BladeBeam V2.3.4 Structural Analysis Software
User and Theory Guide

Gabriel Ackall

January 5, 2026

This document covers the theoretical foundations, implementation details, and usage guidelines for BladeBeam, a Python-based structural analysis application designed specifically for wind turbine blade analysis. The program integrates matrix structural analysis methods with advanced composite section modeling through NuMAD input methodology and the *abdbeam* library to provide structural analyses results for blade structures.

For the latest version of BladeBeam and associated documentation, please visit: https://github.com/gackall/BladeBeam.

Contents

1 Theoretical Foundations

1.1 Matrix Structural Analysis Theory

BladeBeam employs classical matrix structural analysis methods based on the direct stiffness method, following established principles from matrix methods of structural analysis. The fundamental approach treats structures as assemblages of discrete elements connected at nodes, where the relationship between forces and displacements is expressed through stiffness matrices.

1.2 Beam Element Formulation

The program utilizes Euler-Bernoulli beam theory for its primary structural elements. Each beam element possesses six degrees of freedom (three translations and three rotations) at each of its two nodes, resulting in a 12×12 local stiffness matrix. The local stiffness matrix for a beam element is formulated as:

$$
\mathbf{K}_{local} = \begin{bmatrix}
\frac{EA}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{EA}{L} & 0 & 0 & 0 & 0 & 0 \\
0 & \frac{12EI_{zz}}{L^3} & 0 & 0 & 0 & \frac{6EI_{zz}}{L^2} & 0 & -\frac{12EI_{zz}}{L^3} & 0 & 0 & 0 & \frac{6EI_{zz}}{L^2} \\
0 & 0 & \frac{12EI_{yy}}{L^3} & 0 & -\frac{6EI_{yy}}{L^2} & 0 & 0 & 0 & -\frac{12EI_{yy}}{L^3} & 0 & -\frac{6EI_{yy}}{L^2} & 0 \\
0 & 0 & 0 & \frac{GJ}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{GJ}{L} & 0 & 0 \\
0 & 0 & -\frac{6EI_{yy}}{L^2} & 0 & \frac{4EI_{yy}}{L} & 0 & 0 & 0 & \frac{6EI_{yy}}{L^2} & 0 & \frac{2EI_{yy}}{L} & 0 \\
0 & \frac{6EI_{zz}}{L^2} & 0 & 0 & 0 & \frac{4EI_{zz}}{L} & 0 & -\frac{6EI_{zz}}{L^2} & 0 & 0 & 0 & \frac{2EI_{zz}}{L} \\
-\frac{EA}{L} & 0 & 0 & 0 & 0 & 0 & \frac{EA}{L} & 0 & 0 & 0 & 0 & 0 \\
0 & -\frac{12EI_{zz}}{L^3} & 0 & 0 & 0 & -\frac{6EI_{zz}}{L^2} & 0 & \frac{12EI_{zz}}{L^3} & 0 & 0 & 0 & -\frac{6EI_{zz}}{L^2} \\
0 & 0 & -\frac{12EI_{yy}}{L^3} & 0 & \frac{6EI_{yy}}{L^2} & 0 & 0 & 0 & \frac{12EI_{yy}}{L^3} & 0 & \frac{6EI_{yy}}{L^2} & 0 \\
0 & 0 & 0 & -\frac{GJ}{L} & 0 & 0 & 0 & 0 & 0 & \frac{GJ}{L} & 0 & 0 \\
0 & 0 & -\frac{6EI_{yy}}{L^2} & 0 & \frac{2EI_{yy}}{L} & 0 & 0 & 0 & \frac{6EI_{yy}}{L^2} & 0 & \frac{4EI_{yy}}{L} & 0 \\
0 & \frac{6EI_{zz}}{L^2} & 0 & 0 & 0 & \frac{2EI_{zz}}{L} & 0 & -\frac{6EI_{zz}}{L^2} & 0 & 0 & 0 & \frac{4EI_{zz}}{L}
\end{bmatrix}
$$

where E represents the elastic modulus, L is the element length, I_{zz} and I_{yy} are the second moments of area about the principal axes, and A is the cross-sectional area (Ferreira et. al., 2020) (Kassimali, 2011) (Logan, 2017).

Similarly, a local consistent mass matrix is formulated for dynamic purposes as follows:

$$
\mathbf{M}_{local} = m \cdot L \cdot \frac{1}{420}\begin{bmatrix}
140 & 0 & 0 & 0 & 0 & 0 & 70 & 0 & 0 & 0 & 0 & 0 \\
0 & 156 & 0 & 0 & 0 & 22L & 0 & 54 & 0 & 0 & 0 & -13L \\
0 & 0 & 156 & 0 & -22L & 0 & 0 & 0 & 54 & 0 & 13L & 0 \\
0 & 0 & 0 & 140r_x^2 & 0 & 0 & 0 & 0 & 0 & 70r_x^2 & 0 & 0 \\
0 & 0 & -22L & 0 & 4L^2 & 0 & 0 & 0 & 13L & 0 & -3L^2 & 0 \\
0 & 22L & 0 & 0 & 0 & 4L^2 & 0 & -13L & 0 & 0 & 0 & -3L^2 \\
70 & 0 & 0 & 0 & 0 & 0 & 140 & 0 & 0 & 0 & 0 & 0 \\
0 & 54 & 0 & 0 & 0 & -13L & 0 & 156 & 0 & 0 & 0 & 22L \\
0 & 0 & 54 & 0 & 13L & 0 & 0 & 0 & 156 & 0 & -22L & 0 \\
0 & 0 & 0 & 70r_x^2 & 0 & 0 & 0 & 0 & 0 & 140r_x^2 & 0 & 0 \\
0 & 0 & 13L & 0 & -3L^2 & 0 & 0 & 0 & -22L & 0 & 4L^2 & 0 \\
0 & -13L & 0 & 0 & 0 & -3L^2 & 0 & 22L & 0 & 0 & 0 & 4L^2
\end{bmatrix}
$$

where L is the element length, r_x is the polar radius of gyration, and m is element mass.

An additional functionality is implemented to consider shear deformation through the definition of section shear stiffnesses, *kGA*, comprised of the shear stiffness factor, *k*, shear modulus, *G*, and area, *A*. When shear deformation is toggled to be considered, bending components of the stiffness matrix are multiplied by the following factor:

$$
\frac{1}{1+\phi}
$$

with diagonal and off-diagonal rotational terms being multiplied by the additional factors $(4 + \phi)$ and $(2 - \phi)$ respectively, where

$$
\phi = \frac{12EI}{kGA \cdot L^2}.
$$

It is the author's intention to include functionality to calculate shear stiffnesses automatically when beam sections are inputted using the NuMAD methodology, but this is not currently implemented. Currently, whether using NuMAD methodology [Section 1.7] or not, if shear deformation is considered, columns kGA_z and column kGA_y must be filled out, representing shear stiffnesses in the z and y directions, respectively.

1.3 Semi-Rigid Connection Modeling

BladeBeam incorporates semi-rigid connection modeling through spring elements, addressing the limitations of traditional pinned or fixed assumptions. This approach implements rotational spring stiffness values to represent partially restrained connections.

The spring element stiffness matrix connects beam nodes to ground nodes through translational and rotational springs with user-defined stiffnesses:

$$[K_{spring}] = \begin{bmatrix} k_x & 0 & 0 & 0 & 0 & 0 & -k_x & 0 & 0 & 0 & 0 & 0 \\ 0 & k_y & 0 & 0 & 0 & 0 & 0 & -k_y & 0 & 0 & 0 & 0 \\ 0 & 0 & k_z & 0 & 0 & 0 & 0 & 0 & -k_z & 0 & 0 & 0 \\ 0 & 0 & 0 & k_{rx} & 0 & 0 & 0 & 0 & 0 & -k_{rx} & 0 & 0 \\ 0 & 0 & 0 & 0 & k_{ry} & 0 & 0 & 0 & 0 & 0 & -k_{ry} & 0 \\ 0 & 0 & 0 & 0 & 0 & k_{rz} & 0 & 0 & 0 & 0 & 0 & -k_{rz} \\ -k_x & 0 & 0 & 0 & 0 & 0 & k_x & 0 & 0 & 0 & 0 & 0 \\ 0 & -k_y & 0 & 0 & 0 & 0 & 0 & k_y & 0 & 0 & 0 & 0 \\ 0 & 0 & -k_z & 0 & 0 & 0 & 0 & 0 & k_z & 0 & 0 & 0 \\ 0 & 0 & 0 & -k_{rx} & 0 & 0 & 0 & 0 & 0 & k_{rx} & 0 & 0 \\ 0 & 0 & 0 & 0 & -k_{ry} & 0 & 0 & 0 & 0 & 0 & k_{ry} & 0 \\ 0 & 0 & 0 & 0 & 0 & -k_{rz} & 0 & 0 & 0 & 0 & 0 & k_{rz} \end{bmatrix}$$

where k_x, k_y, and k_z correspond with defined linear spring stiffness in their respective directions and k_{rx}, k_{ry}, and k_{rz} correspond with defined rotational spring stiffness in their respective directions.

This formulation allows users to specify boundary conditions that account for foundation flexibility or connection stiffness, providing more accurate structural responses than idealized constraints.

1.4 Dynamic Analysis and Natural Frequencies

The program calculates natural frequencies through eigenvalue analysis of the mass and stiffness matrices. The generalized eigenvalue problem is solved using:

$$[K - \omega^2 M]\{\phi\} = \{0\}$$

where K is the global stiffness matrix, M is the consistent mass matrix, ω represents circular frequencies, and ϕ are the mode shapes. The consistent mass matrix formulation accounts for distributed mass effects more accurately than lumped mass approaches and is thus chosen.

It is important to note that the current formulation includes both the defined self-weight of a structure (should that option be selected) and the imposed loading on the beam converted to an equivalent mass.

The mass/length value at each station used for the dynamic analysis can either be inputted, or when using the NuMAD input, will be calculated automatically by BladeBeam. This is done through summing over the product of each ply's length times its thickness multiplied by the density of the material that makes up the ply for each ply within each segment in the cross section.

1.5 *abdbeam* Theory and Integration into BladeBeam

BladeBeam integrates the *abdbeam* library (Victorazzo, 2023) for advanced composite beam cross-section analysis. *abdbeam* employs classical lamination theory and provides capabilities for calculating section properties of thin-walled composite structures typical in wind turbine blades (Victorazzo et. al., 2016).

abdbeam is used within BladeBeam to calculate cross section stiffness properties of the blade which are used to assemble a Euler-Bernoulli beam stiffness matrix. After structural analysis is performed, beam moments and forces are then applied to each section and *abdbeam* is used to determine the internal forces per unit length for each segment within a cross-section.

The following simplifying assumptions/limitations apply to *abdbeam*:

- Materials are linearly elastic
- Beams are made of thin-walled elements
- Plane sections remain plane when undergoing bending and remain perpendicular to the axis of the beam
- Shear deformation and restrained section warping are neglected

While the effects of restrained warping can reasonably be neglected for wind turbine blades, which are closed section shapes (Pluzsik & Kollar, 2002), shear deformations sometimes must be considered depending on the geometry of the beam being analyzed. Shear deformation functionality has been introduced separately from *abdbeam* [Section 1.2], but shear stiffnesses must still be manually calculated at this time. Previous wind turbine blade analyses using beam methods have used a k factor equal to 1 and GA values based on the projected area of the spar caps and leading/trailing edges of the blade, excluding sandwich panels and blade webs (Gentry et. al. 2024) (Alshannaq et. al. 2021).

1.6 Ply Stress and Strain Calculations

Ply strains and stresses are calculated for each segment within each cross section following basic Classical Laminate Theory (CLT) as outlined in (Nettles, 1994) and (Haynes et. al., 2016). Due to the overwhelming amount of stress and strain data if every ply stress/strain was returned for every segment within every beam cross section along the blade, only the plies with maximum stresses/strains are reported. This methodology has a downside that, depending on the failure criteria used, the ply that fails first may not be the ply deemed to be the critical ply with the maximum stresses. Therefore, engineering judgement should be utilized when determining if the critical ply can be safely used for determining whether the laminate will fail or not. This limitation is acknowledged, and assumed to be adequate for most applications.

Using the segment internal forces calculated by *abdbeam*, midplane strains and curvatures are determined for each segment using the ABD matrix calculated by *abdbeam* for the laminate. Strains and internal forces are coupled following the relationship below:

$$\begin{bmatrix} \mathbf{N} \\ \mathbf{M} \end{bmatrix} = \begin{bmatrix} \mathbf{A} & \mathbf{B} \\ \mathbf{B} & \mathbf{D} \end{bmatrix} \begin{bmatrix} \boldsymbol{\varepsilon}^0 \\ \boldsymbol{\kappa} \end{bmatrix}$$

Ply level strains are then determined according to Kirchhoff-Love assumptions for thin shells, with a laminate z-coordinate system defined with its origin at the midplane. Ply level strains are calculated as follows with a linearly varying strain distribution:

$$\varepsilon_x(z) = \varepsilon_x^0 + z\kappa_x$$

$$\varepsilon_y(z) = \varepsilon_y^0 + z\kappa_y$$

$$\gamma_{xy}(z) = \gamma_{xy}^0 + z\kappa_{xy}$$

These strains are transformed to the ply coordinates for plies at an off angle, θ, to the laminate through the following:

$$\boldsymbol{\varepsilon}_{12} = \mathbf{T}_\varepsilon \boldsymbol{\varepsilon}_{xy}$$

where

$$\mathbf{T}_\varepsilon = \begin{bmatrix} c^2 & s^2 & sc \\ s^2 & c^2 & -sc \\ -2sc & 2sc & c^2 - s^2 \end{bmatrix},$$

$$c = \cos(\theta), \quad s = \sin(\theta)$$

The reduced stiffness matrix of each ply can then be calculated to determine the ply stresses as follows:

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \tau_{12} \end{bmatrix} = \mathbf{Q} \begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{bmatrix}$$

where

$$\nu_{21} = \nu_{12}\frac{E_2}{E_1},$$

$$\Delta = 1 - \nu_{12}\nu_{21},$$

$$\mathbf{Q} = \begin{bmatrix} Q_{11} & Q_{12} & 0 \\ Q_{12} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{bmatrix} = \begin{bmatrix} \frac{E_1}{\Delta} & \frac{\nu_{12}E_2}{\Delta} & 0 \\ \frac{\nu_{12}E_2}{\Delta} & \frac{E_2}{\Delta} & 0 \\ 0 & 0 & G_{12} \end{bmatrix}$$

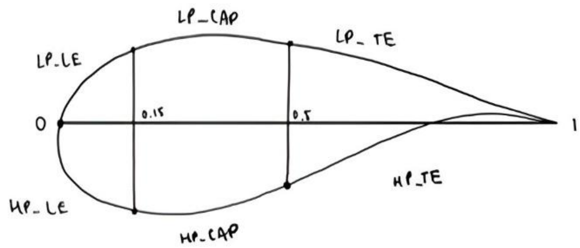
For each segment, the ply with the maximum ply stress (for longitudinal, transverse, or shear stress) is determined and those plies' stresses and strains are saved. These plies are denoted as the critical plies for each segment.

1.7 NuMAD Integration Theory

NuMAD (Numerical Manufacturing and Design) provides a comprehensive framework for wind turbine blade design and analysis (Sandia National Laboratories, 2021). NuMAD uses a structured Excel spreadsheet format to define the complete geometric and material properties of wind turbine blades (Berg et. al., 2012). The methodology divides the blade into discrete spanwise stations, where each station represents a cross-sectional "slice" of the blade at a specific distance from the root. At each station, the blade cross-section is defined by an airfoil shape that is subdivided into multiple structural segments (such as leading edge, trailing edge, spar caps, and aerodynamic shell regions), with each segment assigned a unique composite laminate layup. It should be noted that these segments are different from the segments used by *abdbeam* as their main goal is to assign material laminates to the blade while the *abdbeam* segments are much more numerous to be able to capture the geometry of the blade, while also capturing material properties.

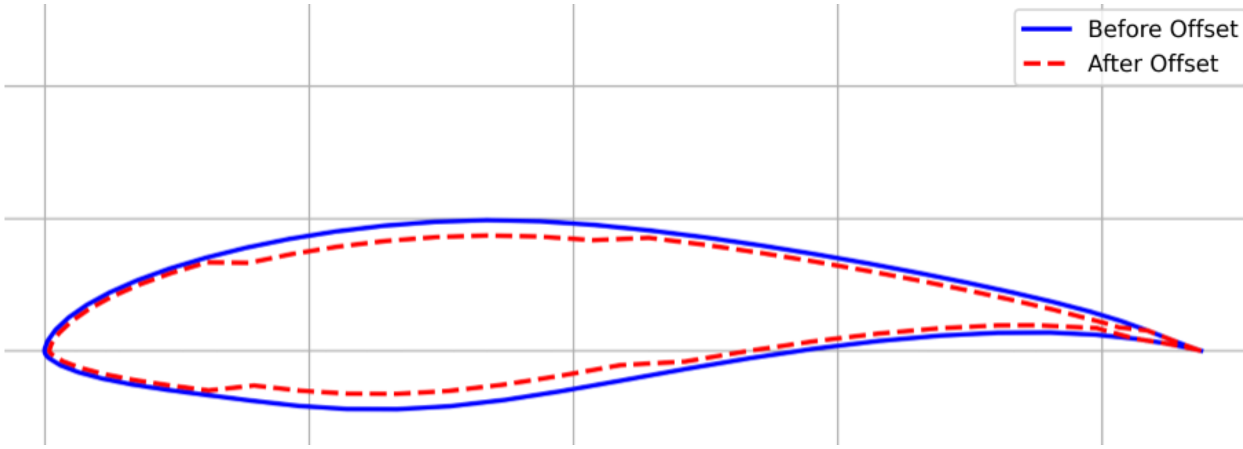
The NuMAD Excel file contains multiple worksheets that work together to fully define the blade structure. The primary worksheets are the Geometry tab and the Materials tab, which contain all essential information about blade shape, structural layout, and composite material properties. The Geometry tab defines the blade's spanwise variation in parameters such as chord length (the width of the airfoil), twist angle (the rotation of the airfoil section for aerodynamic optimization), and the specific airfoil shape file to use at each station. Additionally, this tab specifies how each airfoil cross-section is divided into structural segments and which composite material "stacks" (layup sequences) are assigned to each segment. These segments are defined by their chordwise position along the airfoil, typically normalized from the leading edge (x = 0) to the trailing edge (x = +1).

It should be noted that the NuMAD format specified plies in a stack/laminate starting from the outer surface going towards the inner surface, meanwhile *abdbeam* specifies plies from the bottom (inner) surface towards the top (outer) surface. This is accounted for within BladeBeam and thus when using NuMAD, proper NuMAD convention should be followed, with the outer surface plies being listed first in the order of the laminate.



The program processes airfoil coordinate files and applies geometric transformations including scaling by chord length and rotation for twist. The airfoil coordinates are interpolated to create integration points for shear web connections, ensuring proper structural connectivity in the analysis model.

Airfoils in NuMAD inputs generally specify the outer surface of the wind turbine blade, however, *abdbeam* works by specifying plies from their middle surface. In order to convert airfoil coordinate files defining the exterior surface of the wind turbine blade to one that can be used by *abdbeam*, the airfoil can be offset inwards by half the laminate thickness to the midline of the laminate. This setting by default is toggled on, but can be turned off if the input airfoil shape is already of the midline of the wind turbine blade laminates, or if no offset is desired. It should be noted that this offset can create artifacts where segments with large thickness discrepancies connect and at the trailing edge, where the sharp corner may not be able to be offset by half of the thickness of the laminate without intersecting with the other side. If this occurs at the trailing edge, the program will iterate the level of offset until no intersection is detected. An image showcasing this offset and its artifacts can be seen below. These artifacts can sometimes cause the strong axis (edgewise/lagwise) bending stiffness of the blade to be slightly more inaccurate, but offsetting the wind turbine blade when needed generally leads to much more accurate weak axis (flapwise) bending stiffness and torsional stiffness values. This tradeoff should be considered by the engineer when using the program. As with many other aspects of the program, as laminate thickness decreases relative to the overall size of the wind turbine blade, the effects of offsetting or not offsetting the wind turbine blade geometry become less impactful.



For stations within the NuMAD input excel file that are not populated with chord distance, material laminate data, and etc., BladeBeam uses cubic spline interpolation, consistent with the methodology originally used by NuMAD source code.

2 Input Files and GUI Functionality

2.1 Excel Input File Structure

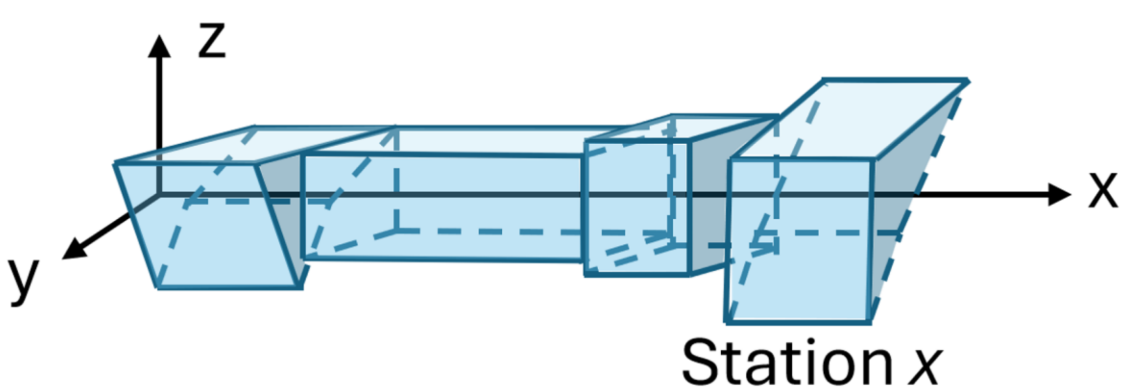
BladeBeam utilizes a structured Excel workbook format with two primary worksheets: 'Setup' and 'Elements'. A ‘Readme’ tab is also provided as additional documentation. This format provides flexibility for defining complex blade geometries and loading conditions while maintaining compatibility with standard spreadsheet software.

In order for the program to function correctly, Excel column headers cannot be modified.

The Setup worksheet defines nodal information and boundary conditions. Key columns include:

Stations: Defines the spanwise positions along the blade structure in the selected length units. These positions correspond to the locations where boundary conditions, loads, or other constraints are applied.

x: Specifies precise positions for point loads and boundary conditions. These should be arranged to match station locations, however, in the case that a point load or boundary condition does not match a station location, it will be applied at the closest station. This allows for flexible placement of supports and applied forces.



Fixity Columns: Define boundary condition types using numerical values:

- 0: Free degree of freedom
- 1: Fixed (infinite stiffness)
- Positive values: Semi-rigid spring stiffness values (semi-rigid setting must be turned on)

The fixity columns correspond to specific degrees of freedom: FixityX (along x-axis), FixityY (along y-axis), FixityZ (along z-axis), FixityT (torsion about x-axis), FixityMY (moment about y-axis), and FixityMZ (moment about z-axis).

Load Columns: Define applied point loads and moments at specified locations. The program supports Loadx (along x-axis), Loady (along y-axis), Loadz (along z-axis), LoadT (torsion about x-axis), LoadMY (moment about y-axis), and LoadMZ (moment about z-axis).

To input a point load or moment at a location that does not have an existing support, add a new row to the spreadsheet at the corresponding x location with all fixity columns set to 0 and load columns populated as desired. It is best practice to ensure that the order of any point loads and boundary conditions is such that the x coordinates are in ascending order.

The Elements worksheet defines the structural properties and distributed loading for each beam element:

Distributed Loads: The program accepts distributed loads in three directions: w_z (vertical), w_y (lateral), and t (torsional). These loads are applied uniformly over each element length and converted to equivalent nodal forces during assembly.

Section Properties: When NuMAD integration is not used, users must specify cross-sectional area (A), second moments of area inertia about the y axis and z axis respectively (I_yy, I_zz), torsional constant (J), elastic modulus (E), and shear modulus (G) for each element. If shear deformations are considered, the user must specify the effective shear stiffnesses in both the y and z directions, (kGA_y) and (kGA_z), where k is an effective shear stiffness factor as commonly used in Timoshenko beam theory.

Mass Properties: Mass per unit length values are used for both dynamic analysis and self-weight calculations when enabled. See the unit selection system below for details on the mass units to be used.

2.2 GUI Interface Design

BladeBeam provides a Tkinter-based graphical user interface organized into four main configuration tabs, ensuring user-friendly operation while maintaining access to advanced analysis options.

The File Selection tab enables users to specify the primary Excel input file containing setup and element definitions. The interface includes browse functionality and file validation to ensure the selected file exists and is accessible. This tab serves as the entry point for all analysis configurations.

The Analysis Settings tab provides controls for fundamental analysis parameters:

Connection Settings: Users can enable semi-rigid moment connections, which activates the spring element formulation for boundary conditions specified with intermediate stiffness values rather than rigid or pinned assumptions.

Unit System Selection: The interface supports both metric (N, m, kg) and imperial (lb-force, inch, lb-mass) unit systems, automatically adjusting all calculations and output formatting accordingly. It should be noted that NuMAD excel input formats use metric units, thus the metric option should always be selected when using NuMAD options.

Self-Weight Analysis: An optional self-weight inclusion feature applies gravitational loading based on the specified mass distribution and user-defined load factors. This capability is particularly valuable for blade analysis where self-weight can significantly influence deflections and stresses.

The NuMAD Settings tab controls the integration with NuMAD blade definition files:

NuMAD File Selection: Users specify the NuMAD Excel file containing blade geometry, materials, and layup information. The program validates file accessibility and format compatibility.

Airfoils Folder: This setting points to the directory containing airfoil coordinate files referenced by the NuMAD definition. The program requires these files for proper cross-section generation.

Base Rotation: A critical parameter that defines the global orientation of the blade cross-sections relative to the analysis coordinate system. The default value of 90 degrees is typically appropriate for blade bending about the strong axis.

The Export Settings tab configures output file generation:

Export Enable/Disable: Users can choose whether to generate Excel output files containing detailed analysis results.

Output File Path: When export is enabled, users specify the location and filename for the results spreadsheet.

Export Critical Ply Stresses/Strains Toggle: Users can choose whether to generate Excel output files containing the critical ply stresses/strains for each segment in each segment of the blade. This setting is only valid if NuMAD functionality is used.

Output File Path: When export is enabled, users specify the location and filename for the critical ply stresses/strains spreadsheet.

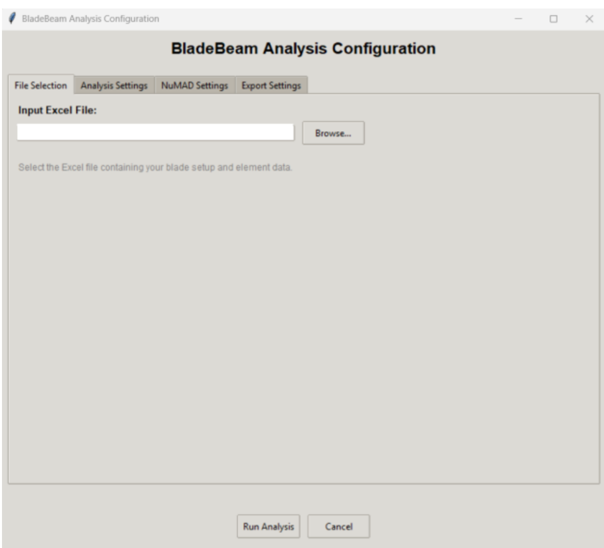
The GUI implements input validation to prevent analysis errors:

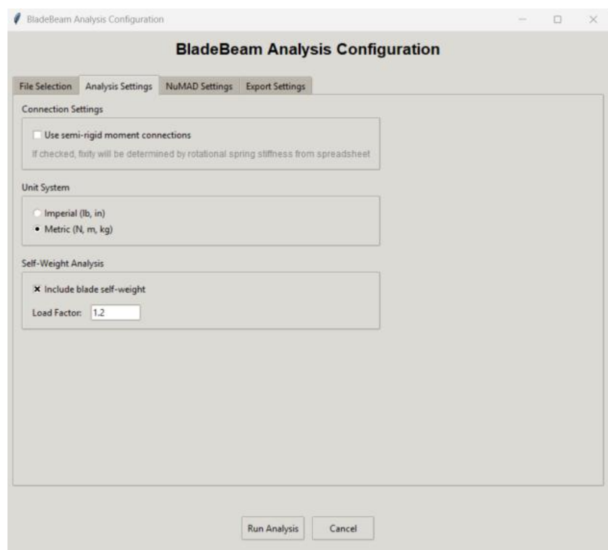
File Existence Checks: All specified files and directories are validated to ensure accessibility before analysis initiation.

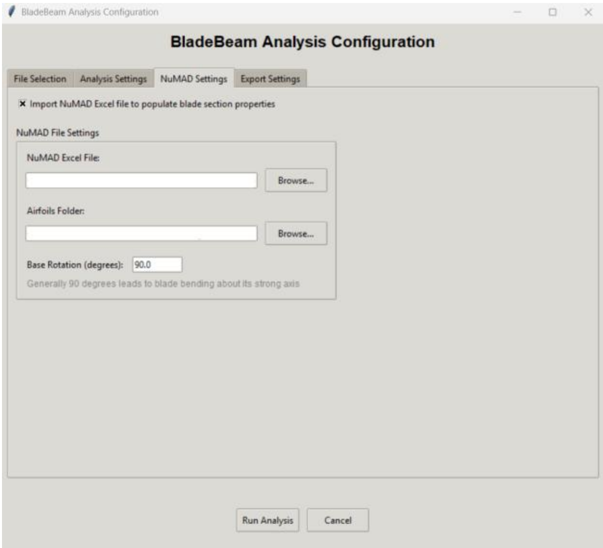
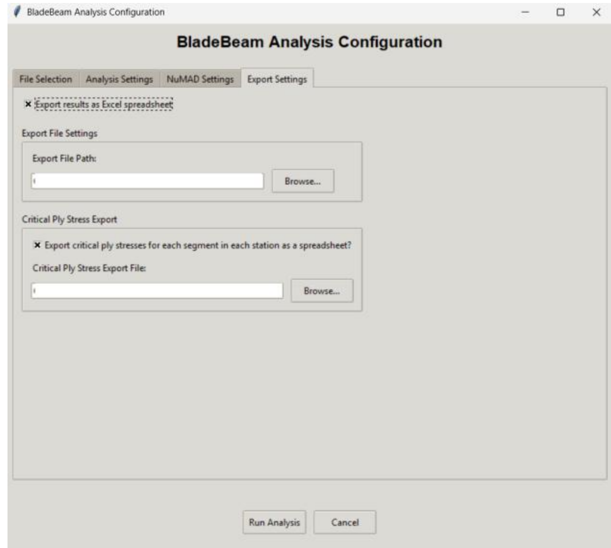
Numerical Validation: User-entered numerical values undergo range and format checking to prevent invalid inputs.

Configuration Consistency: The program verifies that all required settings are properly configured based on the selected analysis options.

The validation system displays error messages in the BladeBeam_log.txt log file when issues are detected, guiding users toward proper configuration. This file can be found in the BladeBeam installation directory. A confirmation dialog summarizes all analysis settings before execution, allowing users to review their choices and make corrections if necessary.






BladeBeam Analysis Configuration
BladeBeam Analysis Configuration
File Selection
Analysis Settings
NuMAD Settings
Export Settings
Import NuMAD Excel file to populate blade section properties
NuMAD File Settings
NuMAD Excel File:
Browse...
Airfoils Folder:
Browse...
Base Rotation (degrees):
90.0
Generally 90 degrees leads to blade bending about its strong axis
Run Analysis
Cancel

BladeBeam Analysis Configuration
BladeBeam Analysis Configuration
File Selection
Analysis Settings
NuMAD Settings
Export Settings
Export results as Excel spreadsheet
Export File Settings
Export File Path:
Browse...
Critical Ply Stress Export
Export critical ply stresses for each segment in each station as a spreadsheet?
Critical Ply Stress Export File:
Browse...
Run Analysis
Cancel

3 Output Analysis and Results Interpretation

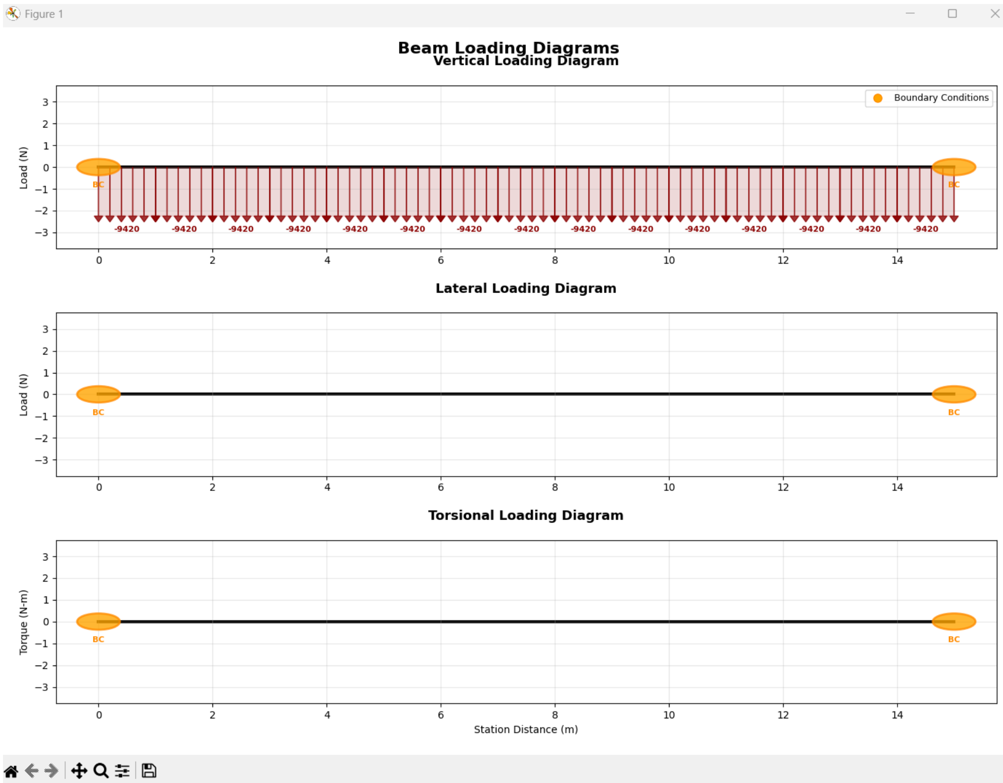
3.1 Structural Analysis Outputs

BladeBeam generates graphical outputs to facilitate understanding of structural behavior and verification of analysis results. This is done through the matplotlib library to create plots with consistent formatting and clear data presentation.

Plots generate in sequential order, once the user is done viewing/saving a plot and would like to move onto the next, they must click the X button on the upper right-hand corner of the window to close it and wait until the next plot is generated. Once all the plots have been clicked through, BladeBeam will export the results to Excel files (if export settings are toggled on) and close automatically.

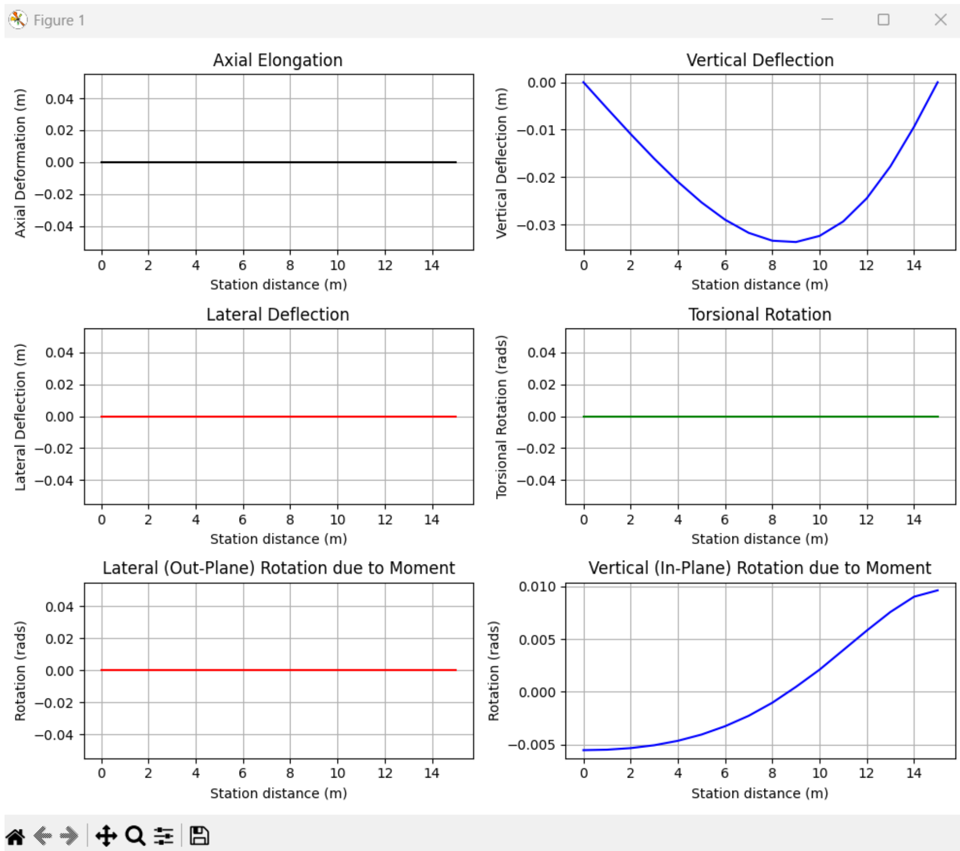
The program automatically generates three loading diagrams showing the applied forces and moments along the blade span. These diagrams distinguish between point loads and distributed loads using different visual representations:

- **Vertical Loading Diagram**: Displays applied vertical forces and distributed loads, with boundary condition markers shown as solid circles. The diagram scales load magnitudes appropriately and includes numerical values when enabled.
- **Lateral Loading Diagram**: Shows lateral forces and their distribution, using consistent color coding to differentiate from vertical loads. The visualization maintains proper element spans for distributed loads based on station positions.
- **Torsional Loading Diagram**: Represents applied torques and distributed torsional moments using circular symbols with directional indicators. This diagram helps verify proper load application and identify potential torsional instabilities.



Six separate plots present the structural response in all degrees of freedom:

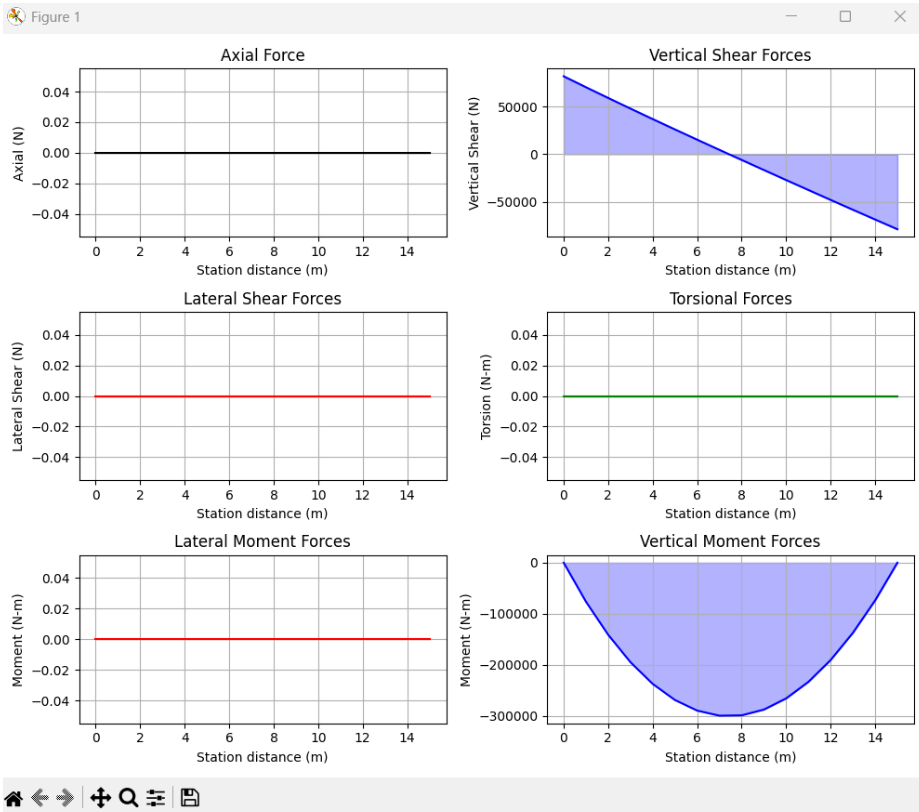
- **Axial Elongation**: Shows axial deformation along the blade span, which is typically minimal for blade structures.
- **Vertical and Lateral Deflections**: These plots display z-axis and y-axis deflections respectively.
- **Rotational Responses**: Three plots show torsional rotation and bending rotations about both axes.



Six force diagram plots illustrate the internal stress resultants throughout the blade structure:

- **Axial Force**: Shows the axial force distribution.
- **Shear Force Diagrams**: Separate plots for vertical and lateral shear forces help identify peak shear locations and verify proper load transfer to supports.
- **Moment Diagrams**: Vertical and lateral moment diagrams are critical for identifying maximum bending stress locations and designing appropriate section properties.

BladeBeam performs modal analysis to determine the dynamic characteristics of the blade structure. The results are organized by directional behavior to provide engineering insight into the dominant vibration modes and exported to the Excel workbook if the export option is toggled.



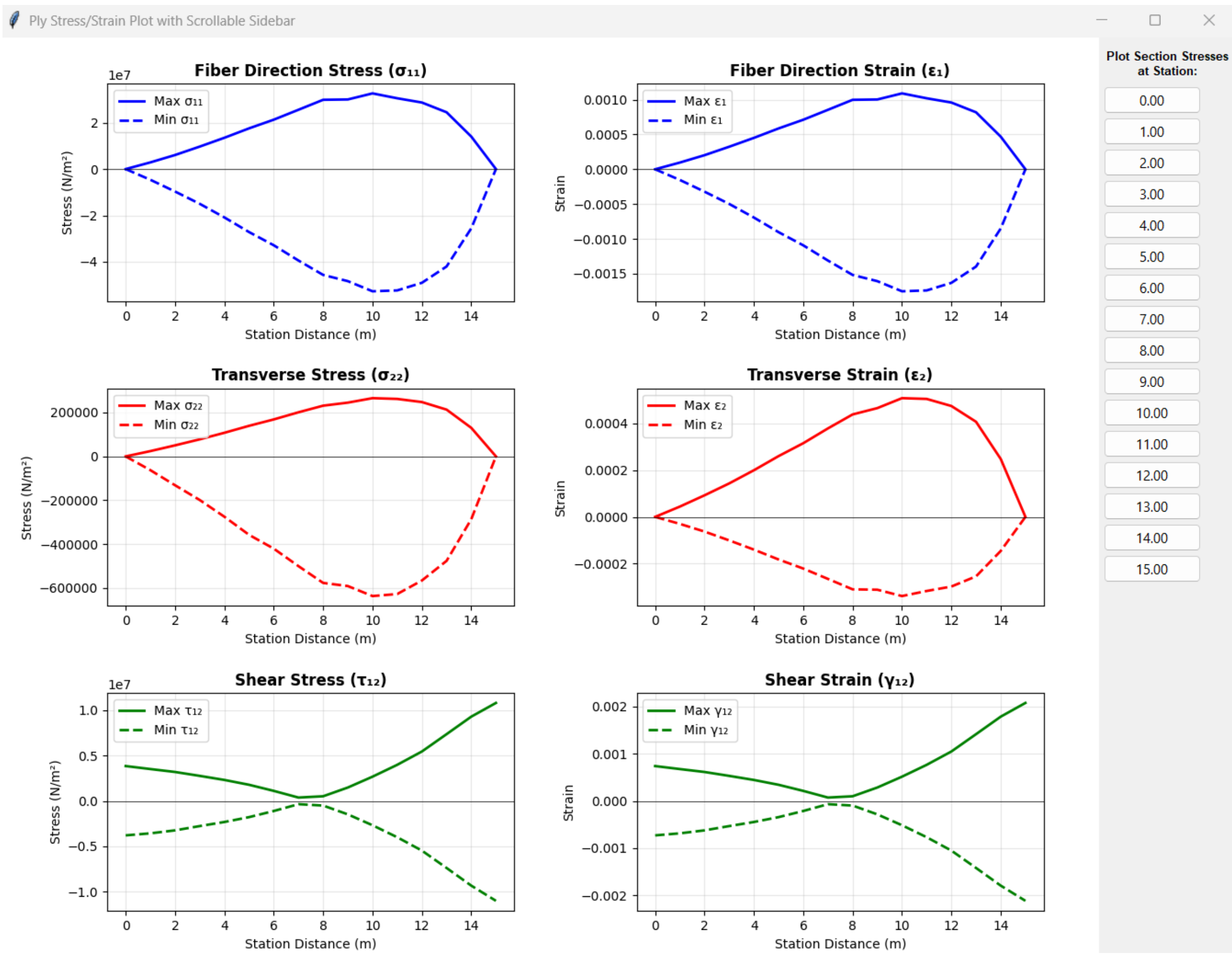
- **Vertical Bending Frequencies**: These represent the primary in-plane vibration modes. The first few vertical bending modes typically dominate the dynamic response.
- **Lateral Bending Frequencies**: These represent out-of-plane vibration modes.
- **Torsional Frequencies**: The program calculates torsional modes, though the documentation notes these results should be considered preliminary due to limitations in the beam element formulation for torsional dynamics.
- **Axial Frequencies**: Calculated for completeness but typically not critical for blade design due to the high axial stiffness relative to bending stiffness. These values should also be considered preliminary and have not been validated.

If a NuMAD input is used to define the wind turbine blade geometry and material properties, a sheet is generated which includes the various section flexural and coupled flexural stiffnesses: EI_{yy}, EI_{zz}, EI_{yz} as well as torsion stiffness, GJ, and axial stiffness, EA. Additionally, the linear mass density (mass/length) of the blade is exported under the column “lin_mass”.

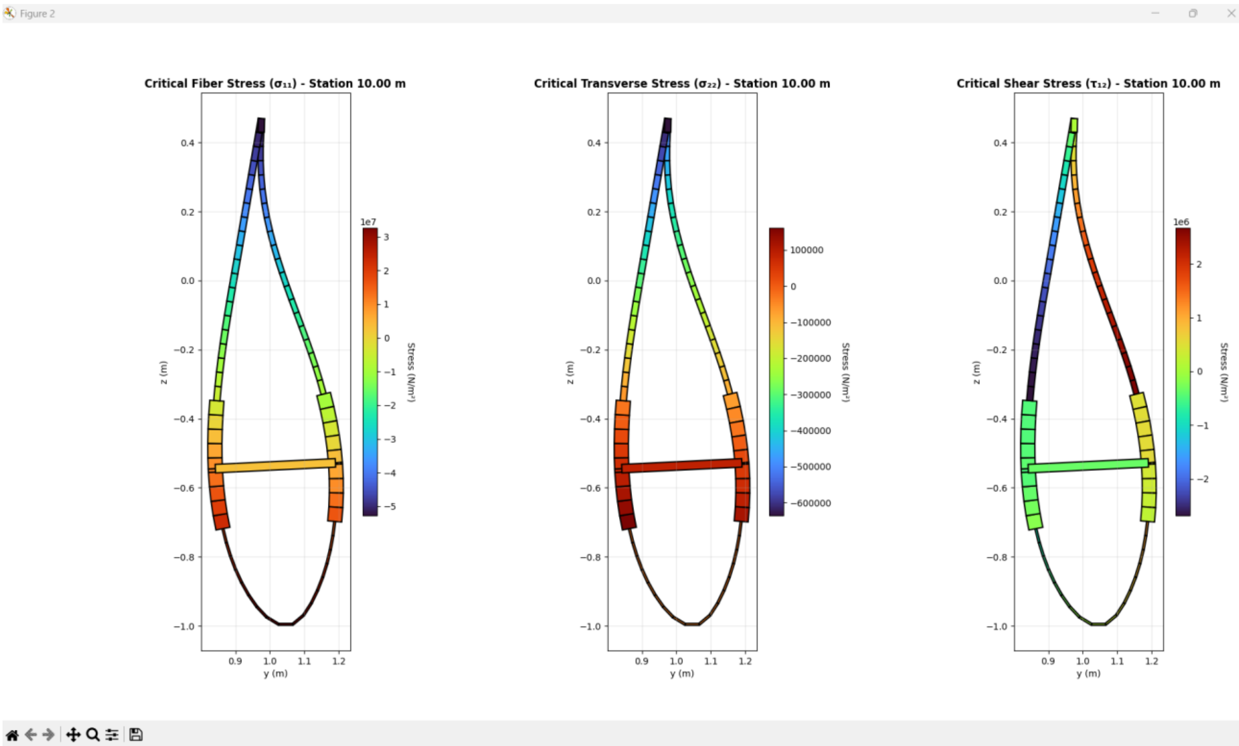
3.2 Stress/Strain Analysis Results

When NuMAD integration is active, BladeBeam provides detailed stress and strain analysis at the ply level for each segment within each blade section.

The program calculates maximum and minimum axial, transverse, and shear stresses for each station along the blade span. These calculations account for the specific laminate configuration and loading state at each location. A plot is generated showing these maximum and minimum stresses/strains along the blade span, and is useful for choosing which station to investigate further for a more detailed stress analysis (described below).



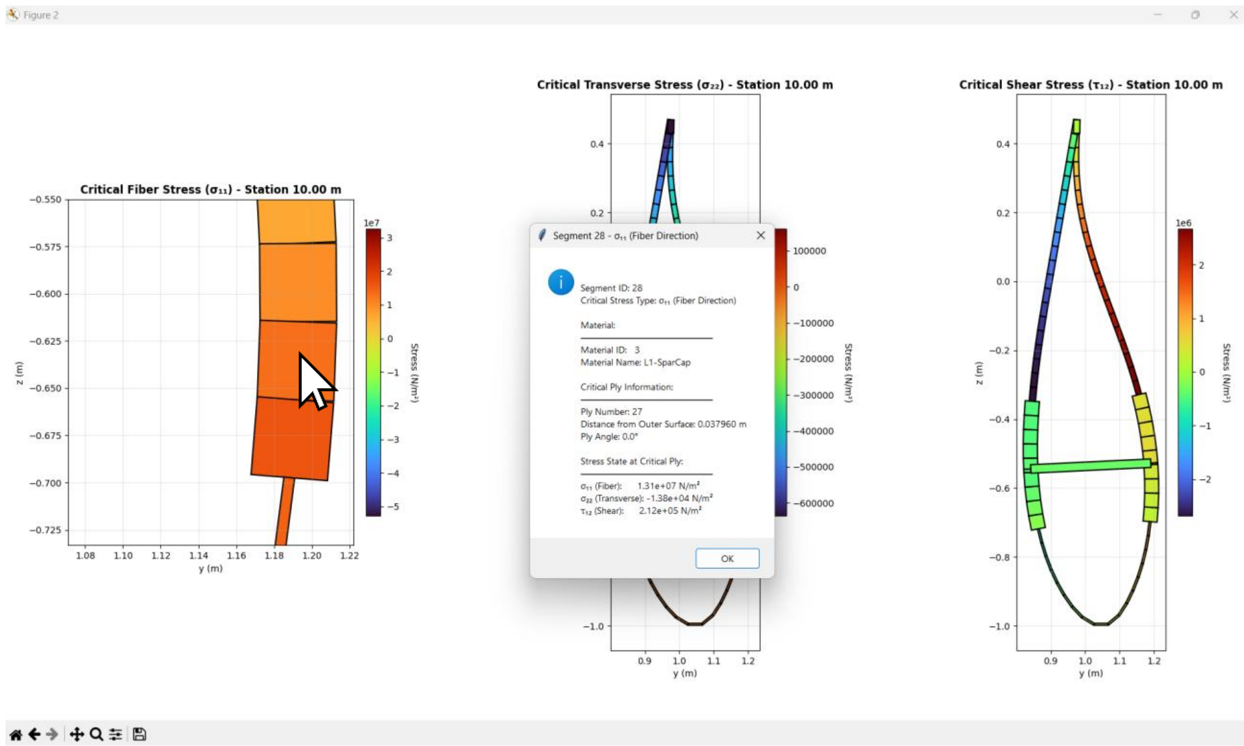
BladeBeam provides an interactive interface for detailed examination of stress distributions at specific blade stations. Users can click on any station from a scrollable right sidebar to generate detailed stress flow visualizations showing the critical ply in each segment within that cross section, with one plot per type of stress (in the ply coordinate system): longitudinal, transverse, and shear.



Users can use the zoom and movement tools in the bottom left corner to zoom in on the specific segment of interest.

	Reset to original plot view
	Move/pan view of plot
	Zoom in on a rectangular selection of the plot
	Configure plot settings
	Save plots as an image

After zooming and panning to the segment of interest, the user can click on the segment to generate a summary of the critical ply in that segment, what material it is made from, and the stresses present within it. It should be noted that the zoom and pan tools must be deselected for the cursor click to open the critical ply summary box. The critical ply values are also included within the critical ply Excel workbook if this option is toggled for the analysis.



3.3 Output File Formats

When export is enabled, BladeBeam generates an Excel workbook containing multiple worksheets with detailed results:

Deflections Sheet: Contains deflections and rotations at all nodes in tabular format, with headers indicating directions and units.

Internal Forces Sheet: Provides internal force results for all elements, organized by force and moment components.

Natural Frequencies Sheet: Lists all calculated natural frequencies organized by directional behavior, with mode numbers for reference.

Blade Section Stiffnesses Sheet: If a NuMAD input is used, a sheet is generated which contains the calculated blade section properties.

Critical Ply Stresses/Strains: When NuMAD integration is active and this option is toggled, an additional Excel workbook is generated with critical plies and their respective stresses/strains for every segment, with each tab in the workbook correlating to a different station along the blade.

3.4 Log File

BladeBeam generates a log file (BladeBeam_log.log) that records analysis steps, warnings, and diagnostic information. This log file can be found in the BladeBeam installation directory and can be opened by any type of notepad or text editor. The log file includes:

Analysis Configuration: Records all user settings and file paths for reproducibility.

Computational Details: Documents matrix dimensions, condition numbers, and solver performance metrics.

Warning Messages: Captures any numerical issues, convergence problems, or data quality concerns that arise during analysis.

Results Summary: Provides high-level summary statistics and key results for quick verification.

The log file serves as both a diagnostic tool for troubleshooting analysis issues and a record for quality assurance in engineering applications. All standard output and error messages are captured, ensuring traceability of the analysis process.

4 Sources

Alshannaq, A. A., Bank, L. C., Scott, D. W., & Gentry, T. R. (2021). Structural analysis of a wind turbine blade repurposed as an electrical transmission pole. *Journal of Composites for Construction*, *25*(4), 04021023.

Berg, J. C., & Resor, B. R. (2012). Numerical manufacturing and design tool (NuMAD v2.0) for wind turbine blades: User's guide (Report No. SAND2012-7028). Sandia National Laboratories. https://www.osti.gov/biblio/1051715

Ferreira, A. J. M., & Fantuzzi, N. (2020). *MATLAB codes for finite element analysis: Solids and structures* (2nd ed.). Springer.

Gentry, R., Ruane, K., & Bank, L. (2024). Numerical Modeling of a BladeBridge in Draperstown, Northern Ireland. In *Proceedings of the Civil Engineering Research in Ireland conference, 29-30 Aug 2024, Galway, Ireland*.

Haynes, R., Cline, J., Shonkwiler, B., & Armanios, E. (2016). On plane stress and plane strain in classical lamination theory. *Composites Science and Technology*, *127*, 20-27.

Kassimali, A. (2011). *Matrix analysis of structures (2nd ed.)*. Cengage Learning.

Logan, D. L. (2017). *A first course in the finite element method (6th ed.)*. Cengage Learning.

Pluzsik, A., & Kollar, L. P. (2002). Effects of shear deformation and restrained warping on the displacements of composite beams. *Journal of reinforced plastics and composites*, *21*(17), 1517-1541.

Nettles, A. T. (1994). *Basic mechanics of laminated composite plates*. Marshall Space Flight Center, National Aeronautics and Space Administration

Sandia National Laboratories. (2021). Numerical Manufacturing And Design Tool (NuMAD) [Computer software]. GitHub. https://github.com/sandialabs/NuMAD

Victorazzo, D. S., & De Jesus, A. (2016). A Kollár and Pluzsik anisotropic composite beam theory for arbitrary multicelled cross sections. *Journal of Composite Materials*, 50(23), 3237-3255. https://doi.org/10.1177/0731684416665493

Victorazzo, D. S. (2023). *abdbeam: A Python package for composite beam cross-sectional analysis* [Computer software]. GitHub. https://github.com/victorazzo/abdbeam