Integrative Numerical Techniques for Fibre Reinforced Polymers - Forming Process and Analysis of Differentiated Anisotropy

Frédéric Waimer[1], **Riccardo La Magna**[2], **Jan Knippers**[3]

[1] Research and Teaching Associate, Ph.D.c., Institute of Building Structures and Structural Design (ITKE), University of Stuttgart, Stuttgart, Germany, *f.waimer@itke.uni-stuttgart.de*
[2] Research and Teaching Associate, Ph.D.c., Institute of Building Structures and Structural Design (ITKE), University of Stuttgart, Stuttgart, Germany, *r.lamagna@itke.uni-stuttgart.de*
[3] Professor, Institute of Building Structures and Structural Design (ITKE), University of Stuttgart, Stuttgart, Partner at Knippers Helbig, Stuttgart, Germany, *j.knippers@itke.uni-stuttgart.de*

Summary: In the current paper, the authors developed two different numerical methods for fibre reinforced polymers. The first method deals with the simulation of an innovative manufacturing process based on filament winding for glass and carbon fibre reinforced polymers. The second developed numerical method aims at modelling a high level of material complexity and allowing reciprocal confrontation with a geometric differentiated global structure. The developed numerical techniques served as a basis for the design and implementation of a Pavilion built on the campus of the University of Stuttgart in 2012 and could thus be tested and proved.

Keywords: *numerical techniques, simulation, fibre reinforced polymers, optimization, manufacturing process*

1. INTRODUCTION

The interest in free-form surface generation in architecture and industrial design is ever more increasing. For complex shaped structures, fibre-based materials are often an obliged choice as they offer numerous advantages compared to traditional building materials and techniques. The wide variety of manufacturing methods allows the production of complex shape components with low effort, but they must be selected accordingly to the application they are meant for. The type of production method strongly affects the radius of the achievable fibre placement, along with the fibre volume content for a unidirectional layer and its resulting stiffness. Unlike isotropic materials, the structural characteristics of composites can be strongly influenced by the fibre morphology. The design and simulation of fibre-based materials is very challenging, since besides the material properties of the individual components also the fibre orientation of each layer plays a decisive role for the structural stiffness distribution within the system. Current developments in the analysis and manufacturing simulation processes are far advanced in the field of aerospace technology and automotive engineering. Nevertheless, such simulation and analysis processes in most cases are non-transferable or unsuitable for architecture and structural engineering purposes. To fully exploit the potential of the material in the construction industry, research must be focused on the development of appropriate methods and techniques rather than attempting to transfer known strategies from other engineering disciplines. Novel approaches are currently being investigated to generate highly efficient structures which take full advantage of the material's properties. It is therefore necessary to develop custom manufacturing processes for the building industry and simulation tools for structural engineers. The application possibilities in architecture and civil engineering are very large and the potential for new structural approaches have not been exhausted yet.

2. COUPLED GEOMETRY AND ANALYSIS

A range of different integrated design tools and workflow methodologies have been developed in recent years. One common procedure is certainly the coupling of parametric geometry models and structural analysis. In the first instance it offers the designer the opportunity to evaluate the structural behaviour and performance at an early design stage [1]. Subsequently however, more precise engineering models and detailed analysis are needed to draw a statement about the ultimate limit state and the serviceability of the structure [2]. Contrary to this established procedure the authors pursue a different aim, namely the development of project-specific tools and solutions with a high degree of accuracy right from the start.

Fig. 1. First architecture built entirely by a robotic winding technique

During the early design stages of the "ICD / ITKE RP12" research prototype [Fig.1] a strong interaction between form, material and manufacturing process was observed. Following this statement, the desire was to combine the reciprocity of geometry, material anisotropy and manufacturing process within a digital model in order to generate the best possible solution. Such global integrative approaches of different process phases are not common in construction [3], although in fields like aerospace, the idea and the desire to view different processes holistically or rather in a multidisciplinary relationship is well established [4]. To enable this kind of workflow strategies, custom simulation packages must be developed depending on the type of project. In order to provide such an integrated and holistic model for the ICD/ITKE RP12, the authors developed and programmed a new framework for an accurate and adaptive simulation. The aim was to combine different complex processes and relationship interfaces for the different aspects involved in the project. Specifically, the analysis system of the prototype includes a structural simulation of the geometry and its anisotropic materiality, along with the definition of the fibres orientation for the manufacturing process [Fig.2]. This allows the engineer to examine different parameters and concurrently to consider the influence these have on the manufacturing process, the bearing capacity and the architectural design. The digital model makes it possible to look at the system in its entirety but it requires a profound knowledge about manufacturing processes, material behaviour and modelling of fibre reinforced polymers.

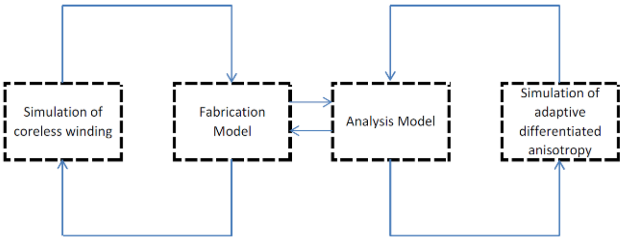

Fig. 2. Correlation of different simulations by geometric data

3. SIMULATION OF CORELESS WINDING PROCESS

3.1. Description of the novel manufacturing technique

In recent years, filament winding has appeared to be a very economic fabrication method [5]. Particularly in the case of elements with low repetitions, very cost-effective results can be achieved. Furthermore, large components can be manufactured and high fibre volume contents can be produced, providing high stiffness and strength. Typical filament winding techniques involve the production of a positive mould onto which the fibres are later laid upon [6]. The mould ensures that the fibres are kept in place and do not assume inconsistent configurations while the polymer matrix is still in the process of drying out. Due to geometric constraints, the production of custom components is mainly limited to synclastic surfaces such as pipes, vessels or aircraft fuselages, besides requiring extra amount of work in the preparation of the mould, often a milled foam core, with the obvious waste of material that this entails. Automated tape and fibre placement methods leverage some of these constraints, allowing the production of surfaces of negative Gaussian curvature (anticlastic). By utilising a composite lay-up end-effector normally installed on a robotic arm, the tape is laid over the surface of the mould in direct contact. Whilst offering greater design freedom, tape laying methods still suffer from the cumbersome necessity of producing a positive core, which also heavily limits the size of the feasible components due to obvious logistic issues. To overcome these major drawbacks, a coreless winding process was developed for the production of a large scale prototype meant to test the potential and advantages of this innovative approach. This approach represents a novelty in the production of double-curved surfaces by winding. So far, the only method not requiring a winding core solely allows the manufacturing of pipes [7].

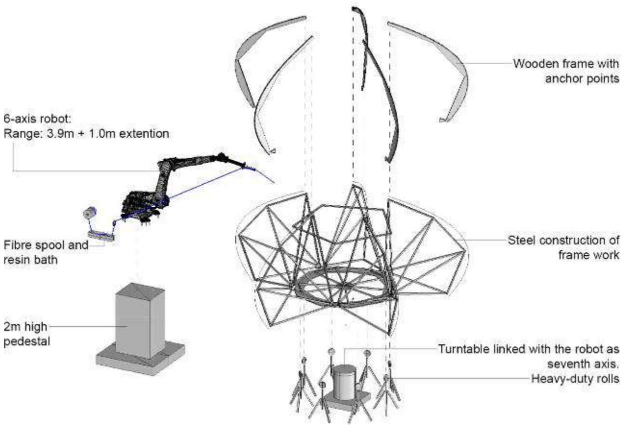

Fig. 3. The construction setup

The main feature consisted in the replacement of the positive, continuous mould with a discrete linear steel frame to hold the fibres in place during the laying process. The frame serves as temporary scaffolding onto which the resin soaked fibres are then tensioned, building step by step a shell structure consisting of individual fibre layers which achieve structural stiffness and bending resistance after the matrix (epoxy) has dried out and the tempering process taken place. To avoid gliding of the placed fibres onto the steel bars, custom milled wooden elements with teeth profiles were attached to the frame, thus ensuring a stable and suitable set of anchor points for the fibres. The complete setup was mounted on an external turntable which provided an extra axis of rotation for a 6-axis industrial robot, equipped with a specifically designed end-effector, which accurately wrapped the resin-saturated carbon and glass fibres together [Fig.3].

Without the constraints of a prefabricated positive mould, this manufacturing process specifically enables the production of anticlastic double curved surfaces as long as the different fibre layers are allowed to undergo relatively large elastic deformations, therefore assuming geometric configurations other than that of a ruled surface. This is achieved by tensioning transverse layers of fibres which would naturally lie on a deeper geometrical plane compared to the previous ones, resulting in a global displacement of the system which gradually tends to a continuous saddle shape as the number of rovings increases. Having replaced the continuous full core with a series of voids and discrete anchoring points, the fibres are allowed to freely deform and assume their natural funicular shape under the load introduced by the overlapping fibres. Furthermore, by differentiating the pre-stressing force within the single fibres it is also possible to control and steer the global shape assumed by the surface, as the deformation within the surface itself can vary locally from point to point. To successfully implement the described manufacturing technique, it was necessary to simulate the entire fabrication event to detect possible faults in the global design and in particular major losses of fibre pre-stress during the wrapping process.

3.2. Geometrical Influences

To accurately predict the final geometric configuration of the prototype, it was necessary to simulate the entire wrapping process to assess in advance the correct position of the fibres. As the resin soaked fibres do not possess any bending resistance whilst the epoxy is still in the process of drying out, successive layers wrapped on the previous ones have the effect of further deforming the system. This implies that the overall design of the shell cannot be entirely dictated and decided beforehand, but it is rather the result of a form-finding process which has to take into account the physical and mechanical behaviour of the different layers which simultaneously react to the external load of the wrapped fibre at each step. The number of degrees of freedom involved in the simulation varies quadratically at each step, as the contact points between the fibres double each time a new roving is introduced in the system. Given the complexity of the simulation model and the increasing number of variables involved at each step, it is not possible to predict in advance the outcome, hence the iterative form-finding process adopted for the simulation. The deformations involved in the process, being of consistent magnitude (meaning that the deformation components of higher order may not be neglected), along with the contact problem between the fibres, result in a highly complex simulation scenario that has to be solved adopting *ad hoc* computational strategies to handle all the aspects involved in the calculation. The main reason which requires the simulation of the fabrication process is to check the pre-stress values of the fibre bundles [8]. As it has later been assessed through the simulation and also observed on the final prototype, the stress distribution greatly varies amongst the fibres. This is due to the particular geometric configuration, as specific areas tend to be overstressed as they absorb most of the load introduced by the later wrapped fibres. Conversely, given the constant change of shape during the wrapping process, other fibres run into the risk of totally losing their pre-stress as a result of the support surface which has further relaxed within a number of wrapping cycles, leading to bundles of fibres lying loosely on the surface. To avoid this dual problem of overstressing and loosening of the fibres, optimal pre-stress values have to be chosen in order to avoid or at least minimise this effect. Especially the loosening of the fibres during curing time represents a major setback in fibre composite products, as the distinct layers do not offer a homogeneous contact surface anymore, giving rise to relative deformations between the layers which, not being coupled together, act independently and lose completely their mechanical characteristic and strength of a compact bond, potentially leading to serious delamination problems during the lifetime of the element.

In the context of the project, besides checking the local stress values the simulation of the forming process was needed to evaluate the shape of the prototype before production start. At an early stage, the Finite Element analysis of the whole structure was performed on a simplified digital model of the single layers which made up the entire prototype. The underlying digital model resulted from the translation of the physical scale prototypes developed during the design phase into 3d-models, which later served as basis for the development of the Finite Element analysis of the layers' differentiated anisotropy. To confirm the results acquired from the analysis of the structure, it was absolutely necessary in the first place to validate the geometric model used for the calculation. Comparing the Finite Element model with the output model obtained from the forming process simulation, resulted in a maximum deviance of 12,5cm which was considered acceptable and within the tolerance boundaries for the purposes of the project. The simulation was finally conducted to correct the laying paths followed by the robotic arm. In filament winding it is known that two major problems may arise in relationship to the followed path, namely uplifting and lateral sliding of the fibres with respect to the underlying surface. Whilst uplifting does not represent an issue for the developed coreless winding technique (no underlying positive mould onto which the fibres lie upon), lateral sliding may still be present and lead to potential damage of the rovings or significant losses of pre-stress. To avoid possible sliding effects, the fibres should lie within a given boundary defined by the geodesic line running between two support points on the reference surface and an offset dependent on the friction value between the fibres. As the path followed by the fibre-laying robotic arm also defines the path taken by the roving, to minimise the side effects of lateral sliding the proposed path is later updated following the newly achieved results. This process required the definition of a preliminary wrapping path along which the intersection points between the fibres are geometrically resolved, updating the mechanical model with the newly defined nodes and elements, solving the system and correcting the path with the found results.

3.3. Simulation Process and Finite Element Model

The whole analysis process relies on two main interfaces, namely the geometric interface and the analysis (Finite Element based) interface. To speed up the process and make the exchange of data viable between the two programs, the workflow was fully customised and automated to allow the automatic generation and analysis of the whole simulation process. In a looping procedure, the 3d-modeling software (Rhino3d) first reconstructs the displaced geometric model which is retrieved directly from the results database of the Finite Element program (SOFiSTiK), along with important mechanical information as stress values, nodal forces, support reactions etc. The model is then updated with the new information deriving from the actual wrapping step, i.e. nodes and cable elements which define the new roving. Following, the generation of the code automatically takes place from the updated geometric setup, and once completed fires an event to the Finite Element program which performs the new calculation and finally saves the results into its native database. The whole simulation process is then looped until the last roving has been set into the model.

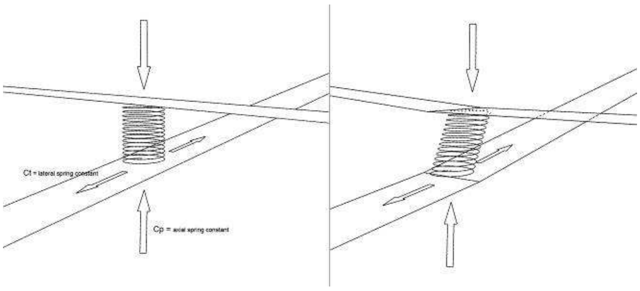

Fig. 4. Mechanical model of the coupling between fibres; relative displacement after the calculation process

Fig. 5. Full scale test windings without resin

The analysis model is based on cable net mechanics. At each step, the intersection nodes between the existing cable elements in the model and the new cable are found geometrically. This process takes place within the 3d-modeling software, where the displaced geometry of the cable net is first imported and rebuilt as a network of polylines. To find the points which make up the new cable element the approximated robot tool path is needed, as the nodes are found by intersecting the plane defined by the two support points and the average point of the robotic arm's spatial trace with the cable net already present. The intersection points define the structural nodes of the next fibre which is later added to the global Finite Element model. Besides defining the new cable element to be added to the system, the newly defined nodes need to be inserted in the definition of the previous cables. In order to successfully update the model, all mechanical information has to be acquired from the results database and processed to accommodate the new nodes which further subdivide the cable elements. As the number of nodes grows quadratically as previously mentioned, to reduce the complexity of the system and the overall computational time required, nodes too close to each other (the threshold value being defined in advance) are merged together, thus keeping the model size more compact and minimising the processing time. Although this process has displayed the production of geometrical artifacts in areas of the model densely cluttered, the mechanical results have been proven to be consistent and the overall required computational time sensibly reduced. As in most cases the fibre does not lie on the geodesic line of the surface, considerable lateral sliding effects may occur during the wrapping process. In order to take into account such sliding effects due to lateral forces which appear if the fibre does not lie on the geodesic line of the surface, the coupling between the structural nodes is modelled through spring elements with infinite axial stiffness and a threshold lateral stiffness which derives from the friction value between soaked fibres previously tested in the laboratory. With a reduced lateral stiffness, the spring is allowed to stretch perpendicularly to the axis of the cable, resulting in a sliding effect on the underlying cable [Fig. 4]. The stretching effect stops as soon as the residual lateral force on the node balances the threshold stiffness of the spring. As springs tend to be computationally intensive elements, the coupling is provided only for the last roving to be simulated in order to find the correct nodal position. Due to the missing bending stiffness of the structure during the wrapping process, the considerable displacements of the fibres have to be taken into account by resorting to a non-linear calculation which considers the internal stress state in the displaced reference configuration. After the calculation is run, the position of the displaced nodes are updated once again in the 3d-model by merging the two nodes and deleting the spring connection, providing a coupled connection between the two cable elements. Finally, the process is looped in the exposed order until the last cable element has been imported and the system calculated, leading to the end geometric configuration with the corresponding stress map [Fig. 6].

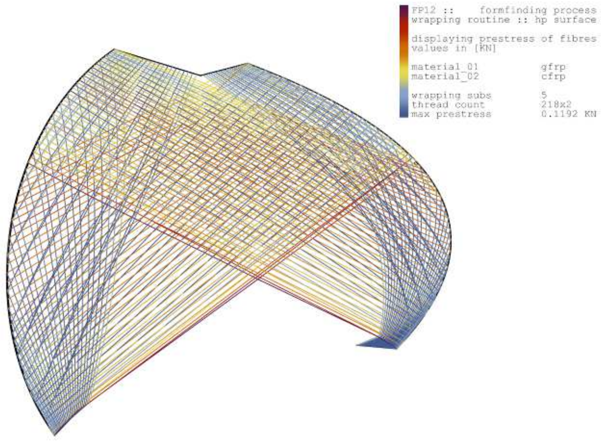

Fig. 6. Stress map of coreless winding

4. ADAPTIVE SIMULATION OF HIGH DIFFERENTIATED ANISOTROPY

The modelling and analysis of fibre-reinforced polymers strongly differs from conventional materials such as timber, reinforced concrete or membrane structures. Simplified approaches are often used in these cases to describe the anisotropic behaviour of the materials and their failure criteria. For the analysis of glass and carbon fibre reinforced polymers in construction, engineers mainly resort to simplified simulation models. The investigation of the Ultimate Limit State and the Serviceability Limit State normally takes place on the component level. This rough and inaccurate approach of analysis is often due to the lack of basic knowledge on behalf of the structural engineer regarding the material behaviour and the modelling on the macro and micro level. The fact that the anisotropic material behaviour is not taken into account for the material design obviously leads to oversized and inefficient structural components. The Finite Element descriptions for the simulation of fibre based materials according to the Classical Laminate Theory (CLT) are currently considered the state of the art and are already implemented in some FE programs [9][10]. The calculation of the stiffness of each element does not constitute a problem but rather the accurate modelling of complex fibre directions of the individual layers of the laminate. The challenge lies in the correct alignment of the varying fibre orientations of the individual layers and is primarily a geometric problem to be solved.

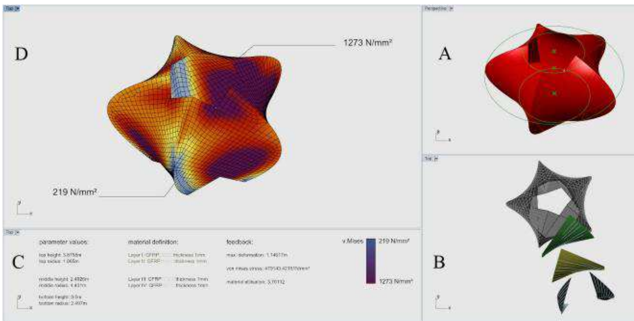

Fig. 7. User Interface: Coupled geometrical and material analysis

4.1. Coupling of geometry and fabrication process to FE-analysis

To take into account and analyse the interaction between geometry and anisotropic materials, an interface between the parametric modeller Grasshopper® and ANSYS was developed and programmed [13]. The user interface thereby is present only within Rhino3d and Grasshopper® and was already used during the investigation of first working models [Fig.7]. From the defined parametric geometry in A and the allocation of the fibre lay-up and number of layers corresponding to the surface section from the manufacturing process in B [Fig. 7], the direct interaction between geometry and fibre direction in the individual layers is considered. This means that by changing the geometry parameters and according to the implementation of the winding logic, the fibres' orientations of the individual layers change too and are consequently re-orientated. The change in orientation of the individual fibres or layers ultimately influences the material properties (rigidity and strength) at each point of the surface. The assignment and the definition of the material properties and thickness of the individual layers along with supports and load actions also take place within the programmed Grasshopper interface. The orientation and the generation of the coordinate systems of the single layers and the following calculation of the Finite Element stiffness by the CLT are processed directly in ANSYS. In section 4.2, this implementation is explained in detail. After the FE calculation, the forces in the element are back-calculated in batch mode with the CLT into the individual layers, and following Puck's criterion~~,~~ the material utilisation is calculated [4]. The distortion condition, the material utilisation or the stress states can then be displayed in C and D [Fig.7].

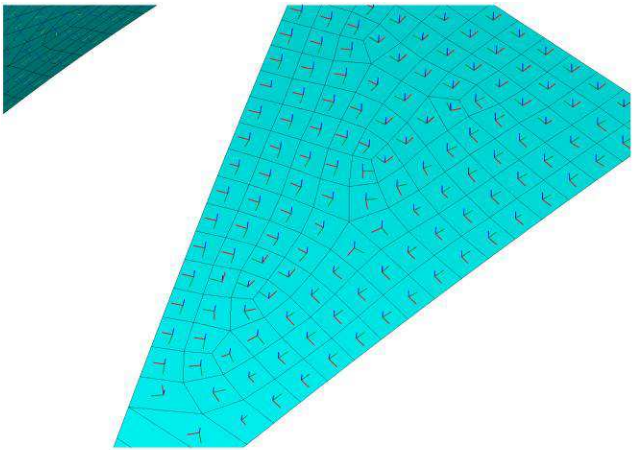

Fig. 8. Example of the element coordinate systems of a curved surface

4.2. Geometric adaptive element stiffness calculation

The interaction of the geometry and the manufacturing process with the anisotropic materiality is generated by geometric parameters. The fibre directions of the individual layers are defined by support curves on the surfaces. For the export of the fibre configuration from Rhino3d in ANSYS a simple geometry file is therefore sufficient. ~~.~~For each area, the number of layers must be defined, and to describe the fibre orientation~~,~~ the corresponding curve along with the appropriate material and thickness which must be assigned for each layer. These parameters are stored in a .txt-file and loaded again in ANSYS [13]. After ANSYS has imported the geometry, the surfaces are meshed. As it can be seen in [Fig. 8], for complex geometries the coordinate systems of single elements are of limited use for the simulation of an anisotropic material behaviour. This differs from software to software. In ANSYS, the orientations of the coordinate systems of the elements are defined by the topology of the mesh, respectively by the first edge of the quad [10]. Thus the element coordinate systems must be re-orientated to be able to map the fibre orientation. This happens as follows:

in a first step, auxiliary geometric coordinate systems on the curve are generated, which define and also reflect the fibre orientation of the individual layer. These auxiliary coordinate systems KS_{Ln} are located on the nodes of the adjacent finite elements [Fig.9]. The coordinate systems define the orientation of a specific laminate layer of a subarea of the entire structure. For each coordinate system of one element, the closest coordinate system on the curve will be defined and rotated accordingly in a second step. The newly defined and designated as KS_{En} coordinate systems are treated for the modelling as superior coordinate systems and are used for the anisotropic stiffness of the element. The coordinate systems KS_{En} likewise mirror the fibre orientation as the coordinate system KS_{En1} of the top layer [Fig.10].

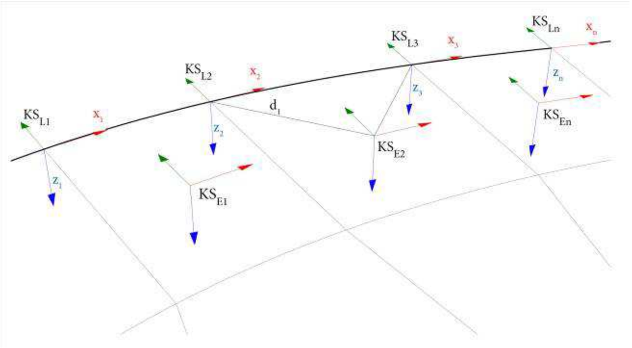

Fig. 9. Generation of KS_{Ln} and KS_{En}

4.3. Simulation and optimization of the layered elements

The orientation of the individual elements is decisive for the subsequent layout modelling of the individual layers of the laminate. For an element which consists of 4 layers for example, a coordinate system for each layer is generated. Thus for each additional layer of the laminate a coordinate system is generated by the help of an additional curve as described in the previous section. These coordinate systems subsequently represent the fibre orientation of each layer [Fig.10]. For each layer a curve on the surface is modelled in Rhino which reflects the orientation of the fibres. The individual coordinate systems are used to determine the exact fibre orientation at each point on the surface. To determine the overall stiffness by the Classical Laminate Theory (CLT), the rotation angles of the coordinate systems of the individual layers must be stored in the element specification. Through the cross product between the x_1-vector of the KS_{En} and the x_1-vector of the KS_{E1n} it is possible to define the precise angular relationship. Now the angle of rotation in the KS_{En} coordinate system of the individual layers can be stored in the layered Finite Element. Subsequently, the effective stiffness of the element can be determined by the CLT and after the definition of the boundary conditions, the structure can be solved.

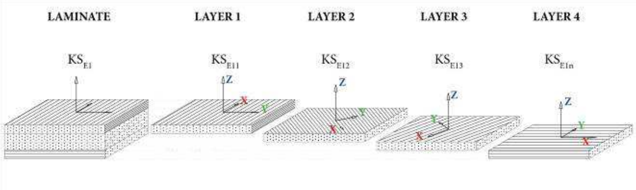

Fig. 10. Generating the overall stiffness of the element

The accurate modelling of the orientations of each layer is of fundamental importance for the subsequent determination of the stiffness and of the assessment of the load bearing capacity. At each point of the structure another layer configuration and a different fibre orientation exists, and a detailed modelling is possible only by the presented routines.

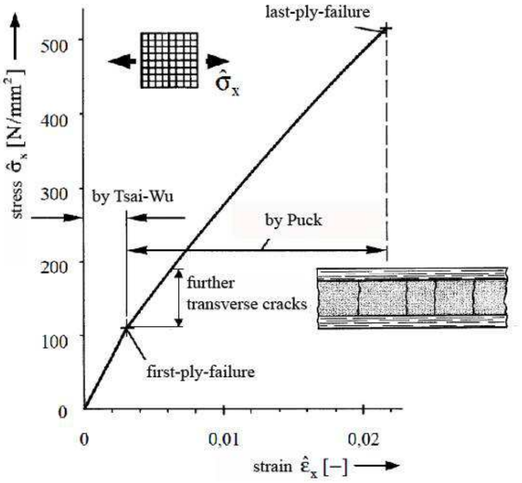

Fig. 11. Consideration of first-ply-failure

In addition to the fibre orientation, the individual layer thicknesses are relevant parameters because on the one hand they influence the distribution of loads in the laminate, and on the other the manufacturing costs. The winding of one layer with a thickness of 0.2mm meant at the ICD/ITKE RP12 a pure robot time of ca. 3 hours. By a gradient-based optimization, the first set laminate thickness was reduced from 10.0mm to a total thickness of 4.5mm [Fig.12]. This approximately corresponds to 82.5 hours of winding time. The maximum deflection of 350mm and a material utilization by the criteria of Tsai-Wu of 1.5 were set as constraints. The utilization was consciously fixed higher than 1.0 due to the more conservative failure criteria of Tsai-Wu compared to the one of Puck. In addition, a non-linear material behaviour of the laminate was considered, which allows a matrix cracking in single layers. This was verified by the failure criterion of Puck after the optimization in detail. Unlike other criteria, the criterion of Puck allows a clear distinction of the five failure modes in the micro-level [11]. This makes it possible to identify a matrix fraction or a first-ply-failure and also to assume load redistribution from the cracked layers to the adjacent layers [Fig.11]. Thus the entire load capacity of the laminate can be increased many times over. By using the load-bearing reserves, the degree of utilization of the laminate could be kept below 1.0.

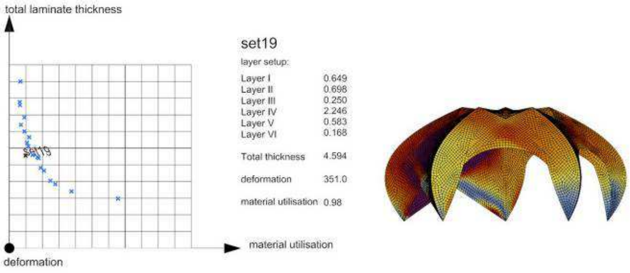

Fig. 12. Material optimisation

5. RESULTS

The mixed laminate consists of epoxy resin and 70% glass fibres and 30% carbon fibres. A total length of 47km glass and 15km carbon rovings were spun. To define the external and internal wind pressure coefficients the DIN EN 1991-1-4:2005 was used. The characteristics of strength and stiffness of the glass fibre laminate and carbon fibre laminate were determined according to DIN EN ISO 527-4. For each laminate three specimens were taken directly from the manufacturing process [tab.1]. By the safety factor on the material side, the fluctuations of the production quality were covered. The assessment took place in accordance to the design concept of the BÜV-Guideline [14].

Table 1: material characteristics from the manufacturing process

	E-Modulus	Strength
GFRP	19514 N/mm²	348 N/mm²
CFRP	105584 N/mm²	576 N/mm²

6. CONCLUSION

To fully exploit the advantages of fibre reinforced polymers and to make them usable in the construction industry, new approaches and ways of thinking are needed. This is the case for the development of appropriate production methods and the development of custom simulation and analysis tools. The two novel developed numerical techniques served as basis for the design and could be tested and proved by the realisation of a full scale prototype. The semi-transparent skin of the Pavilion reveals the system's structural logic through the spatial arrangement of the carbon and glass fibres. Despite its considerable span of 8m by a thickness of 4.6mm [Fig.12] and its remarkable size, the Pavilion has a weight of less than 320kg [Fig.13].

7. REFERENCES

[1] Van de Weerd, B., Rolvink, A., Coenders, J., *StructuralComponents- A Software System for Conceptual Structural Design*. In: IASS-APCS 2012 Proceedings, Seoul (2012)

[2] Fahlbusch, M., Hoffmann, A., Bollinger, K., Grohmann, M., *Skylink at the Frankfurt Airport*. In: IASS-APCS 2012 Proceedings, Seoul (2012)

[3] Knippers, J., *From Model Thinking to Process Design*, Architectural Design 83.2 (2013): 74-81.

[4] Sobieszczanski-Sobieski, Jaroslaw, and Raphael T. Haftka., *Multidisciplinary aerospace design optimization: survey of recent developments*. Structural optimization 14.1 (1997): 1-23.

[5] Bader, Michael G., *Selection of composite materials and manufacturing routes for cost-effective performance*. Composites Part A: Applied science and manufacturing, 913-934 (2002).

[6] Knippers, J., Cremers, J., Gabler, M., & Lienhard, J., *Construction Manual for Polymers+ Membranes*, Birkhäuser (2010).

[7] *Für gewöhnlich das Außergewöhnliche*, Innovation Report, February, CFK-Valley Stade (2011).

[8] Romagna, J. H., *Neue Strategien in der Faserwickeltechnik*. Diss. Eidgenossische Technische Hochschule Zurich, (1997).

[9] Kunststoffe eV, A. I. V., *Handbuch Faserverbundkunststoffe*, Vieweg & Teubner, Wiesbaden (2010).

[10] *ANSYS® Academic Research, Release 14.0, Help System, Coupled Field Analysis Guide, ANSYS, Inc.*

[11] Puck, A., *Festigkeitsanalyse von Faser-Matrix-Laminaten: Modelle für die Praxis*, Carl Hanser (1996).

[12] Schürmann, H., *Konstruieren mit Faser-Kunststoff-Verbunden*, Springer (2007).

[13] Waimer, F, La Magna, R, Reichert, S, Schwinn, T, Menges, A, Knippers, J., *Integrated design methods for the simulation of fibre-based structures*, in: Proceedings of the Design Modelling Symposium Berlin, 2013: p. forthcoming publication, Springer (2013).

[14] BÜV-Empfehlung- *Tragende Kunststoffbauteile im Bauwesen [TKB] - Entwurf, Bemessung und Konstruktion* - Stand 08 / 2010

Fig. 13. A: The glass fibres of the closed surfaces spread the light emitted by the spots (Halbe).

B: The top view shows the areal glass fibre roof and the carbon pressure ring (Halbe).