# Development of a modelling approach for wire harnesses

C. Tamm[1]*, F. Heissler[2], R. d'Ippolito[3], M. Motzer[2], M. Panzeri[3], F. Stöckl[2], G. Stoll[1]

[1]Fraunhofer Institute for Structural Durability and System Reliability LBF, Darmstadt, Germany
[2]Lisa Dräxlmaier GmbH, Vilsbiburg, Germany
[3]NOESIS Solutions N.V., Leuven, Belgium

*contact: christoph.tamm@lbf.fraunhofer.de

**Summary**

The overall requirements on the wire harness design increase with each new car generation as more and more functions are integrated or been electrified. In addition, the available installation space in a car decreases and the harness cannot be installed as assumed during the design phase. This happens particularly to thick bundles with many cable outlets or to tight bending radii. A simulation-based development in the design phase can lead to a reduction of iterations in the manufacturing of prototypes, and hence will reduce production costs and development times.
Wire harnesses pose a particular challenge in numerical simulations due to their flexible composite structures and their complex mechanical behavior. In this contribution an interface to the structured wire harness descriptions in the *Vehicle Electric Container (VEC)* format and the integration of a numeric simulation method for the stiffness calculation of individual harness segments are presented.
Based on the data structure, Finite Element models are obtained through a recursive modelling process. Program code in *ANSYS APDL* and *ANSYS Workbench* is generated and the numerical calculations are performed automatically. The numerical results are validated by experimental data.

**Keywords**

Simulation-based development, Stiffness calculation, Wire harness, Automation, Process integration

## Motivation

As more and more functions are integrated or been electrified in new car generations, engineering tasks in the automotive industry are getting increasingly complex. Paired with the ambition of delivering enhanced products and services, these developments lead to the need of faster and more accurate processes and methods in engineering and design. This certainly applies to the design and development of wire harnesses. Fully or semi-automated process integration, automation of recurrent tasks and comprehensive data management will help to tackle the upcoming challenges.
The cockpit is a challenging design task with multi-disciplinary interdependencies and up to 400 design changes. In order to reduce the development time and costs as well as increase the quality of the design and the product, a holistic representation of a cockpit wiring harness in a graph based, standardized design language including all rules for the automated routing inside the three-dimensional space is pursued [1].
During the geometrical design process of a wire harness the mechanical properties of harness segments have relevant impact. Especially for automated routing or laying simulations the tensile, bending and torsional stiffness will lead to more precise and reliable results [2].
A suitable method for the simulation of the structural dynamics of flexible structures is developed. The method is implemented prototypically and interfaces to other tools in the process chain of the European ITEA3 project IDEaliSM are considered. The output data is used to parametrize laying simulations and to update routing simulations.

## Concept

To consider the mechanical behavior of a wire harness segment in routing simulations, the tensile, bending and torsional stiffness are relevant input parameters. During the design process, segments and their correspondent cross-sections at critical positions in the harness design need to be investigated in detail. Since wire harnesses are compositions of metallic and plastic layered materials they behave non-linear due to their plastic strain and possible relative movements of the layers. For the quantification of all these properties a large number and variety of experimental measurements would be necessary [3].
For practical and economic reasons, it is not feasible to experimentally determine the stiffness for the large variety of cross-sections of realistic wire harnesses. It is more useful and cost effective to simulate the critical cross-sections of the wire harness using Finite Element (FE) Methods [4]. With a sufficient numerical simulation approach the cost and the time needed for the mechanical characterization of wire harnesses could be significantly reduced [5].
The developed calculation concept and modelling approach for wire harnesses is shown in Fig. 1.

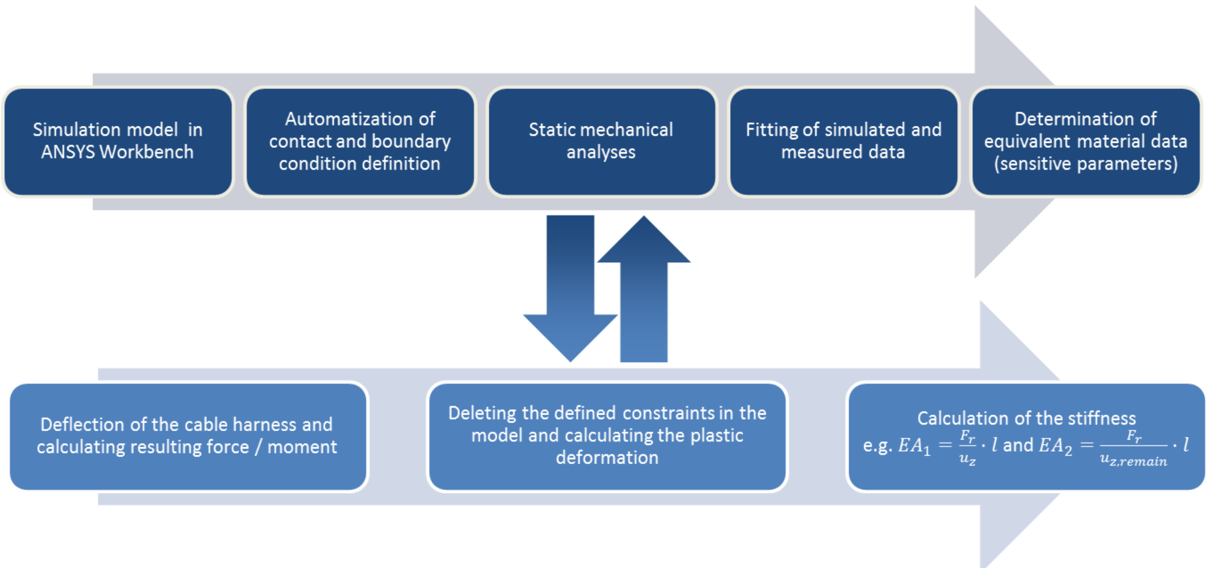



Fig. 1: Concept of stiffness calculation

First, shown in the upper dark blue arrow, a FE simulation model in ANSYS Workbench is built up. The geometrical parameters are read from an existing VEC file and the material parameters are loaded from a user specific data base. The contact and boundary conditions are defined automatically

using the ANSYS Finite Element Modeler. Subsequently two static mechanical analyses are performed. This workflow is shown in the lower arrow in Fig. 1 (see also [6]). An external deflection of the cable harness is applied and the resulting forces or moments are calculated. After deleting the defined constraints in the model, the plastic deformation is computed. Then the stiffness $EA_1$ and $EA_2$ are calculated as depicted in Fig. 1. In an optional step, the simulated data is fitted to measured data, if available. Lastly the user specific material data base is updated with the new matched equivalent material data. By constantly updating the material data base, calculation errors will be reduced and this will lead to more exact results in the future.
The considered load cases are shown in Fig. 2, Fig. 3 and Fig. 4. In order to experimentally validate the simulation results, the measurement and simulation test scenarios are built-up in the depicted way. In Fig. 2 the tensile load case is illustrated. On the left side, the harness segment is clamped; on the right side the tensile load is applied. To avoid slipping between the single wires, the free wire ends of the segment are also clamped.

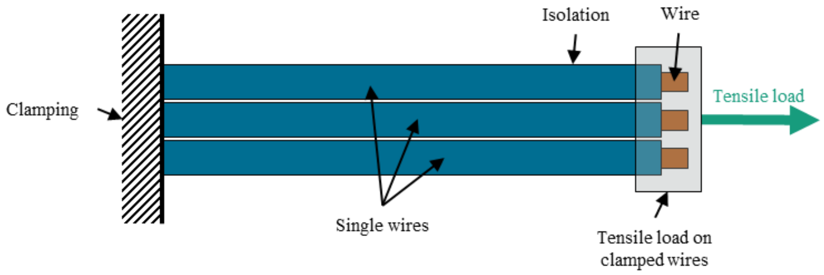



Fig. 2: Schematic diagram of tensile load case (measurement and simulation)

Fig. 3 shows the bending load case. Again, on the left side, the harness segment is clamped and on the right side the load is applied. In this load case the bending load is applied 30 mm from the free end in transversal direction. The deflection, given by the distance between the deformed and undeformed state, and the resulting forces are measured in the same direction.

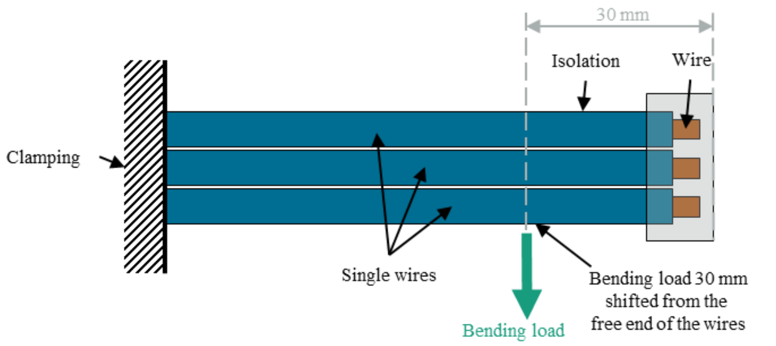



Fig. 3: Schematic diagram of bending load case (measurement and simulation)

In Fig. 4 the torsional load case is illustrated and once more the harness segment is clamped on the one side and the torsional load is applied on the other. The resulting deflection and moment are measured at the free end.

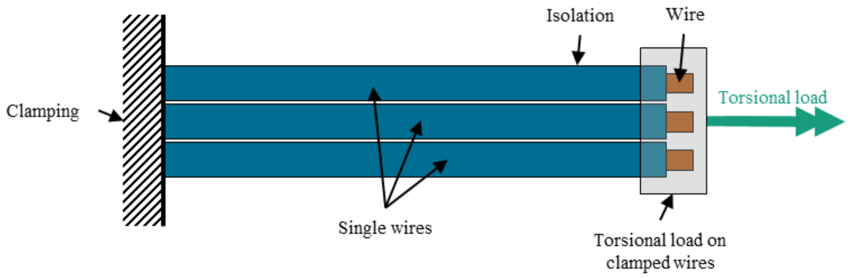



Fig. 4: Schematic diagram of torsional load case (measurement and simulation)

In Fig. 5 the normalized results of a tensile stiffness measurement and calculation are presented. The black line shows the realistic measurement results. During the initial deflection elastic and plastic deformations occur, starting from the unloaded state to the state of maximum force and deflection. With the release of the external load, the force and displacement decreases. Due to plastic deformation a certain deflection remains. The gradient of this curve corresponds directly to an equivalent Young's modulus of an assumed homogenized material.

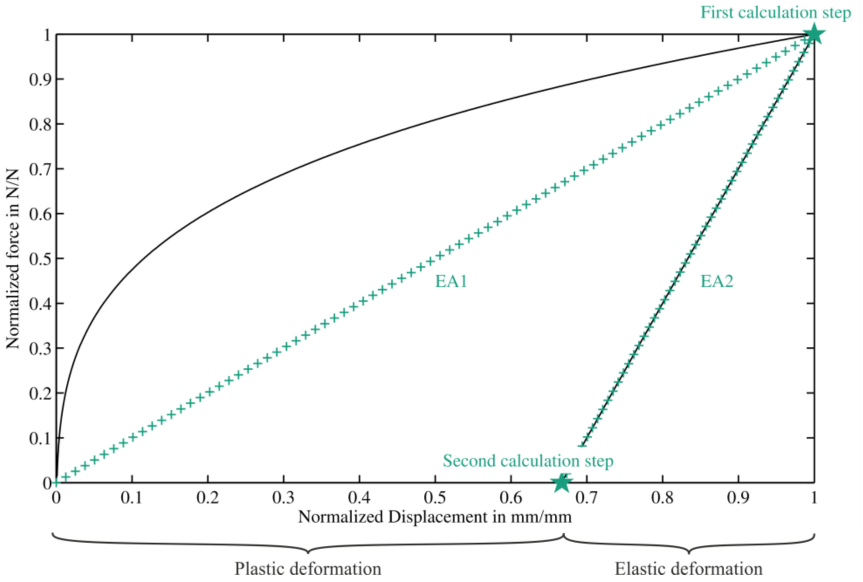



Fig. 5: Results of stiffness calculation

The simulation results are shown in the green dotted curves. In the first calculation step, the stiffness for an elastic-plastic deformation is assessed. The realistic measurement curve is not reproduced exactly, only a length-depended stiffness of the harness segment is calculated. By deleting the defined constraints in the model, the plastic as well as elastic deformation is computed.

## Automation

The workflow of the developed calculation concept is depicted in Fig. 6. The necessary input data is read from a *Vehicle Electric Container (VEC)* file [7], processed in the seven steps of stiffness calculation and the resulting stiffness are again written to the initial VEC file.

On the left hand side the input data (geometry, material parameter) and on the right hand side the output parameter are summarized. The seven steps of stiffness calculation are arranged clockwise in the middle. Firstly, the required harness segment parameters are read and clustered. Wherever possible single cable wire and isolation parts are homogenized and the material parameters are updated. With this step the computational effort can be reduced tremendously. The basic geometry model is then built up using *Mechanical APDL (ANSYS Parametric Design Language)* and the meshing is performed. In the third and fourth step the mesh model is imported into *ANSYS Workbench* and the contacts between the wires, loads, boundary conditions and necessary parameters for the simulation analysis are set up. Contacts are generated as non-linear, frictional contacts and are only considered within the calculation if they are actually closed. Subsequently, two static mechanical analyses are performed (see concept). In the optional sixth step the simulated data is fitted to measured data, if measured data is available. In the final step the material data base is updated and the resulting values are provided.

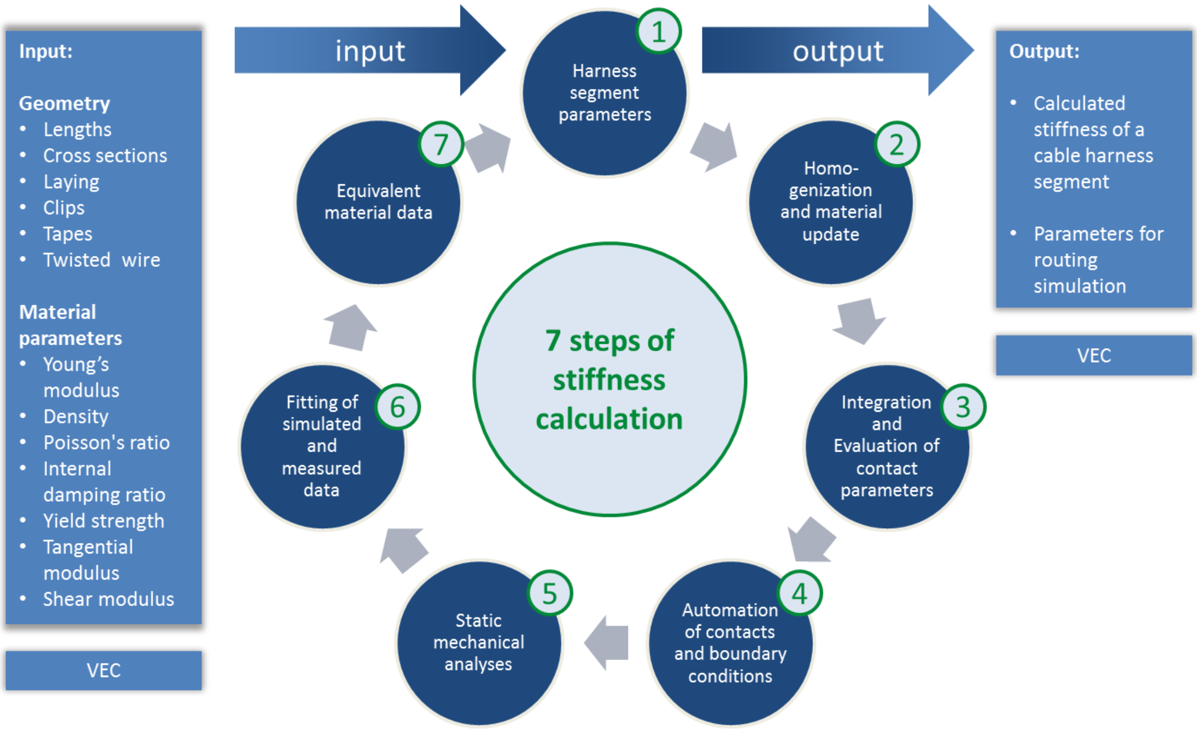



Fig. 6: Seven steps of stiffness calculation, lists of input and output parameters

The completely automated process for the computation of wire harness segment stiffness values is implemented using the Workbench Python scripting interface in combination with custom APDL modelling and simulation code as well as JScript for Workbench Mechanical. The process is either started through a batch command from the command line or from the Workbench user interface using the embedded Python console. The program then sets up the complete simulation environment containing a Mechanical APDL-Block, a Finite Element Modeler-Block and two Static-Structural Analysis blocks.

**Process integration**

Optimus is a process integration and design optimization system environment in which users can automatically visualize and explore the design space and gain the critical insights into the dynamics of a given (multidisciplinary) virtual design problem. The Optimus software environment provides the means to (a) integrate different processes within a unique simulation workflow, (b) automate the tasks required to execute these processes, (c) explore the design space and (d) perform the design optimization.
Integrating different processes in a unique framework requires defining from a conceptual point of view how a set of input (or design) variables is linked to a set of output variables. In Optimus, a Graphical User Interface (GUI) provides the tools required to define input / output variables, files, analyses and all the intermediate activities (e.g., rules defining the mapping / extraction of variables to / from files, batch commands, etc) involved during the simulation of a design process. All these entities are then linked together to form an oriented graph (i.e., the simulation workflow) that allows defining in a logical and quantitative way how the data is transferred from the input to the output variables.
Fig. 7 depicts the simulation workflow used in this study as displayed from the Optimus GUI. In the graph reproduced in Fig. 7, the green icons represent files, the squared orange icons stand for actions (i.e., commands to be executed as batch/shell scripts) while the light-green and red round icons symbolize input and output variables, respectively. From these definitions, it becomes easy to identify from Fig. 7that the main steps entailed by the execution of this workflow are (a) the mapping of the two design input variables (“length” and “forced_displacements”) into the file “Parameters.inp”; (b) the execution of the two analyses (“read_vec” and “wb_batch”), each of which is associated with a given set of input/output files; and (c) the extraction of the output variables “tensile_stiffness_EA1” and “tensile_stiffness_EA2” from the corresponding result files.

A simulation workflow is typically linked with one or more analysis methods. The latter consist of numerical algorithms that are embedded in the Optimus engine and that can be used to:

- investigate the design space in order to explore the model behavior, extract the most significant model features and build surrogate models;
- investigate how the uncertainty associated with the input variables is propagated to the outputs and how the uncertainty in the outputs can be apportioned to different sources of uncertainty in the associated inputs (sensitivity analysis);
- perform the design optimization in order to find the values of the input parameters that minimize or maximize the target output variable of interest under a set constraints.

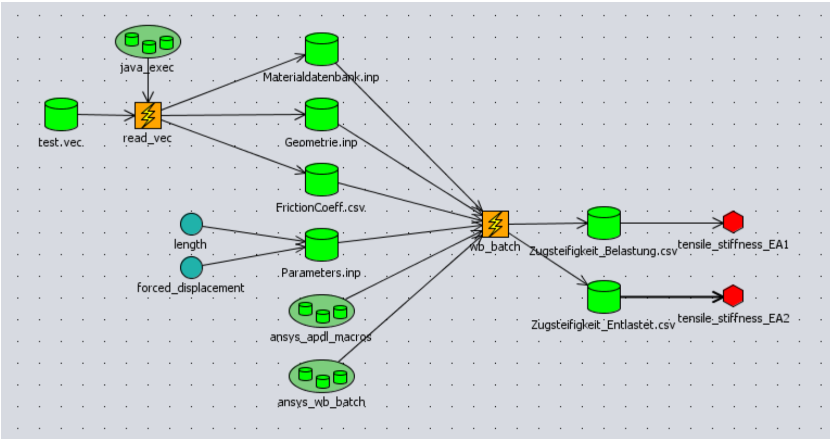


Fig. 7: Optimus graph showing the building blocks (i.e., inputs, files, analyses, outputs) required to link the design variables to the system responses and the connection between these items.

The remainder of this paragraph provides a description of the most interesting functionalities embedded in the Optimus software environment: the Optimus Python SDK module, the Optimus User Customizable Actions (UCAs) and User Customizable Interfaces (UCIs).
The Python SDK module allows creating and accessing every contents of a workflow via the Optimus Python API. This means that for all the features in Optimus, there is a corresponding Python command that the program uses when creating a project. The Optimus Python Development Environment toolkit allows the user to bypass the Optimus graphical user interface and communicate directly with the Workflow Engine and the Design Exploration/Optimization kernel. These python routines provide the core functionalities to automate (a) the authoring of a simulation workflow, (b) the creation, execution, and running of the analysis methods associated with a simulation workflow, and (c) the retrieval of the results produced by these analyses.
The framework provided by the User Customizable Actions (UCAs) allows describing in a central place the specific commands that Optimus will use to launch each user-specific simulation package action involved in the execution of an Optimus workflow. This is an effective solution to cope with each parameter of the simulation action, the IT architecture dependencies (OS, path... etc.), the version of the simulation package and also to deal with the inefficient and error-prone process of defining the commands required to run the simulation package of interest. To achieve this, Optimus relies on the so-called UCA template file, which is an XML-based template file where the user can formalize all the knowledge required to launch a simulation from a single place with a dedicated Optimus dialog. Typical information that can be stored in a UCA template file are, for example, platform dependencies details, executable path or different versions available. The UCA template files are easily portable and can be deployed on any other Optimus installation without specific IT or programming action.
To deal with the fact that Computer Aided Engineering (CAE) packages are generally closed and rigid, Optimus has introduced the User Customizable Interface (UCI). The aim is to deliver a comprehensive set of wrapping capabilities in an open architecture that allow for pervasive customization and integration of third-party simulation file syntax in a unique framework. User Customizable Interfaces (UCI) capture the syntax of any commercial or in-house simulation software. Thanks to a wrapping layer, the user accesses directly the design parameters and the extracted results from the program input or output files. The data exchange between Optimus and the simulation file supports both ASCII and binary file formats. To publish parameters from binary file, the wrapping programmable layer translates the data through a user defined executable, regardless the programming language. To cope with ASCII simulation file, the parameter syntax is incorporated in a simple xml file without any programming. At any time, a new data block can be added to support a broader set of values in the

xml. Each UCI, enabled in a local or central place, appears automatically on all Optimus seat without having to modify the Optimus deployment Thanks to the UCI, Optimus provides an easy-to-use graphical drag and drop interface to quickly create a multi-disciplinary workflow. Parsing and accessing the parameters is achieved from an Optimus dialog without any syntax related complexity. Substitution and extraction of design parameters, respectively results, rely on the same wrapper technology.

## Validation

The proposed modelling approach for wire harnesses is validated in two different use case scenarios. First the sensitivities of selected parameters of a generic wire harness are assessed and second the simulation results are compared to experimentally determined test data.
The sensitivity analysis is performed on the wire harness shown in Fig. 8. On the left, the figure shows the FE model of the wire harness with taping, on the right without it.

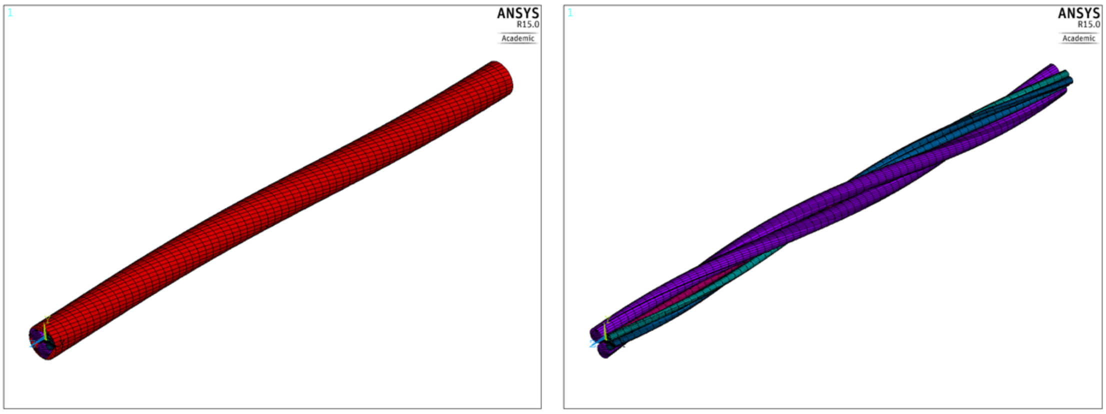



Fig. 8: FE model of a wire harness with (left) and without (right) taping

Several parameters are varied and the simulation results are investigated on the consistency to expectable or analytical solutions. In Fig. 9 and Fig. 10 exemplary results of the sensitivity analysis are depicted. The figures show the correlation between segment length and tensile stiffness of the elastic plastic deformation ($EA_1$) and the correlation between segment length and tensile stiffness of the pure elastic deformation ($EA_2$).

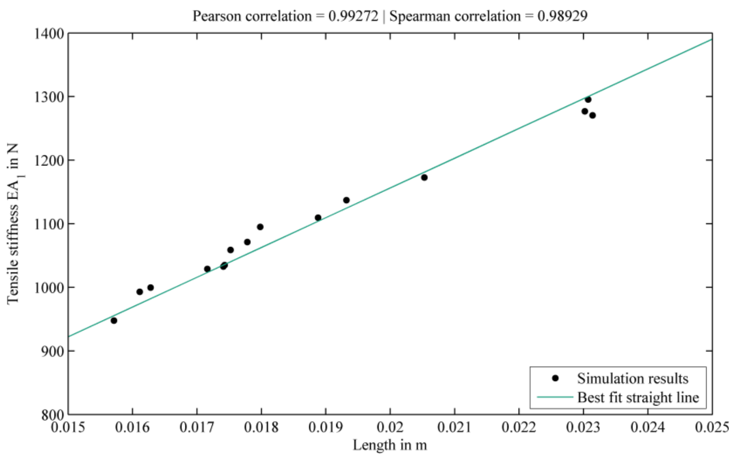



Fig. 9: Correlation between segment length and tensile stiffness (elastic plastic deformation)

With increasing wire harness length, the length-related stiffness $EA_1 = \frac{F_{res}}{u} \cdot l$ increases (with the resulting force $F_{res}$, the forced deflection $u$ and the wire harness length $l$) (see also [6]). The Pearson and Spearman correlation coefficients for this parameter variation exhibit a value near +1, which underlies the expectable linear dependence between the segment length and the length-related stiffness. Generally, correlation coefficients are a measure of the linear dependence between two variables, giving a value between +1 and −1 inclusive, where +1 is total positive linear correlation, 0 is no linear correlation, and −1 is total negative linear correlation. The deviation to the green best fit straight line probably occurs due to the influence of the automatically generated mesh on the simulation results.

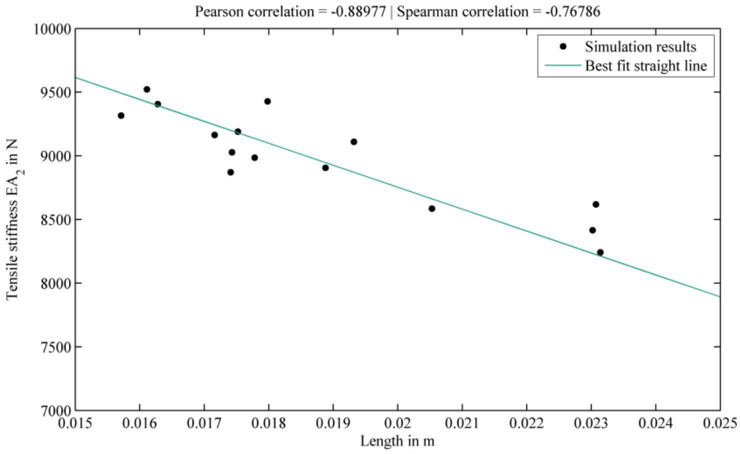
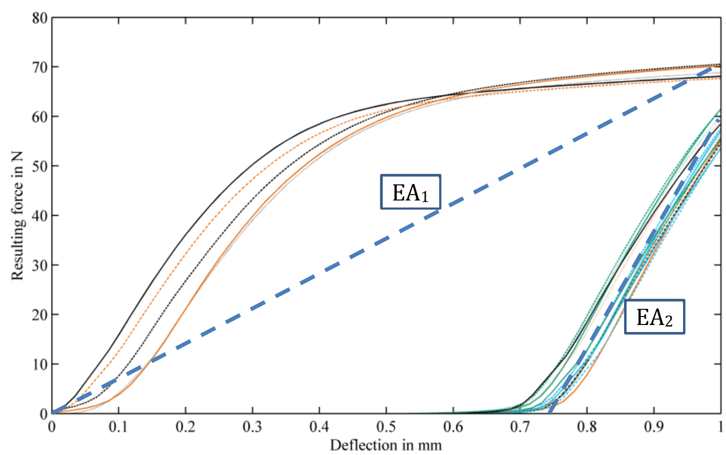



Fig. 10: Correlation between segment length and tensile stiffness (pure elastic deformation)

In Table 1 representative results for the tensile load case of the experimentally determined data and the corresponding simulation results are depicted. The figure on the left shows the measured force (in N) plotted against the deflection (in mm). Based on the slope of the graph, the tensile stiffness of the wire harness is determined. This value of the slope is multiplied by the sample length, in order to calculate the length-related stiffness. This stiffness value is significantly relevant for a subsequent laying simulation. The calculation steps and the experimentally determined stiffness in comparison to the simulated ones are listed on the right.




Approximated stiffness (test data)

$$EA_1 = \frac{70\ N}{1\ mm} \cdot 100\ \mathrm{mm} = 7000\ \mathrm{N}$$

$$EA_2 = \frac{55\ \mathrm{N}}{0.3\ \mathrm{mm}} \cdot 100\ \mathrm{mm} = 18333\ \mathrm{N}$$

Simulation results with adapted equivalent material data (cupper, PVC)

$EA_1 = 6500\ \mathrm{N}$

$EA_2 = 16500\ \mathrm{N}$

Table 1: Experimentally determined data (left), approximated stiffness from measurements and simulated results of a wire harness segment

## Conclusion

The presented modelling approach for wire harnesses is a basic step to reduce time and costs in development processes involving wire harness routing and laying. The automation of the process allows the realization of large test series as well as the parallelization of routing simulations and the assessment of mechanical properties of harness designs. The modelling approach is validated by test data and the performance is demonstrated.

Future work will focus on more efficient numerical models by homogenized material models and data. Additionally the statistical scatter caused by the manual manufacturing process of wire harnesses will be analyzed and taken into account.

The implementation of the developed and proposed methods will give Fraunhofer LBF the opportunity to provide an add-on software solution to interested customers. It is expected that the project results from IDEaliSM will not only be utilized in the automotive industry. An extended and modified application can be provided to other target groups, i.e. aerospace and other engineering industries.

**Acknowledgment**

The authors would like to express their gratitude to the consortium members of the European research project IDEaliSM for their support and contributions. The research leading to these results was performed within the European ITEA3 project IDEaliSM (#13040) as part of the Eureka cluster program.

**References**

[1] Rudolph, S.; Beichter, J.; Eheim, M.; Hess, S.; Motzer, M. and Weil, R.: “On multi-disciplinary architectural synthesis and analysis of complex systems with graph-based design languages”, DGLR Jahrestagung 2013, Stuttgart, Germany, September 10-12, 2013.

[2] Atzrodt, H.: “Mehradrige Kabel in der Verlegesimulation”, In: Fraunhofer LBF Annual Report 2014, Publisher: Fraunhofer-Institut für Betriebsfestigkeit und Systemzuverlässigkeit LBF, Bartningstraße 47, 64289 Darmstadt, 2014, S. 76–77, ISSN: 1864-0958.

[3] Ghoreishi, S.; Messager, T.; Cartraud, P. and Davies, P.: “Validity and limitations of linear analytical models for steel wire strands under axial loading, using a 3D FE model”, International Journal of Mechanical Sciences, vol. 49, no. 11, 2007, pp. 1251-1261.

[4] Bathe, K.J.: “Finite Element Procedures in Engineering Analysis”, Prentice-Hall civil engineering and engineering mechanics series, Prentice-Hall, isbn 9780133173055, 1982.

[5] Stoll, G.; Pöllmann, J.; Atzrodt, H.; Schmidgall, G.: “An Automated Process for Numerical Evaluation of Cable Stiffnesses”, In: Benchmark the international magazine for engineering designers & analysts from NAFEMS. Backford Street, Hamilton, Lanarkshire, ML3 0BT, UK: NAFEMS Beckford Business Centre, S. 10–18. ISSN: 0951 6859, 2014.

[6] Pilkey, W. D.: “Formulas for Stress, Strain and Structural Matrices”, John Wiley & Sons, INC., Holboken, New Jersey, US, ISBN 0-471-03221-2, 2005.

[7] VDA-Recommendation 4968: “Vehicle Electric Container (VEC)”, 2nd edition, February 2014 http://www.vda.de/de/publikationen/publikationen_downloads/detail.php?id=1025