Static and Buckling Analyses of Fiber Reinforced Composites in Siemens NX 10

Ralph Kussmaul

Zurich, 07-October-2016
Contents

1 Introduction .................................................................................................................. 4
   1.1 NX 10 License ........................................................................................................ 4
   1.2 Task definition ......................................................................................................... 4
2 Pre-Processing ............................................................................................................. 7
   2.1 Geometry ............................................................................................................... 7
   2.2 Meshing .................................................................................................................. 11
   2.3 Material definition ................................................................................................. 14
   2.4 Constraints ............................................................................................................ 20
3 Solving ........................................................................................................................ 27
4 Post-Processing ........................................................................................................... 29
5 Buckling Analysis ......................................................................................................... 32
1 Introduction

1.1 NX 10 License

In order to activate the Advanced Simulation module in NX 10 an adaption of the license environment variable is needed:

- Go to: Start/Systemsteuerung/System/Erweiterte Systemeinstellungen/UmgebungsvARIABLEN
- Choose system variable: UGS_LICENSE_BUNDLE
- Change value to: “ACD10;ACD11”
- Confirm changes

![Systemvariable bearbeiten](image)

Afterwards the Advanced simulation module of NX 10 can be used.

1.2 Task definition

In this NX introduction the simulation of a fiber reinforced composite beam subjected to 3-point-bending is presented.

The beam under study is inspired from a crash element used in a Smart Fortwo 501 compact car.
This cross beam element shall be modelled using layered shell elements and is therefore abstracted to the following surface geometry:

![Diagram](image)

The two supports are modeled as half cylinders. The impactor geometry is found below.

![Diagram](image)
As material a glass fiber reinforced composite is chosen. The material properties of a GFRP unidirectional layer are as follows:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_1$</td>
<td>35.5</td>
<td>GPa</td>
</tr>
<tr>
<td>$E_2$</td>
<td>8</td>
<td>GPa</td>
</tr>
<tr>
<td>$G_{12}$</td>
<td>4.1</td>
<td>GPa</td>
</tr>
<tr>
<td>$\nu_{12}$</td>
<td>0.25</td>
<td>-</td>
</tr>
<tr>
<td>$\rho$</td>
<td>1900</td>
<td>kg/m$^3$</td>
</tr>
<tr>
<td>XT1</td>
<td>0.030</td>
<td>-</td>
</tr>
<tr>
<td>XC1</td>
<td>0.021</td>
<td>-</td>
</tr>
</tbody>
</table>

The beam consists of a (45/-45/0/0/0/45/-45)$_s$ laminate. The single layer thickness is 0.38 mm, hence a total thickness of 5.32 mm is obtained.

A load of 26440 N is applied on the impactor in negative z-direction.
2 Pre-Processing

After starting NX 10 a new model is created by clicking on “File” – “New”. Select “Model”, assign a suitable name and click OK. Thus, a new CAD part file .prt is created.

2.1 Geometry

In order to define the profile of the beam, click on “Sketch” and select of the yz-plane. Draw the beam profile and after entering its dimensions according to Chapter 1.1 click “Finish Sketch”.
Next, use the “Extrude” function to create the surfaces. To do so, select the sketch, define extrusion direction to “Xc” and set “Start” and “End” distance to -500 mm respectively 500 mm. Set “Body Type” to “Sheet” and click OK.

Thus, the abstracted geometry model of the beam is defined. Then create a new part file and generate the half cylinder featuring as support.
Create a new part file and generate the impactor for the beam.

Now, click “New” – “Model” – “Assembly”, give a suitable name. In the “Add component” to load the beam, a first support element and the impactor in the assembly file.
Click “Move component”, select the support and then use the function “Distance” to move the support 450 mm along the x-axis. Afterwards use the function “Angle” to turn the support by 180° around the y-axis. Move the support in negative z-direction by 15 mm.

Click “Add” to insert a second support element, turn by 180° and move it -450 mm in x- and -15 mm in z-direction.

Finally, move the impactor 55 mm in the z-direction.
Now, the modelling of the geometry is completed.

2.2 Meshing

After defining the geometry, create a new FEM file by clicking “New” – “Simulation” – “NX Nastran FEM”. Give a suitable name and click “OK” to create the .fem file.

In the appearing window select your assembly file as Master Part and click “OK”.

Now the structure can be meshed. Click on “2D Mesh”. First, select the five faces of the beam. Select “CQUAD4” as “Element Type” and an “Element Size” of “5 mm”.
Next, click “Edit Mesh Associated Data”. Select “Material Orientation Method”: “Vector” and specify the “Xc”-axis. Thus, the material direction (0°-direction) is defined to be along the x-axis. Click “OK”.

Click on “Show Result” to get a preview of the mesh. If satisfied, click “OK”.

Repeat the steps above for the two half cylinders and the impactor. They can be meshed at once.

By meshing the assembly in two steps, there are two separate 2D collectors created in the Simulation Navigator, making it easier to define different materials to the surfaces in a later step.
Right click on “2D Collectors” – “Check all” - “Material Orientation”, select the whole model, check “Shell Orientation” and click “Display Element Material Orientation” to verify proper material orientation.
Right click on “2D Collectors” – “Check all” - “Element Normals”, select the whole model and click “Display Normals”. The element normal vectors are important for the application of laminates and the definition of contact surfaces in a later stage.

In order to check the continuity of the mesh, click “Checks and Information” - “Duplicate Nodes”. Select all the elements of the beam and click “List nodes”. The appearing information window states that no duplicate nodes are found and thus the mesh is continuous. Otherwise, the solving process of the model would fail or yield incorrect results.

Now, the meshing process is completed.

2.3 Material definition

First step of the material definition is to create a new material set for the GFRP UD (unidirectional) layer.

To do so, click “Manage Materials”, select “Orthotropic” and click “Create Material”.
Give a suitable name and enter mechanical and strength data according to the values given in Chapter 1. As we want to design the beam for fiber failure only, set the strain limits other than XT1 and XC1 to arbitrary, high values. Click “OK” – “Close”.

Now, a material must be assigned to the beam. Double-click on the 2D collector of the beam (here ThinShell(1)) . As Type select “Laminate”.

Click on “Create Physical” to open the Laminate Modeler. Check “Stress or Strain Output Request”. As Output Format choose “PCOMP”. The Stacking Sequence is “Inherited from layup”, the Ply Failure Theory is “Maximum Strain”, Interlaminar Failure Theory is “None”. Set an arbitrarily high value for “Shear Stress for Bonding”. Click “OK”.

After that, the laminate layup can be built up and assigned to the beam. This is done in the so-called ply-based modeling approach. Click “Laminates” – “Global Layup”. Click “Create New Ply” repeatedly to create the 14 plies of the symmetric (45/-45/0/0/0/45/-45)\_s layup. For each ply, select the proper material, the thickness and the ply angle.
Now select all plies and click “Define Draping Input”. In this step the laminate is placed as desired on the structure. As the beam consists of a uniform layup, select all five faces. Select a central node of the beam as starting point for the draping simulation. The 0°-direction is defined by “Primary Alignment”. Select the x-axis of the model. As “Draping Mesh Properties” select “Percentage of Mesh Size” = 1.00.

Click “OK” – “OK” to close the Global Layup card.
In the Simulation Navigator, open the folder “Layups”. Here, the Layup Offset can be defined. The offset is a distance which defines in which direction the laminate is applied on the shell element as can be seen in the following figure.

![Laminate Diagram]

In this case we want to stack the laminate on the outside of the surface of the beam, respectively in negative direction of the element normal vector (see Chapter 2.2). Thus the offset “Top” (-5.32 mm) is chosen. Double-click “Layup Offset – ‘Top”, select the 5 faces of the beam and click “OK”.

Next, the draping simulation is carried out. Right-click on “Layup 1” – “Update” executes the draping simulation.
After the draping simulation is finished, the fiber angle of every layer in every element in the beam is defined. Now, right-click on “Zones” – “Compute Zones”.

Now, the material for the supports and the impactor is defined. Double-click on “ThinShell(2)” – “Edit” – Material 1 “Choose material” and select “Steel” from the list. Enter a Default Thickness of “1 mm”. Click “OK” – “OK”.
Now, the materials are fully defined and assigned to the elements.

### 2.4 Constraints

In order to apply the constraints on the model, the creation of a .sim file is required. To do so click “New” – “NX Nastran Sim”, give a suitable name and click “OK”. In the opening window click “OK”.
In the opening “Solution” window give a suitable name to your simulation job and check “Ignore Material Temperature Dependences”. The solution type is “SOL 101 Linear Statics”. Click “OK.

Now, the constraints can be defined. Uncheck “2D Collectors” in the Simulator Navigator to have a better view only on the geometry. If possible, always apply constraints and loads to geometric entities and leave the distribution on the nodes to the FEM program.

Click “Constraint Type” – “Fixed Constraint”, select the polygon faces of the two support cylinders and click “OK”. Now all six degrees of freedom (DoF) of the supports are constrained.
Select “User Defined Constraint”, select the polygon face of the impactor and constrain the DoF [12456]. Click “OK”.

The so far applied constraints to the cylinders constrain the beam only in z-translation and xy-rotation. Thus, rigid body motion can still occur, making the global stiffness matrix singular.
That is why additional constraints are needed. Due to symmetry, nodes on the yz-plane (x = 0) should not move in x-direction. Therefore, they can be constrained. Use the selection filter “Node” to select all nodes of the beam which lie on the yz-plane and constrain their DoF [1].

Now, rigid body motions can only occur in y-direction. To suppress them, the one node on the top of the xy-parallel surface in the middle of the beam is constrained in DoF [2].
To apply the load on the impactor click “Load type” – “Force”. Select the polygon face of the impactor, set the force magnitude to “26440 N” and specify the vector to “-Zc”. Click “OK”. The total load of 26440 N will now be automatically distributed to the nodes of the impactor.

In the last step, the contacts between supports and beam, and between impactor and beam have to be defined. Click “Simulation Object Type” – “Surface-to-Surface-Contact”. Click “Create Face Pairs” and then select all faces that are in contact with other faces. Choose a Distance Tolerance of “1 mm”. Using the preview function, it can be checked if the proper contacts are found. Click “OK” – “OK”.
Now, in the Simulation Object Container, four Face Contacts are created. In NX the correct definition of the contact sides is of great importance for contact detection during the simulation run. Every shell element has its TOP side in the direction of its normal vector and its BOTTOM side in the negative direction of its normal vector.

Face Contact(1) is the contact between the beam’s upper side with the impactor. According to the shell element normal vectors, the the bottom side of the beam is in contact with the bottom side of the impactor. Double-click on Face Contact(1). Click on Source Region “Edit”, select Surface “Bottom” and click “OK. Click on Target Region “Edit”, select Surface “Bottom” and Type “Rigid”. Thus the impactor is treated as an ideally rigid part. Click “OK” – “OK”.
Face Contact(2), Face Contact(3), Face Contact(4) account for the contact between beam and the supports. According to the element normals, the TOP face of the beam is in contact with the BOTTOM face of the supports. Define the contact surfaces in Face Contact(2), Face Contact(3), Face Contact(4) accordingly as it was shown for Face Contact(1). Set the Type to “Rigid” for the supports.

Finally, all constraints and the loads are defined and model can be solved.
3 Solving

To properly prepare the FEM analysis run, right-click on the job “Solution 1” and choose “Edit”. In the “Solution” window navigate to the card “Case control”.

Select “Output Requests” – “Edit”. Here the requested results of the analysis can be selected. Enable “Contact Results” and “Strain”. “Displacement” and “Stress” are already enabled by default. Click “OK”.

Click “Global Contact Parameters” – “Edit”. Select “Initial Penetration” - “Set to zero”, so it is ensured that possible initial penetrations between the contact partners will not produce stresses in the structure. Click “OK” – “OK”.

To execute the solution click “Solve” – “OK”.

After solving the equation system for the unknown nodal displacements, a follow-up calculation is executed in order to determine the requested output results like strains, stresses or failure indices.
4 Post-Processing

To visualize the simulation results double-click on “Results” in the Simulation Navigator. Now, in the Post Processing Navigator the available results are listed.

Select “Displacement Nodal” – “Z”. Now a magnified illustration of the deformed beam is displayed. The magnification factor can be adjusted by double-clicking on “Post View” in the Post Processing Navigator. There select “Deformation” – “Results” and the set “Scale” to an appropriate value.

![Deformation settings](image)

As can be seen, a maximum z-displacement in the middle of the beam of -26.32 mm is calculated.
To assess the strain results click on “Ply Strain – Elemental”. Select the desired laminate ply and the strain direction. The directions “11”, “22” and “12” indicate the strains in fiber-direction, transverse direction and shear direction of the ply.

E.g. “Ply 3” is referring to the first 0°-layer of the beam. The fiber direction “11” of the 0°-layer is equivalent to the previously defined material direction, which is the global x-axis. The results indicate the compressive strains in fiber direction at the load introduction side and the tensile strains at the bottom of the beam. Notice that the magnitude of the tensile strains at the bottom is higher than the magnitude of compressive strains at the top. That is plausible for this open beam profile, whose bottom side has a bigger distance from the neutral axis than the top side.

To display the exploitation of the material according to the maximum strain criterion click on “Ply Failure Index – Elemental” and select the desired ply. The biggest ply failure index 0.638 is due to compressive strains in fiber direction and is located in Ply 3 (a 0°-layer) at the upper side of the beam. The maximum strain criterion is linear, thus the current load of 26440 N on the beam could be increased by a factor of $1/0.638 = 1.57$ until fiber failure has to be expected.
5 Buckling Analysis

Whenever compressive or shear loads occur in a structure, its stability has to be analyzed. In the following it is presented how a linear buckling analysis is carried out in NX 10.

Click on “Solution” and select “SOL 105 Linear Buckling” as solution type. Click “OK”.

A second solution is added in the Simulation Navigator and set to active. It consists of the subcase “Buckling Loads” and the subcase “Buckling Methods”. In the first, the loads on the structure are applied. There a type SOL 101 Linear Statics analysis is carried out. This linear static analysis is needed in order to calculate the so-called stress stiffness matrix. In the second, the buckling analysis is carried out. It contains solving the linear buckling analysis from the eigenvalue problem:

\[(K + \lambda K_\sigma)\phi = 0\]

Select all contacts, constraints and loads that were previously defined. Right-click and select “Add to active solution or step”.

---

IMES-ST/2016-10-07 Analyses of Fiber Reinforced Composites in Siemens NX 10 32 / 34
Click “Solve” to carry out the buckling analysis. Double-click on “Results” and open “Subcase - Buckling Method”. Per default, the first ten buckling modes of the structure are calculated.

As can be seen, Modes 1 – 8 have negative eigenvalues. These modes can only occur if the load on the structure would be reversed (in +Xc-direction). They can thus be ignored. Mode 9
is found to have a positive eigenvalue of 1.568. The corresponding buckling load is obtained by multiplying the eigenvalue with the applied loads. Hence, the buckling load for Mode 9 would be 41,771 N. Since the buckling load is higher than the applied load, buckling failure of the beam must not be expected.

To display the buckling mode shape, double-click on “Displacement – Nodal”.

As can be seen, the buckling is concentrated on the top part of the beam where the highest compressive forces occur. Thus, the result is plausible.