Modelling of Wind Turbine Blades with ABAQUS

Bitsche, Robert

Publication date: 2015

Document Version
Peer reviewed version

Citation (APA):
Modelling of Wind Turbine Blades with ABAQUS

Robert D. Bitsche
Composites Seminar, Senior Scientist
March 12, 2015
DTU Risø Campus
Modelling of Wind Turbine Blades: State of the Art

- Element types:
  - Layered shell elements

- Typical model size:
  - ca. 500,000 degrees of freedom

- Load application:
  - small number of concentrated forces and moments
  - applied through distribution coupling constraints

- Typical analysis procedures:
  - Static stress/strain analysis (geometrically linear and non-linear)
  - Natural frequency analysis
  - Eigenvalue buckling analysis
Modelling of Wind Turbine Blades: Shell Node Offset

- Wind turbine blade shell models usually have the finite element nodes offset to the exterior surface.
- However, the offset creates a systematic error in the torsional stiffness of the blade.
- Benchmark example: Thin-walled beam with rectangular cross-section made from an isotropic material:

<table>
<thead>
<tr>
<th>Abaqus Element Type</th>
<th>Mesh</th>
<th>Normalized angle of twist</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shell S4</td>
<td>28x40</td>
<td>0.995</td>
</tr>
<tr>
<td>Shell S4R</td>
<td>28x40</td>
<td>0.995</td>
</tr>
<tr>
<td>Shell S4, nodes offset</td>
<td>28x40</td>
<td>1.069</td>
</tr>
<tr>
<td>Shell S4R, nodes offset</td>
<td>28x40</td>
<td>1.069</td>
</tr>
</tbody>
</table>

Note: Benchmark study from 2009
Aeroelastic Analysis

• The structural design of wind turbine blades is based on loads that are derived using an dynamic, aeroelastic model of the turbine (e.g. HAWC2).

• For efficiency reasons the aeroelastic model usually relies on beam theory to describe the structural response of the blades.

• In ABAQUS input parameters for a beam model of the blade can be computed using “Meshed beam cross-sections”. Limitations are:
  – No off-axis orientations of anisotropic materials
  – No consideration of coupling effects like bend-twist coupling
Aeroelastic Analysis: BECAS

- BECAS is DTU Wind Energy’s cross section analysis software.
- It is similar to the “meshed beam cross-sections” in Abaqus, but allows for any material orientation and computes the full 6x6 cross section stiffness matrix of a beam cross section.
- More information: http://www.becas.dtu.dk

\[
\begin{bmatrix}
T_x \\
T_y \\
T_z \\
M_x \\
M_y \\
M_z
\end{bmatrix}
= 
\begin{bmatrix}
K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16} \\
K_{21} & K_{22} & K_{23} & K_{24} & K_{25} & K_{26} \\
K_{31} & K_{32} & K_{33} & K_{34} & K_{35} & K_{36} \\
K_{41} & K_{42} & K_{43} & K_{44} & K_{45} & K_{46} \\
K_{51} & K_{52} & K_{53} & K_{54} & K_{55} & K_{56} \\
K_{61} & K_{62} & K_{63} & K_{64} & K_{65} & K_{66}
\end{bmatrix}
\begin{bmatrix}
\tau_x \\
\tau_y \\
\tau_z \\
\kappa_x \\
\kappa_y \\
\kappa_z
\end{bmatrix}
\]
BECAS and ABAQUS

- BECAS is integrated with ABAQUS.
- “Shellexpander” automatically generates BECAS input files (2D cross section meshes) based on the information contained in an ABAQUS finite element shell model.
**BECAS and ABAQUS**

- Use a slice of an ABAQUS 3D solid finite element model and pass the FE stiffness matrix of the slice to BECAS.
- The command *SUBSTRUCTURE MATRIX OUTPUT* can be used for that purpose.
- Limitation: Elements may only have translational degrees of freedom (Continuum elements or continuum shell elements).
High Resolution Non-Linear Analysis

- 3D slice of a wind turbine blade: 20,000 nodes!
High Resolution Non-Linear Analysis

- Modelling trick:
  In pure bending a beam section exhibits no out-of-plane warping deformation, even if geometrical non-linearity is included.

- Kinematic coupling constraint applied only to the out-of-plane degree of freedom.

- If the axis of bending does not coincide with one of the principal axes of the section, further consideration are required to guarantee equilibrium.
High Resolution Non-Linear Analysis: Energy Release Rate at the Trailing Edge
Simulation of Wind Turbine Blade Full Scale Tests

- Wind turbine blade full scale tests (fatigue and ultimate) are a part of the blade certification process.
- **Movie of a full scale test** in the DTU Wind Energy blade test facility (the fun is at 1:16)
- During static tests concentrated loads are applied to the blade to approximate a certain bending moment distribution.
- If loads are applied through cables, the loads application direction changes significantly due to blade deformation.
- In ABAQUS connector elements of type “AXIAL” can be used to model this effect.
- The connector element can be “actuated”, i.e. a relative motion can be prescribed or a load can be applied to the element.
Simulation of Wind Turbine Blade Full Scale Tests

- Good match between full scale test results and numerical simulation.

Simulation of debonding at the trailing edge using cohesive elements or VCCT

Courtesy of Philipp Haselbach PhD Student phih@dtu.dk
A Short ABAQUS Wish List

- A better implementation of the “node offset” option for shell elements.
- An efficient tool for creating 2D cross section finite element meshes of composite beams.
  → Composite Modeller for ABAQUS
- A beam element that accepts a fully populated 6x6 cross section stiffness matrix as input.

\[
\begin{bmatrix}
T_x \\
T_y \\
T_z \\
M_x \\
M_y \\
M_z
\end{bmatrix} =
\begin{bmatrix}
K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16} \\
K_{21} & K_{22} & K_{23} & K_{24} & K_{25} & K_{26} \\
K_{31} & K_{32} & K_{33} & K_{34} & K_{35} & K_{36} \\
K_{41} & K_{42} & K_{43} & K_{44} & K_{45} & K_{46} \\
K_{51} & K_{52} & K_{53} & K_{54} & K_{55} & K_{56} \\
K_{61} & K_{62} & K_{63} & K_{64} & K_{65} & K_{66}
\end{bmatrix}
\begin{bmatrix}
\tau_x \\
\tau_y \\
\tau_z \\
\kappa_x \\
\kappa_y \\
\kappa_z
\end{bmatrix}
\]
Thank You.

Questions?