SIEMENS

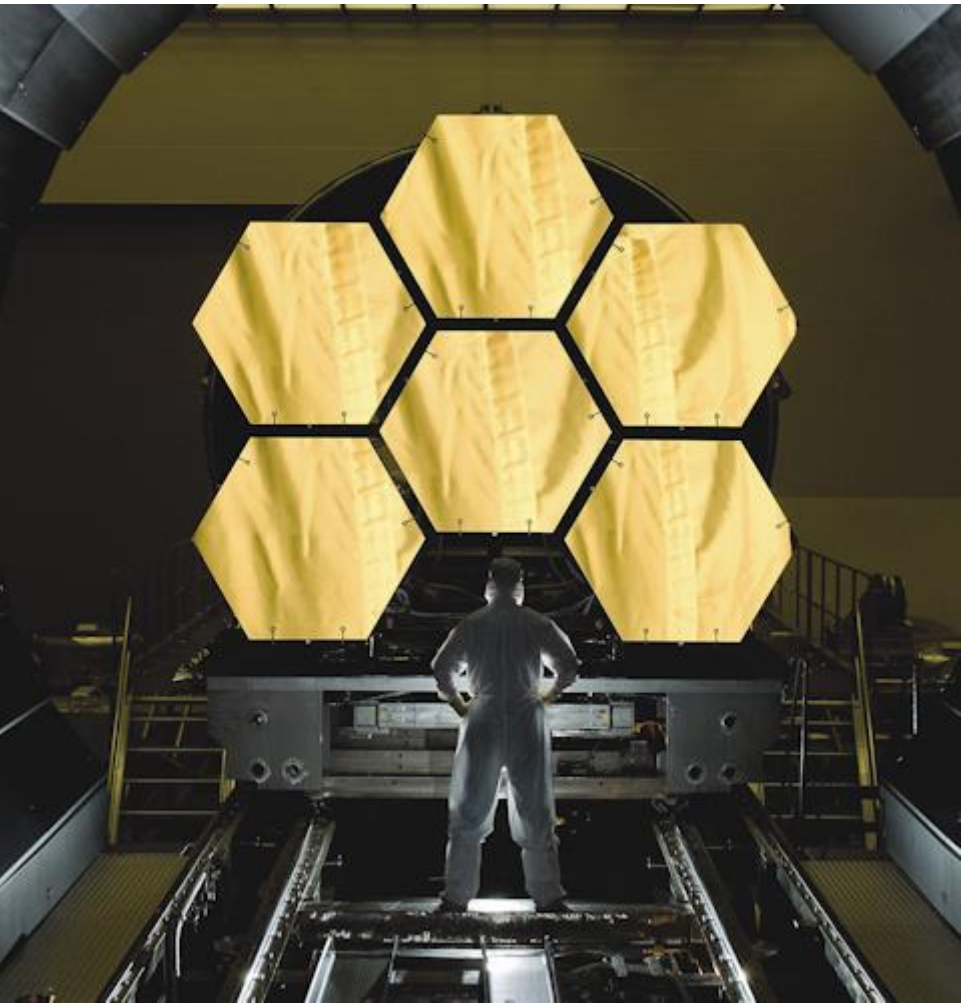
Patrick Kriengsiri

Using the FEMAP Beam Calculator

FEMAP SYMPOSIUM 2014
Discover New Insights

Femap Symposium 2014
May 14-16, Atlanta, GA, USA

Agenda

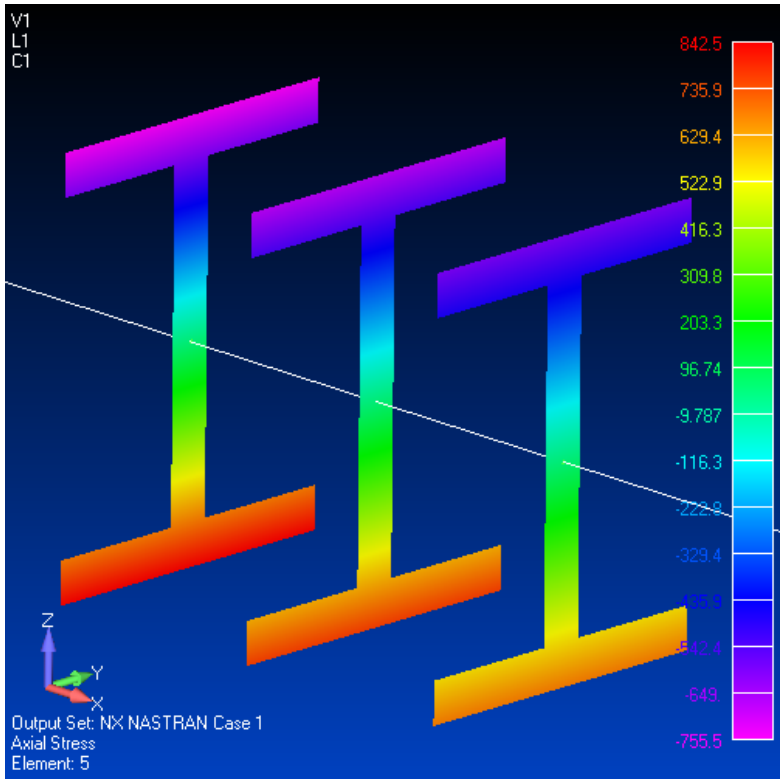


- What is the FEMAP Beam Calculator
- Using the FEMAP Beam Calculator via the GUI
  - Options
  - Visualization
  - Compared to NASTRAN Results
- Using the FEMAP Beam Calculator API
  - Creating Beam Processor Objects
  - Extracting Results Data from the Beam Processor Object

What is the FEMAP Beam Calculator

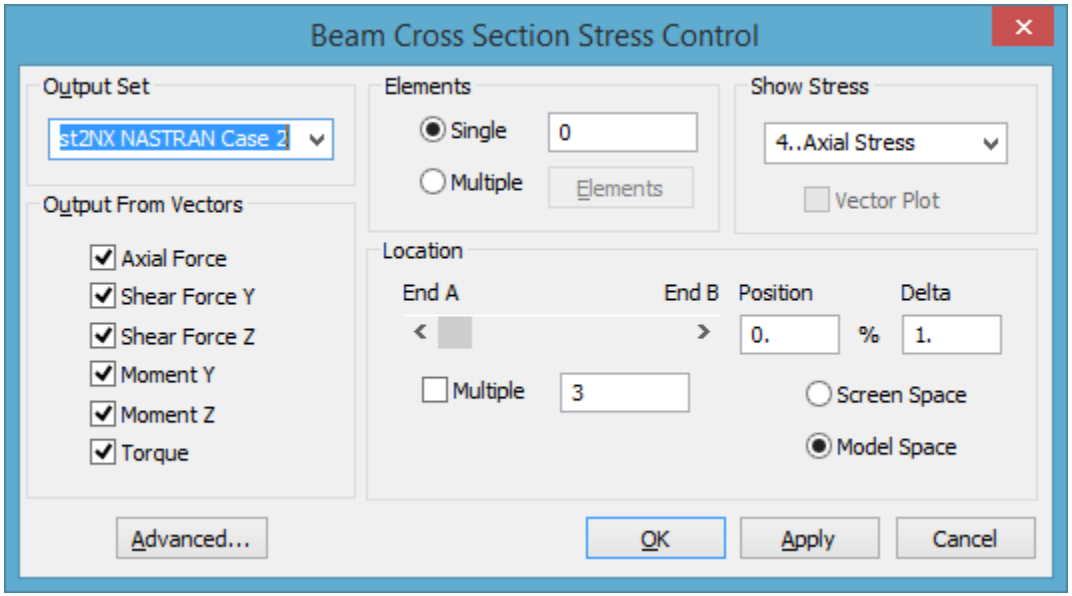
The FEMAP Beam Calculator provides detail section stresses on a line element using beam forces recovered via a standard analysis

- A local finite-element solution is used to calculate section stresses
- Any solver may be used as a “source” for the forces used in the calculations
- Multiple stress calculations are available, including von Mises, max shear, max/min principal, axial, Y/Z shear and combined shear
- Requires line elements with a section



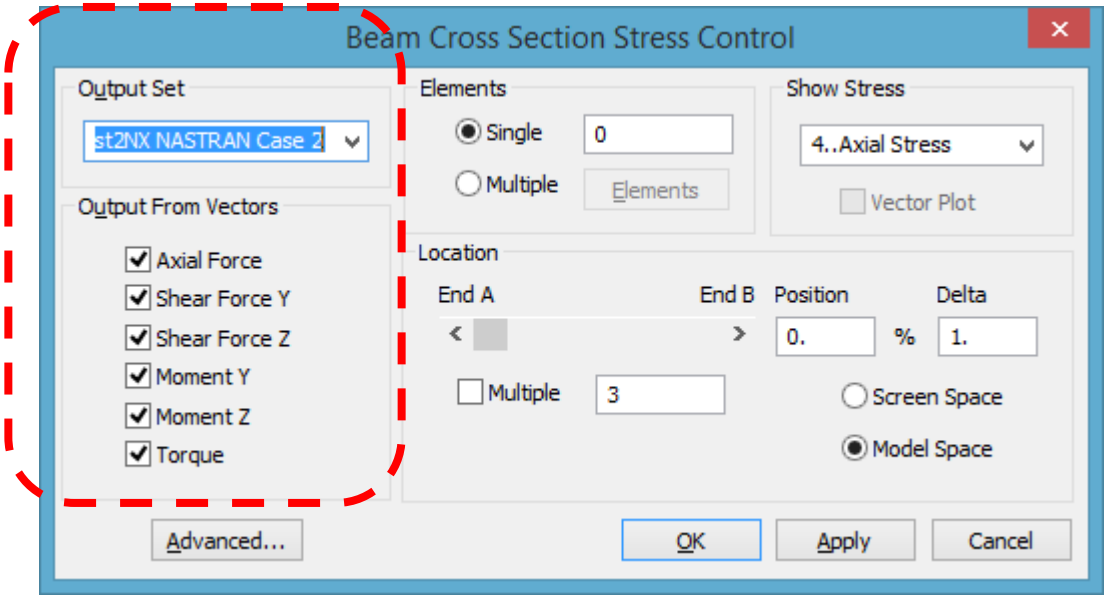
Using the FEMAP Beam Calculator

The UI for the FEMAP Beam Calculator is accessed via “View->Advanced Post->Beam Cross Section”



FEMAP Beam Calculator Basic Setup

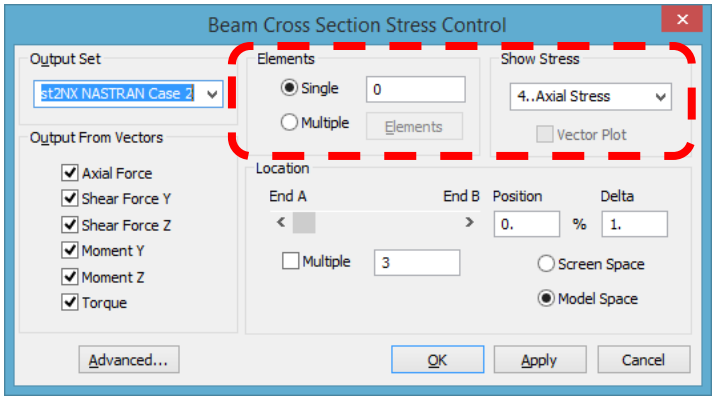
- Select Output Set
- Select forces from Output Set to include in calculations

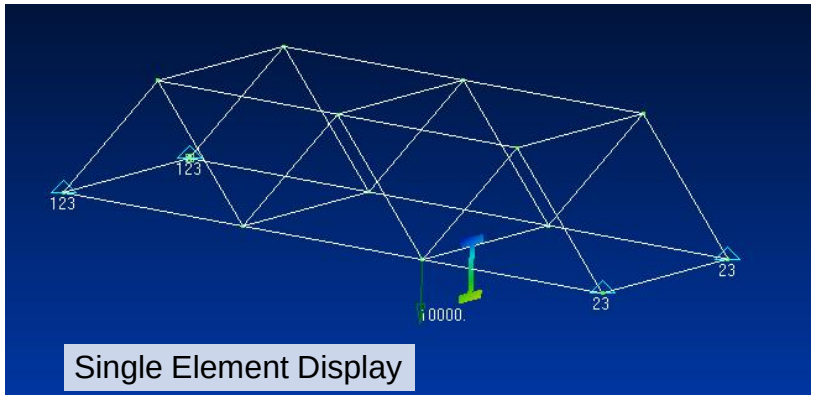


Info	
*	Vector IDs used for forces are dependent on element type (bar, beam, etc) as well as solver (Nastran, Abaqus, Ansys, etc) and are automatically selected by FEMAP

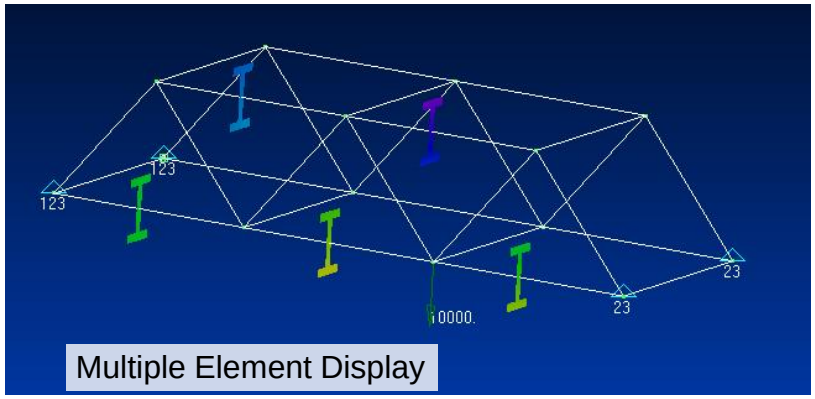
FEMAP Beam Calculator Basic Setup

- Select element(s) to display; either single or multiple elements can be displayed
- Select stress component to display





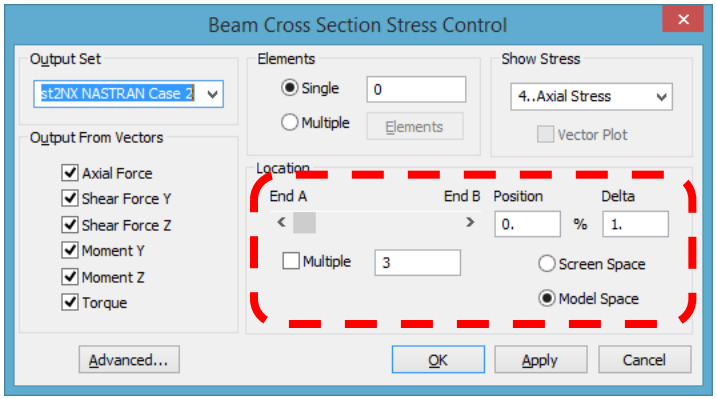
Single Element Display



Multiple Element Display

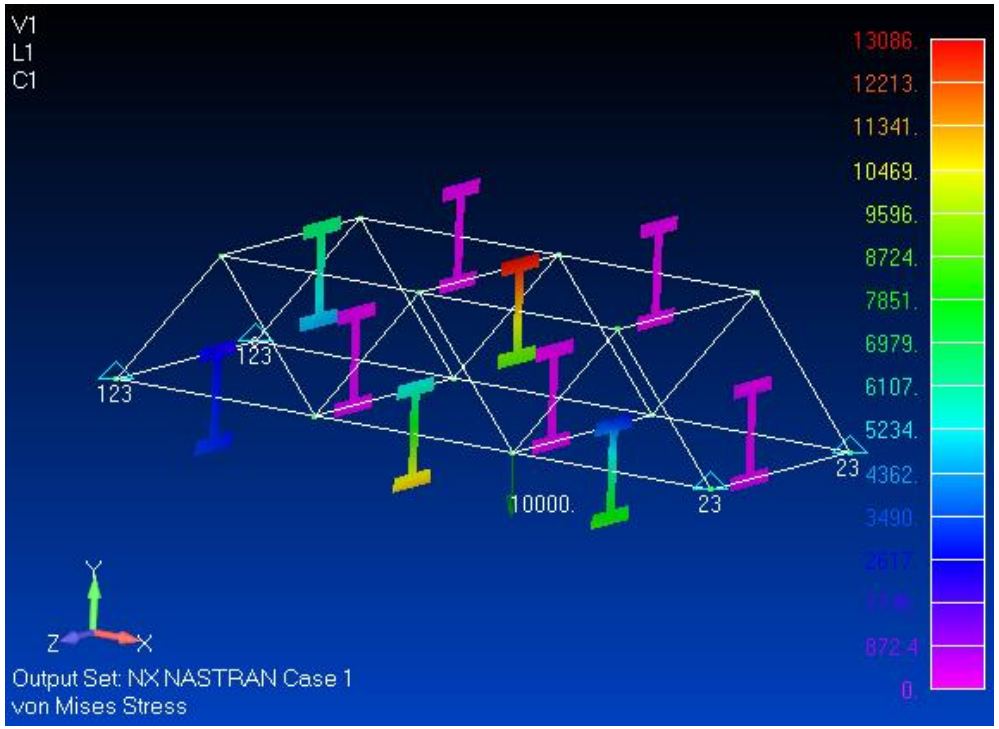
FEMAP Beam Calculator – Section Location

The section location (from end A to end B) can be modified using the slider, or manually specifying an option

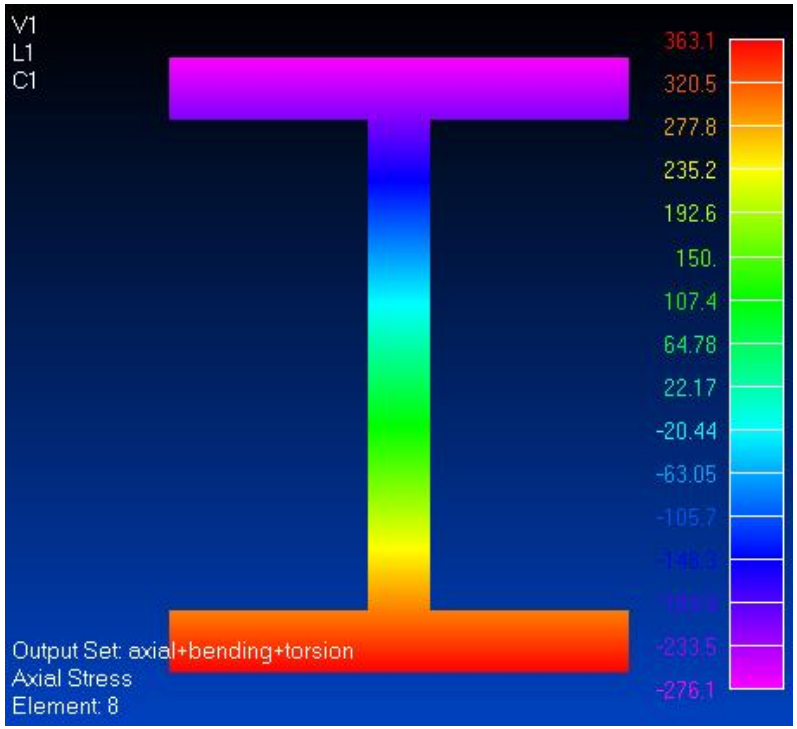


- By default, only a single section is displayed at the selected location
  - Enabling multiple sections will perform calculations / display at end A, end B and the equally-spaced points in between (2 to 10)
    - Default of 3 displays sections at end A, the midpoint and end B
- Contour plots can be displayed either in screen space or model space
  - When displaying in screen space, only a single section for a single element can be displayed

FEMAP Beam Calculator – Section Location



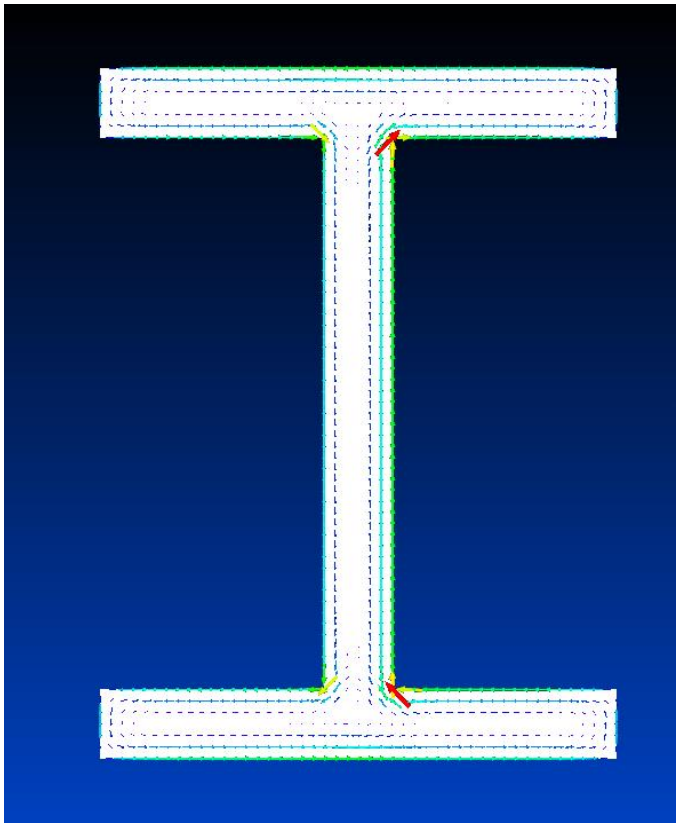
Model Space



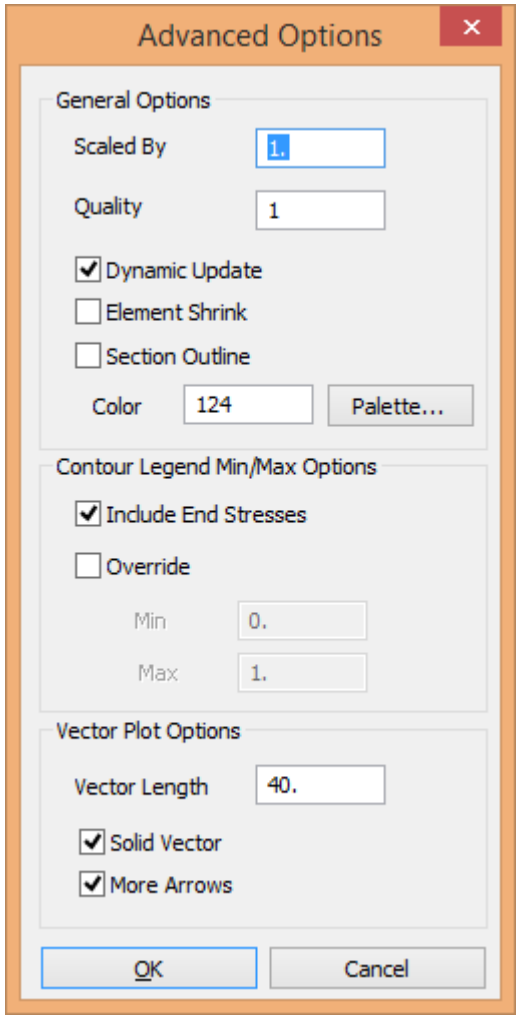
Screen Space

FEMAP Beam Calculator – Combined Shear Vector Plot

When displaying Combined Shear Stress quantities, the results may be viewed as a contour vector plot to visualize the direction of shear flow



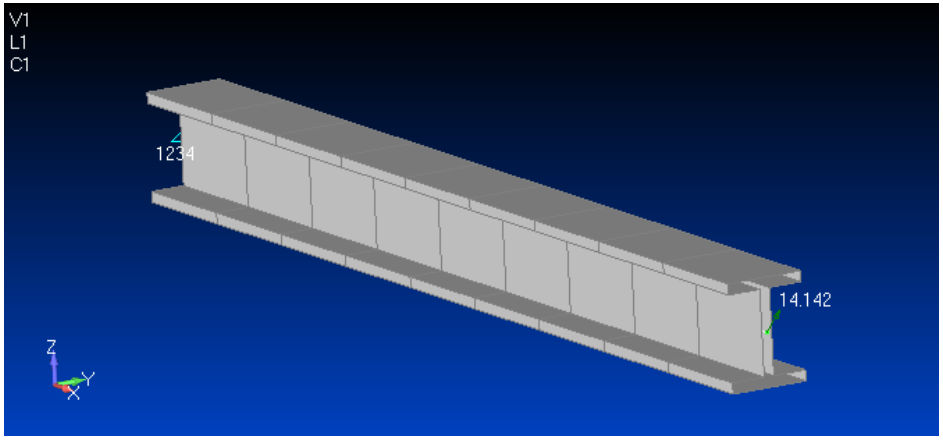
Advanced Beam Calculator Options



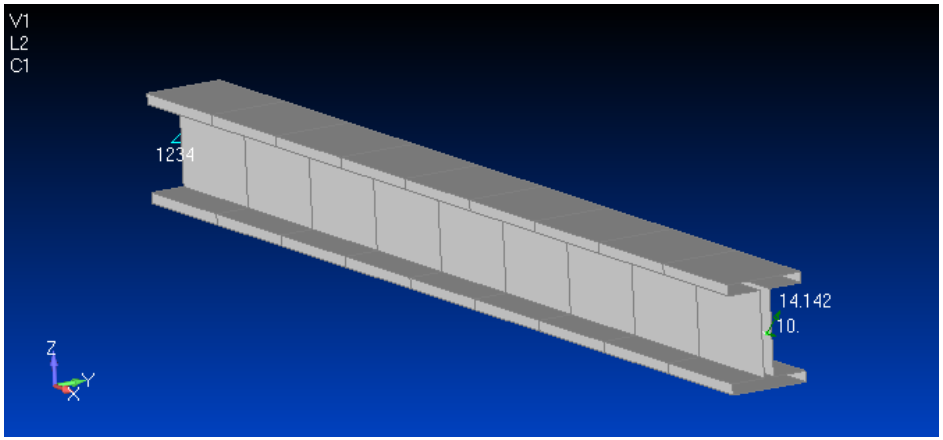
- **Scaled by** – scale factor for *display* of beam cross section display, from 0.1 to 100. Only used for model space
- **Quality** – section mesh quality, value from 1 (coarse) to 5 (fine)
- **Dynamic Update** – automatically update viewport when visualization options are changed, if unchecked, “Apply” must be manually clicked
- **Element Shrink** – enable element shrink in viewport to aid in visualization of end sections when multiple beams are selected
- **Include End Stresses** – contour levels include end stresses at end A and end B
- **Override** – manually specify contour levels

Compared to NASTRAN Results

Analysis Set 1: Axial + Bending



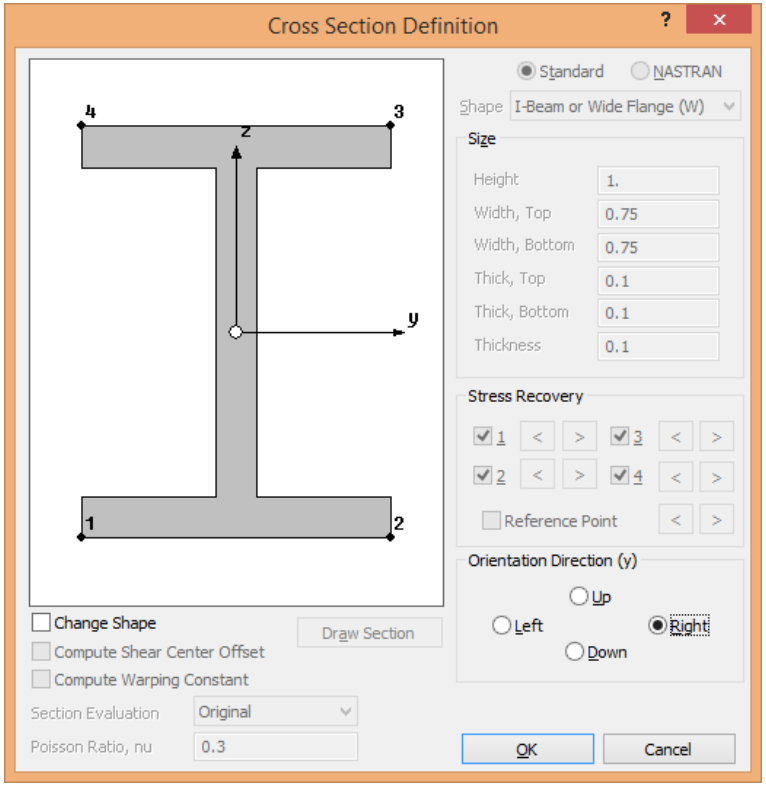
Analysis Set 2: Axial + Bending + Torsion



NASTRAN Stress Results

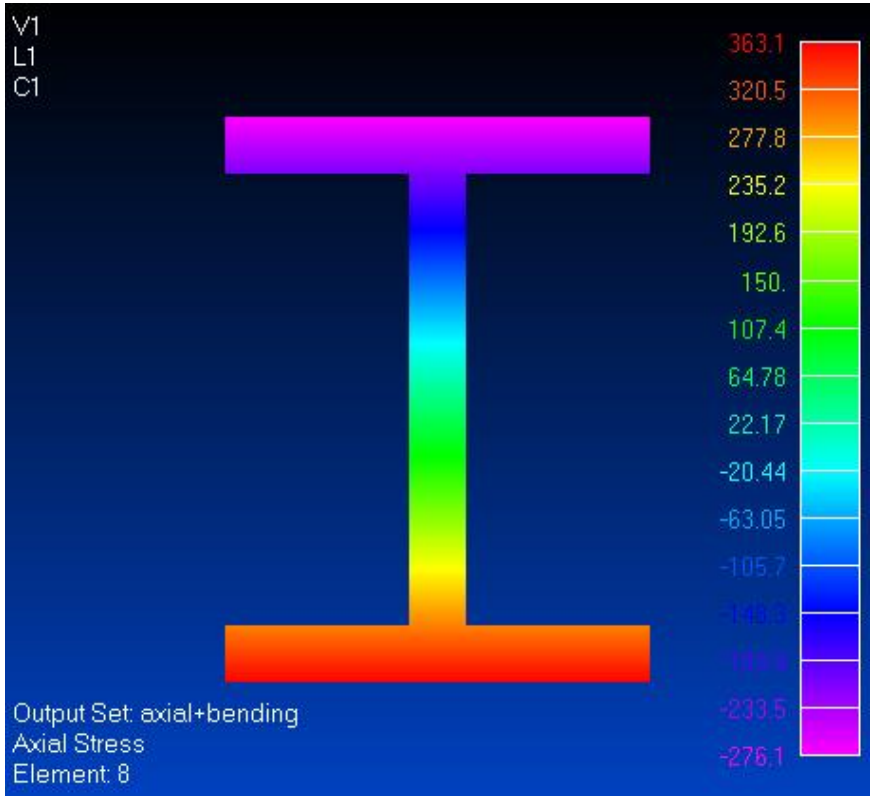
Data Table

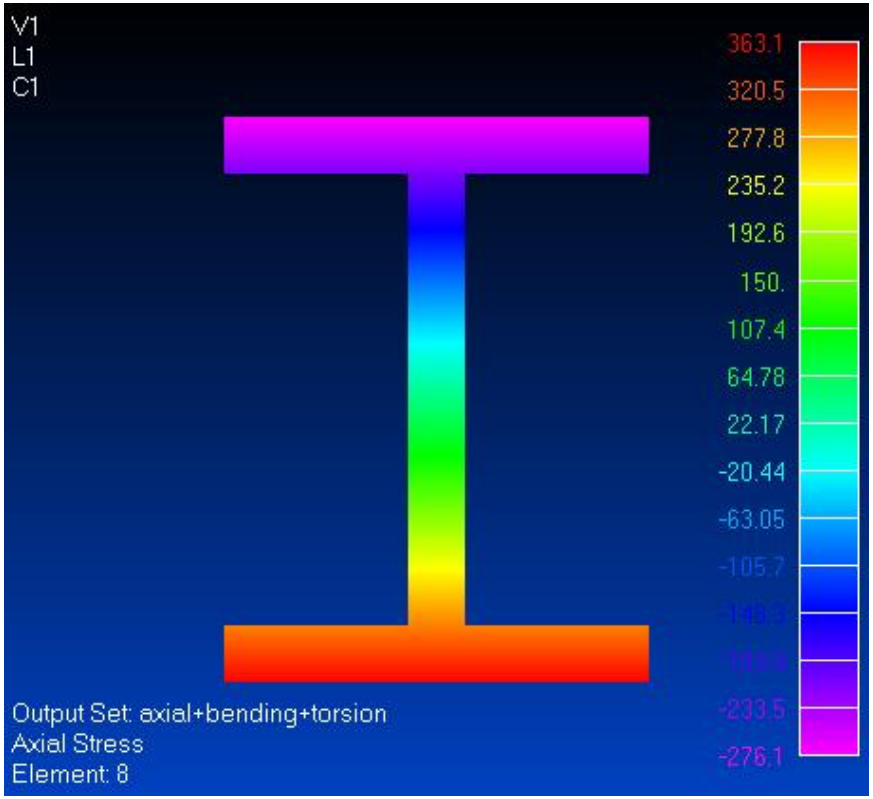
ID	CSys ID	Vector ID	Title	1..NX NASTRAN Case 1	2..NX NASTRAN Case 1
8	0	3139	Beam EndA Pt1 Comb Stress	363.0688	363.0688
8	0	3140	Beam EndA Pt2 Comb Stress	363.0688	363.0688
8	0	3141	Beam EndA Pt3 Comb Stress	-276.1123	-276.1123
8	0	3142	Beam EndA Pt4 Comb Stress	-276.1123	-276.1123
8	0	3151	Beam EndB Pt1 Comb Stress	203.2736	203.2736
8	0	3152	Beam EndB Pt2 Comb Stress	203.2736	203.2736
8	0	3153	Beam EndB Pt3 Comb Stress	-116.317	-116.317
8	0	3154	Beam EndB Pt4 Comb Stress	-116.317	-116.317
8	0	3164	Beam EndA Max Comb Stress	363.0688	363.0688
8	0	3165	Beam EndA Min Comb Stress	-276.1123	-276.1123
8	0	3166	Beam EndB Max Comb Stress	203.2736	203.2736
8	0	3167	Beam EndB Min Comb Stress	-116.317	-116.317



FEMAP Beam Calculator Results

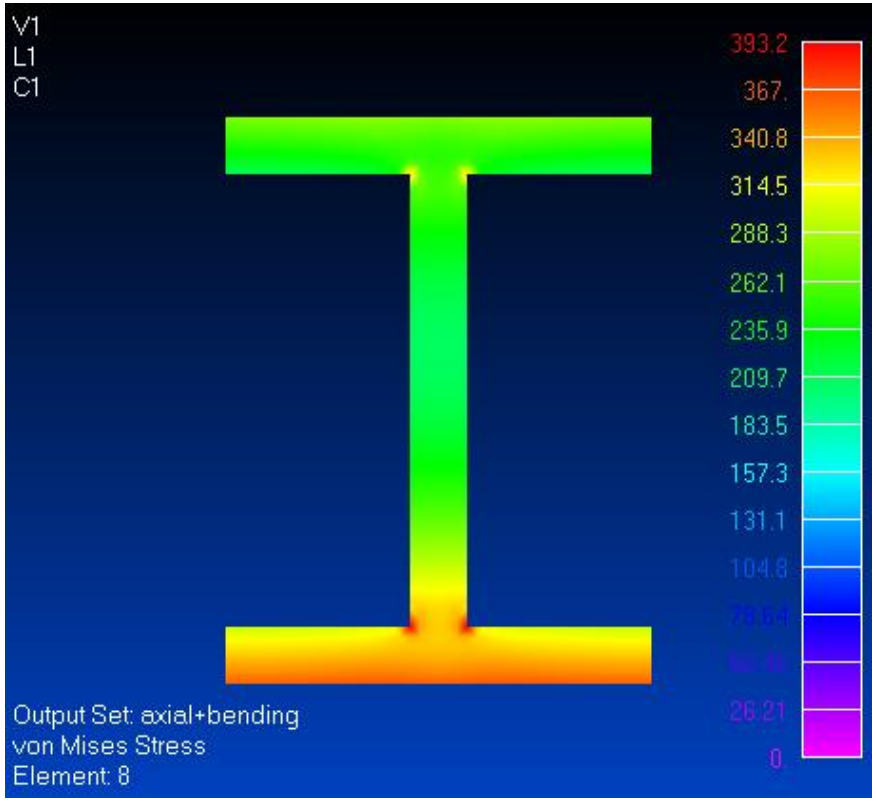
Axial Stress, Element 8, End A

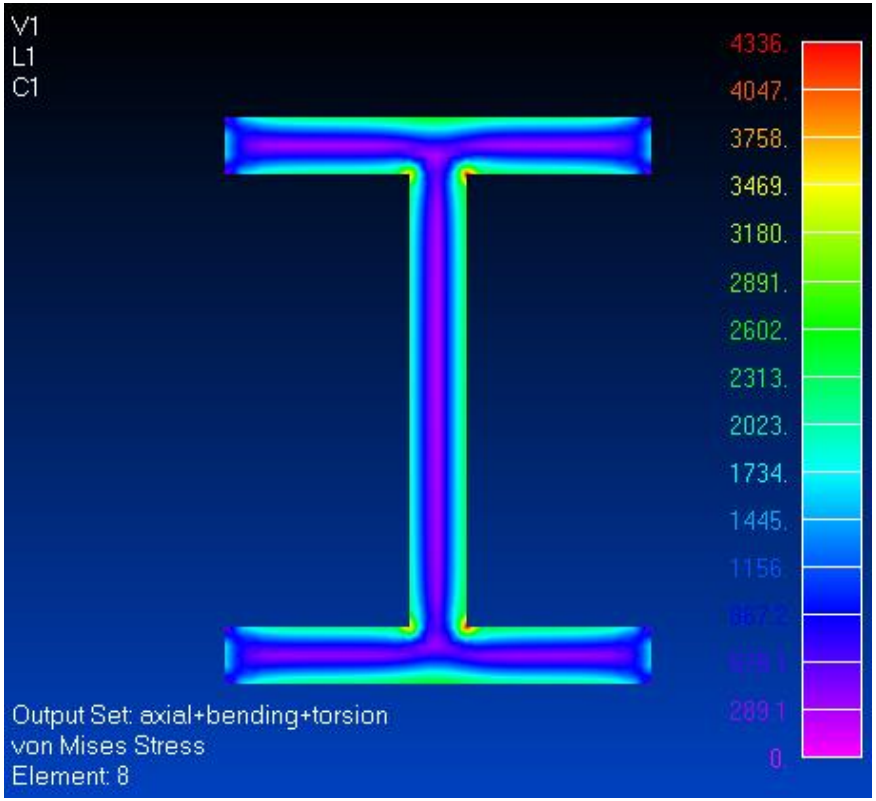




FEMAP Beam Calculator Results

von Mises Stress, Element 8, End A

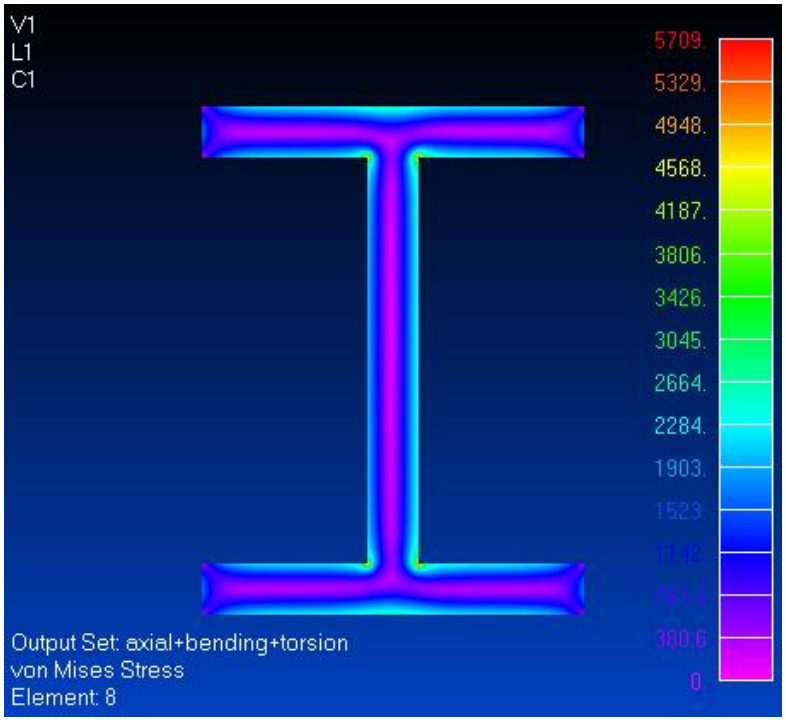




Stress Concentrations

Due to the nature of finite element analysis, stress concentrations will appear in the stress results

- It is up to the analyst to understand the nature of the stress concentration and make the ultimate determination of if it is a “real” stress
- As mesh refinement, increases, stress at a sharp point will also increase



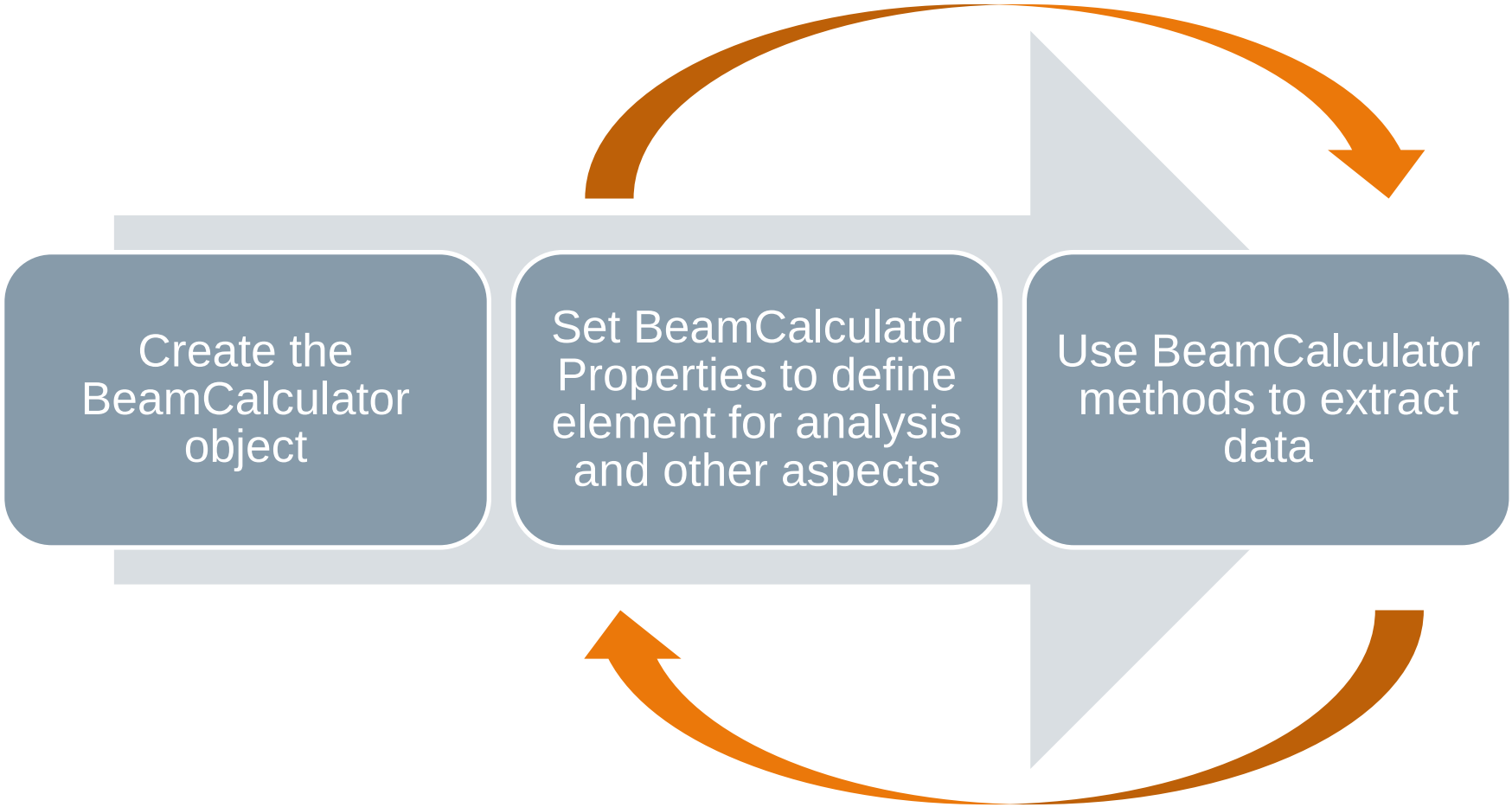
Using the FEMAP Beam Calculator API

The computational aspects of the FEMAP Beam calculator is handled through the BeamCalculator API object

- The BeamCalculator object
  - Was added in FEMAP v11.1.1
  - Is a “tool” object, similar to Sets and the Data Table. Properties and methods of the BeamCalculator are unique to that object and not inherited from a parent class
  - Works on a single beam element at a time, across one or more output sets
  - Can be used to extract
    - Element section property data
    - Underlying mesh data
    - Calculated stress information
    - Max/Min stress data
- Any number of BeamCalculator objects can be created / used

Using the FEMAP Beam Calculator API

Workflow



Using the FEMAP Beam Calculator API
BeamCalculator Properties

SIEMENS

Property	Description	Default
`INT4 Element`	Beam element to analyze	n/a
`REAL8 Position`	Position along beam, 0.0 (end A) to 1.0 (end B)	0.0
`INT4 MeshFactor`	Mesh refinement factor, 1.0 (coarse) to 5.0 (fine)	1.0
`BOOL IncludeAxialForce`	Include axial force in stress calculations	TRUE
`BOOL IncludeShearForceY`	Include Y shear force in stress calculations	TRUE
`BOOL IncludeShearForceZ`	Include Z shear force in stress calculations	TRUE
`BOOL IncludeMomentY`	Include moment about Y in stress calculations	TRUE
`BOOL IncludeMomentZ`	Include moment about Z in stress calculations	TRUE
`BOOL IncludeTorque`	Include torque in stress calculations	TRUE

Info
*

The only property that is must be set is the Element property. Values for each property will be used for all subsequent method calls until changed.

Using the FEMAP Beam Calculator API
BeamCalculator Methods

SIEMENS

`GetElementProperties()` – Returns section properties for the specified element	
`OUT – REAL8 pdArea`	Area
`OUT – REAL8 pdIzz`	Izz
`OUT – REAL8 pdIyy`	Iyy
`OUT – REAL8 pdIyz`	Iyz
`OUT – REAL8 pdJ`	J
`OUT – REAL8 pdShearAreaY`	Y shear area
`OUT – REAL8 pdShearAreaZ`	Z shear area
`OUT – REAL8 pdWarp`	Warping constant
`OUT – REAL8 pdNeutralAxisOffsetY`	Y neutral axis offset
`OUT – REAL8 pdNeutralAxisOffsetZ`	Z neutral axis offset

Info *	If the beam calculator in the Property dialog was used (and not modified by hand), these values should match the values on the Property object

Using the FEMAP Beam Calculator API
BeamCalculator Methods

SIEMENS

`GetMeshInfo()` – Returns node location and element connectivity information for the associated section mesh	
`OUT – INT4 pnNumNode`	Number of nodes in mesh
`OUT – REAL8 pvCoord [0..(pnNumNode)*2–1]`	Y and Z coordinates of each node
`OUT – pnNumElem`	Number of elements
`OUT – INT4 pvNodes [0..(pnNumElem)*9–1]`	Node indices for 9-noded quad (node 9 is at the element centroid)

Info	
*	The coordinate (0.0, 0.0) is located at the centroid

Using the FEMAP Beam Calculator API
BeamCalculator Methods

SIEMENS

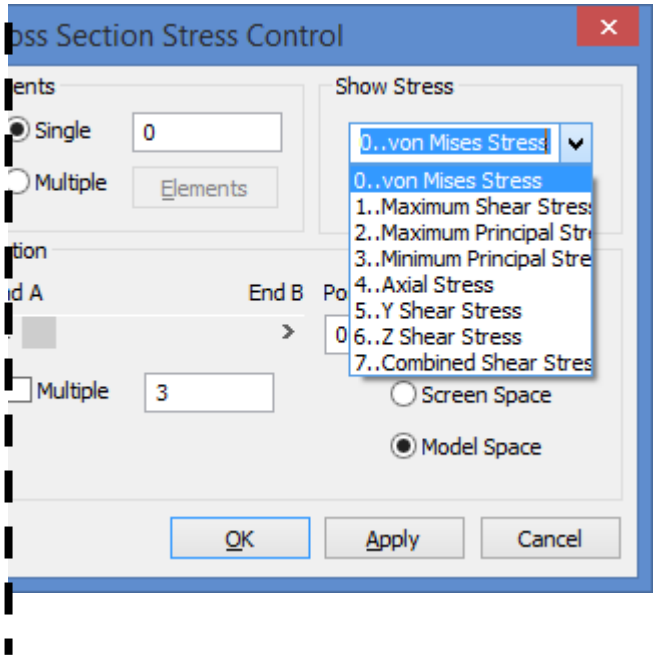
`CalculateStress()` – calculates beam section stresses based on forces from one or more results sets. Use the Include* properties to specify which components are considered in the calculations	
`INT4 nOutputSetID`	ID of FEMAP Set object containing result set IDs. Alternatively, specify a negative number for a single output set
`enum enComponent`	Stress component
`OUT - INT4 pnNumNodes`	Number of nodes returned
`OUT - REAL8 pvCoordinates [0..(pnNumNodes*2)-1]`	Y and Z coordinates of each node returned
`OUT - REAL8 pvStresses [nOS*nComp*pnNumNodes-1]`	Stresses at each node. Organized by - Output sets - Stress components - Stress at each node

Using the FEMAP Beam Calculator API
BeamCalculator Methods

SIEMENS

`CalculateStress()` – continued	
`OUT – REAL8 pvMaxStress [0..(0 or 7)]`	Maximum calculated stress for requested stress component. If all components were selected, an array is returned containing the max stress for each component
`OUT – INT4 pvMaxStressInfo [0..(0 or 7)]`	Information about maximum calculated stress for requested stress component. If only one output set was specified, this value is the node index corresponding to the max stress. If multiple output sets were specified, this value is the corresponding output set ID
`OUT – REAL8 pvMinStress [0..(0 or 7)]`	Minimum calculated stress for requested stress component(s)
`OUT – INT4 pvMinStressInfo [0..(0 or 7)]`	Information about minimum calculated stress for requested stress component(s)

Using the FEMAP Beam Calculator API
BeamCalculator Methods



enum zBeamCalculatorStressComponent	
FBMC_SC_ALL	FBMC_SC_AXIAL
FBMC_SC_VONMISES	FBMC_SC_YSHEAR
FBMC_SC_MAXSHEAR	FBMC_SC_ZSHEAR
FBMC_SC_MAXPRIN	FBMC_SC_COMBSHEAR
FBMC_SC_MINPRIN	

Using the FEMAP Beam Calculator API
BeamCalculator Methods

SIEMENS

`CalculateStressFromForces()` – Identical to CalculateStress() method, however user-specified forces are used, rather than ones from a result set. When using CalculateStressFromForces(), all Include* properties are assumed to be TRUE	
`REAL8 dForcesA[0…5]`	Forces at end A 0 – Axial force 1 – Y shear force 2 – Z shear force 3 – Moment, Y 4 – Moment, Z 5 – Torque
`REAL8 dForcesB[0..5]`	Forces at end B (see array for end A)
`enum enComponent`	Stress component to return
`OUT – All output values are identical to CalculateStress()`	

Using the FEMAP Beam Calculator API
BeamCalculator Methods

SIEMENS

`FindMaxMinStress()` – returns the max/min stress along with associated stress component and related location for an element across one or more output sets	
`INT4 nOutputSetID`	FEMAP Set ID with output sets / -(output set)
`enum enComponent`	Stress component
`OUT – INT4 nMaxSetID`	Output set ID for max stress
`OUT – enum enMaxComponent`	Stress component for max stress
`OUT – REAL8 dMaxLocation`	Max stress location (0.0, 0.5, or 1.0)
`OUT - REAL8 dMaxStress`	Max stress value
`OUT - INT4 nMinSetID`	Output set ID for min stress
`OUT – enum enMinComponent`	Stress component for min stress
`OUT – REAL8 dMinLocation`	Min stress location (0.0, 0.5, or 1.0)
`OUT – REAL8 dMinStress`	Min stress value

Example

Find Max von Mises Stress Across Multiple Elements:

```
Sub Main
    Dim App As femap.model
    Set App = feFemap()

    Dim fbc As femap.BeamCalculator
    Dim fsB As femap.Set
    Dim fsO As femap.Set

    Dim dMax As Double
    Dim dMin As Double
    Dim dMaxLoc As Double
    Dim dMinLoc As Double
    Dim eMax As zBeamCalculatorStressComponent
    Dim eMin As zBeamCalculatorStressComponent
    Dim nMax As Long
    Dim nMin As Long

    Set fbc = App.feBeamCalculator
    Set fsB = App.feSet
    Set fsO = App.feSet

    fsB.AddRule( FET_L_BEAM, FGD_ELEM_BYTYPE )
    fsB.AddRule( FET_L_BAR, FGD_ELEM_BYTYPE )
    fsO.AddAll( FT_OUT_CASE ) fsB.Reset
```

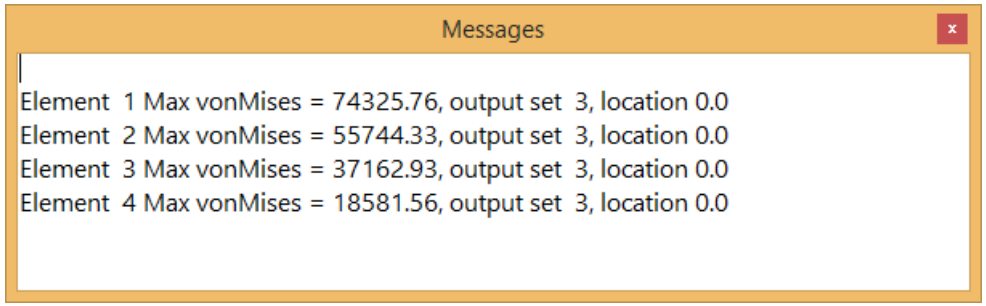
Example

Find Max von Mises Stress Across Multiple Elements:

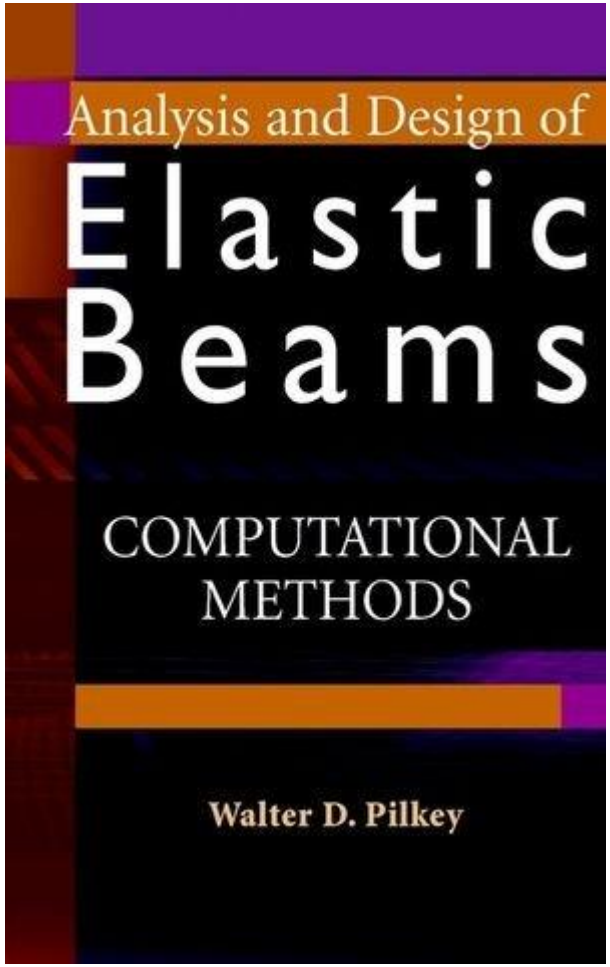
```
    While fsB.Next() <> FE_FAIL
        fbc.Element = fsB.CurrentID
        fbc.FindMaxMinStress( fsO.ID, FBMC_SC_VONMISES, _
                                      nMax, eMax, dMaxLoc, dMax, _
                                      nMin, eMin, dMinLoc, dMin )

        App.feAppMessage( FCM_NORMAL, "Element " + Str$( fsB.CurrentID ) + _
                                      " Max vonMises = " + Format$( dMax, "#.00" ) + _
                                      ", output set " + Str$( nMax ) + _
                                      ", location " + Format( dMaxLoc, "0.0" ) )
    Wend

End Sub
```

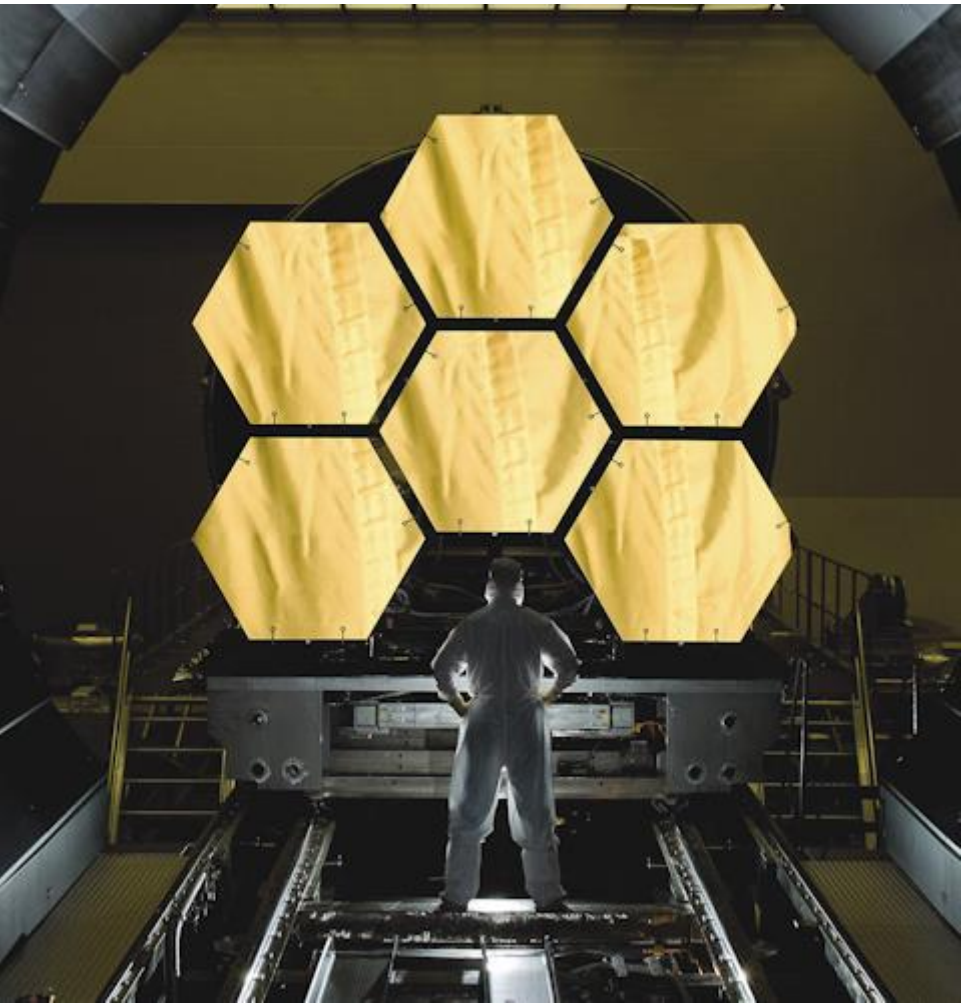


References



Analysis and Design of Elastic Beams: Computational Methods, Walter D. Pilkey

Using the FEMAP Beam Processor



Patrick Kriengsiri
FEMAP Development

411 Eagleview Blvd
Exton, PA, 19341

Phone: 404-353-6596

E-mail:
patrick.kriengsiri@siemens.com