

- [1] Dassault Systemes, *Simulia Abaqus and Abaqus/CAE Documentation*, version 6.12-1, 2012.
- [2] Vanderplaats Research & Development, *Design and Analysis Reference Manuals*, version 12.2, 2012.
- [3] Modelica Association, *Modelica Language Specification*, version 3.2, 2010