Created in COMSOL Multiphysics 6.1

Stress and Modal Analysis of a Wind Turbine Composite Blade

Introduction

Wind turbines are an increasingly popular source of renewable energy. As such, the design, analysis and manufacture of wind turbines are important to the energy industry.

The turbine blades are critical components of a wind turbine. When generating electric power through rotation, they have to withstand different types of loads, such as wind, gravitational, and centrifugal loads. The sheer size of a blade necessitates light and strong materials, and composites are well suited for this.

This example shows how to analyze a composite wind turbine blade using a mixture of carbon-epoxy, glass-vinylester and PVC foam. The blade is constructed as a sandwich structure where the PVC foam core is sandwiched between carbon-epoxy and glass-vinylester.

First, a stress analysis of the blade is performed in which it is subjected to a combination of gravitational and centrifugal loads. The tip displacement, maximum stress values, and through-thickness stress distribution at a particular point on the blade are computed for different load cases. Second, a prestressed eigenfrequency analysis is performed for a range of operating speeds. A Campbell diagram depicting the variation of eigenfrequencies with rotation speed is generated.

Model Definition

The geometry consists of a wind turbine blade of 61.5 m length as shown in Figure 1. This is a blade geometry used in the NREL 5MW wind turbine (Ref. 1 and Ref. 2). The front and top views of the geometry are shown in Figure 2.

The blade geometry has 19 different sections, where each section is defined by an airfoil shape. The details of the type of airfoil together with the maximum chord and the twist in each section is given in Table 1.

Essentially there are six different types of airfoils which are used to build the full wind turbine blade as shown in Figure 3. Note that the NACA 64-618 is the best airfoil from aerodynamic point of view and hence used near the tip region whereas the DU 99-W-405 is good from a structural point of view and hence used near the root of the blade. In between there are other airfoils of DU family which are used for smooth transition between the two extreme airfoil shapes. More details about the DU family airfoils can be found in Ref. 3.

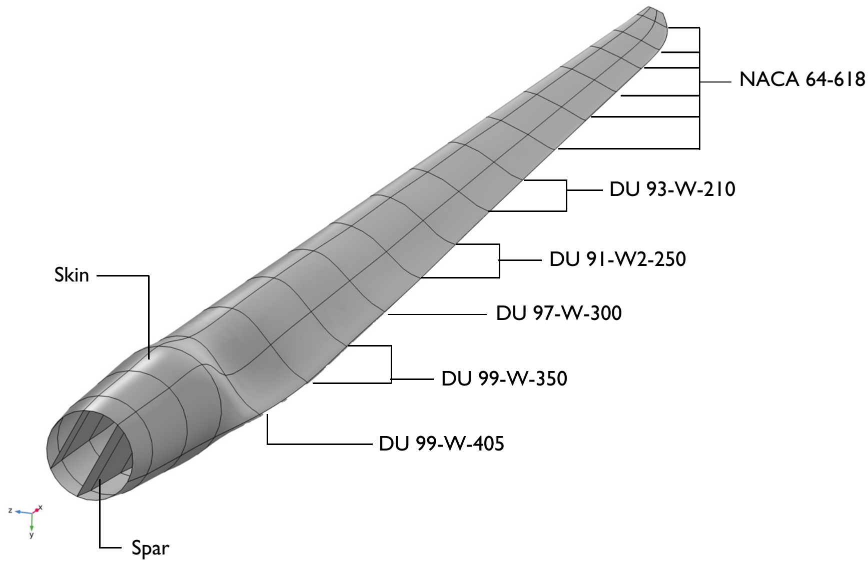


Figure 1: Geometry of a wind turbine blade for the NREL 5MW wind turbine. The different airfoils used at various sections are also shown.

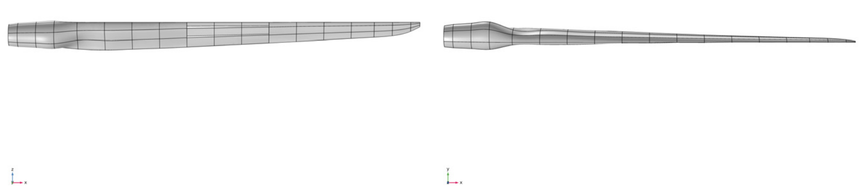


Figure 2: Front view (zx-plane) and top view (xy-plane) of the wind turbine blade.

TABLE 1: DIRECTIONAL INFORMATION OF WIND TURBINE BLADE GEOMETRY.

No	Blade span (m)	Chord (m)	Twist (deg)	Airfoil
1	0	3.2	13.08	Circular airfoil
2	1.36	3.54	13.08	Circular airfoil

TABLE 1: DIRECTIONAL INFORMATION OF WIND TURBINE BLADE GEOMETRY.

No	Blade span (m)	Chord (m)	Twist (deg)	Airfoil
3	4.1	3.85	13.08	Circular airfoil
4	6.83	4.167	13.08	Circular airfoil
5	10.25	4.55	13.08	DU 99-W-405
6	14.35	4.652	11.48	DU 99-W-350
7	18.45	4.458	10.16	DU 99-W-350
8	22.55	4.249	9.011	DU 97-W-300
9	26.65	4.007	7.795	DU 91-W2-250
10	30.75	3.748	6.544	DU 91-W2-250
11	34.85	3.502	5.361	DU 93-W-210
12	38.95	3.256	4.188	DU 93-W-210
13	43.05	3.01	3.125	NACA 64-618
14	47.15	2.764	2.319	NACA 64-618
15	51.25	2.518	1.526	NACA 64-618
16	54.67	2.313	0.863	NACA 64-618
17	57.4	2.086	0.37	NACA 64-618
18	60.13	1.4	0.16	NACA 64-618
19	61.5	0.7	0	NACA 64-618

There are two important parts of a wind turbine blade:

- Skin
- Spars

The skin consists of the outer curved boundaries. It carries essentially the entire loading. In order to increase bending and torsional stiffness, the blade is reinforced using spars, which are internal vertical members.

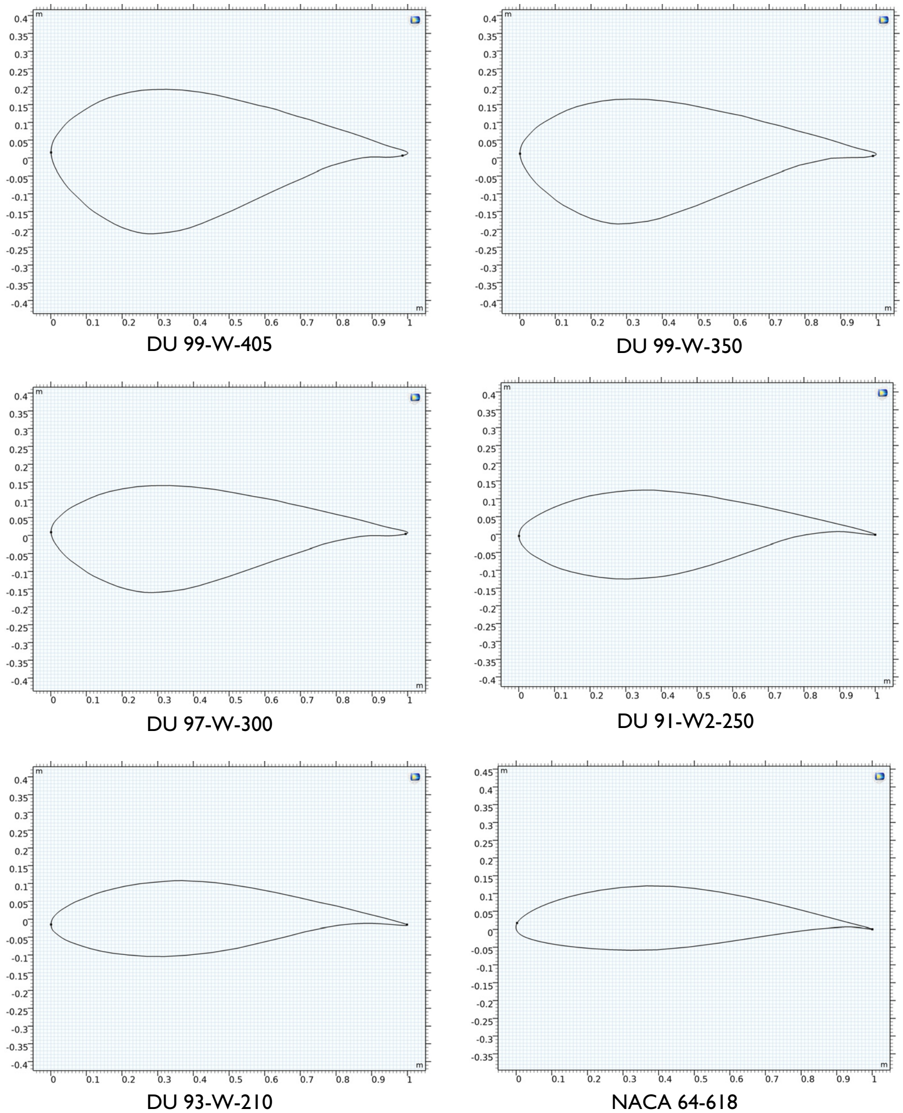


Figure 3: Different types of airfoils used at various sections of the wind turbine blade.

LOADS AND BOUNDARY CONDITIONS

The left end of the blade is connected to the rotor hub, and it is fixed, as shown in Figure 4. The loads acting on the structure are the self-weight of the blade and the centrifugal force. The aerodynamic or wind loads are not considered.

Two types of analyses are performed:

- A Stationary analysis: This analysis is performed for the gravity load case, centrifugal force load case, and a combination of the gravity and centrifugal force load cases, for a single blade-RPM value (15 rpm).
- A Prestressed Eigenfrequency analysis: This analysis is performed for centrifugal load case for a range of blade RPMs (0–30 rpm).

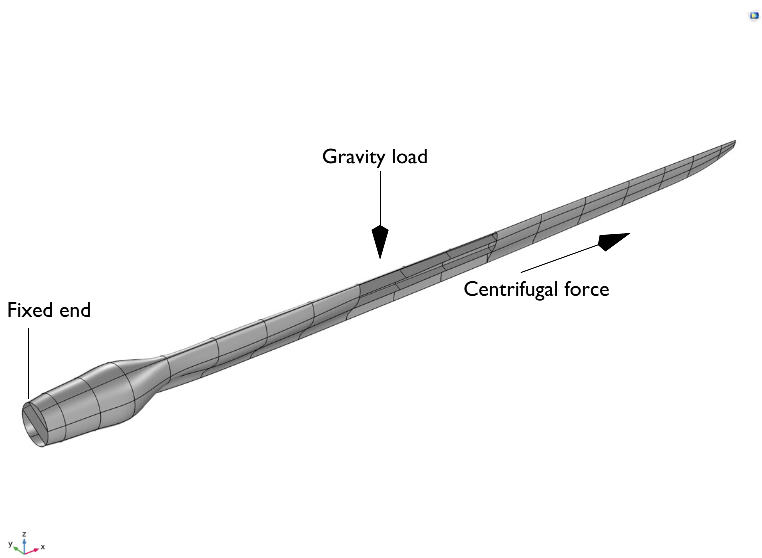


Figure 4: The geometry showing boundary conditions and loads acting on the structure.

LAMINA MATERIAL PROPERTIES

The analyzed wind turbine blade is a sandwich structure consisting of three different layered material types as shown in Figure 5.

The material properties of different laminae is taken from Ref. 4.

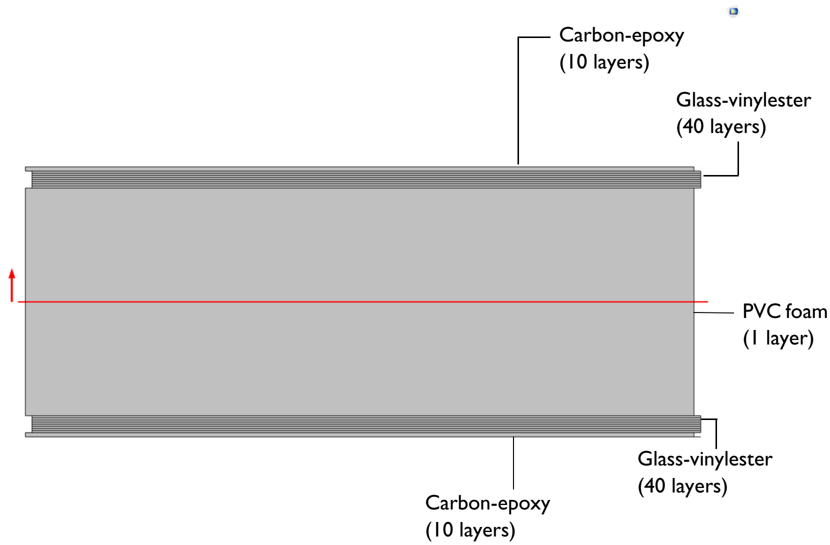


Figure 5: Sandwich arrangement of different layered materials used in the skin as well as in the spar of the blade.

Carbon-Epoxy Laminate

The outer part of the sandwich structure is a carbon-epoxy laminate having 10 layers, each of thickness 0.28 mm and oriented at 0 degrees to the first axis of the laminate coordinate system. The density of the lamina is taken as 1560 kg/m^3. The orthotropic material properties of the lamina are given in Table 2.

TABLE 2: MATERIAL PROPERTIES OF CARBON-EPOXY MATERIAL.

Material Property	**Value**
$\{E_1, E_2, E_3\}$	{139,9,9}(GPa)
$\{G_{12}, G_{23}, G_{13}\}$	{5.5,5.5,5.5}(GPa)
$\{\nu_{12}, \nu_{23}, \nu_{13}\}$	{0.32,0.32,0.32}

Glass-Vinylester Laminate

The next part of the sandwich structure is a glass-vinylester laminate. The density of this lamina is taken as 1890 kg/m^3. The orthotropic material properties are given in Table 3.

TABLE 3: MATERIAL PROPERTIES OF GLASS-VINYLESTER MATERIAL.

Material Property	Value
$\{E_1, E_2, E_3\}$	{41,9,9}(GPa)
$\{G_{12}, G_{23}, G_{13}\}$	{4.1,4.1,4.1}(GPa)
$\{\upsilon_{12}, \upsilon_{23}, \upsilon_{13}\}$	{0.3,0.3,0.3}

This laminate is made of 40 layers, each of 0.28 mm thickness with the stacking sequence shown in Table 4 and Figure 6.

TABLE 4: FIBER ORIENTATION IN GLASS-VINYLESTER LAYERED MATERIAL.

Layer Number	Fiber Orientation
1-5	0
6-10	45
11-15	-45
16-20	90
21-25	90
26-30	-45
31-35	45
36-40	0

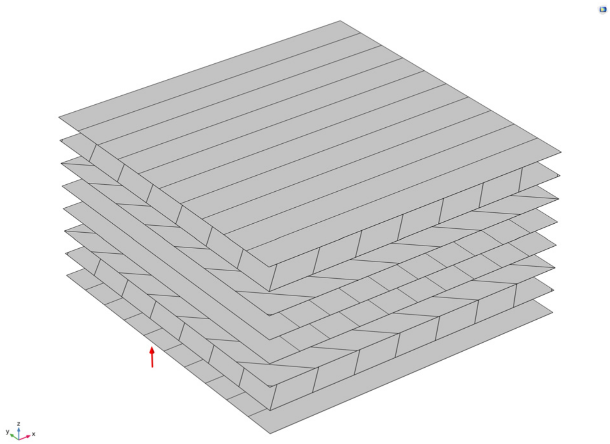


Figure 6: Stacking sequence of the glass-vinylester laminate, showing the fiber orientation in each layer, from bottom to top.

PVC Foam

The core material of the sandwich structure is made of PVC Foam of thickness 15 cm. The density of the material is taken as 200 kg/m^3. The values of Young's modulus, shear modulus and Poisson's ratio for the material are taken as 250 MPa, 92.6 MPa, and 0.35, respectively.

LAMINATE COORDINATE SYSTEM

The orientation of the laminate coordinate system in which the laminate material properties are specified is shown in Figure 7.

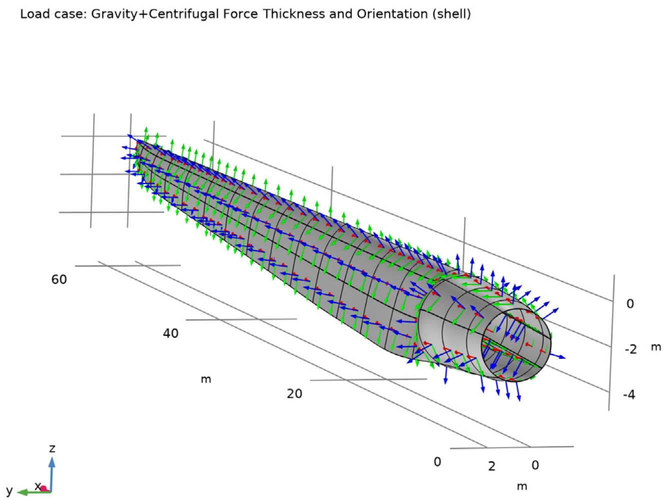


Figure 7: The laminate coordinate system used to define material properties in the wind turbine blade.

FINITE ELEMENT MESH

The structure is discretized using a mapped mesh, as shown in Figure 8.

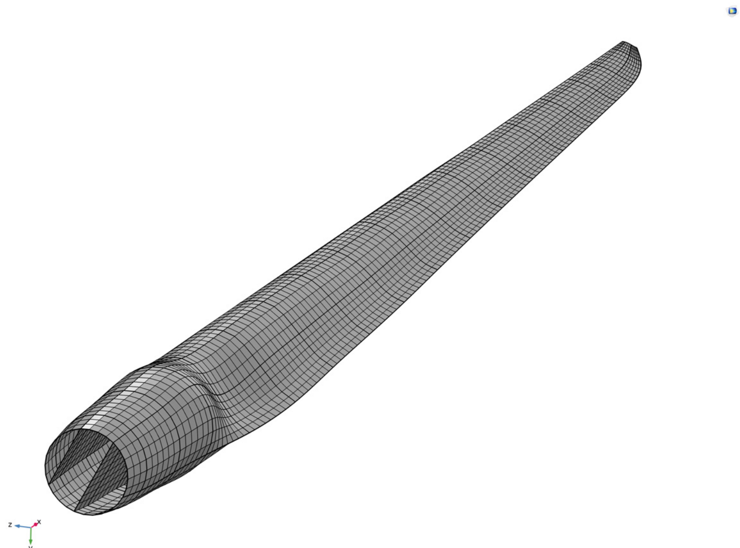


Figure 8: The mapped mesh for the wind turbine blade.

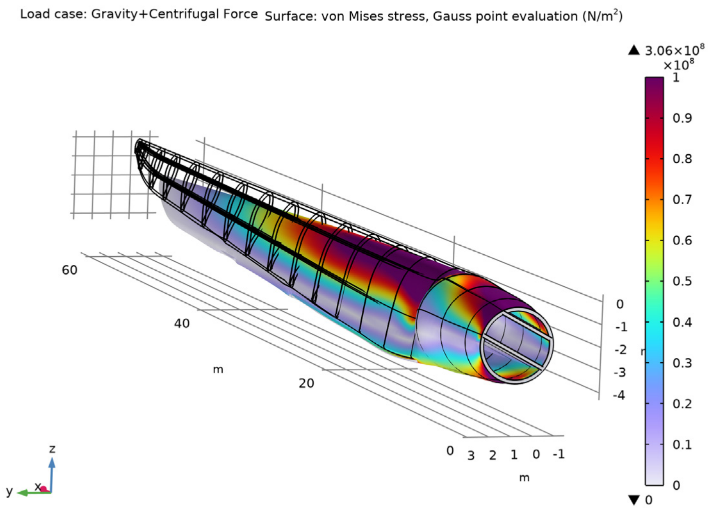


Figure 9: The von Mises stress distribution in the skin for combined gravitational and centrifugal loads.

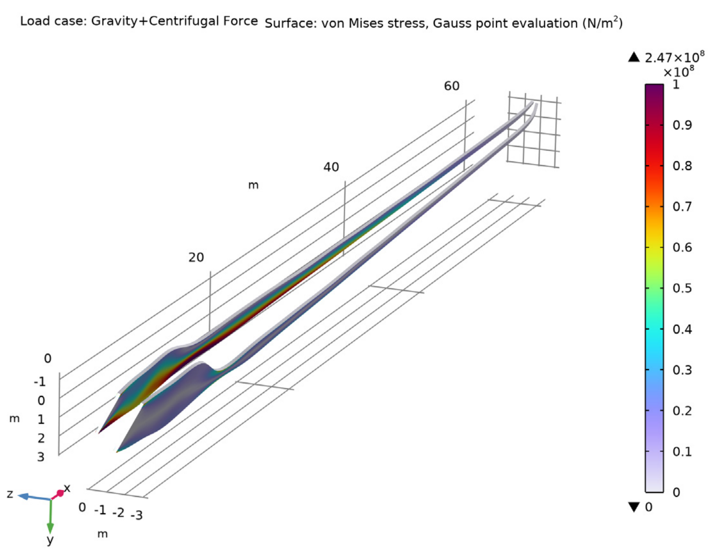


Figure 10: The von Mises stress distribution in the spars for combined gravitational and centrifugal loads.

Figure 9 shows the distribution of von Mises stress in the skin and spars for a combined load case of gravitational and centrifugal forces. High stresses are present near the root of the blade and in the junction between the circular and airfoil cross sections. The stress distribution for the spars is shown separately in Figure 10. The stress distribution in the outermost carbon-epoxy laminate is shown in Figure 11 for the three load cases.

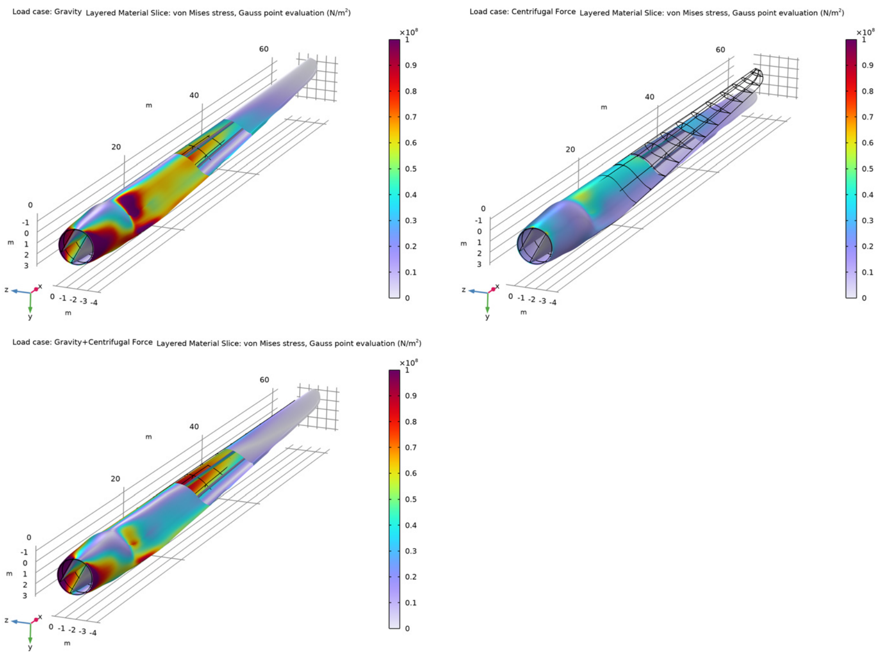


Figure 11: The von Mises stress distribution in the outermost carbon-epoxy layer for the three different load cases.

The through-thickness variation of von Mises stress at a particular point on the wind turbine blade is shown in Figure 12 for different load cases. Stress levels vary between laminates as well as between plies inside the different laminates. The highest stress levels can be seen in the carbon-epoxy that forms the outermost layer of the sandwich structure.

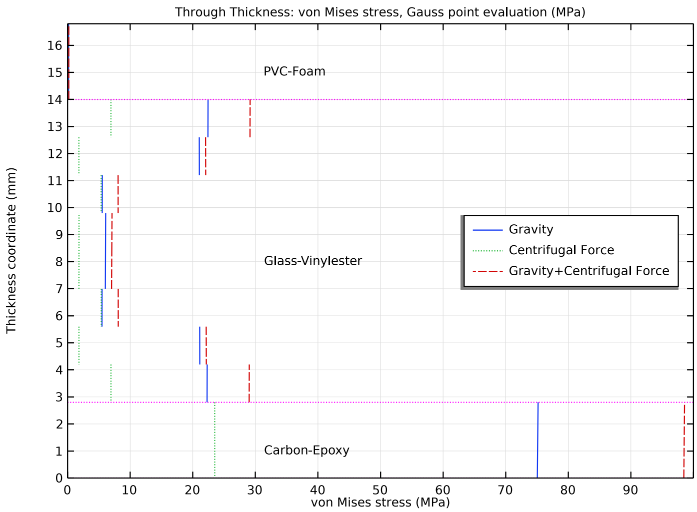


Figure 12: Through-thickness variation of von Mises stress for different load cases.

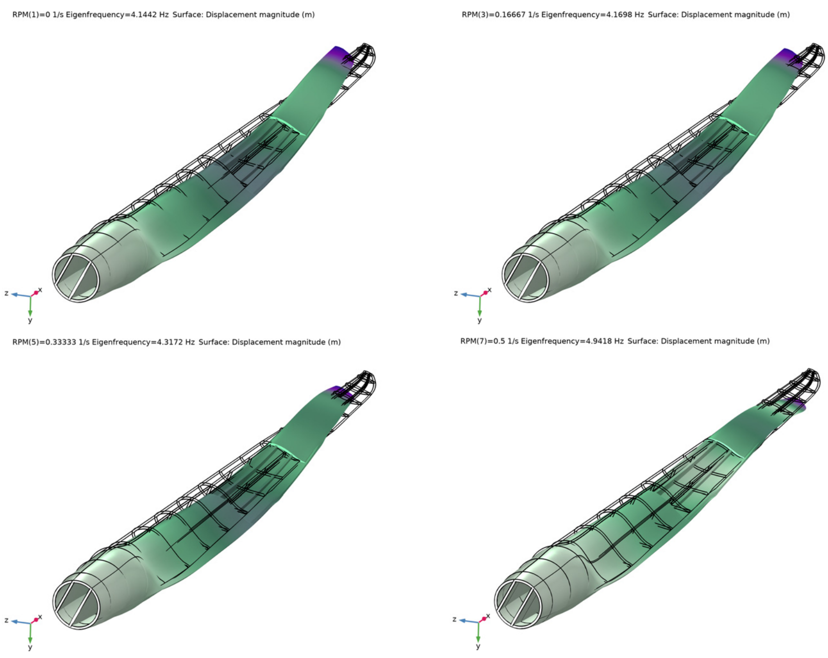


Figure 13: The fourth mode shape of the blade for different blade speeds.

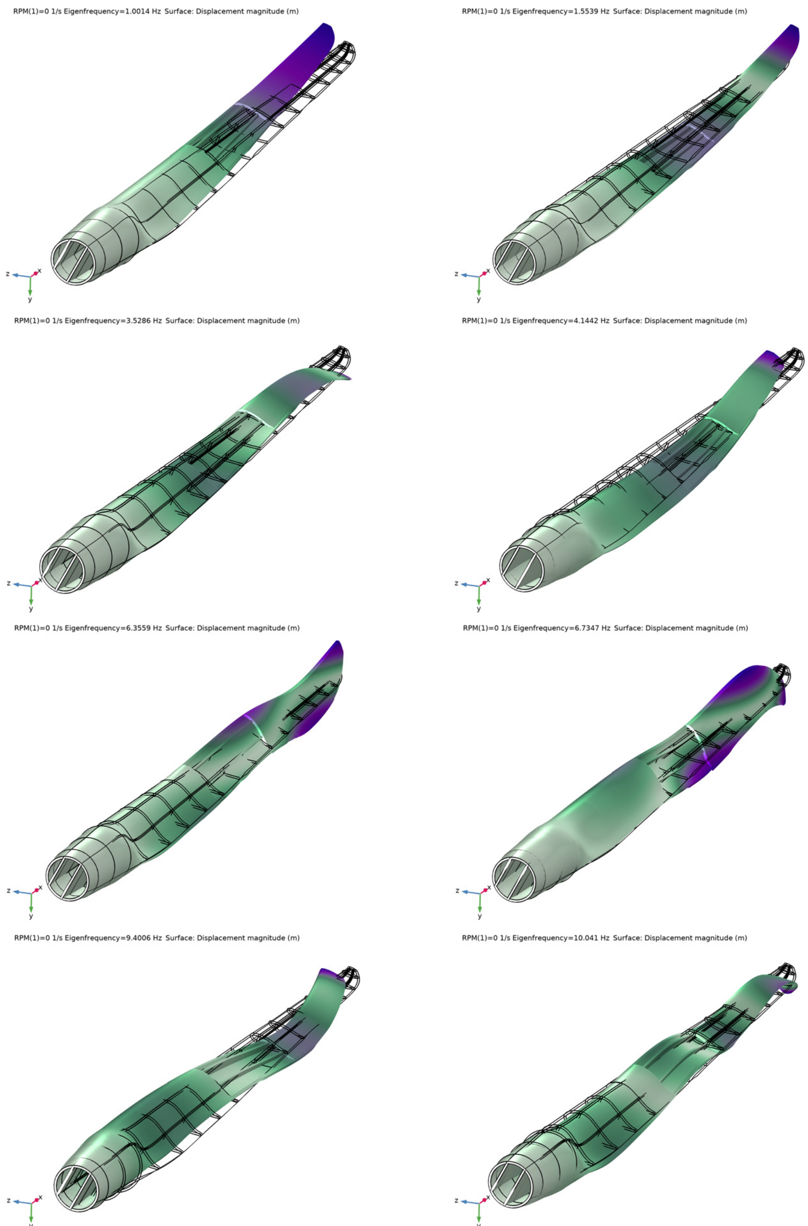


Figure 14: First eight mode shapes of the blade when it is not rotating.

In the second part of the example, a prestressed eigenfrequency analysis is carried out to compute eigenfrequencies and corresponding mode shapes of the blade under centrifugal forces. Different mode shapes of the blade as well as the effect of centrifugal force on the mode shape are shown in Figure 14 and Figure 13.

A Campbell plot is created in order to understand the variation of eigenfrequencies with respect to the blade rotation speed as shown in Figure 15. The eigenfrequencies increase with an increase in the blade RPM. This is due to the centrifugal stiffening effect.

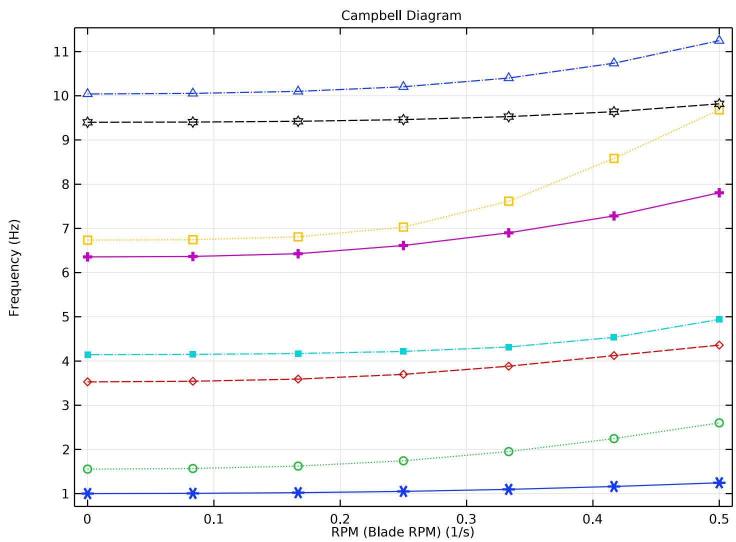


Figure 15: Campbell plot showing the variation of eigenfrequencies with an increase in the blade RPM.

Notes About the COMSOL Implementation

- Modeling a composite laminated shell requires a surface geometry (2D), in general called a base surface, and a **Layered Material** node which adds an extra dimension (1D) to the base surface geometry in the surface normal direction. You can use the **Layered Material** functionality to model several layers stacked on top of each other having different thicknesses, material properties, and fiber orientations. You can optionally specify the interface materials between the layers and control mesh elements in each layer.

- From a constitutive model point of view, you use the *equivalent single layer (ESL)* theory based formulation in **Layered Linear Elastic** material model of Shell interface.

References

1. J. Jonkman, S. Butterfield, W. Musial, and G. Scott, *Definition of a 5-MW Reference Wind Turbine for Offshore System Development, Technical Report,* NREL/TP-500-38060, 2009.

2. M.K. Yeh, and C. H. Wang, *Stress Analysis of Composite Wind Turbine Blade with Different Stacking Angle and Different Skin Thickness,* ICMSEA and MCEBM, 2017.

3. W.A. Timmer, and R. P. J. O. M. van Rooij, *Summary of the Delft University Wind Turbine Dedicated Airfoils,* J. Sol. Energy Eng 125(4), 488-496, 2003.

4. K. Kox and A. Echtermeyer, *Structural Design and Analysis of a 10MW Wind Turbine Blade,* Energy Procedia 24, 194-201, 2012.

Application Library path: `Composite_Materials_Module/Dynamics_and_Vibration/wind_turbine_composite_blade`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
th	0.28[mm]	2.8E-4 m	Layer thickness
thc	15[cm]	0.15 m	Core thickness
RPM	15[rpm]	0.25 1/s	Blade RPM
omega	2*pi[rad]*RPM	1.5708 rad/s	Blade angular speed

GEOMETRY 1

Import 1 (imp1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1** node.

2 Right-click **Geometry 1** and choose **Import**.

3 In the **Settings** window for **Import**, locate the **Import** section.

4 From the **Source** list, choose **COMSOL Multiphysics file**.

5 Click **Browse**.

6 Browse to the model's Application Libraries folder and double-click the file `wind_turbine_composite_blade.mphbin`.

7 Click **Import**.

You can adjust the view of the geometry for better visualization.

DEFINITIONS

Skin Boundaries

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Skin Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 50 only.

5 Select the **Group by continuous tangent** check box.

Spar Boundaries

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Spar Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 3 and 5 only.

5 Select the **Group by continuous tangent** check box.

Fixed Edges

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Fixed Edges` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Edge**.

4 Select Edge 10 only.

5 Select the **Group by continuous tangent** check box.

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Edge**.

4 From the **Selection** list, choose **Fixed Edges**.

5 Locate the **Advanced** section. From the **Frame** list, choose **Material (X, Y, Z)**.

Boundary System 1 (sys1)

1 In the **Model Builder** window, click **Boundary System 1 (sys1)**.

2 In the **Settings** window for **Boundary System**, locate the **Settings** section.

3 Find the **Coordinate names** subsection. From the **Axis** list, choose **x**.

Reverse Normal 1

1 Right-click **Boundary System 1 (sys1)** and choose **Reverse Normal**.

2 In the **Settings** window for **Reverse Normal**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Skin Boundaries**.

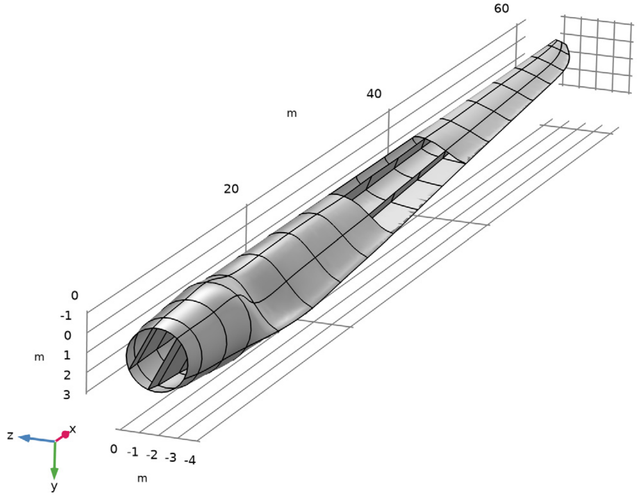
GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Graphics** window toolbar, click ▼ next to **Select Objects**, then choose **Select Boundaries**.

3 Click the Click and Hide button in the **Graphics** toolbar.

You can hide parts of the skin to visualize the spars inside the blade.



GLOBAL DEFINITIONS

Material: Carbon-Epoxy

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Material: Carbon-Epoxy` in the **Label** text field.

Layered Material: CE-[0]_10

1 Right-click **Materials** and choose **Layered Material**.

2 In the **Settings** window for **Layered Material**, type `Layered Material: CE-[0]_10` in the **Label** text field.

3 Locate the **Layer Definition** section. In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
`Layer 1`	Material: Carbon-Epoxy (mat1)	`0.0`	`th*10`	1

Material: Glass-Vinylester

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Material: Glass-Vinylester` in the **Label** text field.

Layered Material: GV-[0_5/45_5/-45_5/90_5]_s

1 Right-click **Materials** and choose **Layered Material**.

2 In the **Settings** window for **Layered Material**, type `Layered Material: GV-[0_5/45_5/-45_5/90_5]_s` in the **Label** text field.

3 Locate the **Layer Definition** section. In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Layer 1	Material: Glass-Vinylester (mat2)	0.0	th*5	1

4 Click **Add**.

Add six additional layers so that the material has a total of eight layers.

5 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Layer 2	Material: Glass-Vinylester (mat2)	45	th*5	1
Layer 3	Material: Glass-Vinylester (mat2)	-45	th*5	1
Layer 4	Material: Glass-Vinylester (mat2)	90	th*5	1
Layer 5	Material: Glass-Vinylester (mat2)	90	th*5	1
Layer 6	Material: Glass-Vinylester (mat2)	-45	th*5	1
Layer 7	Material: Glass-Vinylester (mat2)	45	th*5	1
Layer 8	Material: Glass-Vinylester (mat2)	0	th*5	1

6 Click to expand the **Preview Plot Settings** section. In the **Thickness-to-width ratio** text field, type `0.6`.

7 Locate the **Layer Definition** section. Click **Layer Stack Preview** in the upper-right corner of the section.

Material: PVC Foam

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Material: PVC Foam` in the **Label** text field.

Layered Material: PF-[0]

1 Right-click **Materials** and choose **Layered Material**.

2 In the **Settings** window for **Layered Material**, type `Layered Material: PF-[0]` in the **Label** text field.

3 Locate the **Layer Definition** section. In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
`Layer 1`	Material: PVC Foam (mat3)	`0.0`	`thc`	1

MATERIALS

Create a **Layered Material Stack** in which the previously created layered materials can be arranged to finalize the sandwich structure of the composite.

Layered Material Stack 1 (stlmat1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Layers>Layered Material Stack**.

Layered Material Link: Carbon-Epoxy

In the **Settings** window for **Layered Material Link**, type `Layered Material Link: Carbon-Epoxy` in the **Label** text field.

Layered Material Link: Glass-Vinylester

1 Right-click **Layered Material Link: Carbon-Epoxy** and choose **Duplicate**.

2 In the **Settings** window for **Layered Material Link**, type `Layered Material Link: Glass-Vinylester` in the **Label** text field.

3 Locate the **Link Settings** section. From the **Material** list, choose **Layered Material: GV-[0_5/45_5/-45_5/90_5]_s (lmat2)**.

Layered Material Link: PVC Foam

1 Right-click **Layered Material Link: Glass-Vinylester** and choose **Duplicate**.

2 In the **Settings** window for **Layered Material Link**, type `Layered Material Link: PVC Foam` in the **Label** text field.

3 Locate the **Link Settings** section. From the **Material** list, choose **Layered Material: PF-[0] (lmat3)**.

Layered Material Link: Glass-Vinylester 1 (stlmat1.stllmat4)

In the **Model Builder** window, under **Component 1 (comp1)>Materials> Layered Material Stack 1 (stlmat1)** right-click **Layered Material Link: Glass-Vinylester (stlmat1.stllmat2)** and choose **Duplicate**.

Layered Material Link: Carbon-Epoxy 1 (stlmat1.stllmat5)

In the **Model Builder** window, under **Component 1 (comp1)>Materials> Layered Material Stack 1 (stlmat1)** right-click **Layered Material Link: Carbon-Epoxy (stlmat1.stllmat1)** and choose **Duplicate**.

Layered Material Stack 1 (stlmat1)

1 In the **Settings** window for **Layered Material Stack**, click to expand the **Preview Plot Settings** section.

2 Clear the **Shows labels in cross-section plot** check box.

3 Locate the **Layered Material Settings** section. Click **Layer Cross-Section Preview** in the upper-right corner of the section.

SHELL (SHELL)

Layered Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Shell (shell)** and choose **Material Models>Layered Linear Elastic Material**.

2 In the **Settings** window for **Layered Linear Elastic Material**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Linear Elastic Material** section. From the **Material symmetry** list, choose **Orthotropic**.

5 Select the **Transversely isotropic** check box.

Add the material properties of carbon-epoxy, glass-vinylester and PVC foam.

GLOBAL DEFINITIONS

Material: Carbon-Epoxy (mat1)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Material: Carbon-Epoxy (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	{Evect1, Evect2}	{139e9, 9e9}	Pa	Transversely isotropic
Poisson's ratio	{nuvect1, nuvect2}	{0.32, 0.32}	1	Transversely isotropic
Shear modulus	Gvect1	5.5e9	N/m²	Transversely isotropic
Density	rho	1560	kg/m³	Basic

Material: Glass-Vinylester (mat2)

1 In the **Model Builder** window, click **Material: Glass-Vinylester (mat2)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	{Evect1, Evect2}	{41e9, 9e9}	Pa	Transversely isotropic
Poisson's ratio	{nuvect1, nuvect2}	{0.3, 0.3}	1	Transversely isotropic
Shear modulus	Gvect1	4.1e9	N/m²	Transversely isotropic
Density	rho	1890	kg/m³	Basic

Material: PVC Foam (mat3)

The PVC foam is isotropic. You just need to set the Young's modulus and Poisson's ratio. Orthotropic material properties will be synchronized from them.

1 In the **Model Builder** window, click **Material: PVC Foam (mat3)**.

2 In the **Settings** window for **Material**, locate the **Material Properties** section.

3 In the **Material properties** tree, select **Solid Mechanics>Linear Elastic Material>Young's Modulus and Poisson's Ratio**.

4 Click + **Add to Material**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	250e6	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.35	1	Young's modulus and Poisson's ratio
Density	rho	200	kg/m³	Basic

6 Click the **Show More Options** button in the **Model Builder** toolbar.

7 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

8 Click **OK**.

SHELL (SHELL)

Layered Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Shell (shell)** click **Layered Linear Elastic Material 1**.

2 In the **Settings** window for **Layered Linear Elastic Material**, click to expand the **Shear Correction Factor** section.

3 From the list, choose **User defined**.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Edges** and choose **Fixed Constraint**.

2 In the **Settings** window for **Fixed Constraint**, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Fixed Edges**.

Gravity 1

1 In the **Physics** toolbar, click **Global** and choose **Gravity**.

2 Click **Load Group** and choose **New Load Group**.

GLOBAL DEFINITIONS

Load Group: Gravity

1 In the **Model Builder** window, under **Global Definitions>Load and Constraint Groups** click **Load Group 1**.

2 In the **Settings** window for **Load Group**, type `Load Group: Gravity` in the **Label** text field.

3 In the **Parameter name** text field, type `lgG`.

SHELL (SHELL)

Rotating Frame 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Rotating Frame**.

2 In the **Settings** window for **Rotating Frame**, locate the **Rotating Frame** section.

3 From the **Axis of rotation** list, choose **User defined**. Specify the $\mathbf{r}_{\mathrm{bp}}$ vector as

aveop1(X)	x
aveop1(Y)	y
aveop1(Z)	z

4 Specify the $\mathbf{e}_{\mathrm{ax}}$ vector as

0	x
1	y
0	z

5 From the **Rotational direction** list, choose **Clockwise**.

6 In the Ω text field, type `omega`.

7 Locate the **Frame Acceleration Effect** section. Clear the **Spin softening** check box.

8 In the **Physics** toolbar, click **Load Group** and choose **New Load Group**.

GLOBAL DEFINITIONS

Load Group: Centrifugal Force

1 In the **Model Builder** window, under **Global Definitions>Load and Constraint Groups** click **Load Group 2**.

2 In the **Settings** window for **Load Group**, type `Load Group: Centrifugal Force` in the **Label** text field.

3 In the **Parameter name** text field, type `lgCF`.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Size 1

1 Right-click **Mapped 1** and choose **Size**.

2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

3 Locate the **Element Size** section. Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Maximum element size** check box. In the associated text field, type `0.5`.

6 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY: STATIC

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Study: Static` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Study: Static** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Define load cases** check box.

4 Click **Add**.

Add two additional load cases.

5 In the table, enter the following settings:

Load case	lgG	Weight	lgCF	Weight
`Load case: Gravity`	√	`1.0`		`1.0`
`Load case: Centrifugal Force`		`1.0`	√	`1.0`
`Load case: Gravity+Centrifugal Force`	√	`1.0`	√	`1.0`

6 In the **Home** toolbar, click **Compute**.

RESULTS

Layered Material 1

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets** and choose **More Datasets>Layered Material**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Spar Boundaries**.

Tip Displacement

1 In the **Results** toolbar, click 8.85 e-12 **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, type `Tip Displacement` in the **Label** text field.

3 Select Point 110 only.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`shell.disp`	`m`	`Displacement magnitude`

5 Click **Evaluate**.

Maximum Stress

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Maximum>Volume Maximum**.

2 In the **Settings** window for **Volume Maximum**, type `Maximum Stress` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material 1**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`shell.mises`	`MPa`	`von Mises stress`

5 Click **Evaluate**.

Use the following instructions to plot the von Mises stress distribution on the skin and spar boundaries of the blade as shown in Figure 9.

Stress: Skin+Spar

1 In the **Model Builder** window, under **Results** click **Stress (shell)**.

2 In the **Settings** window for **3D Plot Group**, type `Stress: Skin+Spar` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 1

1 In the **Model Builder** window, expand the **Stress: Skin+Spar** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, click to expand the **Range** section.
3 Select the **Manual color range** check box.
4 In the **Maximum** text field, type `1e8`.
5 In the **Stress: Skin+Spar** toolbar, click **Plot**.

Use the following instructions to plot the von Mises stress distribution on the spar boundaries as shown in Figure 10.

Stress: Spar

1 In the **Model Builder** window, right-click **Stress: Skin+Spar** and choose **Duplicate**.
2 In the **Settings** window for **3D Plot Group**, type `Stress: Spar` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material 1**.
4 Locate the **Plot Settings** section. From the **View** list, choose **Automatic**.
5 Clear the **Plot dataset edges** check box.

Surface 1

1 In the **Model Builder** window, expand the **Stress: Spar** node, then click **Surface 1**.
2 In the **Stress: Spar** toolbar, click **Plot**.

   Use the following instructions to plot the von Mises stress distribution on the layered material slice as shown in Figure 11.
3 In the **Home** toolbar, click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.
2 In the tree, select **Study: Static/Solution 1 (sol1)>Shell>Stress, Slice (shell)**.
3 Click **Add Plot** in the window toolbar.

RESULTS

Stress, Slice (Carbon-Epoxy)

1 In the **Settings** window for **3D Plot Group**, type `Stress, Slice (Carbon-Epoxy)` in the **Label** text field.
2 In the **Model Builder** window, expand the **Stress, Slice (Carbon-Epoxy)** node.

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Results>Stress, Slice (Carbon-Epoxy)> Layered Material Slice 1** node, then click **Layered Material Slice 1**.

2 In the **Settings** window for **Layered Material Slice**, click to expand the **Range** section.

3 Locate the **Through-Thickness Location** section. In the **Local z-coordinate [-1,1]** text field, type `1`.

4 Locate the **Range** section. Select the **Manual color range** check box.

5 In the **Maximum** text field, type `1e8`.

6 In the **Stress, Slice (Carbon-Epoxy)** toolbar, click **Plot**.

Use the following instructions to plot the von Mises stress distribution on different layered material slices.

Stress, Slice

1 Right-click **Layered Material Slice 1** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Stress, Slice` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

4 From the **View** list, choose **New view**.

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Stress, Slice** node, then click **Layered Material Slice 1**.

2 In the **Settings** window for **Layered Material Slice**, locate the **Through-Thickness Location** section.

3 From the **Location definition** list, choose **Physical**.

4 In the **Local z-coordinate** text field, type `5*th, 17*th, 30*th, 50*th+0.5*thc, 70*th+thc, 83*th+thc, 95*th+thc`.

5 Locate the **Layout** section. From the **Displacement** list, choose **Linear**.

6 In the **Relative z-separation** text field, type `0.3`.

7 Select the **Show descriptions** check box.

8 In the **Relative separation** text field, type `0.35`.

Table Annotation 1

1 In the **Model Builder** window, right-click **Stress, Slice** and choose **Table Annotation**.

2 In the **Settings** window for **Table Annotation**, locate the **Data** section.

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	z-coordinate	Annotation
80	0	0	Carbon-Epoxy
80	0	18	Glass-Vinylester
80	0	37.5	Glass-Vinylester
80	0	56	PVC-Foam
80	0	76	Glass-Vinylester
80	0	94	Glass-Vinylester
80	0	112	Carbon-Epoxy

5 Locate the **Coloring and Style** section. Clear the **Show point** check box.

6 From the **Orientation** list, choose **Vertical**.

Stress, Slice

1 In the **Model Builder** window, click **Stress, Slice**.

2 In the **Stress, Slice** toolbar, click **Plot**.

Use the following instructions to plot the through-thickness variation of von Mises stress for different load cases as shown in Figure 12.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study: Static/Solution 1 (sol1)>Shell>Stress, Through Thickness (shell)**.

3 Click **Add Plot** in the window toolbar.

RESULTS

Stress, Through Thickness (shell)

1 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

2 From the **Parameter selection (Load case)** list, choose **All**.

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** check box. In the associated text field, type `von Mises stress (MPa)`.

5 Select the **y-axis label** check box. In the associated text field, type `Thickness coordinate (mm)`.

6 Locate the **Axis** section. Select the **Manual axis limits** check box.

7 In the **x minimum** text field, type `0`.

8 In the **x maximum** text field, type `100`.

9 In the **y minimum** text field, type `0`.

10 In the **y maximum** text field, type `60*1000*th`.

11 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

Through Thickness 1

1 In the **Model Builder** window, expand the **Stress, Through Thickness (shell)** node, then click **Through Thickness 1**.

2 In the **Settings** window for **Through Thickness**, locate the **Selection** section.

3 Click **Clear Selection**.

4 Select Point 26 only.

5 Locate the **x-Axis Data** section. From the **Unit** list, choose **MPa**.

6 Locate the **y-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `shell.xd`.

8 From the **Unit** list, choose **mm**.

9 Find the **Interface positions** subsection. From the **Show interface positions** list, choose **Interfaces between layered materials**.

10 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

11 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

12 In the table, enter the following settings:

Legends
`Gravity`
`Centrifugal Force`
`Gravity+Centrifugal Force`

Stress, Through Thickness (shell)

In the **Model Builder** window, click **Stress, Through Thickness (shell)**.

Table Annotation 1

1 In the **Stress, Through Thickness (shell)** toolbar, click **More Plots** and choose **Table Annotation**.

2 In the **Settings** window for **Table Annotation**, locate the **Data** section.

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
30	5*1000*th	Carbon-Epoxy
30	30*1000*th	Glass-Vinylester
30	55*1000*th	PVC-Foam

5 Locate the **Coloring and Style** section. Clear the **Show point** check box.

6 In the **Stress, Through Thickness (shell)** toolbar, click **Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study: Static/Solution 1 (sol1)>Shell>Shell Geometry (shell)**.

3 Click **Add Plot** in the window toolbar.

4 In the tree, select **Study: Static/Solution 1 (sol1)>Shell>Thickness and Orientation (shell)**.

5 Click **Add Plot** in the window toolbar.

6 In the **Home** toolbar, click **Add Predefined Plot**.

RESULTS

Shell Geometry (shell)

1 In the **Model Builder** window, under **Results** click **Shell Geometry (shell)**.

2 In the **Shell Geometry (shell)** toolbar, click **Plot**.

Use the following instructions to plot the layered material coordinate system as shown in Figure 7.

Thickness and Orientation (shell)

1 In the **Model Builder** window, click **Thickness and Orientation (shell)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **View 3D 3**.

Thickness

1 In the **Model Builder** window, expand the **Thickness and Orientation (shell)** node, then click **Thickness**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Gray**.

5 In the **Thickness and Orientation (shell)** toolbar, click **Plot**.

Now you can add an **Eigenfrequency, Prestressed** study.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Eigenfrequency, Prestressed**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY: EIGENFREQUENCY

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study: Eigenfrequency` in the **Label** text field.

You can perform a parametric sweep for a range of blade RPMs, from 0 to 30 rpm.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
RPM (Blade RPM)	`range(0,5,30)[rpm]`	`1/s`

Step 1: Stationary

1 In the **Model Builder** window, click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.

3 Select the **Define load cases** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Load case	lgG	Weight	lgCF	Weight
`Load Case: Centrifugal Force`		`1.0`	√	`1.0`

Step 2: Eigenfrequency

1 In the **Model Builder** window, click **Step 2: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

3 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `8`.

4 From the **Eigenfrequency search method around shift** list, choose **Larger real part**.

5 In the **Study** toolbar, click **Compute**.

Use the following instructions to plot the mode shapes for different RPMs as shown in Figure 13.

RESULTS

Mode Shape (shell)

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

2 From the **Eigenfrequency (Hz)** list, choose **4.9418**.

3 From the **Parameter value (RPM (1/s))** list, choose **0**.

4 Click the **Go to XY View** button in the **Graphics** toolbar.

5 Click the **Show Grid** button in the **Graphics** toolbar.

6 In the **Mode Shape (shell)** toolbar, click **Plot**.

Use the following instructions to plot the mode shapes for different RPMs as shown in Figure 15.

Campbell Diagram

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Campbell Diagram` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Eigenfrequency/Parametric Solutions 1 (sol4)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

5 Locate the **Plot Settings** section.

6 Select the **y-axis label** check box. In the associated text field, type `Frequency (Hz)`.

7 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 Right-click **Campbell Diagram** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`freq`	`1/s`	`Frequency`

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

5 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Scene** section.

3 From the **Subject** list, choose **Thickness and Orientation (shell)**.

4 Locate the **Animation Editing** section. From the **Sequence type** list, choose **Dynamic data extension**.

5 Locate the **Frames** section. In the **Frame number** text field, type `25`.

6 Click the **Play** button in the **Graphics** toolbar.