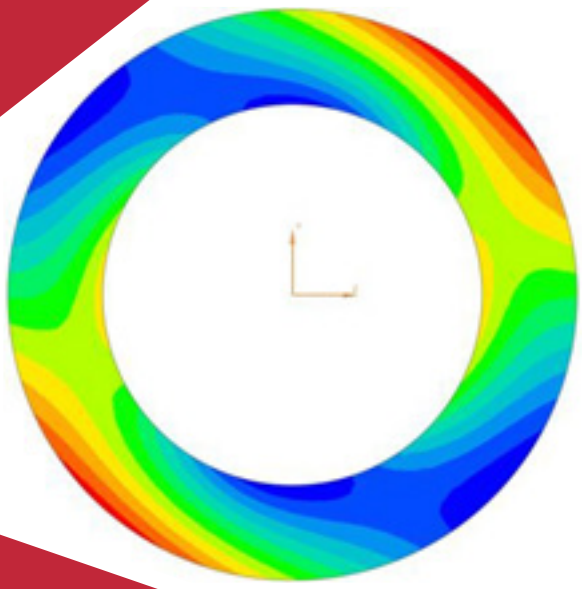
NX 9

WHITEPAPER

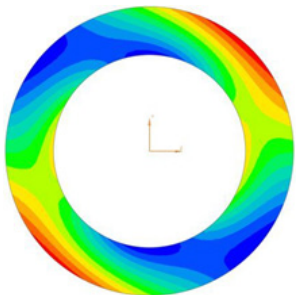
Beam Post-Processing with Cross-Section Views in NX



ATA ENGINEERING, INC.

www.ata-plmsoftware.com
844-756-7638 (844-PLM-SOFT)

Beam Post-Processing with Cross-Section Views in NX



Software:
NX 9 Advanced Simulation

Overview

Visualizing stresses in beam cross-sections in NX can be important during beam post-processing. Beam output defaults to four locations on the cross-section, which may not include the highest-stress location. This document explains how to create and orient beam cross-sections, how to request outputs for the beams, and how to view cross-sectional stresses of beams in NX 9.

Note: There are two types of elements used to represent beam-type structures in NX: CBEAMs and CBARs. The differences between these two elements are not discussed in this paper. However, post-processing CBEAM and CBAR elements in NX is similar, so these elements are collectively referred to as beam elements in this paper.

This whitepaper is part of a series of free Siemens PLM Software training resources provided by ATA. For more whitepapers, tutorials, videos, and macros, visit ATA's PLM Software website: http://www.ata-plmsoftware.com/resources.

Beam Post-Processing with Cross-Section Views in NX

Beam Creation and Orientation

The 1D Element Section command in the Mesh toolbar is used to define the cross-section of beams. In this tool, beam dimensions can be defined in multiple ways: select from a list of solver-specific standard sections, create a cross-section from an existing face, or create a cross-section from scratch via sketch geometry. The cross-sections are then applied to 1D meshes via the Physical Property Table Manager in the Properties toolbar (along with the 1D mesh's material and any nonstructural mass). A constant cross-section beam requires a cross-section to be defined in the Fore Section line of the Physical Property Table only, but tapered beams require both a fore and aft cross-section definition.

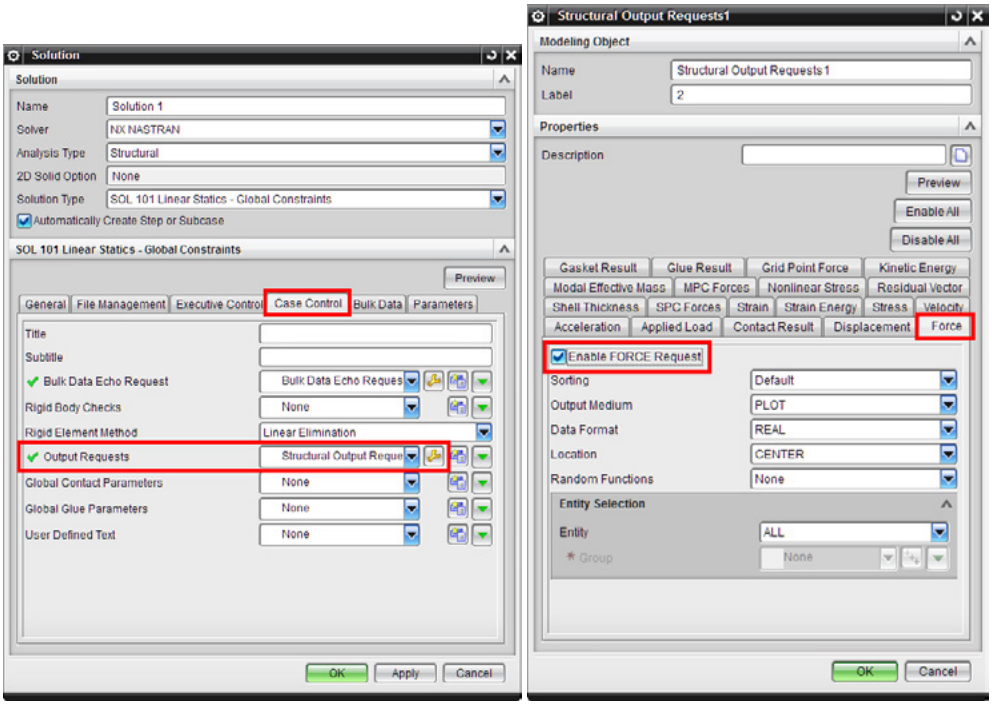
It is important to have consistent beam orientation when analyzing beam elements. Inconsistent beam orientations may not affect a model's stiffness if the beam cross-section is symmetric, but they will cause difficulty in post-processing, as beam forces and stresses will change sign. Consistent element orientation can be achieved by selecting the Auto Chain Selection option in the 1D Mesh command. This option automatically connects the B node from one element to the A node of the next element when meshing adjacent edges or curves. After the mesh is created, it may be necessary to reorient and offset the mesh from its default position. This is accomplished by right-clicking on the mesh object in the Simulation Navigator and selecting Edit Mesh Associated Data. The beam's local Y or Z axis or offset can be edited to achieve the desired orientation. Beam orientation can be displayed by right-clicking on the mesh collector and selecting "edit display" and then checking "section orientation."

Beam Output Requests

Beam stress output is requested by default in NX, but it includes stress results at the four stress-recovery points only. Force output, on the other hand, is not requested by default and therefore must be explicitly requested. When requesting beam force output, one gets beam force/moment resultants. NX can then use this information to calculate stress across the entire cross-section of the beam. To request element force output, create/edit a Structural Output Requests modeling object from the Case Control tab in the solution dialog, then click on the yellow wrench next to Output Requests to edit the current modeling object, and select the Force tab and check the Enable FORCE Request checkbox. Figure 1 shows the page sequence.

Beam Post-Processing with Cross-Section Views in NX

Figure 1: ▶
Page sequence to request bar force output.

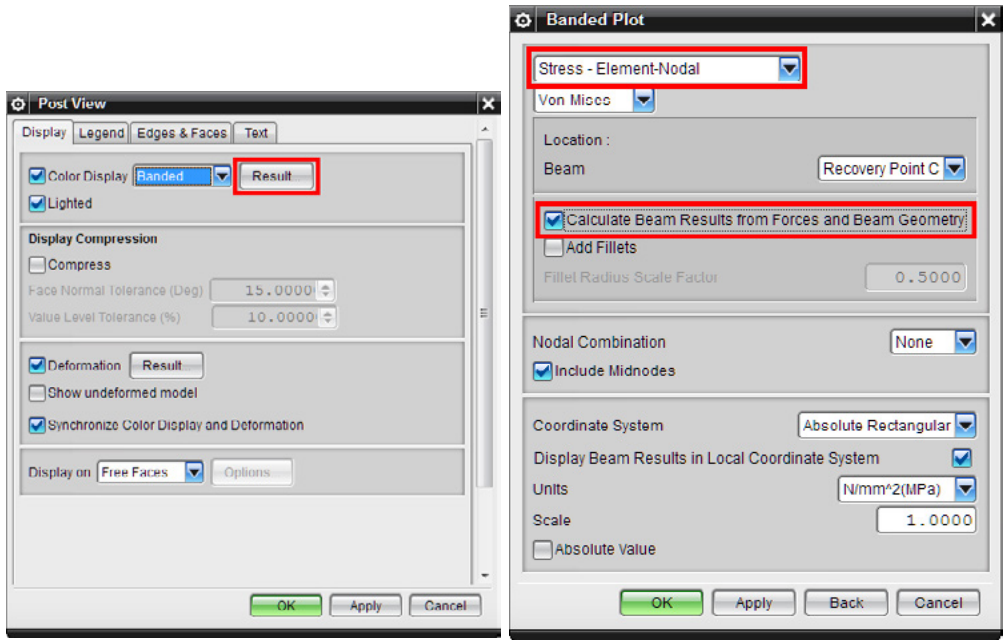


Visualizing Beam Cross-Section During Post-Processing

As previously discussed, element force output is used by NX to calculate stress across the entire cross-section. NX creates a shell-type mesh of a beam cross-section and applies the forces and moments from Nastran output to create a contour plot of stress across the entire beam cross-section. When plotting beam stress, the default behavior is to plot the maximum stress at the four stress-recovery points only. As a result, not all stress datasets are available for post-processing, and derived results such as von Mises stress may be inaccurate. To enable NX to calculate and display a complete cross-sectional stress result, click Edit Post View when viewing an Element-Nodal Stress output. From this window, click on the Result button and check the Calculate Beam Results from Forces and Beam Geometry checkbox in the plot dialog box. Figure 2 walks through this process.

Beam Post-Processing with Cross-Section Views in NX

Figure 2: ▶
Page sequence to correctly calculate beam stresses.



Since beam stresses vary both over the length of the beam element and across the beam cross-section, stresses can be displayed in two ways: (a) creating a deformed line display of stresses along a fixed cross-sectional location, or (b) creating a contour display of stresses across the beam cross-section at the fore or aft end of the beam. A cross-section display can be created by clicking the Beam Cross-Section View command after a standard post view has been created and then selecting an element and specifying whether the fore or aft end is desired. The cross-section dialog box also allows fillets to be added to beam cross-sections. Right-click on the cross-section view in the post-processing navigator, select Set Result, and check Add fillets; this multiplies the fillet radius scale factor by a minimum beam wall thickness to determine the fillet radius, modifies the section geometry, remeshes the cross-section, and recalculates the stress components. As a result, potentially artificial stress concentrations at interior corners of beam cross-sections are reduced (Figure 3).

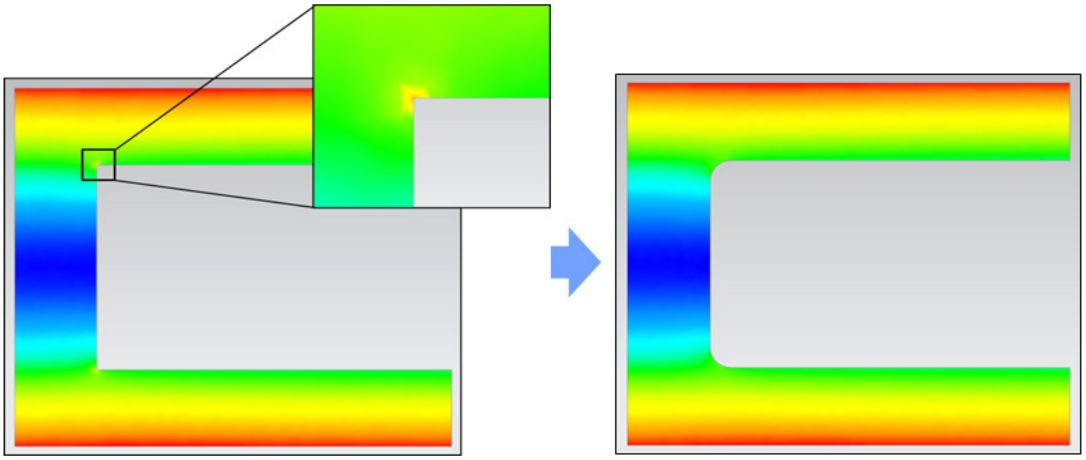


Figure 3: ▲
Adding fillets to a beam cross-section reduces corner stress concentrations.

www.ata-plmsoftware.com

ATA Engineering

ATA ENGINEERING, INC.

Solution Partner

PLM

SIEMENS

www.ata-e.com

ata-engineering

@ataengineering

sales@ata-e.com

858.480.2000

www.ata-plmsoftware.com

San Diego
Corporate Headquarters

Albuquerque

Denver

Huntsville

Los Angeles

San Francisco

Washington, D.C.