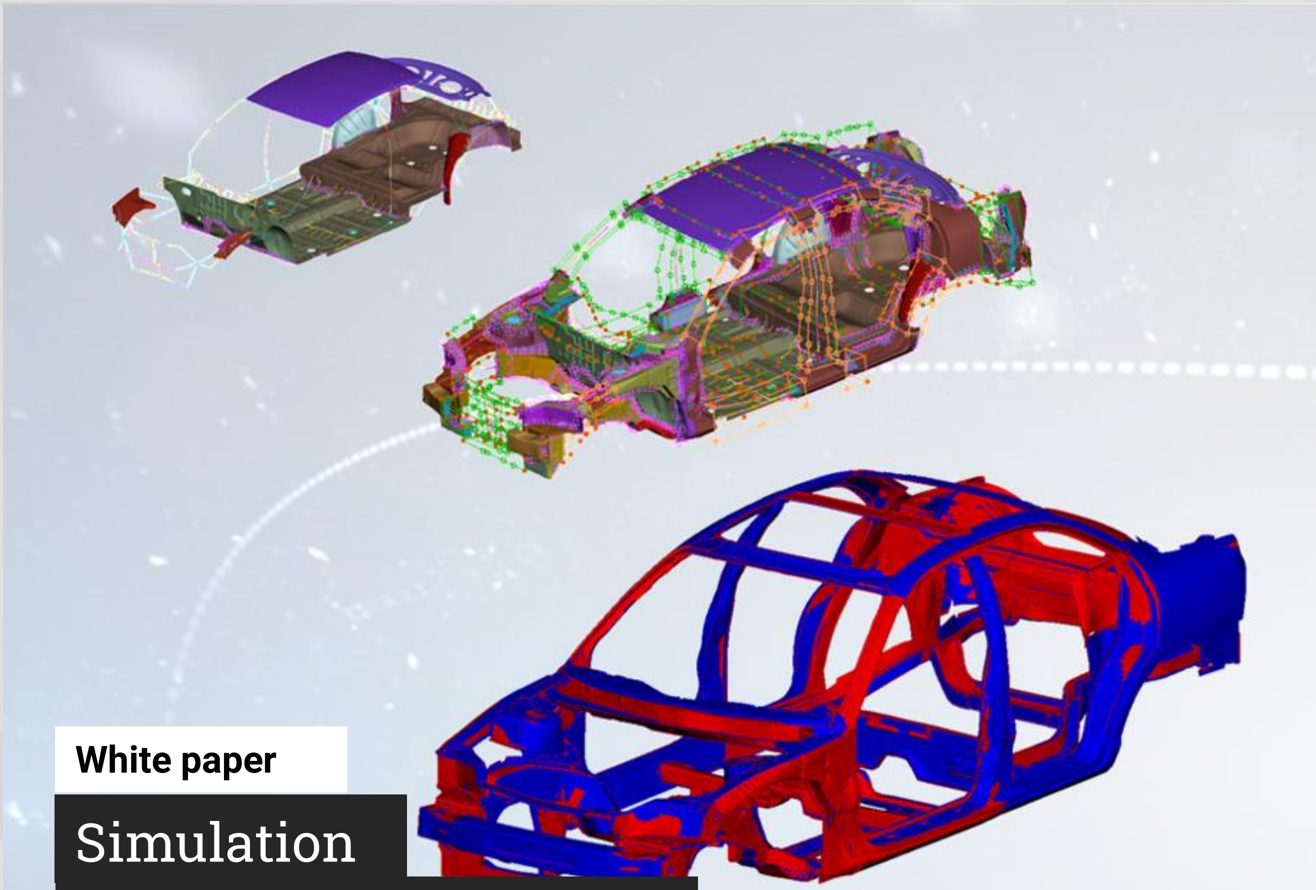
BETA
SIMULATION SOLUTIONS

physics on screen



White paper

Simulation enabling technologies

Beam Section Optimization tool: A BiW Optimization case

A new tool automatically simplifies a detailed Finite Element model to a beam-panel model assembly. Then, a Beam Cross section Optimization is performed on the simplified model. The optimum solution is automatically applied on the detailed model using mesh morphing methods, producing an updated detailed model used to verify the analysis' accuracy.

Performing Cross Section Optimizations in complex finite element models requires significant computational and time resources, even in the concept phase. With the aid of reduced models, engineers drastically decrease the optimization time while same time maintain the structural characteristics of the fully detailed model. However, the transition from a detailed shell model to a beam element model that is structurally sound and ready for optimization needs to provide control and to be robust and efficient.

In this case we convert a fully detailed ready-to-solve model to a reduced model consisting of beam elements, shell elements, and possible matrix elements. This is achieved through the Beam Section Optimization tool which provides the automation and the required options for the model reduction. We then proceed with a beam section optimization.

Several optimization cases are defined for the important cross sections of the vehicle's "members" that run much faster than the optimization of the fully detailed model. Several model reduction configurations are also defined and tested. These optimization cases are defined using either EPILYSIS SOL200 optimization on cross sections, or the DoE tool and an external optimizer software.

For validation purposes, the results of the optimization are then applied on the initial detailed model using morphing methods.

Model Reduction

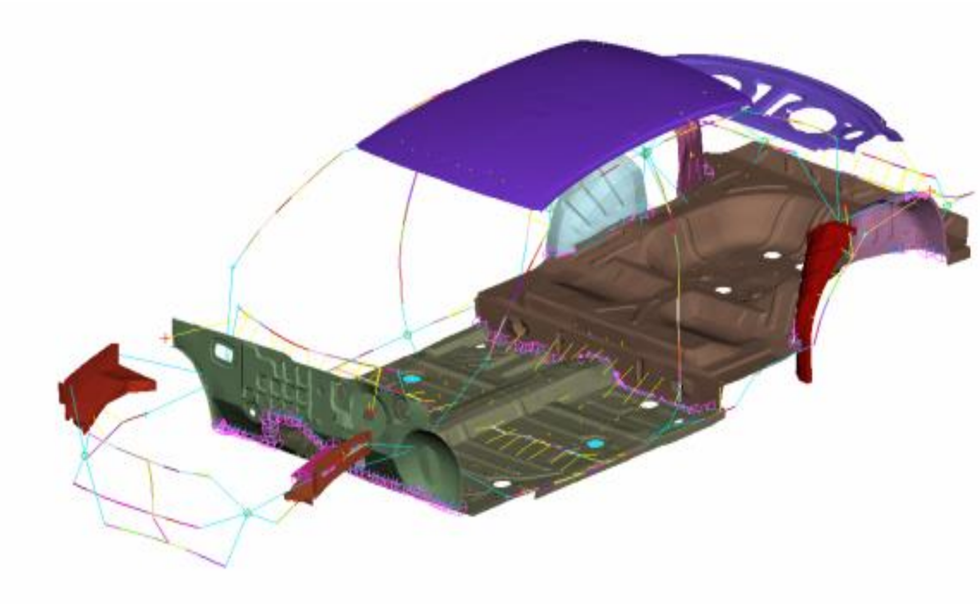


Figure 1: Reduced Optimization model

The main concern in the definition of the beam section optimization case is to reduce the model complexity and size to achieve faster analysis while maintaining the model's stiffness and structural behavior.

In our case, it was possible to reduce the model from a fully detailed finite element model with defined load cases, to a model consisting of beam elements (Figure 1). Also, several options allowed for use of body panels or junctions (referred as Nodes) in the reduced models. These models are referred to as optimization models.

It was expected that the optimization models would not accurately simulate the detailed finite element models, as by converting to beam element with different properties, different connections, and different representations at the "junctions", the behavior could not be the same.

A special tool was created to easily produce optimization models used for the beam cross section optimization. These were defined with the assumption that the optimization of such models could produce results that could be applied on the detailed models, resulting in improved models.

Model Options

More precisely, it was possible to create optimization models that consisted solely of beam elements replacing the beam-like sections of the model (referred to as Members), or a model that along with the beams, contains the shell element body panel areas.

Initially, the Members of the BiW were identified using the Wrap Morph boxes created around them (Figure 2). This way, the areas of the models that would be converted to beam elements were isolated and easily converted to beam elements by utilizing each member’s cross section.

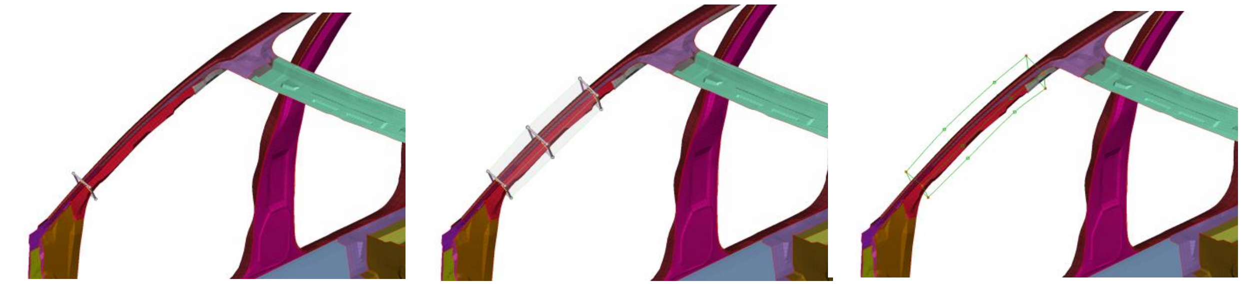


Figure 2: Wrap morphing boxes identify the Members

In case the model was selected to contain beams and body panels, the connection between the two, could automatically be applied with RBE2 or RBE3 elements to account for the connection between the members and panels of the detailed model (Figure 3).

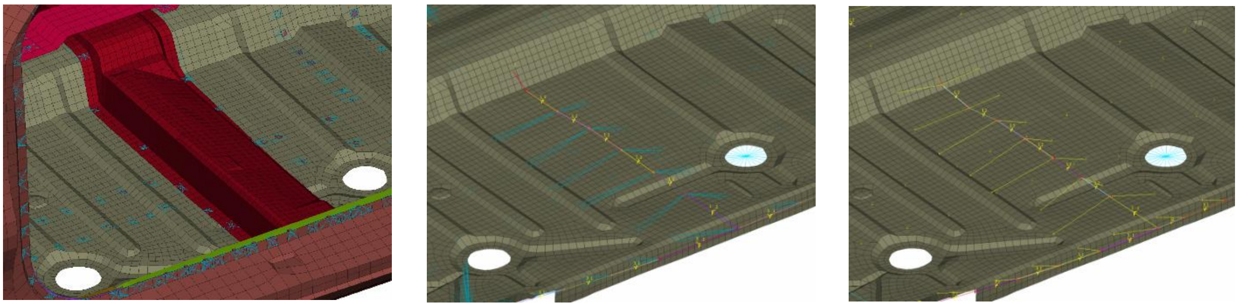


Figure 3: Original, RBE3 and RBE2 connections of Beams to Panels

Node Options

For the Nodes (junctions) of the model, there were three options. The first option was to maintain the shell element description of the original model. The second option was to discard the shell elements and connect the beam elements with a rigid RBE2 element. The third option was to use EPILYSIS Super Element creation capabilities to convert the Nodes into Super Elements (Figure 4). The fourth option was to discard the shell elements and connect the beam elements with CBUSH elements. This created BUSH properties with default stiffness so it was not used for this study. This option was more appropriate when the node stiffness is known.

Selecting the Shell description option had the advantage of a fast definition of the optimization model and increased accuracy in the area of the junction. Both stiffness and mass of each Node were correct and the Nodes were connected with the BEAM elements using RBE3 elements. Maintaining the shell element description produced an optimization model with a few thousand shell elements that affected the solution time during optimization.

Selecting the Rigid description option also had the speed advantage during the definition of the optimization model and reduced optimization time. The BEAM elements were directly connected with the RBE2 element and the mass of the removed Node elements was added at the center of the rigid element. Using a rigid element to connect all the beam elements in a junction increased the stiffness of the junction. This “artificial” extra stiffness produced an optimization model that behaved stiffer than the original model it replaced.

The matrix element – Super Element description option, initially required more time during the definition of the optimization model. Stiffness and Mass matrices were automatically calculated for each Node and applied in the optimization model as Super Elements (include files). This resulted in a smaller, lighter optimization model that maintained the accuracy of the original model and ran faster during the optimization cycles.

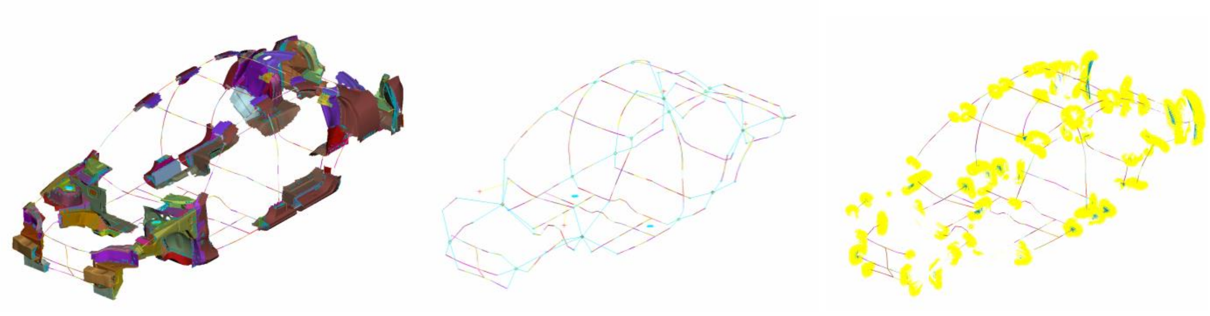


Figure 4: Shell, Rigid, Super Element Node options

Cross Section Options

Regarding the cross section of the beam element, there were two options concerning the effect of the body panel in the beam stiffness. The Closed cross section option used the Panel area, neighboring the Member structure, to create and calculate a closed section. On the other hand, the Open cross section option did not use the Panel area and created and calculated an Open section using only the Member structure (Figure 5).

This option provided the ability to create Optimization models that did not contain the Panel structures. Such models, free of shell elements, were able to run much faster during the optimization runs and the Closed cross sections option was necessary to account for the loss in stiffness.

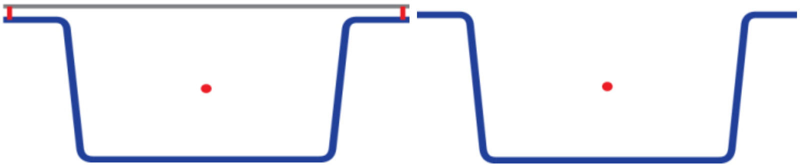


Figure 5: Closed and Open cross section options

Beam Property Options

The beam-like structures of the model could be converted to beam elements of three types of Beam element properties.

- Simple PBEAM property that calculates the properties of the current cross-section (Area, Moments of Inertia, Torsional stiffness, etc.) and applies these properties on the CBEAM element that is created. This type of property could not be used for optimization with the SOL200 Optimization method. It was used for DOE and parametric optimization studies.

- PBEAML_BOX property that calculates the properties of the current cross-section and converts the results into an equivalent box section. The shape of this equivalent box is controlled by dimension entities, like width, height, thickness, that can be used as design variables. This gave the ability to use this property for optimization with the dimensions (width and height) of the cross-section as design variables of a SOL200 optimization.

- PBMSECT property defines the shape of any current arbitrary cross-section and calculates its properties. This property provides Width, Height and Thickness entities of a current cross section’s segments that can be used as design variables. In this paper Width and Height parameters were defined for each cross section and used for a SOL200 optimization.

Optimization model

Regardless of all the mentioned options, optimization models as seen in Figure 6 were automatically defined, creating design variables that control the width and height of each cross section.

The general concept was that during the optimization's iterations (SOL200, DoE or parametric) the design variables of each Beam property would be modified, thus changing the element's properties (Area, Moments of Inertia, Torsional stiffness, etc.). This would lead to a new model with different behavior. Eventually, an optimum design would be created.

With this concept in mind, it was possible to create the Optimization models for SOL200 Optimization, using PBEAML_BOX or PBMSECT properties. In this type of optimization, the design variables that control the dimensions of each cross section were controlled by the solver internally, fact that decreased optimization time significantly. After a number of cycles, the optimization converged, based on the defined objective and constraints.

It was also possible to create optimization models that could be used for DoE studies or a parametric optimization with external optimization software. This type of models utilized morphing boxes, in order to modify the dimensions of the cross-sections and alter each beam's properties. An Optimization tool was used to perform the DoE and connect the pre-processor to an external optimization software.

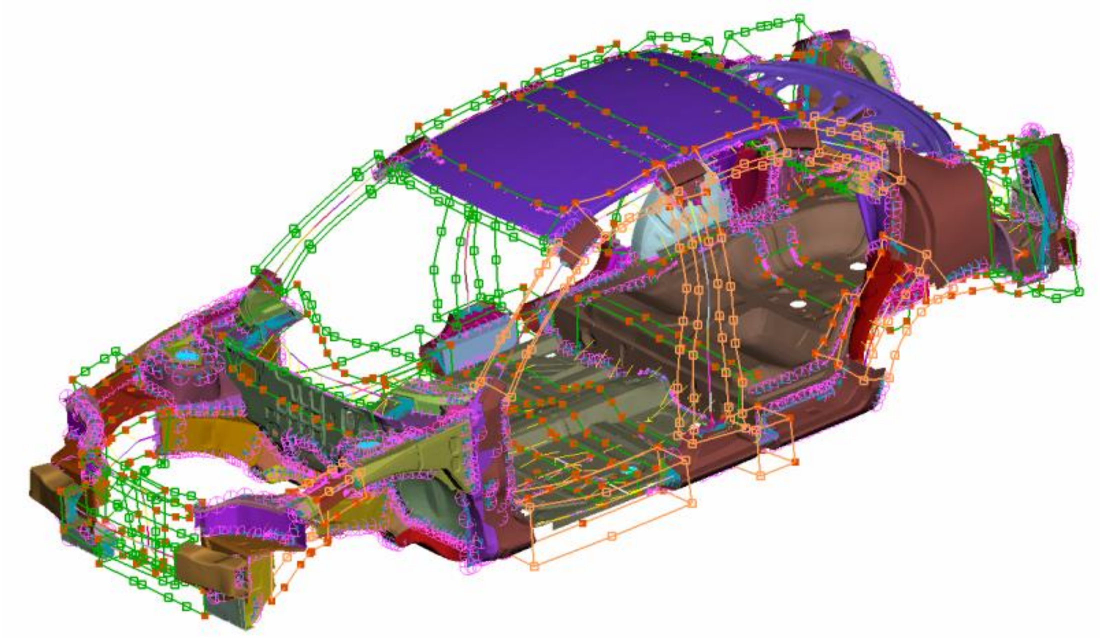


Figure 6: Ready to run Optimization model

Design of Experiments (DoE) / Parametric Optimization

With the DoE/Parametric option, two design variables were created for each box of each member, controlling the members' width and height (Figure 7).

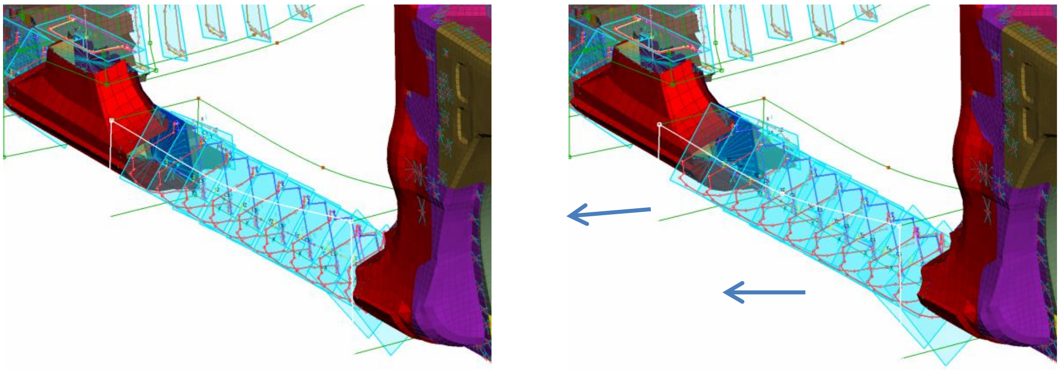


Figure 7: Controlling cross section shape using morphing boxes

All the design variables for the shape of the cross sections were included in the Optimization tool (optionally, thickness design variables for each Member could also be added). The design variable bounds were added as a percentage of the width or height of each cross section. Each design variable was connected with a morphing box and could morph and modify the curves of each cross section of a Member, while the cross section's properties were automatically recalculated. This way, a new cross section shape was defined for each member at each cycle.

The DoE was defined using ANSA, EPILYSIS solver for the analysis, and META for post-processing. An algorithm was used, in order to generate the experiments. Each experiment ran independently from the others and the results appeared in lists and charts, allowing the quick detection of the best design.

SOL 200 Optimization

During the definition of the model for the SOL200 Optimization, specific design variables (DESVAR) and solver entities that control the properties (DVPREL) were automatically defined for all cross sections. Design variables were created for each Member's width and height and controlled the respective DVPREL entities of each of its cross sections.

In the same time, the bounds of the design variables were calculated as a percentage of the current cross section's width and height. The design variables were used by EPILYSIS SOL200 to optimize the cross sections of each member. The objective function of the optimization was to minimize the compliance of the model when subjected to two static loading conditions, simulating Torsion and Bending.

Update model

In order to evaluate the performance of the process and the results of the solver, the DoE or SOL200 optimization results were applied on the initial model.

Using a specific functionality, the initial model was automatically morphed and acquired an updated shape, according to the design variable values of the best design of the DoE or the Optimization's final iteration. Advanced morphing methods were automatically used to morph the Nodes (junctions), in order to provide a smooth and seamless transition between the adjacent Members of the BiW (Figure 8).

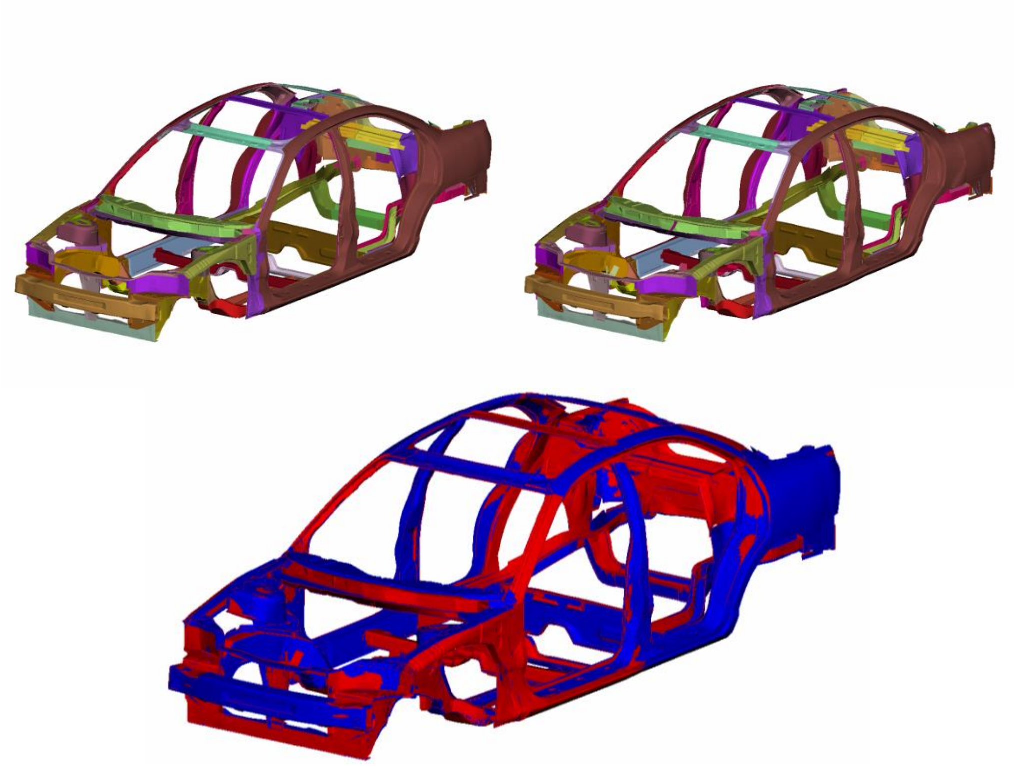


Figure 8: *Original, Updated model and overlay*

The final/updated model was analyzed under the same loading conditions, in order to be compared with the original model and assess the validity of the process.

Case Study

This part of the study was focused on identifying the configuration of the optimization model that would provide the best optimization results.

Body in White

The configurations that appear in Table 1 produced reduced optimization models that behaved similar to the initial model, and also produced the best Optimization results.

Table 1: Optimization model configurations

Model Description					
Model id	**Property Type**	**Section Type**	**Beams to Panels**	**Node Type**	**Model Type**
1	PBEAM	Open	RBE3	Super Element	Frame w Body Panels
2	PBMSECT	Open	RBE2	Shell	Frame w Body Panels
3	PBMSECT	Open	RBE3	Shell	Frame w Body Panels
4	PBEAML	Open	RBE3	Super Element	Frame w Body Panels
5	PBEAML	Closed	RBE2	Super Element	Frame w Body Panels
6	PBEAML	Closed	RBE3	Super Element	Frame w Body Panels
7	PBEAM	Open	RBE2	RBE2	Frame w Body Panels
8	PBEAML	Closed	RBE2	SHELL	Frame w Body Panels
9	PBEAML	Closed	-	Super Element	Frame
10	PBEAML	Closed	-	SHELL	Frame

Results

Initial detailed model vs Optimization models

The model was subjected to two structural load cases, Torsion and Bending, and one normal mode analysis. Displacements of the nodes, where the load was applied, were used in order to evaluate the stiffness of the model. In Table 2, the comparison between the original detailed shell model and the various optimization models is displayed.

The results from the optimization models should be as close as possible to the results of the initial model.

Table 2: Optimization models initial results

	SOL 101-Subcase 1 Displacement (mm)	SOL 101-Subcase 2 Displacement (mm)	1st elastic mode Frequency(Hz)
Shell model	**3.833**	**5.783**	**21.18141**
1	5.675	9.0987	22.00288
2	5.153	8.242	20.76343
3	5.058	7.848	19.2318
4	5.0533	7.9876	19.05221
5	5.1897	8.137	18.21233
6	4.9133	7.7295	19.07317
7	1.3	2.286	42.78797
8	5.2596	8.4819	17.72121
9	12.356	18.977	12.4667
10	13.811	21.835	12.05859

It is clear that while there are some expected differences between the initial model and the reduced optimization models, the majority of them provided valid models to be used for optimization purposes.

Sol 200 Optimization and DOE

According to each model's configuration, the optimization models were then used for optimization with SOL2OO or a DOE study for twenty cycles. For this part of the study, only the structural load cases were used in the Optimization analysis and not the normal modes analysis.

Responses from the analysis could be used as design objectives and design constraints, for all the models that run for optimization with SOL200. The compliance of the model was used as the objective function, in order to be minimized, aiming to increase the BiW's stiffness. This way, the Optimization algorithm changed the shape of the cross sections of the BiW's members, searching for a better/stiffer result. Weight, displacement or other responses could be used as constraints. In this study, no constraints were used, to achieve faster convergence. The results of the optimization runs are listed in the Table 3. In the end, the optimum results were applied on the initial shell models using morphing methods, as described in chapter 5.

Table 3: SOL200 optimization results

	SOL 101-Subcase 1 Displacement (mm)	**SOL 101-Subcase 2 Displacement (mm)**	**SOL 200 S1 Displacement (mm)**	**SOL200 S2 Displacement (mm)**	**Average Improvement %**
Shell model	**3.833**	**5.783**	-	-	-
2	5.153	8.242	4.79	7.75	6.5
3	5.058	7.848	4.85	7.65	3.31
4	5.0533	7.9876	4.93	7.82	2.26
5	5.1897	8.137	5.05	7.96	2.43
6	4.9133	7.7295	4.87	7.7	0.63
8	5.2596	8.4819	4.91	7.97	6.34
9	12.356	18.977	12.012	18.535	2.55
10	13.811	21.835	12.597	20.2661	7.98

For the models where DoE was selected, initially, twenty experiments were performed, collecting the displacement responses at crucial areas of the model, to evaluate its structural behavior. The results of the DoE runs appear in Table 4.

Table 4: DoE results

	SOL 101-Subcase 1 Displacement (mm)	**SOL 101-Subcase 2 Displacement (mm)**	**DOE S1 Displacement (mm)**	**DOE S2 Displacement (mm)**	**Average Improvement %**
Shell model	**3.833**	**5.783**	-	-	-
1	5.675	9.0987	5.664	8.938	0.98
7	1.3	2.286	1.23	2.21	4.35

Update models and validation

Result files from the SOL200 optimization, the DoE, or the parametric optimization were used in the update process. In an automated way, the design variable values for the shape of the cross sections of a BiW Members were applied on morphing boxes of the original model, which controlled the shape of the FE mesh of these Members. Each Member got an updated shape. The Nodes (junctions) were also updated, smoothly, following the movement of the adjacent Members, resulting in a naturally flowing shape without discontinuities.

The same structural load cases were used for the updated models as well, in order to acquire the results and evaluate the process. The results from the updated shell models are listed in Table 5.

Table 5: Updated models results

	Updated SOL 101-Subcase 1 Displacement (mm)	**Updated SOL 101-Subcase 2 Displacement (mm)**	**Average Improvement %**
Shell model	**3.833**	**5.783**	-
1	3.7537	5.6756	1.963021557
2	3. 532	5.473	6.60
3	3. 652	5.496	4.84
4	3.807	5.7239	0.838400236
5	3.6175	5.4492	5.697159315
6	3.9304	5.7988	-1.407152562
7	3.6345	5.5018	5.020619646
8	3.9068	5.784	-0.97133844
9	3.904	5.7658	-1.074879234
10	3.896	5.764	-0.98608519

It is clear from Table 5, that most models managed to improve under both load cases, exhibiting less displacement at the measuring areas, evidence of improved stiffness, which was the objective.

Specifically, Models 2, 5 and 7 achieved the best results, reducing the displacement by more than 5% under both Torsion and Bending load cases.

The improvement may seem small, however, only twenty cycles of optimization were selected for each model, to reduce the optimization time to less than ten minutes.

The respective parametric studies (DoE or Optimization) required eighty to ninety minutes, providing similar results, indicating the potential of the SOL200 Optimization for beam section optimization. In comparison, a similar type of optimization with the fully detailed shell model would require a few hours due to increased analysis time.

Parametric Optimization with external optimizer

In continuation of the parametric DoE studies, and in order to evaluate the non-parametric SOL200 performance against the parametric optimization, external optimizer software was used to get the optimum results from this method. The two models with PEAM properties (Model 1 and Model 7) were also used for an optimization run with an external optimizer (Figure 9).

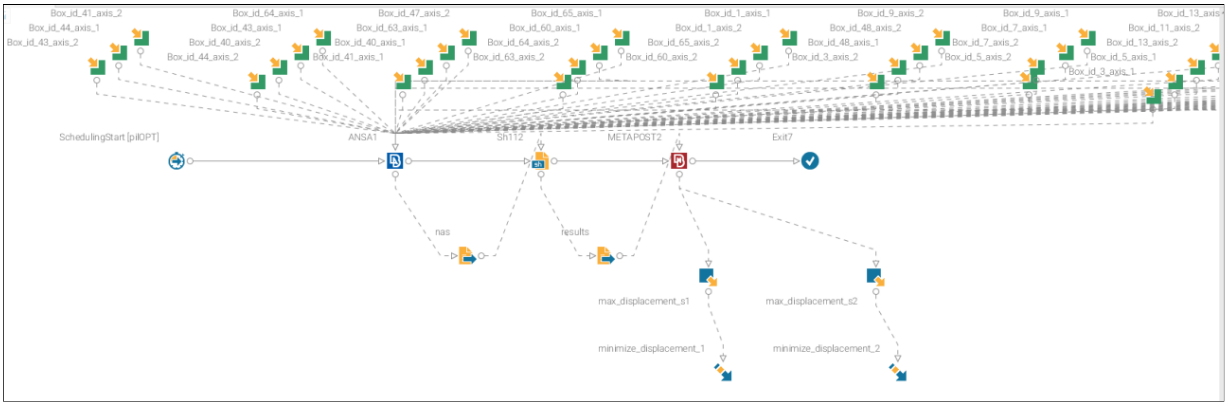


Figure 9: Parametric Optimization

In Table 6, the results of the optimization models are listed compared with the original shell model and the initial reduced models. Both models had slightly increased performance, however due to the nature of the study, the number of iterations of the optimization was kept very small. Ten designs of an optimization algorithm and twenty iterations were not sufficient for a problem with that many design variables like the current one. The trend of the optimization showed that more iterations would provide better results.

Table 6: External Optimizer results

Model	SOL 101-Subcase 1 Displacement (mm)	SOL 101-Subcase 2 Displacement (mm)	Optimum s1	Optimum s2
Shell	**3.833**	**5.783**	-	-
1	5.675	9.098	5.615	8.872
7	1.3	2.286	1.19	2.1

The results of the updated morphed models using the results of the Optimization are listed in Table 7. The improved performance of the optimum models did pass through to the initial model; however, a larger number of iterations would provide more significant improvements. Model number 1 showed the biggest improvement of all configurations, using the external optimization software.

Table 7: Updated optimized models results

Model	Updated SOL 101-Subcase 1 Displacement (mm)	Updated SOL 101 Subcase 2 Displacement (mm)	Average Improvement %
Shell	**3.833**	**5.783**	-
1	3.526	5.40	7.316126114
7	3.829	5.767	0.190514954

RSECTBT element

Research is ongoing concerning the entities that connect the beams to the residual structure. It was observed that these connection elements significantly affect the structural behavior of the optimization models. Improvements in this area are expected to increase the accuracy of the optimization models.

New R type element

A new R type element is introduced for Epilysis solver models called RSECTBT, in order to increase precision, as a connection between a beam and the shell structure.

Until now, the connection between the beam elements and the residual structure was done by RBE2 or RBE3 elements (Figure 10).

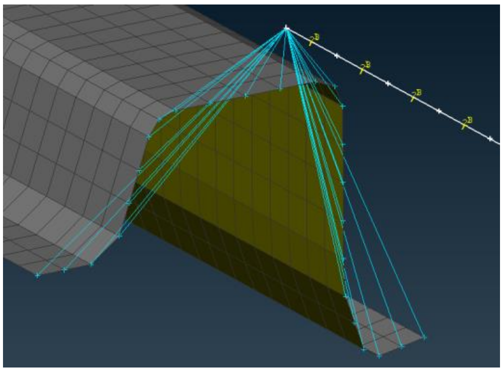


Figure 10: Beam connected to the shell structure with RBE2 element

The main restriction of the RBE2 elements is that all the relative displacements between dependent nodes are restricted. This induces an extra stiffness to the connection, which is not ideal.

RBE3 elements with weighting factors describe a motion of a single dependent node as a weighted average of the degrees of freedom (d.o.f.) of multiple nodes. As such, the RBE3 constitutes a more flexible connection than the original shell structure.

The new suggested RSECTBT is a Multi Point Constraint element consists of six d.o.f. that describe the rigid body motion of the cross section and six d.o.f. that describe the six other types of deformation, one for each load case (Figure 11). This enables all nodes of the cross section to Translate and Rotate according to six different types.

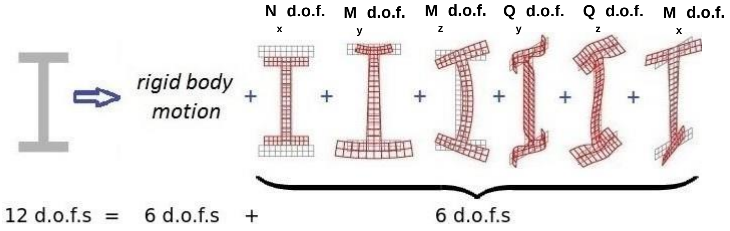


Figure 11: New element degrees of freedom

To incorporate these extra unknown factors, a new cross section solver engine was developed, aiming to calculate in and out of plane deformations for all load cases.

The solver calculates the coordinates of the mass and shear center and the orientation of the principal coordinate system. It also calculates the moments of inertia for bending about major and minor principal axis, torsional stiffness (I_t), and Normal and Shear stresses for all cases of applied loads.

Our studies showed that the results of using the RSECTBT on the reduced model are much closer to the original structure than both RBE2 and RBE3 elements. A limited stiffening effect with respect to the original structure is introduced.

RSECBT Element on BiW

In this study, one of the previously used models of the BiW was used, in order to evaluate the behavior of the new RSECTBT element. The initially created model uses RBE3 elements for the connection of the beams with the residual structure. The model was analyzed under static loads, with a Torsion and a Bending load case, and with a normal modes analysis.

The same model was automatically modified in a way that all RBE3 elements, that connected the beam elements with the residual structure, were converted to RBE2 elements. The same analyses ran for this model as well, for comparative purposes.

Finally, the new RSECTBT element was used to replace the RBE2 elements in the reduced BiW model.

RSECTBT element results

The results in Table 8 and Figure 12 show that the models with the RSECTBT elements connecting beams to the Nodes, exhibits an improved behavior, over the behavior of both models with RBE2 and RBE3 elements. As expected, the RBE2 elements add stiffness to the model and this is visible in both structural and normal modes analyses. The RBE3 creates a softer connection between the beam element and the shell structure, which results in a more compliant model.

Table 8: R-type element comparison results

	Shell	RBE2	RBE3	R-Type
Static Loadcase 1 Torsion (mm)	3.83	2.29	5.479	3.1
Static Loadcase 2 Bending (mm)	5.78	3.45	8.79	5.2

The RSECTBT element seems to add some stiffness, resembling the RBE2 element, but in all load cases, the results are much closer to the fully detailed shell model.

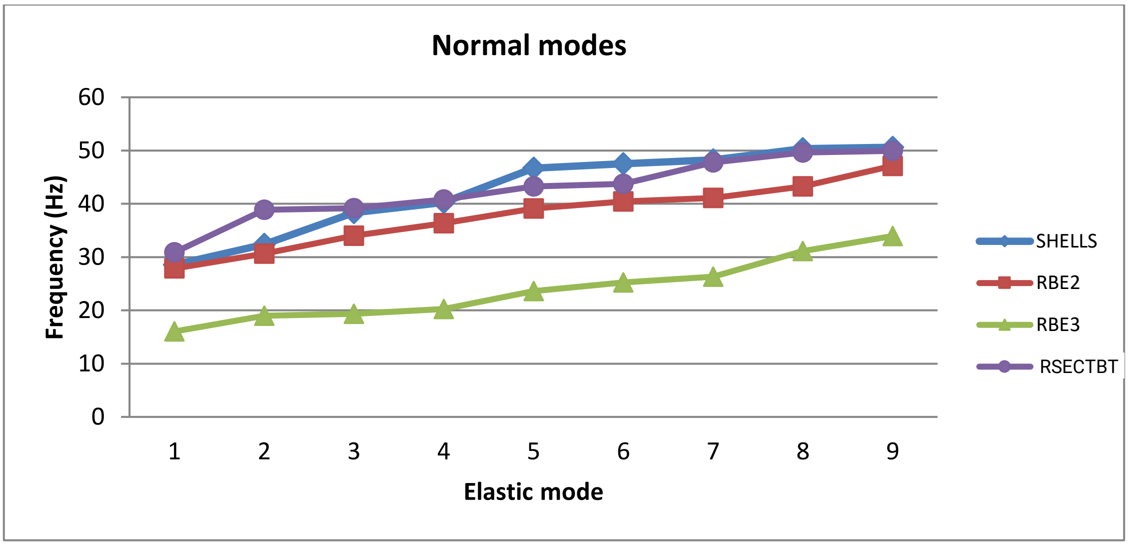


Figure 12: Elastic modes 1 to 9 for the three configurations compared to the shell model

Conclusions

The results of this study confirm that it is possible to use reduced beam element models, to optimize the cross sections of the members of a body in white. Choosing the reduced beam element models for optimization, provides fast definition of multiple optimization models and also faster optimization runs. Reduced beam element models can simulate the behavior of detailed finite element models, within logical expectations, taking into consideration the important simplifications that the model was subjected to.

The multiple configurations used to simplify the detailed model and the fast definition of the optimization models allowed for the creation of multiple optimization models. Out of these optimization models, the ones that better simulated the original model were used for optimization analyses.

All optimization options provided improved results. However, SOL 200 Optimization run much faster than the DoE (Design of Experiments) or the optimization with external optimization software. On the other hand, the external optimization software delivered the best results.

With the ability to update the original detailed finite element model using the optimization results, it was possible to produce new designs of detailed models, which had improved performance over the original model.

About BETA CAE Systems International AG
BETA is a simulation solutions provider, dedicated to the development of state of the art software systems for CAE. For almost 30 years, we have been developing tools and delivering services for the front-runners in numerous sectors by listening to their needs and taking up even the most demanding challenges. For more information on BETA CAE systems, our products, and our services, visit www.beta-cae.com

Headquarters
D4 Business Village Luzern, Platz 4
CH-6039 Root D4,
Switzerland
+41 415453650

BETA
SIMULATION SOLUTIONS

physics on screen

www.beta-cae.com