SIMBEAM Reloaded

The first versions of SIMPACK in the early 1990's could already model flexible Bodies consisting of beam elements without an interface to finite element software. The first available preprocessor was called BEAM, which could model straight beams based on the differential equations of the Euler-Bernoulli beam theory. BEAM was frequently used to model leaf springs and to consider the basic modes of a railway carbody, among other tasks. However, as BEAM did not allow users to model beam structures with an arbitrary topology and geometry (such as a stabilizer or a chassis sub frame), Intec GmbH (now named SIMPACK AG) decided to develop SIMBEAM in 2001. Right from the start, SIMBEAM was based on the finite element approach. Euler-Bernoulli, Timoshenko beam elements, rigid body elements and arbitrary topology and geometry could be used for setting up flexible bodies.

This made SIMBEAM the basis for modeling several flexible bodies in drivetrain, engine and wind energy applications. With the completely new implementation of SIMBEAM in SIMPACK 9.4, we seized the opportunity to expand its capabilities towards an automated set-up of the equations of motion, full parameterization by SubVars, and the use of scripting for setting up and modifying SIMBEAM models. This allows the user to carry out automatic parametric studies of entire systems, including variants of the flexible structures. Extra utilities, such as ASCII input decks for quickly setting up flexible towers for wind turbines or suspension leaf springs, are no longer necessary.

"SIMBEAM allows the user to carry out automatic parametric studies of entire systems, including variants of the flexible structures."

MODULE DESCRIPTION, LICENSING

In the SIMBEAM module, the beam structure — consisting of nodes, elements, cross section and material properties — can be set up in the SIMPACK Pre(-processor). The modal representation of the flexible Body is instantaneously generated and updated when any parameter or property of the beam structure is changed. If requested, the SIMPACK Post(-processor) can display element stress as a contour plot
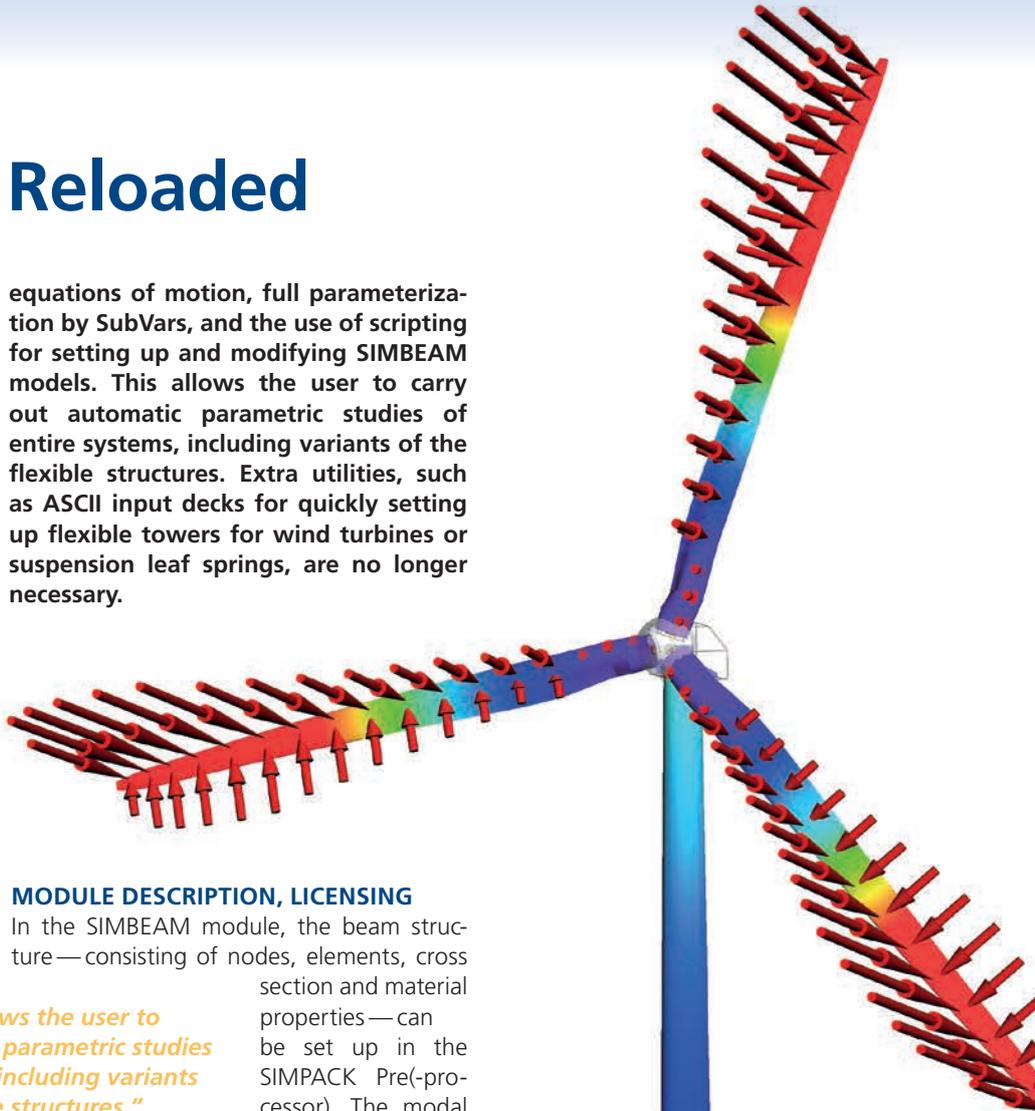

Fig. 1: Definition of cross sections for a drive shaft with conical sections

Fig. 2: Offshore wind turbine. SIMBEAM structures: rotorblades, tower and jacket
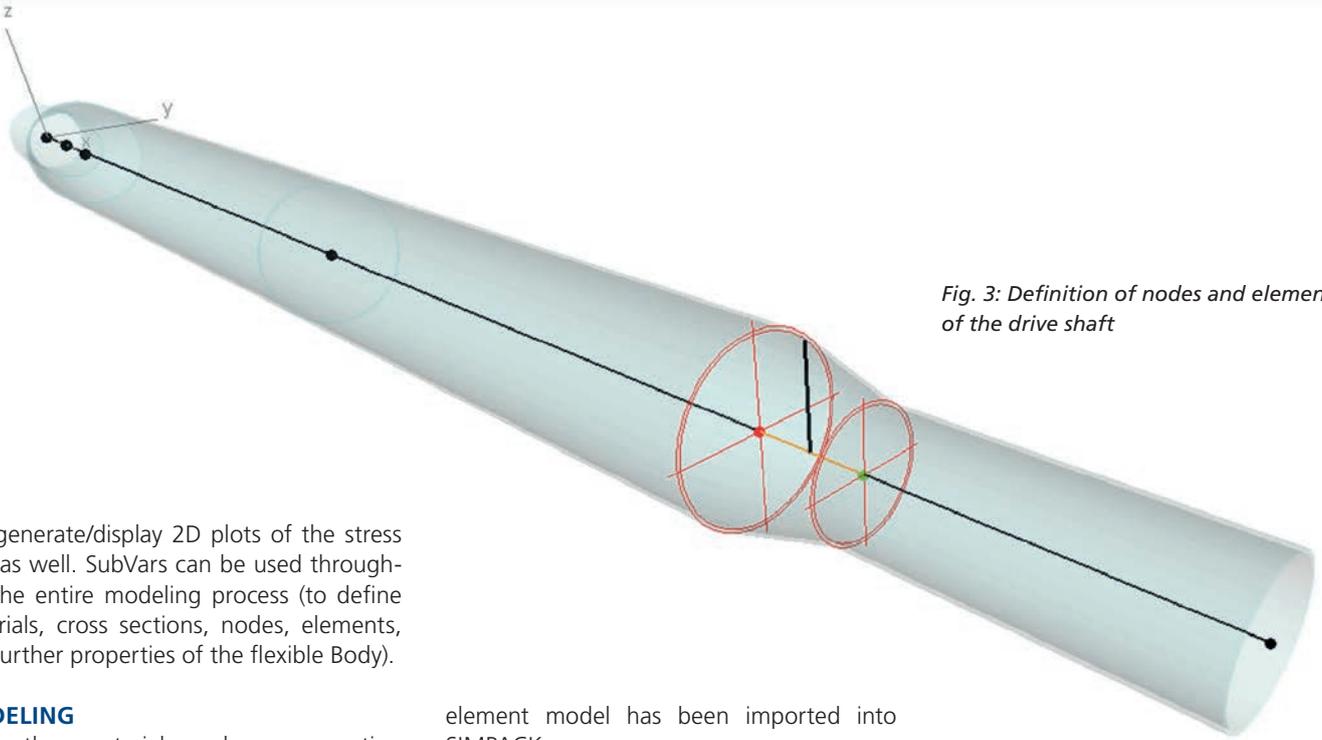
Fig. 3: Definition of nodes and elements of the drive shaft

and generate/display 2D plots of the stress data as well. SubVars can be used through-out the entire modeling process (to define materials, cross sections, nodes, elements, and further properties of the flexible Body).

MODELING

Once the material and cross section properties are set up, beam elements can be defined between nodes. Nodes were introduced into SIMPACK 9.4 as a new modeling element in order to make a clear distinction between nodes and Markers, and to enable a model that is consistent with flexible Bodies imported via FlexModal (FE-interface to NASTRAN, ABAQUS, ANSYS etc.). Finally, in the equations of motion, the deformation of SIMBEAM Bodies is represented by a modal approach; the same approximation that is used when a finite

"With the completely redesigned graphical user interface, getting started and setting up models is easier than ever."

element model has been imported into SIMPACK.

Linear isotropic material is defined by Young's modulus, Poisson's ratio and the shear modulus. A combination of two of these parameters is used to define the material. The density is used to set up the mass matrix. The de-fined materials appear in the model tree.

Standard cross section types like circles, rectangles and ellipses (either full or hol-low) are defined by their respective physical dimensions. The values of the physical cross section parameters (cross section area, area

moments of inertia, torsional stiffness) are computed from the dimensions. The geom-etry and physical cross section parameters are shown in an element plot in the Cross Section property dialog. Further cross sec-tions can be represented by the "General" cross section types. The "General Basic" type enables the use of arbitrary double symmetric cross section geometries with homogeneous mass density, whereas the "General Advanced" Cross Section type can additionally incorporate twist-bend coupling and a nonhomogeneous mass density into
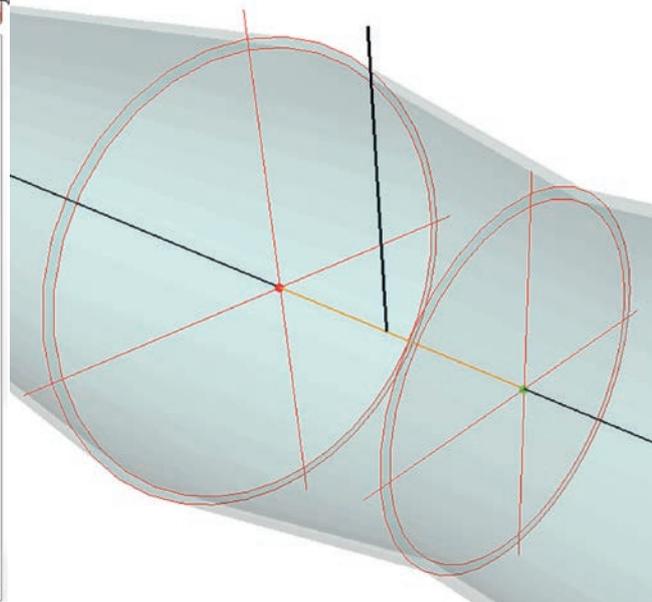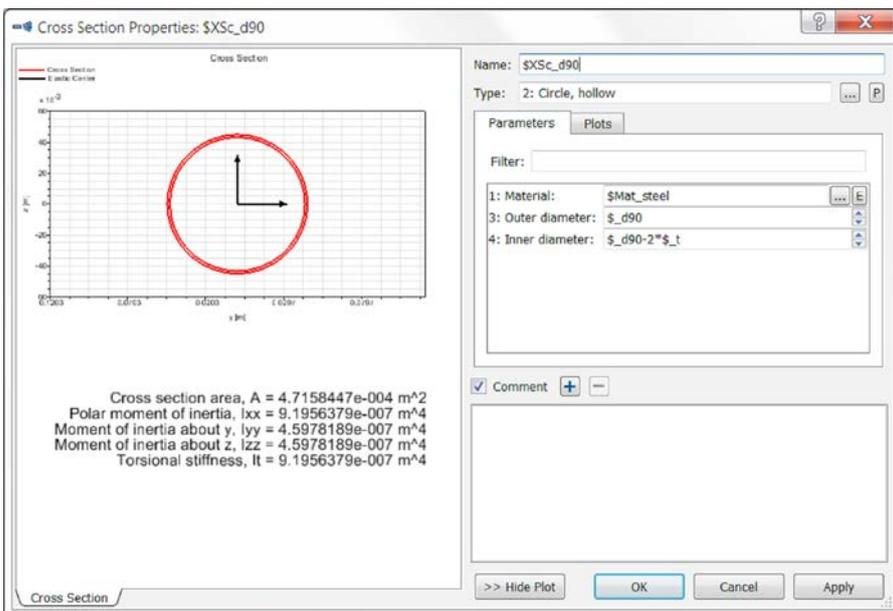

Fig. 4: The finished drive shaft model — cross section definitions are shown for elements selected in the user interface

the beam elements. The "General Basic" Cross Section requires as input the cross section area, the moments of inertia and the torsional stiffness. In the case of the "General Advanced" Cross Section type, these values are input in combination with their respective material parameters. If one of the "General" Cross Section types is used, geometry information for the 3D Page is generated based on a polyline that can be specified in the Cross Section dialog.

Switching the Body type to SIMBEAM yields a new layout of the user interface. The first tab, "SIMBEAM", is used to set up nodes and elements.

The positions of the nodes are specified with respect to the Body Reference Frame. Once defined in the node table, their position is displayed on the 3D Page. The data can be pasted from Excel sheets into the node list. Otherwise, the node list may be exported from the body property dialog into a comma separated file.

A usual beam element definition — consisting of connectivity, element type, cross section type, and cross section orientation — can be defined in the element list of the Body property dialog. The new SIMBEAM enables the user to assign two different cross sections of the same standard cross section type to the nodes of the same element, assuming a linear change of the cross section dimensions along the beam element axis. This new feature allows a considerable increase in accuracy with a smaller number of elements, and therefore, an easier set-up of the model, e.g., for drive shafts with conical sections and leaf springs with variable cross section height over the length of the longitudinal axis. For the elements being defined and selected, the 3D Page shows their connectivity, their assigned cross sections, and the cross section orientation.

For non-selected elements, the 3D Page shows the element axes. Once valid, node and beam definitions are applied to the SIMBEAM
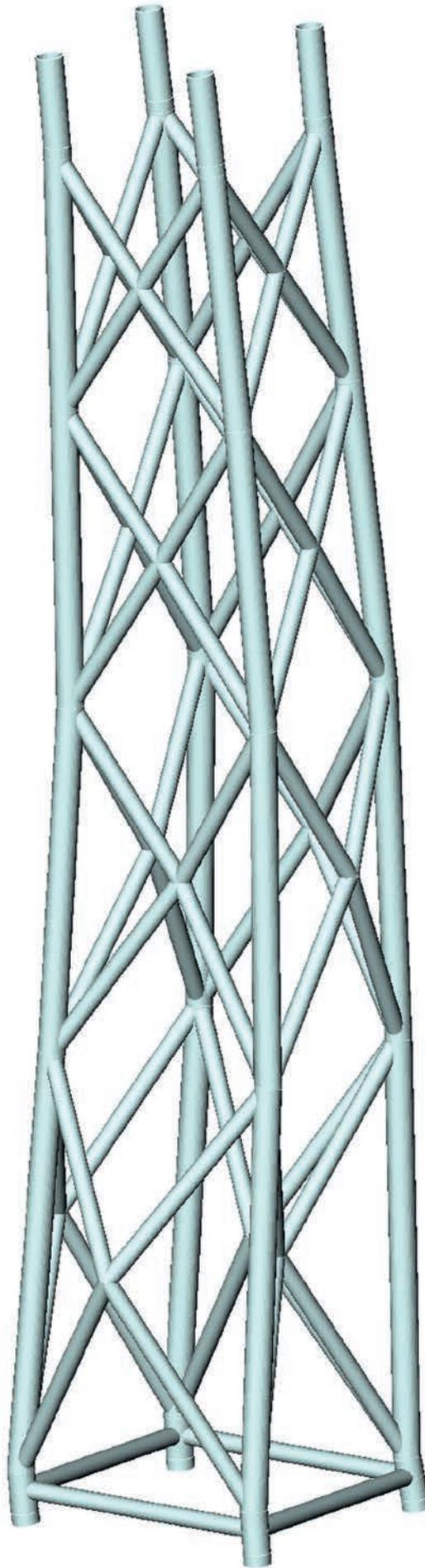
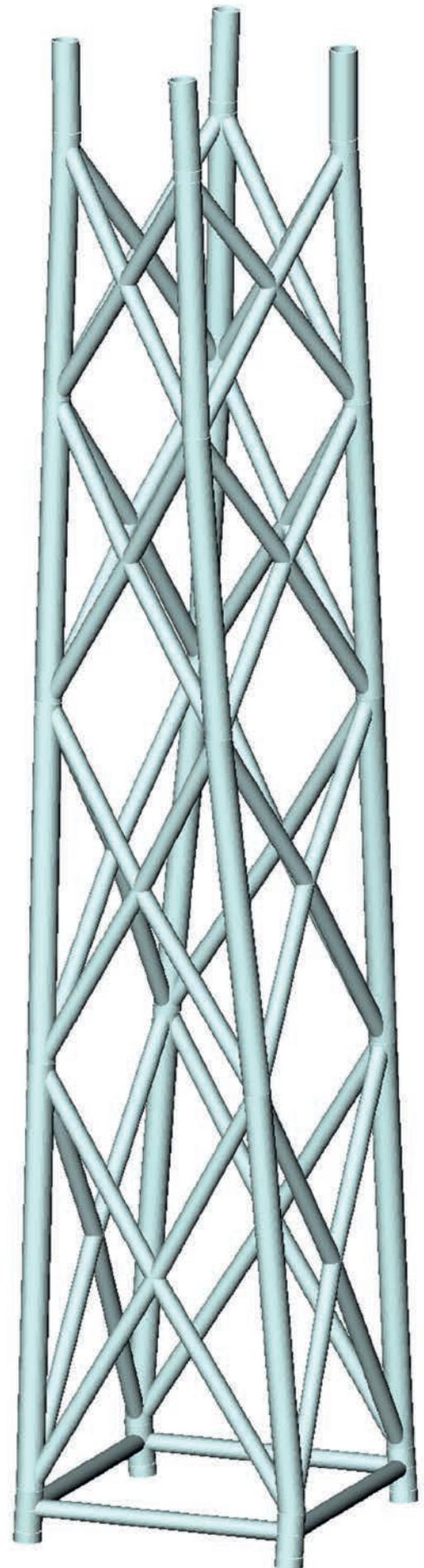*Fig. 5: Bending eigenmode of the offshore jacket*

Fig. 6: Torsional eigenmode of the offshore jacket

Bending moment about Z

x 10³

1.20
1.12
1.04
0.96
0.88
0.80
0.72
0.64
0.56
0.48
0.40
0.32
0.24
0.16
0.08
0.00
-0.08
-0.16
-0.24
-0.32
-0.40
-0.48
-0.56
-0.64
-0.72
-0.80
-0.88
-0.96
-1.04
-1.12
-1.20

Node: 4 (Refernced by 476)
Bending moment about Z:
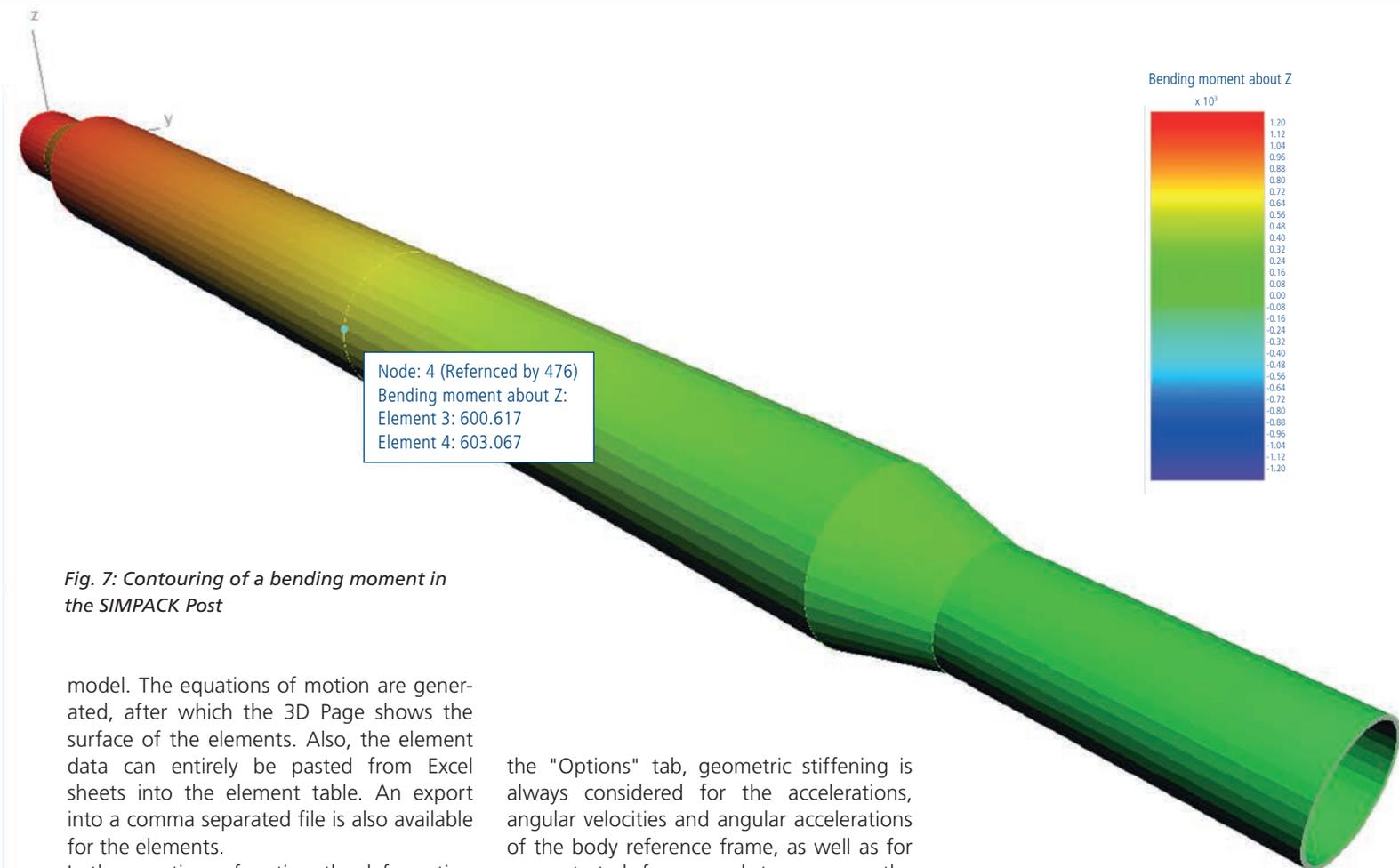Element 3: 600.617
Element 4: 603.067

Fig. 7: Contouring of a bending moment in the SIMPACK Post

model. The equations of motion are generated, after which the 3D Page shows the surface of the elements. Also, the element data can entirely be pasted from Excel sheets into the element table. An export into a comma separated file is also available for the elements.

In the equations of motion, the deformation of the SIMBEAM Body is represented by a modal approach consisting of eigenmodes and interface modes. Eigenmodes are considered for a user-defined frequency range that is relevant to the respective application. Interface modes are used to obtain an accurate response to loads that act on a Body via Joints, Force Elements and Constraints.

Either inertia relief modes or frequency response modes may be used for this purpose. These interface load cases are generated for nodes which have a connection to Force Elements, Constraints and Joints via the Marker-node connection types (rigid body link, interpolation, etc.) if the interface mode generation is in the "automatic" mode. Manual interface mode generation for selected directions is available in the Marker dialog.

The tabs "Mass properties", "Modes" and "Options" have the same functionality as in FlexModal, except for the check box labeled "Stiffness depends on loads", which enables or disables the geometric stiffening terms in the equations of motion. Geometric stiffening represents the preloads, which can act on the structure and lower or increase the stiffness of the flexible Body. If enabled in

"...everything can be described by SubVars, and every modeling step can be achieved by scripting commands, making it easy to generate variants..."

the "Options" tab, geometric stiffening is always considered for the accelerations, angular velocities and angular accelerations of the body reference frame, as well as for concentrated forces and torques on the nodes which are subjected to loads. These terms are automatically considered for the aforementioned interface load cases regarding concentrated forces and torques.

Stress recovery data is generated if requested in the "Loads" tab. Then, the cross section forces (i.e., the normal force, the shear forces, the bending moments and the torsional torque) can be displayed as contour plots in the SIMPACK Post. Corresponding 2D plots over time can also be generated and displayed. By default, the 3D Page and the SIMPACK Post show the beam model according to the assumption that cross sections are always perpendicular to the respective beam axis in the undeformed state, which yields a graphical representation with gaps if the beam axis of two adjoining elements is kinked. This makes it easier to detect modeling errors. However, for presentation purposes, this can be replaced by a CAD representation of the beam structure which can be loaded optionally into the flexible Body Primitive.

FUTURE DEVELOPMENTS

Beyond SIMPACK version 9.4, SIMBEAM will be extended towards an easier set-up of detailed models. This could be achieved by

subdivision of one element into a number of elements in order to easily increase the accuracy of the representation of inertia forces within the structure. Conversely, unnecessary details could be removed from the modeling by merging a group of elements into a smaller number of elements. Preliminary tests revealed significant added value from the variable cross section approach. It seems promising to extend this approach to the "General" Cross Section types as well.

BENEFIT OF THE NEW SIMBEAM

With the completely redesigned graphical user interface, getting started and setting up models is easier than ever. The physical dimensions of a SIMBEAM Body, and everything else, can be described by SubVars. Every aforementioned step can be achieved by scripting commands, making it easy to generate variants thereof with a solver script. This makes it possible to adjust the properties of flexible structures in automated calculations in order to achieve an improved design. Users can now easily set up parameterized flexible components, e.g., wind turbine towers or suspension leaf springs. Thus, the new SIMBEAM is the final link required for fully automated DoE of multi-body systems, including variations of flexible structures.