SIMPLIFIED REPRESENTATION OF COMPLEX STRUCTURAL COMPONENTS FOR FINITE-ELEMENT-ANALYSIS

L. Reichert[1], J. Krieglsteiner[1], C. Schmidt[2], and P. Horst[1]

[1]Institute of Aircraft Design and Lightweight Structures, Technische Universität Braunschweig,
Ottenbecker Damm 12, 21684 Stade, Germany
Email: lisa.reichert@tu-braunschweig.de, Web Page: http://www.hpcfk.de
[2]Institute of Production Engineering and Machine Tools, Leibniz Universität Hannover,
Ottenbecker Damm 12, 21684 Stade, Germany

Keywords: aerospace, aircraft fuselage design, composite design, simplified representation, stiffened shell structures, structural design, omega stiffener

Abstract
Simplified representations of stiffened shell structures are needed for the FE-analysis of aircraft fuselage structures on a global level. Two methods are presented to derive surrogate stiffness parameters of complex structural components. The first method is based on FE-simulations, the second uses an analytical approach. Both methods are compared to a conventional analytical approach.
For the FE-based approach, the structural response to characteristic load cases of a detailed model is used to calculate beam properties. These properties are used to set up a simplified FE-model. To validate the results, the structural behavior of both models is compared. A conventional analytical approach does not respect the increased complexity of the component and leads to a mean error of 30 % compared to the validated FE-results. Therefore, a second, refined analytical approach is developed, which takes the complexity into account by discretizing the component into simple segments. From this, parameters of the entire component are derived using a spring analogy. This approach is shown to reduce the mean error to 2 %.

Symbols

ABD	*stiffness matrix*	h	*height*	w	*width*
D	*deviation*	l	*length*	x, y, z	*coordinate (ce system)*
EA	*extensional stiffness*	m	*moment flux*	Φ	*rotation*
EI	*bending stiffness*	n	*force flux*	ε	*strain*
M	*moment*	t	*thickness*	ζ, η	*coordinate (origin system)*
N	*normal force*	u	*displacement*	κ	*curvature*

Indices

N	*normal load*	ce	*center of elasticity*	s	*start, support*
M	*bending load*	cs	*shear center*	start	*start*
all	*overall system*	e	*end*	skin	*skin*
appl	*applied load*	l	*load introduction*	x, y	*x, y axis*
beam	*stiffener beam*	mid	*middle section*	1, 2	*load case 1, 2*

1. Introduction

The need for weight reduction in structural systems can cause an increased complexity in their design. This complexity refers to the system's components' geometry and material. Composite materials such as carbon fiber reinforced plastics (CFRP) introduce a new level of design variability resulting from orthotropic laminates and new production technologies providing fewer limitations for component

geometry. One example for this is the double curved stringer introduced by [1] for use in aircraft fuselage structures. Another example for a fuselage stiffening structure design as proposed by [2] includes several complex components such as slanted stiffeners and a grid structure, which requires complex stiffener intersections.

The design process of large structures requires numerical modeling at different detail levels. In a global model, simple representations of the structural components are used to determine deformations required as boundary conditions for local models. The local model allows a more detailed representation of each component and it is used in the component design. In order to correctly determine the deformation by utilizing the global model, the behavior/stiffness of each component has to be modeled with sufficient accuracy using only one or a few elements. For conventional, simple structural components, an analytical approach can be used to directly convert their geometrical and material characteristics into properties of a simple single beam element. For more complex components, this analytical approach is not directly applicable.

To extend this analytical approach, the characteristics of the beam element have to be derived from the complex geometry and/or possible non-homogeneous lay-ups. For CFRP stiffeners with a complex but constant profile this was already done in [3]. As an extension to this, the focus of the presented work is placed on obtaining the surrogate stiffness of complex components with variable cross sections along their main axis, as for example in a double curved stiffener.

The different structural components that are analyzed in this paper are all used to stiffen thin skin segments. As the stiffening component interacts with the skin underneath it, both are included in the model. The focus of the presented work is to find a way to model the stiffening component on top of the skin element using only a single beam element while keeping the structural behavior of the overall system comparable. Therefore, all models that are considered in this paper include a stiffening component, for which a simplified representation is derived together with a skin segment underneath. A finite element (FE) mesh of skin segment stiffened by a double curved stiffener is shown in Figure 1.
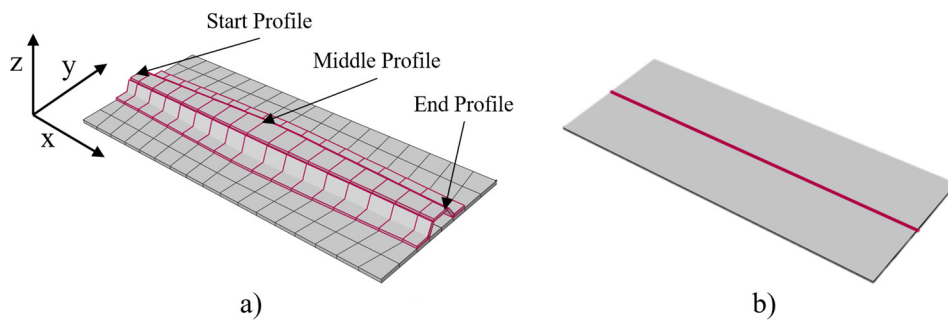

Figure 1: FE-mesh of the detailed model (a) and the simplified model (b)

The specific methods used in this paper are only applicable for stiffeners, which are symmetrical to the x-z plane. A schematic sketch for an omega cross section used in this paper is shown in Figure 2. The different coordinate systems, which are used are also indicated.
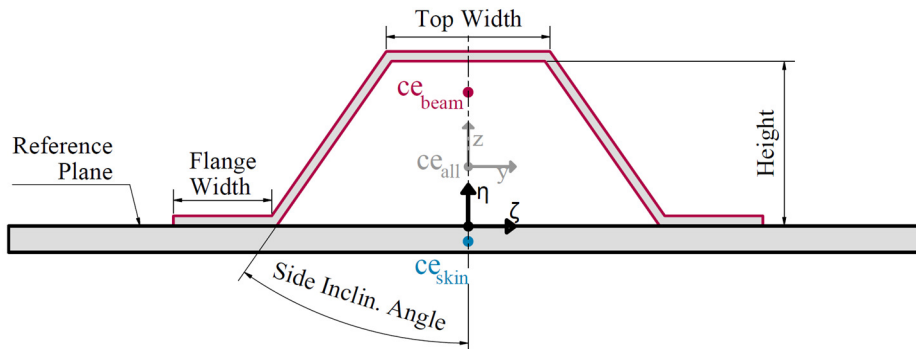

Figure 2: Schematic sketch of a skin stiffened by an omega stiffener as used in the detailed FE-model. The centers of elasticity of the skin, the beam and the overall system are also indicated.

L. Reichert, J. Krieglsteiner, C. Schmidt, and P. Horst

The beam element used for the simplified model assumes a constant cross section along its main axis and is described by only a few parameters. In Abaqus® a Bernoulli beam element can be defined with a closed profile, which requires the following parameters: center of elasticity (ζ_{ce}, η_{ce}), shear center (ζ_{cs}, η_{cs}), extensional stiffness (EA_x), bending stiffness (EI_y, EI_z), and torsional stiffness (GI_x) [5]. Due to their increased importance in the design process of stiffened structures, the presented work only focuses on the extensional stiffness EA_x, the bending stiffness EI_y and the height of the center of elasticity η_{ce}.

The goal of the presented work is to derive these parameters from the detailed geometry definition of the stiffening element. In order to find a suitable analytical method to do this, the following steps are carried out.

In a first approach, different FE-simulations are carried out to derive the required parameters from a detailed model of the structure.. These parameters are validated by applying them to the beam element in the simplified model and comparing its behavior to the detailed model. The parameters obtained by using this first approach are based on an additional FE-simulation for every different component configuration. For the application in design optimization, however, a less time consuming analytical approach is needed. The second approach is based on a method to mechanically characterize stiffener profiles as introduced in [3], which is applied to the different sections of the stiffener along its main axis. The resulting parameters are compared to the validated FE-results.

2. Characterization: FE-Approach

2.1. Modeling

For the analysis, two different FE-models are required as shown in Figure 1. The difference between the two is the level of detail, in which the geometry of the structure is modeled. They both represent the same geometry and have the same size, which corresponds to a model on a local level. In the detailed model, both, skin and stiffener, are modeled using several shell elements. In the simplified version, the skin is modeled using two shell elements, one on each side of the stiffener, while the entire stiffener is reduced to one single beam element. This model represents the desired simplification of the structural component. The detailed model is used to characterize the structure and to calculate the target values for the surrogate stiffness parameters.

In order to calculate the required stiffness data from the detailed model, two different load cases are applied. They are chosen in such a way that it is possible to deduce the stiffness parameters from the structural response. All loads are applied to the load master node, which is indicated in Figure 3a. For load case 1, the following unit loads are applied: $M_{y,appl,1} = 1 \, \text{Nmm}$, $N_{x,appl,1} = 0$. For load case 2, the loads are: $M_{y,appl,2} = 0$, $N_{x,appl,2} = 1 \, \text{N}$.
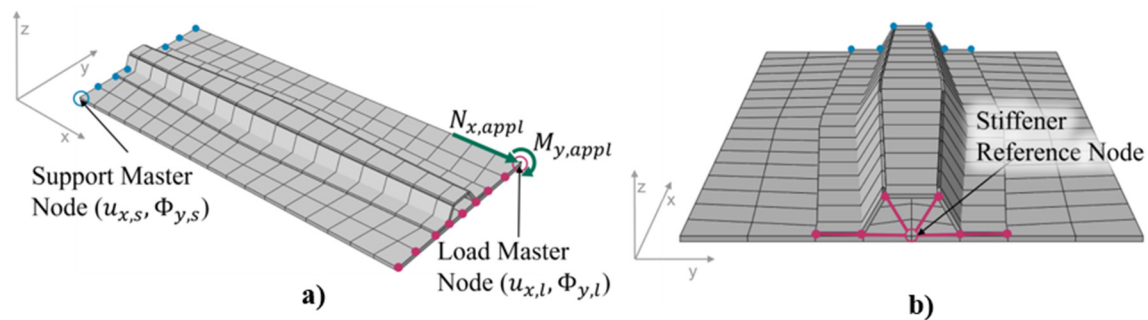

Figure 3: Load introduction and boundary conditions of the detailed FE-model

The remaining nodes on the load master node side of the skin form the *load node set* (indicated in red in Figure 3a). Each of these nodes is coupled to the load master node using the following equations:

$$u_{x,i} = u_{x,l}, \qquad for \; i \; \in load \; node \; set$$

L. Reichert, J. Krieglsteiner, C. Schmidt, and P. Horst

$$\Phi_{y,i} = \Phi_{y,l}, \qquad for \ i \ \in load \ node \ set$$

On the opposite side, the support master node is defined. For this node, every degree of freedom is fixed. The remaining nodes on this side, at the reference plane, form the *support node set* (indicated in blue in Figure 3a). The following equations are applied to each of these nodes to couple them to the support master node:

$$u_{x,j} = u_{x,s} = 0, \qquad for \ j \ \in support \ node \ set$$

$$\Phi_{y,j} = \Phi_{y,s} = 0, \qquad for \ j \ \in support \ node \ set$$

So far, only the application of the loads and the boundary conditions (BC) to the skin were discussed. In order to apply both to the stiffener profile nodes, an additional stiffener reference node is defined on each side. This node is located at the reference plane (as indicated in Figure 3b) and part of the *load node* or the *support node set*. Therefore, it is coupled to the respective master node as described above. The remaining stiffener nodes on each side (see Figure 3b) are kinematically coupled to the respective stiffener reference node.

2.2. Calculation of Stiffness Parameters

The overall model is treated as a single beam with the following properties: $\eta_{ce,all}$, $EA_{x,all}$, $EI_{y,all}$. These parameters are directly calculated from the results of the FE-simulations of the detailed model.
In general, the loads acting on the beam have to be differentiated from the loads applied to the model. As the normal force $N_{x,appl}$ is not applied to the center of elasticity, it also contributes to the total bending moment M_y.

$$N_x = N_{x,appl} \qquad M_y = M_{y,appl} + N_{x,appl} * \eta_{ce,all} \qquad (1), (2)$$

The following equations are set up for the known resulting master load node displacements u_x and rotations Φ_y. The total displacement u_x, as read from simulation results, is composed of the reaction to the normal force $u_{x,N}$ and the reaction to the bending moment $u_{x,M}$.

$$u_x = u_{x,N} + u_{x,M} \qquad \Phi_y = \Phi_{y,M} \qquad (3), (4)$$

The following equations are used to calculate the stiffness parameters from the known applied loads and the resulting displacements/rotations:

$$EI_{y,all} = \frac{M_y * l}{\Phi_y} \qquad (5)$$

$$EA_{x,all} = \frac{N_x * l}{u_{x,N}} \qquad (6)$$

For load case 1, since $N_{x,appl,1}=0$, the second part of equation (2) equals zero, too. Therefore, the bending moment $M_{y,1}=M_{y,appl,1}$ and the rotation $\Phi_{y,1}$ as read from the simulation results can be used to calculate $EI_{y,all}$.
For load case 2, the superposition of the resulting displacements described in equation (3) has to be considered. Due to the linearity of the FE-simulations, the bending induced component $u_{x,2,M}$ can be calculated using the resulting displacement and rotation from the first load case.

$$u_{x,2,M} = u_{x,1} \frac{\Phi_{y,2}}{\Phi_{y,1}} \qquad (7)$$

With the known bending induced component, the required force induced displacement is calculated using equation (3). This can directly be inserted into equation (6) to calculate the extensional stiffness $EA_{x,all}$.

L. Reichert, J. Krieglsteiner, C. Schmidt, and P. Horst

$$EA_{x,all} = \frac{N_{x,2} * l}{u_{x,2} - u_{x,1} \dfrac{\Phi_{y,2}}{\Phi_{y,1}}} \tag{8}$$

In order to calculate the center of elasticity $\eta_{ce,all}$, the bending moments for both load cases $M_{y,1}$, $M_{y,2}$ and the resulting rotations $\Phi_{y,1}$, $\Phi_{y,2}$ are required. Since the bending stiffness has to be constant, equation (5) can be set up for both load cases. Combined with equation (2) set up for load case 2, $\eta_{ce,all}$ is calculated as follows:

$$\eta_{ce,all} = \frac{-M_{y,1}}{N_{x,2}} * \frac{\Phi_{y,2}}{\Phi_{y,1}} \tag{9}$$

The stiffness data of the skin is calculated analytically from the given geometry and the laminate that is used. Since only symmetrical lay-ups are considered:

$$\eta_{ce,skin} = -\frac{t_{skin}}{2} \tag{10}$$

The stiffness of the skin is obtained from the stiffness matrix [**ABD**]. A characteristic load is applied to the matrix: a bending moment flux m_y to calculate the bending stiffness and a force flux n_x to calculate the extensional stiffness, respectively. Just like in the boundary conditions of the FE-model, transverse contraction effects are not restricted in the following formulas.

$$\kappa_y = [\mathbf{ABD}]^{-1}_{5,5} * m_y \tag{11}$$

$$\varepsilon_x = [\mathbf{ABD}]^{-1}_{1,1} * n_x \tag{12}$$

The resulting distortion, combined with the applied load, is used to calculate the bending and extensional stiffness:

$$EI_y = \frac{m_y}{\kappa_y} * w_{skin} \tag{13}$$

$$EA_x = \frac{n_x}{\varepsilon_x} * w_{skin} \tag{14}$$

In order to derive the beam parameters from the overall and skin parameters, the stiffener and skin are treated as two separate beams in a parallel connection. The following equations for the stiffener beam parameters are derived:

$$EA_{beam} = EA_{all} - EA_{skin} \tag{15}$$

$$\eta_{ce,beam} = \frac{EA_{all} * \eta_{ce,all} - EA_{skin} * \eta_{ce,skin}}{EA_{beam}} \tag{16}$$

$$EI_{beam} = EI_{all} - EI_{skin} - EA_{skin} * z^2_{ce,skin} - EA_{beam} * z^2_{ce,beam} \tag{17}$$

For the calculation of the extensional stiffness and center of elasticity, all parameters are known and the values can be calculated directly. In order to calculate the bending stiffness, the coordinates of the components' height of the center of elasticity is required. For this, the known coordinates in the principal coordinate system are transformed using the following equation:

$$z_{ce,comp} = \eta_{ce,comp} - \eta_{ce,all} \tag{18}$$

The resulting beam parameters are then used to define the beam element in the simplified model.

L. Reichert, J. Krieglsteiner, C. Schmidt, and P. Horst

2.3. Validation Beam-model

To validate the model, the described analysis is carried out for several double curved stiffener configurations, in which the following stiffener parameters are varied: height, top width, and side inclination angle for the start/end profile and the middle profile. The deviation D between the resulting overall stiffness parameters of the detailed and simplified model are summarized in Table 1. For all parameters, the deviation between the results of the detailed and simplified model is significantly below 0.001 %. The obtained beam parameters are used as the reference for the analytical approach.

Table 1: Comparison of the detailed and simplified FE-results

	EA_{all}	EI_{all}	$\eta_{ce,all}$
$|D|_{max}$ [%]	0.0003	0.0005	0.0008
$|D|_{min}$ [%]	0.0002	9.8e^{-6}	1.8e^{-6}
$|D|_{mean}$ [%]	0.0003	0.0002	0.0006

3. Characterization: Analytical Approach

As a baseline for this analytical approach, the deviation between the FE-based beam stiffness parameters obtained in Chapter 2 and the analytically calculated beam stiffness parameters is analyzed. For the analytical calculation, the conventional approach is used. For this, the stiffener profile is assumed to be constant. The values of the start/end profile are used. The resulting deviations are listed in Table 2.

Table 2: Comparison of conventional analytical and validated FE-results

	EA_{all}	EI_{all}	$\eta_{ce,all}$
$|D|_{max}$ [%]	10.2	69.0	25.0
$|D|_{min}$ [%]	0.7	1.0	0.6
$|D|_{mean}$ [%]	6.9	32.2	12.9

It is clearly visible that a conventional analytical approach does not lead to applicable simplified beam properties. Especially the bending stiffness can cause issues with an error of up to 69 %, depending on the configuration of the stiffener geometry. By applying the previously developed FE-method this error can be avoided. The next goal is to develop an improved/refined analytical method to obtain the stiffness parameters with a reduced error. Just as in the conventional approach, the interaction of stiffener and skin is not considered. As a main difference to the conventional approach, the refined approach takes non-constant profiles along the stiffener's main axis into account.

At first, the stiffener is discretized into several segments (see Figure 4). A double curved stiffener for example is discretized into five segments, one in the middle and two on each side respectively.
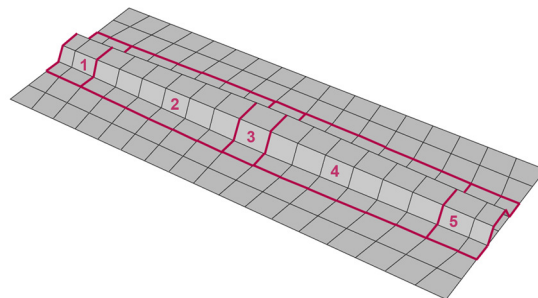

Figure 4: A double curved stiffener discretized into five segments

L. Reichert, J. Krieglsteiner, C. Schmidt, and P. Horst

For each segment, the start- and end-profiles stiffness parameters $EI_{y,s}$, $EI_{y,e}$, $EA_{x,s}$, $EA_{x,e}$, $\eta_{ce,s}$, and $\eta_{ce,e}$ are calculated analytically as presented in [3]. The tapered beam model shown in [5] is used to calculate segment stiffness parameters from the respective start and end profile parameters. The following equations are derived:

$$EI_{y,section} = \frac{EI_{y,s} + \sqrt[4]{EI_{y,s}^3 * EI_{y,e}} + \sqrt{EI_{y,s} * EI_{y,e}} + \sqrt[4]{EI_{y,s} * EI_{y,e}^3} + EI_{y,e}}{5} \qquad (19)$$

$$EA_{x,section} = \frac{EA_{x,s} + \sqrt{EA_{x,s} * EA_{x,e}} + EA_{x,e}}{3} \qquad (20)$$

$$\eta_{ce,section} = \frac{\eta_{ce,s} + \eta_{ce,e}}{2} \qquad (21)$$

In the next step, the stiffener is treated as a series connection of springs, where each segment i is an individual spring. For this, the different segment parameters are summed up to obtain the parameters for the final stiffener beam model. Using this analogy, the following equations for the beam stiffness parameters are derived.

$$EI_{y,beam} = \frac{l_{beam}}{\sum_i \frac{l_i}{EI_{y,i}}} \qquad (22)$$

The extensional stiffness is calculated in the same way. Secondary bending effects are not considered here.:

$$EA_{x,beam} = \frac{l_{beam}}{\sum_i \frac{l_i}{EA_{x,i}}} \qquad (23)$$

To calculate the stiffener beam's center of elasticity, a normal force N_x is assumed at the origin of the η/ζ-coordinate system. The resulting rotation of each segment is calculated and summed up and from this the center of elasticity of the stiffener beam is calculated (compare equations (2) and (5)).

$$\Phi_{y,beam,N} = N_x * \sum_i \frac{\eta_i * l_i}{EI_{y,i}} \qquad (24)$$

$$\eta_{ce,beam} = \frac{\frac{EI_{y,beam} * \Phi_{y,beam,N}}{l_{beam}}}{N_x} \qquad (25)$$

The analytically computed stiffness parameters are compared with the validated stiffness parameters based on the FE-results. .

The results for the extensional and bending stiffness show only a slight deviation between the FE- and analytical results. The deviation depends on the ratio between the profiles used for the start/end and middle segment of the stiffener as shown in Figure 5 using the ratio of profile heights h_{mid}/h_{start} as an example. The maximum deviation of the bending stiffness results (4%) is slightly higher than for the extensional stiffness (2%). This could be due to the larger error in calculating the center of elasticity, which is at 4%. Just as for the stiffness, the error is dependent on the ratio between the start and end profile. Overall, in the refined approach, the mean deviation is reduced to 2% with the considered stiffener configurations.
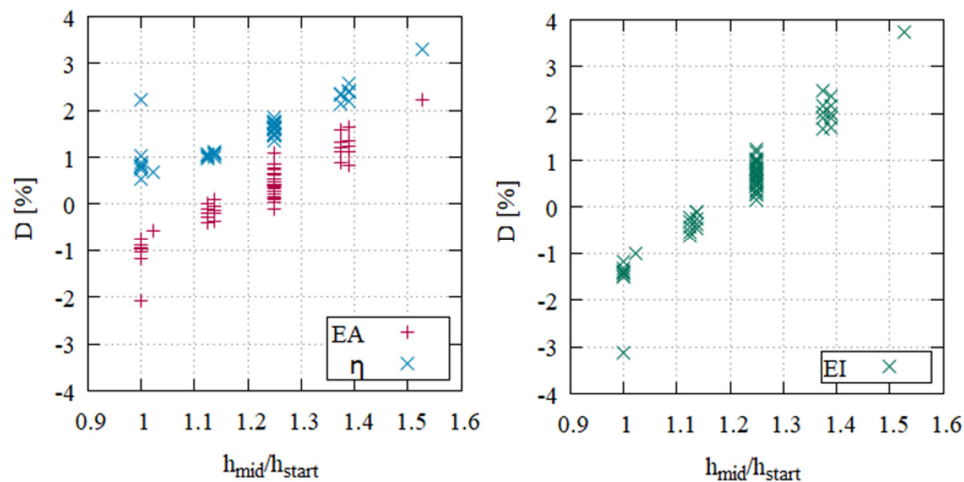
L. Reichert, J. Krieglsteiner, C. Schmidt, and P. Horst

Figure 5: Comparison of the analytical and numerical results. Mean deviations of $|D|_{EA,mean}=0,7\,\%$, $|D|_{\eta,mean}=2,2\,\%$ and $|D|_{EI,mean}=1,0\,\%$ are achieved.

4. Conclusions

It is shown how the beam properties for complex stiffeners are determined through FE-analysis. By using these properties in simplified models, the system can be represented with a small error ($<0,001\,\%$) in linear FE-analyses.

The conventional analytical approach does not respect the increased complexity of the stiffener and leads to high mean deviations compared to the reference values ($32\,\%$). A refined analytical approach respects the complexity of the stiffener and leads to a decreased error ($2\,\%$). The refined analytical approach requires less computing resources than the FE-approach. It is, therefore, suitable for an application in analyses during optimization processes.

By using one of the two presented approaches, the error in the simplified stiffness parameters can be reduced by about $30\,\%$ compared to a conventional analytical approach. To further reduce the error of the analytical approach, it should be extended to take into account the interaction of stiffener and skin. Moreover, the approaches should be extended for torsion and shear loads and tested for different design features.

Acknowledgments

References

[1] B. Kolesnikov and A. Frink. Semimonocoque-Bauweise, Double Curved Stringers. *Patent DE 10 2008 057 247 B3* Jan 28, 2010.

[2] O. Deniz, P. Horst and C. Schmidt. Production-based Multi-criteria Design Optimisation of an Unconventional Composite Fuselage Side Panel by Evolutionary Strategies and a Surrogate Model of Manufacturability Analysis. *Proceedings of the 11th World Congress on Structural and Multidisciplinary Optimisation, Sydney, Australia*, June 07-12 2015.

[3] J. Krieglsteiner, P. Horst and C. Schmidt. Characterization of Fiber-Reinforced Stiffener Profiles for Aircraft Fuselage Preliminary Structural Design. *Proceedings of the 16th European Conference on Composite Materials, Seville, Spain*, June 22-26 2014.

[4] H. Schürmann. *Konstruieren mit Faser-Kunststoff-Verbunden*. Springer, 2007.

[5] Dassault Systèmes. *Abaqus Analysis User's Guide*. Version 6.14. 2014