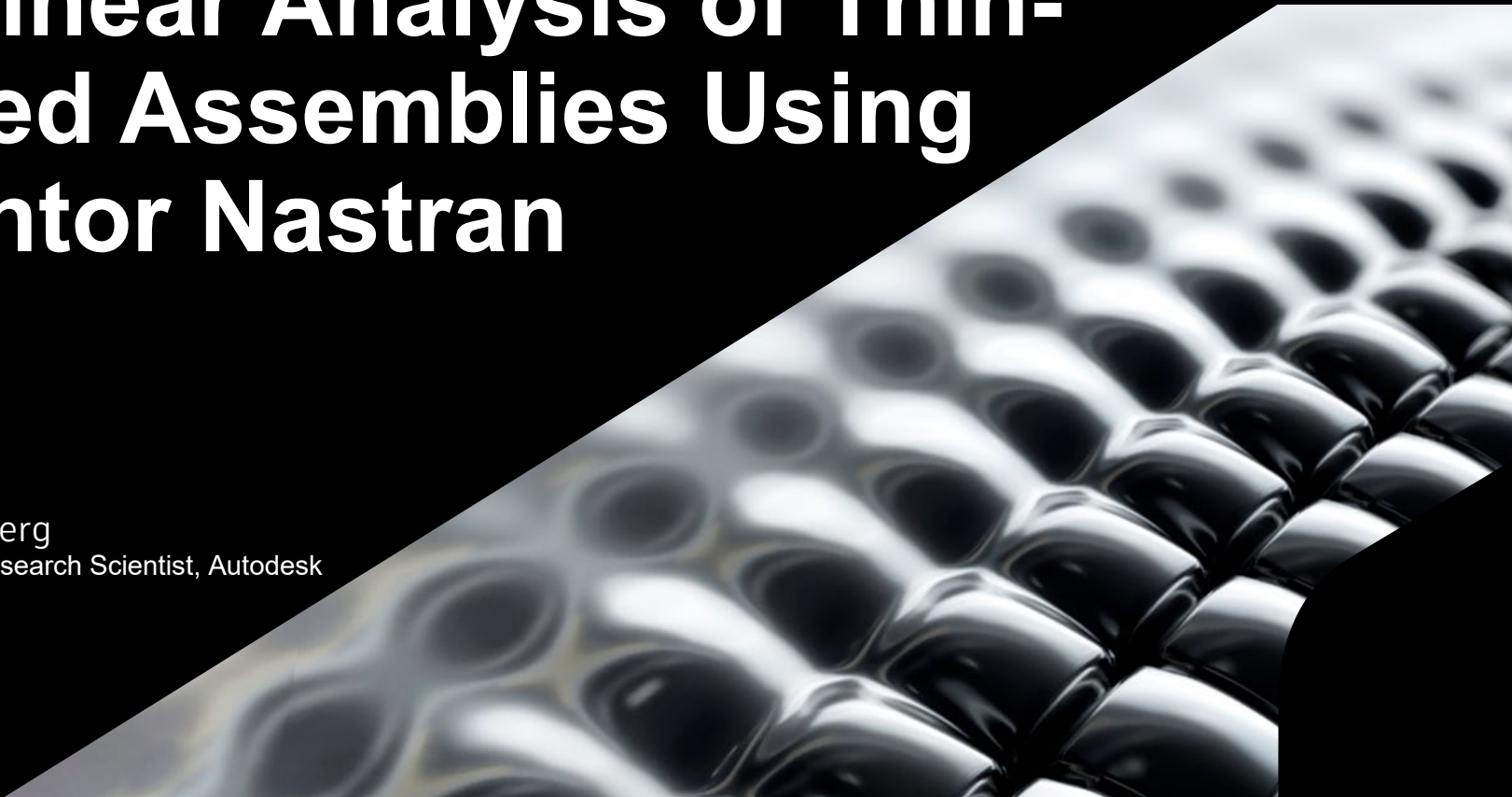




Nonlinear Analysis of Thin-Walled Assemblies Using Inventor Nastran

David Weinberg
Distinguished Research Scientist, Autodesk



David Weinberg



- Distinguished Research Scientist with Autodesk
- Autodesk Product Development and Manufacturing Solutions (PDMS), Nastran Simulation and Generative Design group
- Primary developer for Autodesk Nastran and Inventor Nastran
- Currently lead the team of developers for Autodesk Nastran
- Over 35 years' experience in FEA simulation working both as a user for several large Aerospace companies and as a developer
- Retired USAF aircraft commander/pilot

Safe Harbor Statement

We may make forward-looking statements regarding planned or future development efforts for our existing or new products and services and statements regarding our strategic priorities. These statements are not intended to be a promise or guarantee of business results, future availability of products, services or features but merely reflect our current plans and are based on factors currently known to us. These planned and future development efforts may change without notice. Purchasing and investment decisions should not be made based upon reliance on these statements.

A discussion of factors that may affect future results is contained in our most recent Form 10-K and Form 10-Q filings available at www.sec.gov, including descriptions of the risk factors that may impact us and the forward-looking statements made in these presentations. Autodesk assumes no obligation to update these forward-looking statements to reflect events that occur or circumstances that exist or change after the date on which they were made. If this presentation is reviewed after the date the statements are made, these statements may no longer contain current or accurate information.

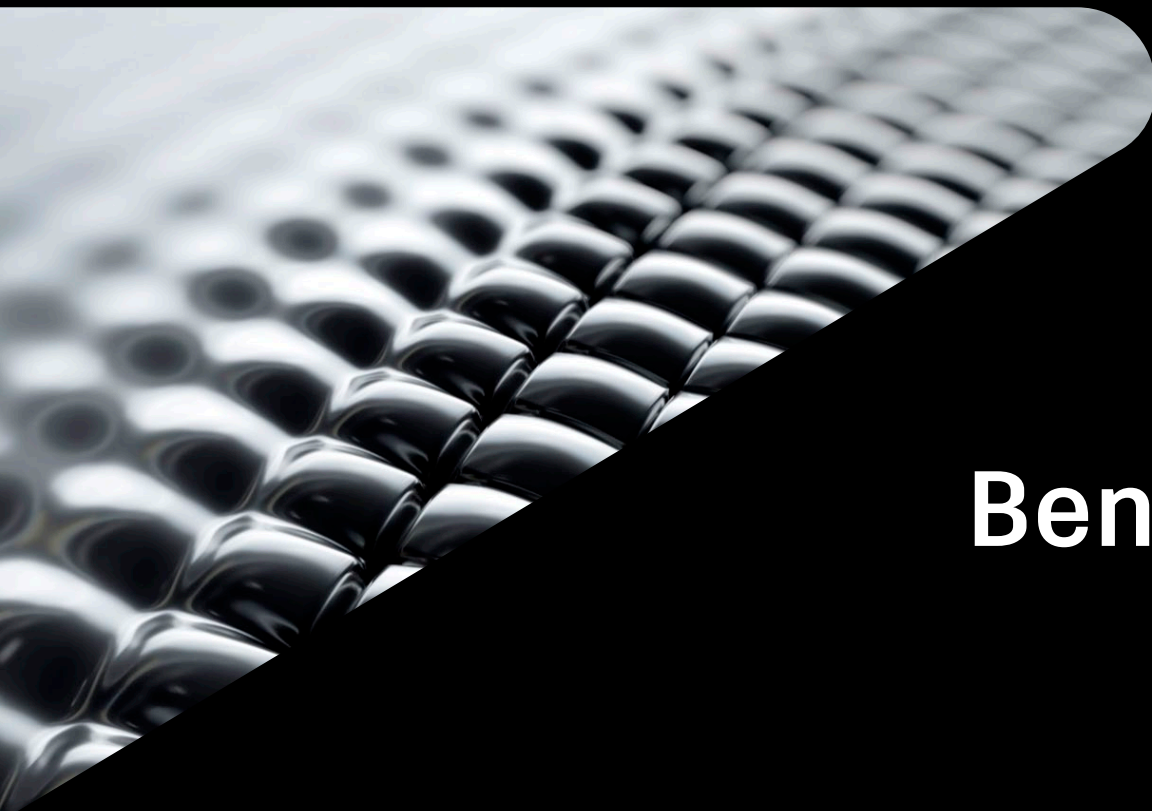
This presentation also contains information, opinions and data supplied by third parties and Autodesk assumes no responsibility for the accuracy or completeness of such information, opinions or data, and shall not be liable for any decisions made based upon reliance on any such information, opinions or data.

Autodesk's partners frequently compete against each other in the marketplace, and it is critically important that all participants in this meeting observe all requirements of antitrust laws and other laws regarding unfair competition. Autodesk's long insistence upon full compliance with all legal requirements in the antitrust field has not been based solely on the desire to stay within the bounds of the law, but also on the conviction that the preservation of a free and vigorous competitive economy is essential to the welfare of our business and that of our partners, the markets they serve, and the countries in which they operate. It is against the policy of Autodesk to sponsor, encourage or tolerate any discussion or communication among any of its partners concerning past, present or future prices, pricing policies, bids, discounts, promotions, terms or conditions of sale, choice of customers, territorial markets, quotas, inventory, allocation of markets, products or services, boycotts and refusals to deal, or any proprietary or confidential information. Communication of this type should not occur, whether written, oral, formal, informal, or "off the record." All discussion at this meeting should be strictly limited to presentation topics.

PLEASE NOTE: AU content is proprietary. Do Not Copy, Post or Distribute without expressed permission.

Learning Objectives

- Learn how to mid-surface and mesh solid geometry to generate a shell mesh.
- Learn how to lightweight an existing shell meshed design using Autodesk Nastran SIMP Topology Optimization.
- Learn how to define nonlinear materials and large displacement effects and perform a nonlinear analysis.
- Learn how to connect shell meshed parts in an assembly using contact.



3-Point Tube Bending Example

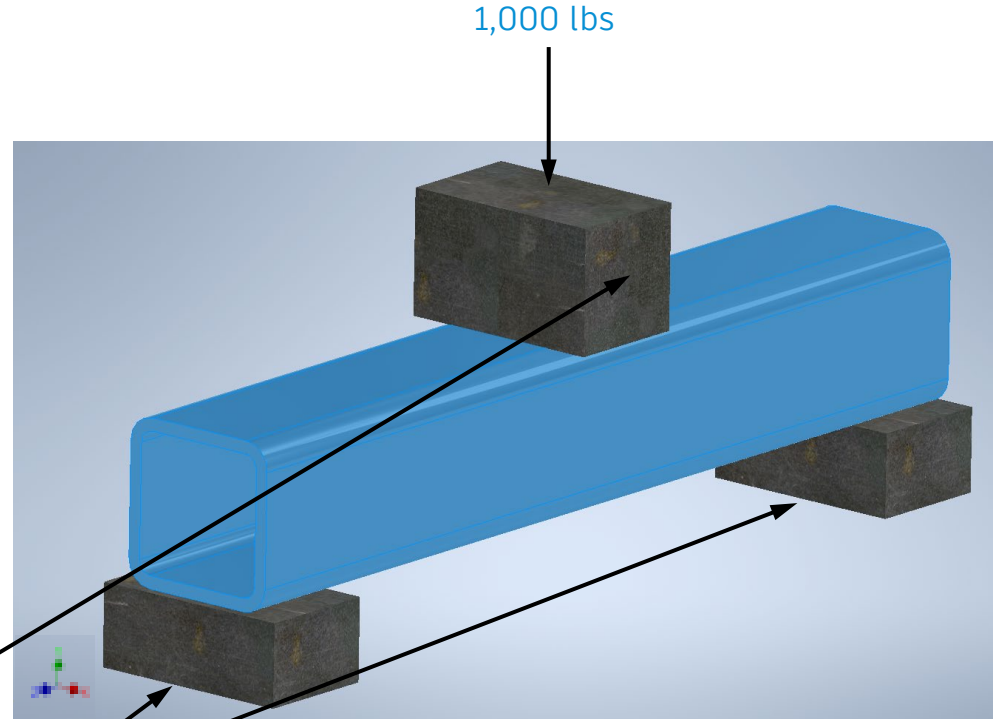
Solid Elements

Steel Tube Model

- Tube: AISC Steel:
 - 4 x 4 x 24 inches
 - 5/16 inches thick
 - Yield point 30ksi
- Blocks: Steel
- Constrained at the base of pads and side of pusher block

Constrained on one face in x-z direction, free movement in y

Fully constrained at base of pads



Steel Tube Material Definition

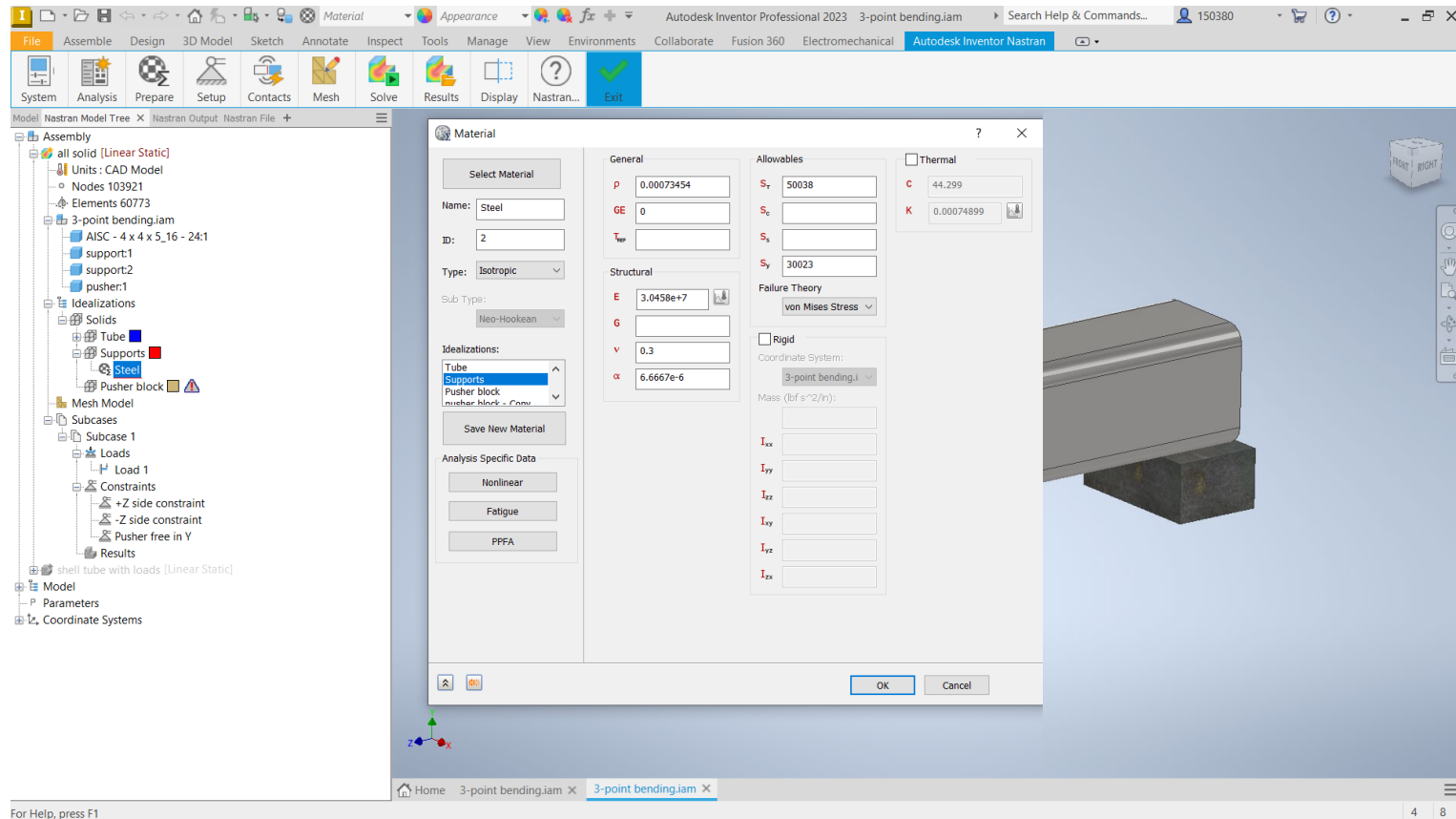
The screenshot displays the Autodesk Inventor Professional 2023 interface. The main window shows a 3D model of a steel tube assembly. A 'Material' dialog box is open, allowing the user to define the material properties for the selected 'Steel, Mild' material.

Material Dialog Box Fields:

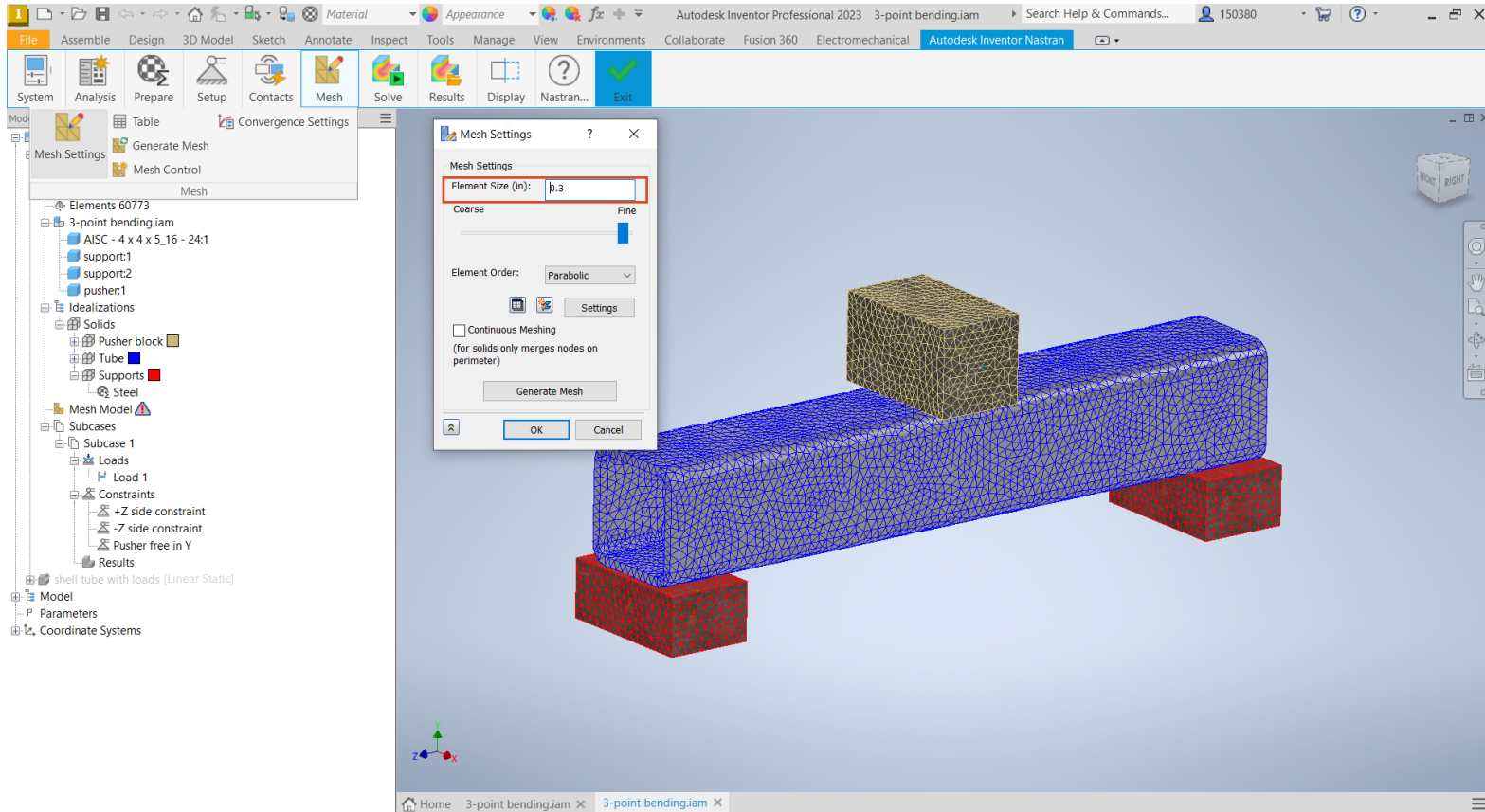
- General:**
 - Name: Steel, Mild
 - ID: 1
 - Type: Isotropic
 - Sub Type: Neo-Hookean
 - Idealizations: Tube, Supports, Pusher block, Pusher block - Conn.
 - Save New Material
- Analysis Specific Data:**
 - Nonlinear
 - Fatigue
 - PPFA
- General Properties:**
 - P : 0.00073454
 - GE : 0
 - T_{α}
- Structural:**
 - E : 3.1908e+7
 - G
 - ν : 0.275
 - α : 6.6667e-6
- Allowables:**
 - S_r : 50038
 - S_y
 - S_x
 - S_y : 30023
- Failure Theory:**
 - von Mises Stress
- Thermal:**
 - C : 44.299
 - K : 0.00060186
- Other:**
 - ☐ Thermal
 - ☐ Rigid
 - Coordinate System: 3-point bending.i
 - Mass (lbf s²/in):

The background shows a 3D model of a steel tube assembly, including a tube, supports, and a pusher block. The model is rendered in a blue and grey color scheme.

Steel Supports and Pusher Material Definition



Model Mesh Size Definition



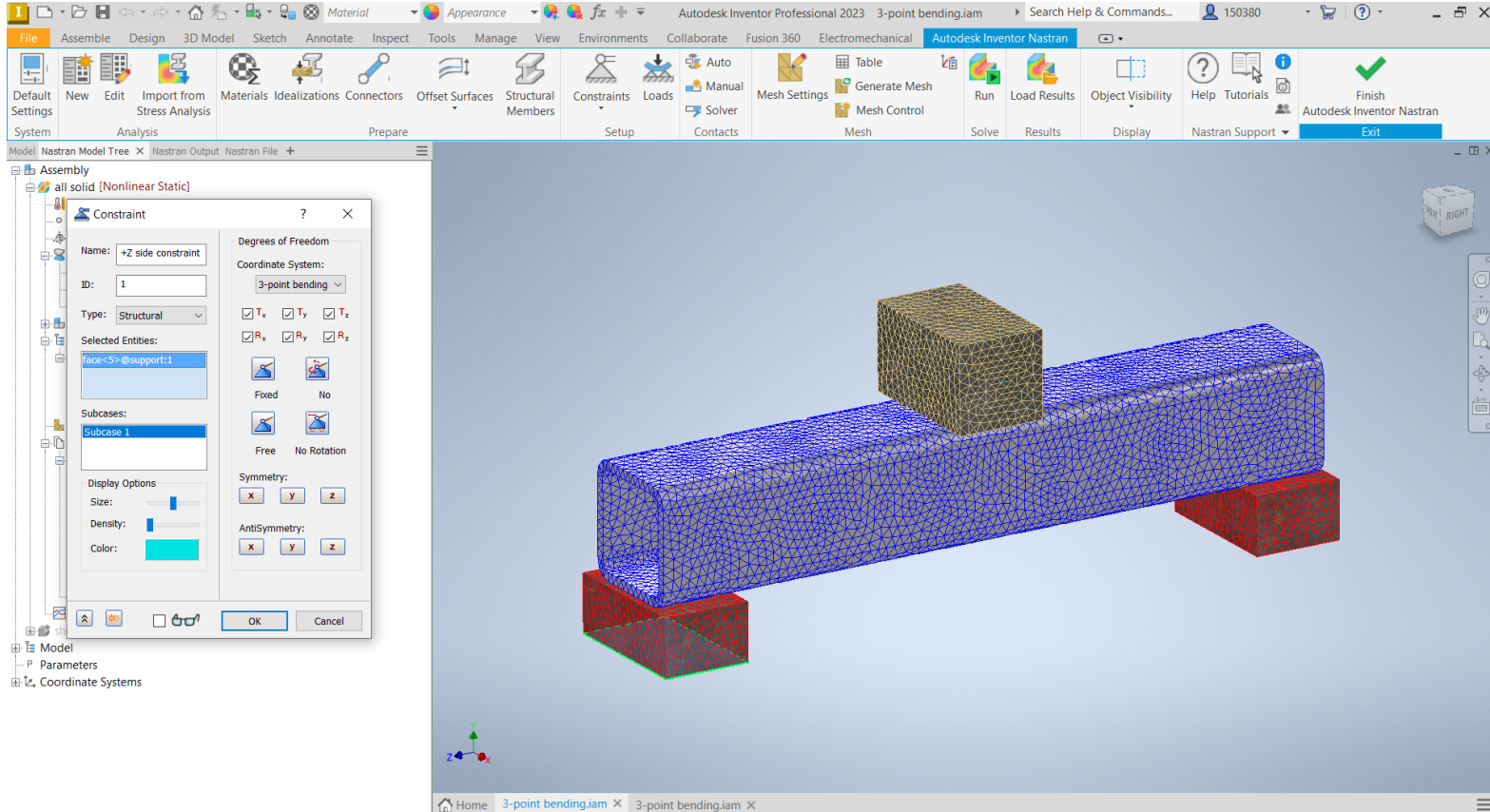
Load Definition

The screenshot displays the Autodesk Inventor Nastran interface for a 3-point bending analysis. The main 3D view shows a blue meshed beam supported by two red blocks, with a green meshed cube applying a downward force. The 'Load' dialog box is open, showing the following details:

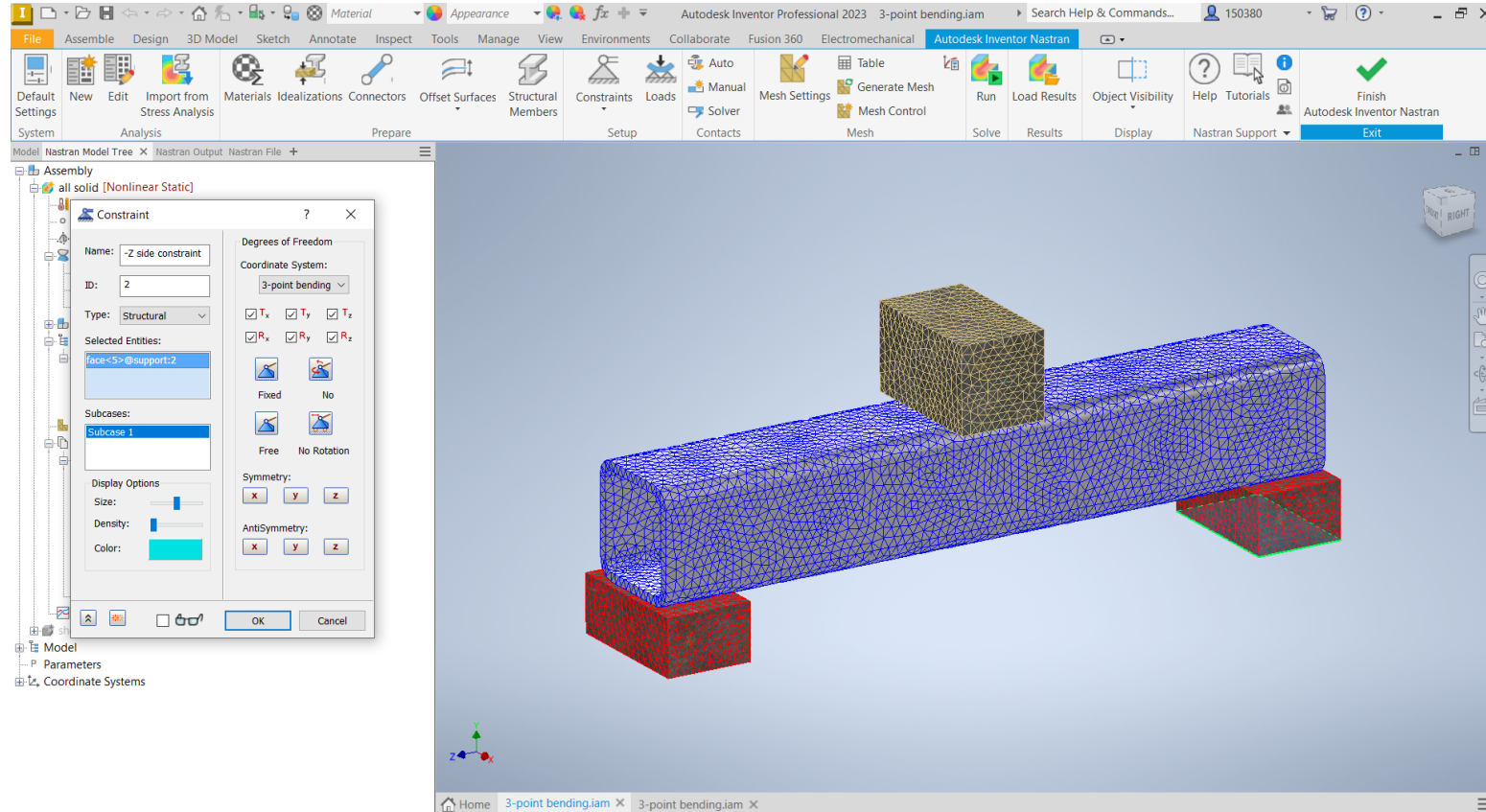
- Name:** Load 1
- ID:** 1
- Type:** Force
- Sub Type:** (empty)
- Selected Entities:** face<5>@pusher:1
- Subcases:** Subcase 1
- Display Options:** Size (slider), Density (slider), Color (green)
- Advanced Options >>**
- Load Definition:**
 - Direction:** Components
 - Coordinate System:** 3-point bending.1
 - Total Force:** ☐
 - Magnitude (lbf):**
 - F_x : 0
 - F_y : -1000
 - F_z : 0

The bottom status bar indicates 'For Help, press F1' and the active file is '3-point bending.iam'.

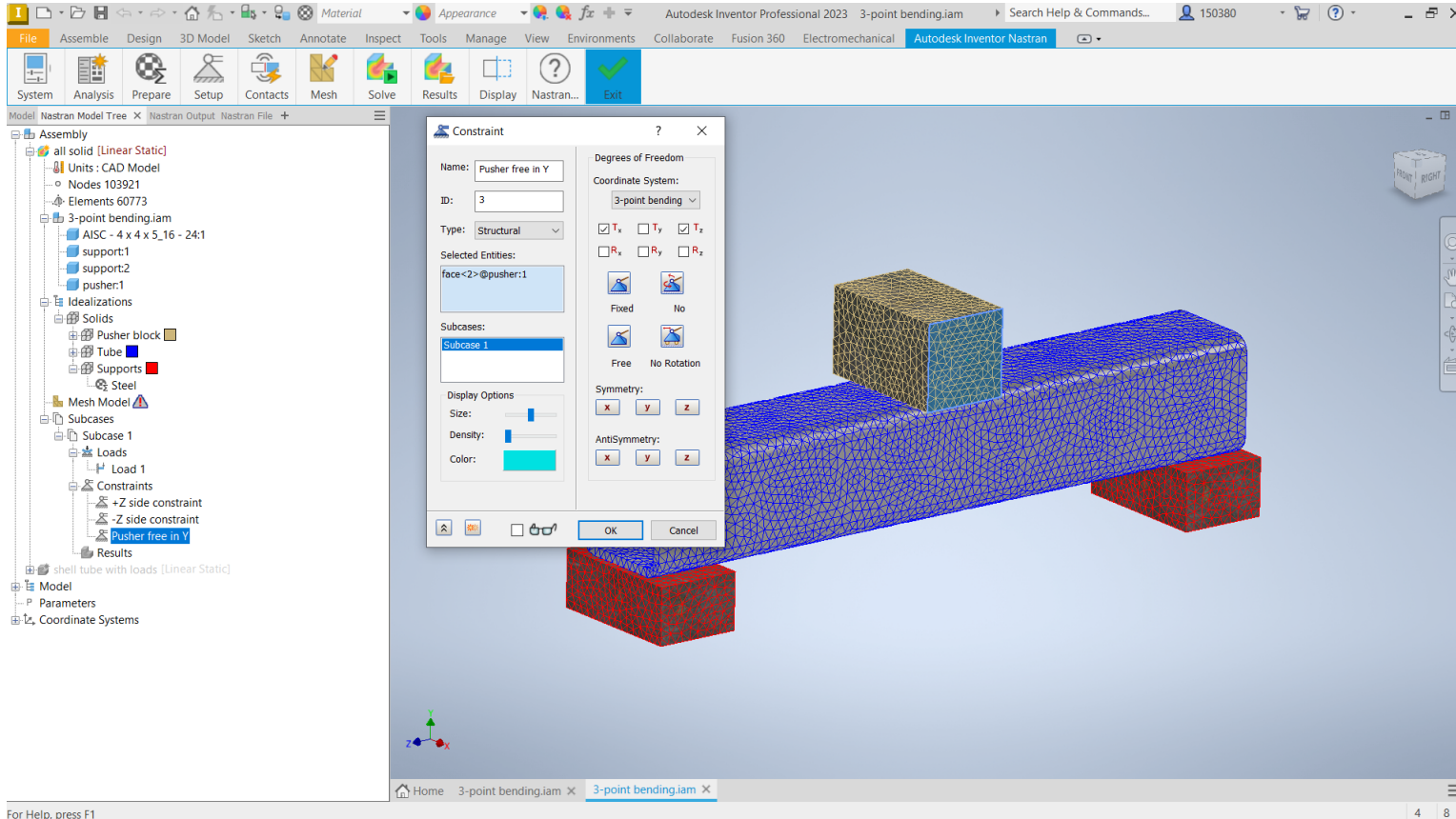
+Z Support Constraint Definition



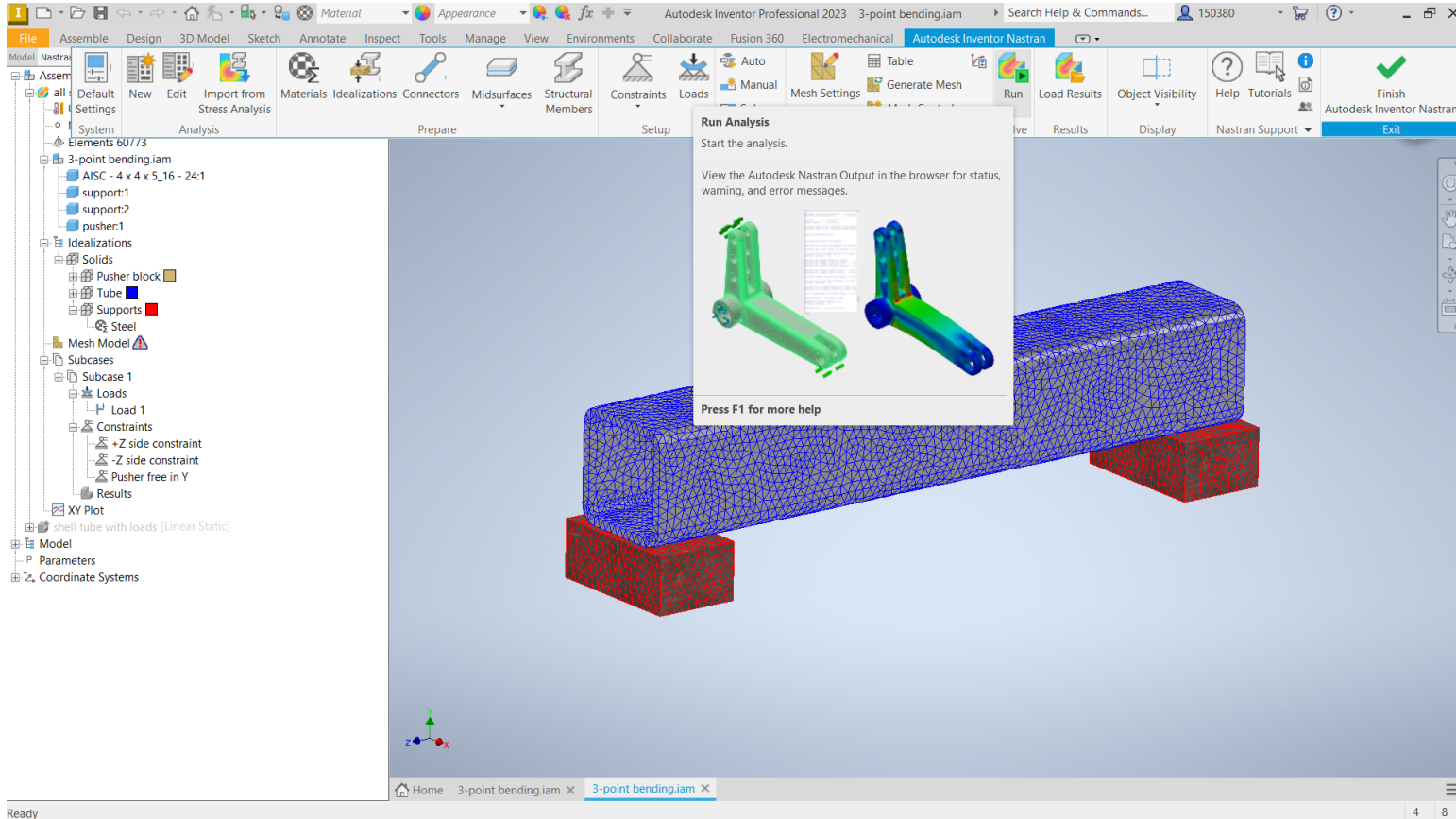
-Z Support Constraint Definition



Pusher Constraint Definition



Running Linear Statics



Fatal Error! Why?

Autodesk Inventor Professional 2023 3-point bending.iam

File Assemble Design 3D Model Sketch Annotate Inspect Tools Manage View Environments Collaborate Fusion 360 Electromechanical Autodesk Inventor Nastran

Model Nastran Model Tree Nastran Output X Nastran File +

MASS DISTRIBUTION BY TYPE

STRUCTURAL MASS = 1.534951E-01
NONSTRUCTURAL MASS = 0.000000E+00
CONCENTRATED MASS = 0.000000E+00
OTHER MASS = 0.000000E+00

CENTER OF MASS RELATIVE TO ORIGIN OF BASIC SYSTEM

X-COORDINATE = -7.179453E-02
Y-COORDINATE = -5.45151E-01
Z-COORDINATE = 2.012342E-05

WRITING OUT GRID POINT MASS PROPERTIES
PAGES WRITTEN: 1

CALCULATING PART DEFINITIONS
PERCENT COMPLETE: 100

WRITING OUT PART DEFINITIONS
PAGES WRITTEN: 1

LINEAR SOLUTION PROCESSOR MODULE

SOLUTION SEQUENCE FOR SUBCASE 1

FETCHING GLOBAL STIFFNESS MATRIX
PERCENT COMPLETE: 100

MAXIMUM STIFFNESS MATRIX DIAGONAL = 1.7264E+08 AT GRID 884
MINIMUM STIFFNESS MATRIX DIAGONAL = 1.7841E+06 AT GRID 924

ASSEMBLING GLOBAL LOAD VECTOR FOR SUBCASE 1

ASSEMBLING GRID POINT LOADS
PERCENT COMPLETE: 100

AVAILABLE VIRTUAL MEMORY = 4410259432 WORDS 33647.6 MB
AVAILABLE PHYSICAL MEMORY = 10335661568 WORDS 78854.8 MB

DRIVE C: DISK SPACE = 19371185152 WORDS 147790.4 MEGAB

SOLVING FOR DISPLACEMENTS FOR SUBCASE 1
PERCENT COMPLETE: 100

GLOBAL STIFFNESS MATRIX ITERATIVE SOLUTION STATISTICS

NUMBER OF SOLUTION ITERATIONS = 28
FINAL CONVERGENCE FACTOR = 6.030E-01

FATAL ERROR E5004 STIFFNESS MATRIX SINGULAR OR NON-POSITIVE

VIRTUAL MEMORY = 4412497920 WORDS 33664.7 MEGABYTES
REAL MEMORY = 10338014208 WORDS 78872.8 MEGABYTES

DRIVE C: DISK SPACE = 19371184128 WORDS 147790.4 MEGAB

MODEL ANALYSIS TIME SUMMARY

TOTAL CPU TIME = 12.86 SECONDS
WALLCLOCK TIME = 12.96 SECONDS

EXECUTION TERMINATED DUE TO DISPLAYED ERROR(S)

TOTAL WARNINGS = 0
TOTAL FATAL ERRORS = 1

Autodesk Inventor Nastran

Nastran Solution Failed.

OK

Home 3-point bending.iam X 3-point bending.iam X

Ready 4 8

Getting Help

Autodesk Inventor Professional 2023 3-point bending.iam

File Assemble Design 3D Model Sketch Annotate Inspect Tools Manage View Environments Collaborate Fusion 360 Electromechanical Autodesk Inventor Nastran

Model Nastran Model Tree Nastran Output X Nastran File +

MASS DISTRIBUTION BY TYPE

STRUCTURAL MASS = 1.534951E-01
NONSTRUCTURAL MASS = 0.000000E+00
CONCENTRATED MASS = 0.000000E+00
OTHER MASS = 0.000000E+00

CENTER OF MASS RELATIVE TO ORIGIN OF BASIC SYSTEM

X-COORDINATE = -7.179453E-02
Y-COORDINATE = -6.545151E-01
Z-COORDINATE = 2.012342E-05

WRITING OUT GRID POINT MASS PROPERTIES
PAGES WRITTEN: 1

CALCULATING PART DEFINITIONS
PERCENT COMPLETE: 100

WRITING OUT PART DEFINITIONS
PAGES WRITTEN: 1

LINEAR SOLUTION PROCESSOR MODULE

SOLUTION SEQUENCE FOR SUBCASE 1

FETCHING GLOBAL STIFFNESS MATRIX
PERCENT COMPLETE: 100

MAXIMUM STIFFNESS MATRIX DIAGONAL = 1.7264E+08 AT GRID 984
MINIMUM STIFFNESS MATRIX DIAGONAL = 1.7841E+06 AT GRID 924

ASSEMBLING GLOBAL LOAD VECTOR FOR SUBCASE 1

ASSEMBLING GRID POINT LOADS
PERCENT COMPLETE: 100

AVAILABLE VIRTUAL MEMORY = 4410258432 WORDS 33647.6 MB
AVAILABLE PHYSICAL MEMORY = 10335661568 WORDS 78854.8 MB

DRIVE C: DISK SPACE = 19371185152 WORDS 14739.4 MEGAB

SOLVING FOR DISPLACEMENTS FOR SUBCASE 1
PERCENT COMPLETE: 100

GLOBAL STIFFNESS MATRIX ITERATIVE SOLUTION STATISTICS

NUMBER OF SOLUTION ITERATIONS = 28
FINAL CONVERGENCE ERROR = 6.030E-01

FATAL ERROR E5004 STIFFNESS MATRIX SINGULAR OR NON-POSITIVE

VIRTUAL MEMORY = 4412497920 WORDS 33664.7 MEGABYTES
REAL MEMORY = 10338014208 WORDS 78872.8 MEGABYTES

DRIVE C: DISK SPACE = 19371184128 WORDS 147790.4 MEGAB

MODEL ANALYSIS TIME SUMMARY

TOTAL CPU TIME = 12.86 SECONDS
WALLCLOCK TIME = 12.96 SECONDS

EXECUTION TERMINATED DUE TO DISPLAYED ERROR(S)

TOTAL WARNINGS = 0
TOTAL FATAL ERRORS = 1

Ready

Home 3-point bending.iam X 3-point bending.iam X

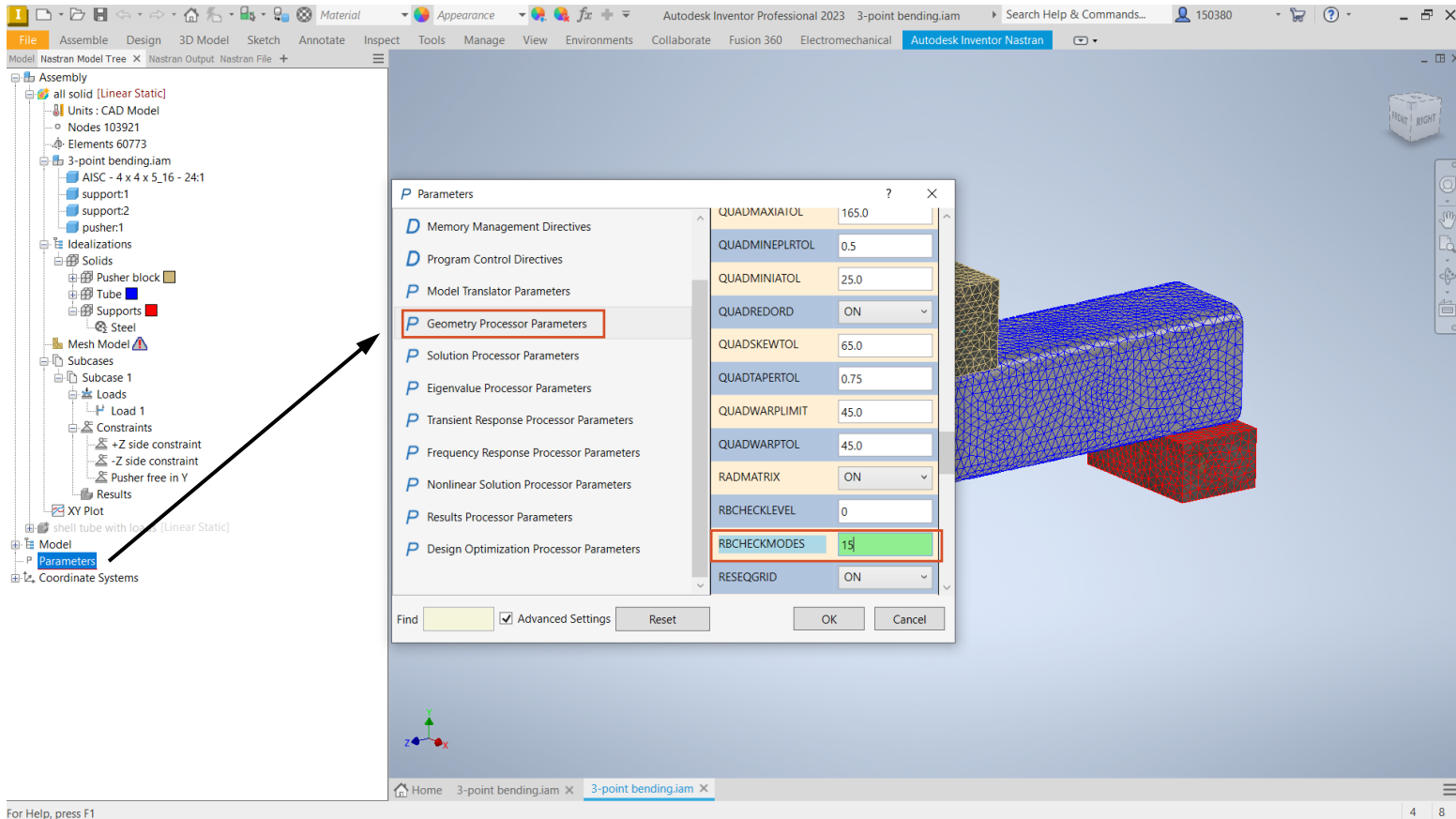
4 8

E5004

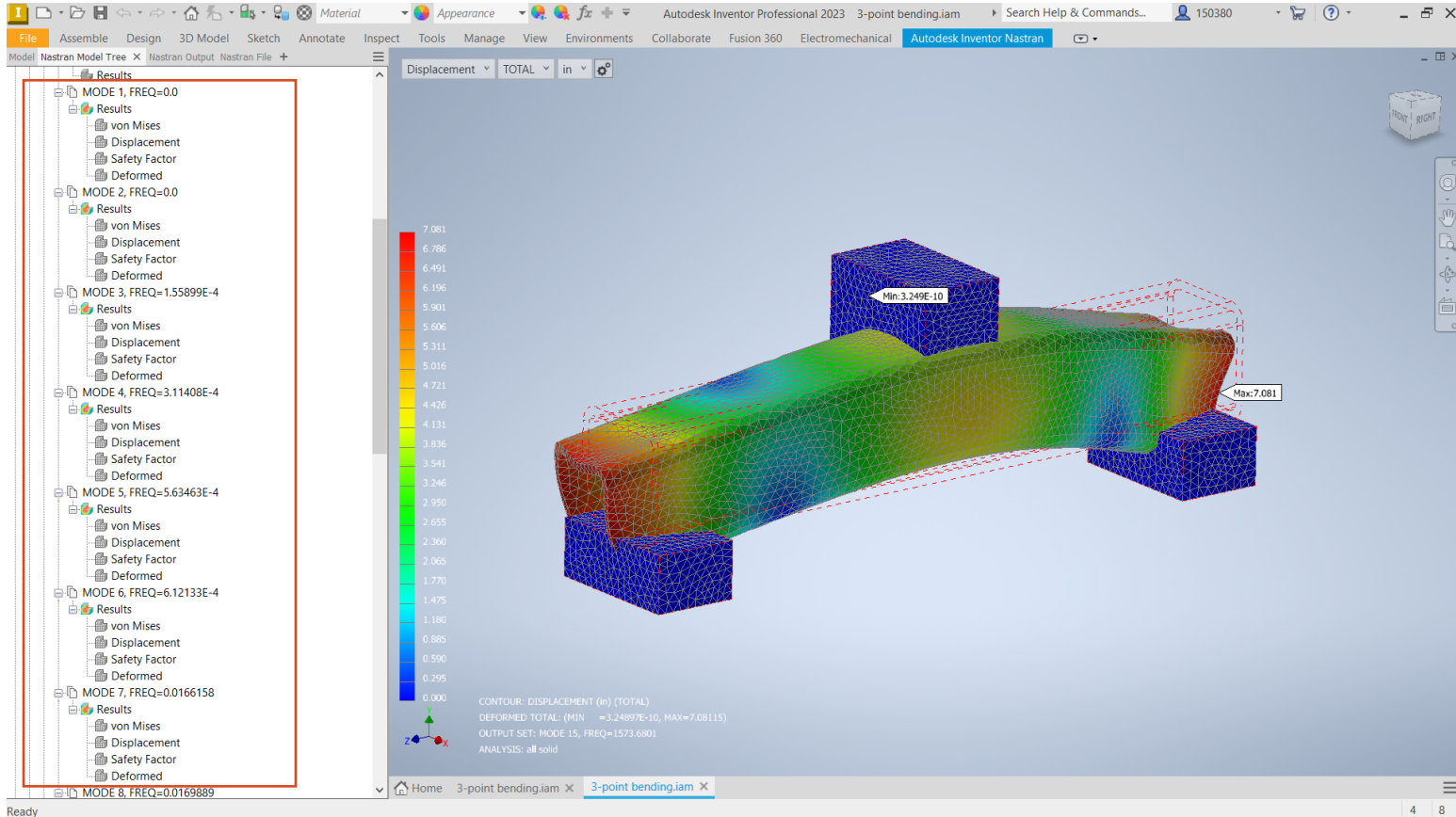
SHARE

E5004	FATAL ERROR: E5004	STIFFNESS MATRIX SINGULAR OR NON-POSITIVE DEFINITE
Cause:	A singularity or non-positive definite has been detected in the stiffness matrix during the preconditioning phase of the iterative solver.	
Action:	Investigate the model for a lack of constraint. If using shell elements, either set the K6ROT model parameter to a value between 1.0 and 100.0, or set the SHELLRNODE model parameter to ON. Also, check for elements with bending stiffness (line and shell elements) improperly connected to elements without (solids). If you are unable to locate the source of the singularity, consider using the sparse direct solver.	
Remarks:	See Autodesk Nastran Reference Guide, Section 5, Parameters , for more information on the K6ROT and SHELLRNODE parameters.	

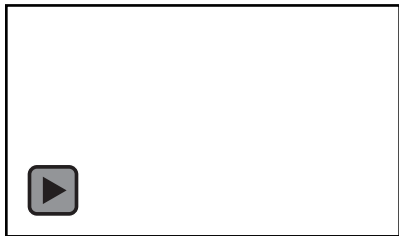
Isolating the Issue...RBCHECKMODES Parameter



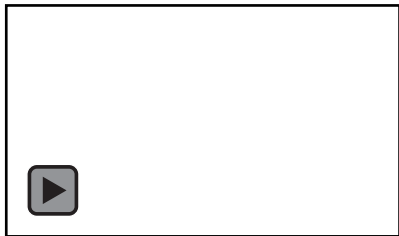
Isolating the Issue



Rigid Body Modes Results



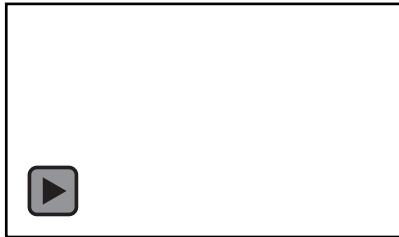
Mode 1 – Rigid Body



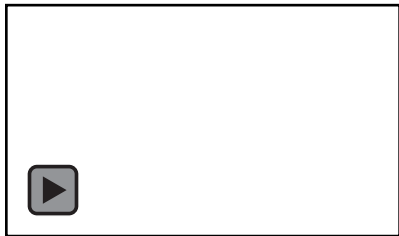
Mode 4 – Rigid Body



Mode 2 – Rigid Body



Mode 5 – Rigid Body



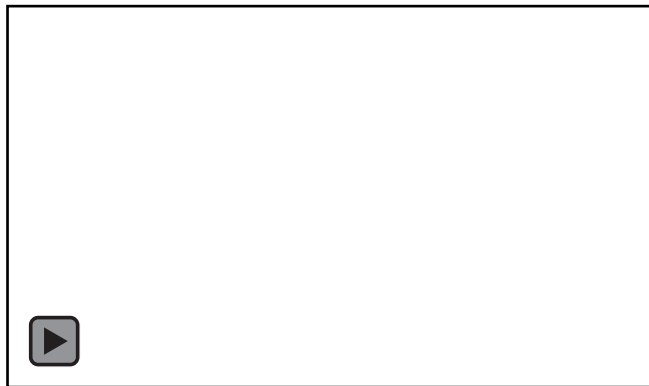
Mode 3 – Rigid Body



Mode 6 – Rigid Body



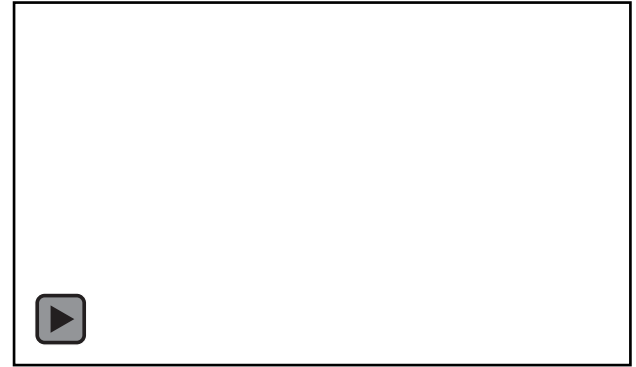
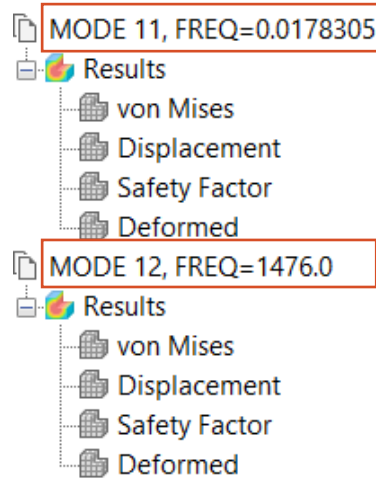
Mode 7 – Rigid Body – Flexible Expected



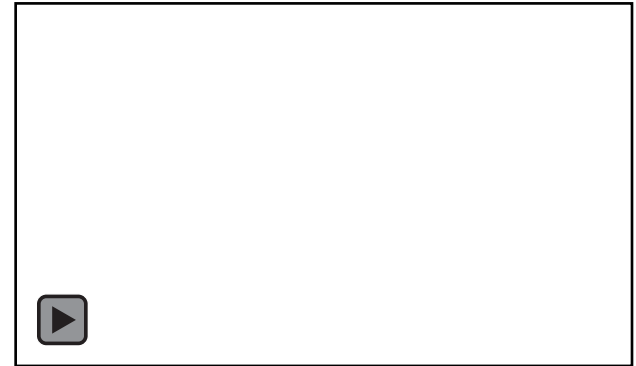
Mode 8 – Rigid Body – Flexible Expected

Modes 7 and On Should be Flexible Modes

- Modes 1-6 are rigid body modes as expected
- Modes 7-11 are also rigid body which means there is a lack of constraint in the assembly
- If the frequency is near zero, then it is most likely a rigid body mode

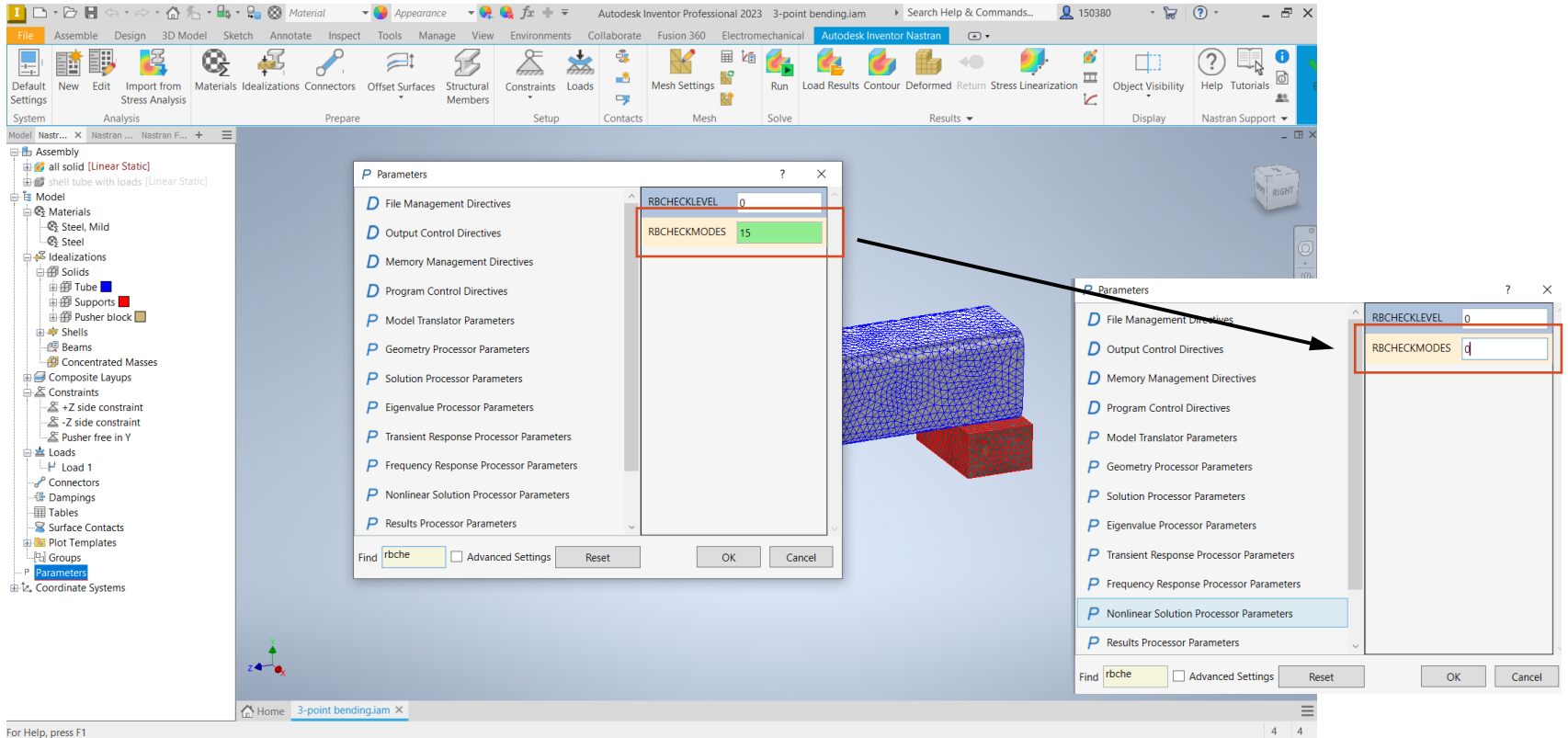


Mode 11 – Rigid Body

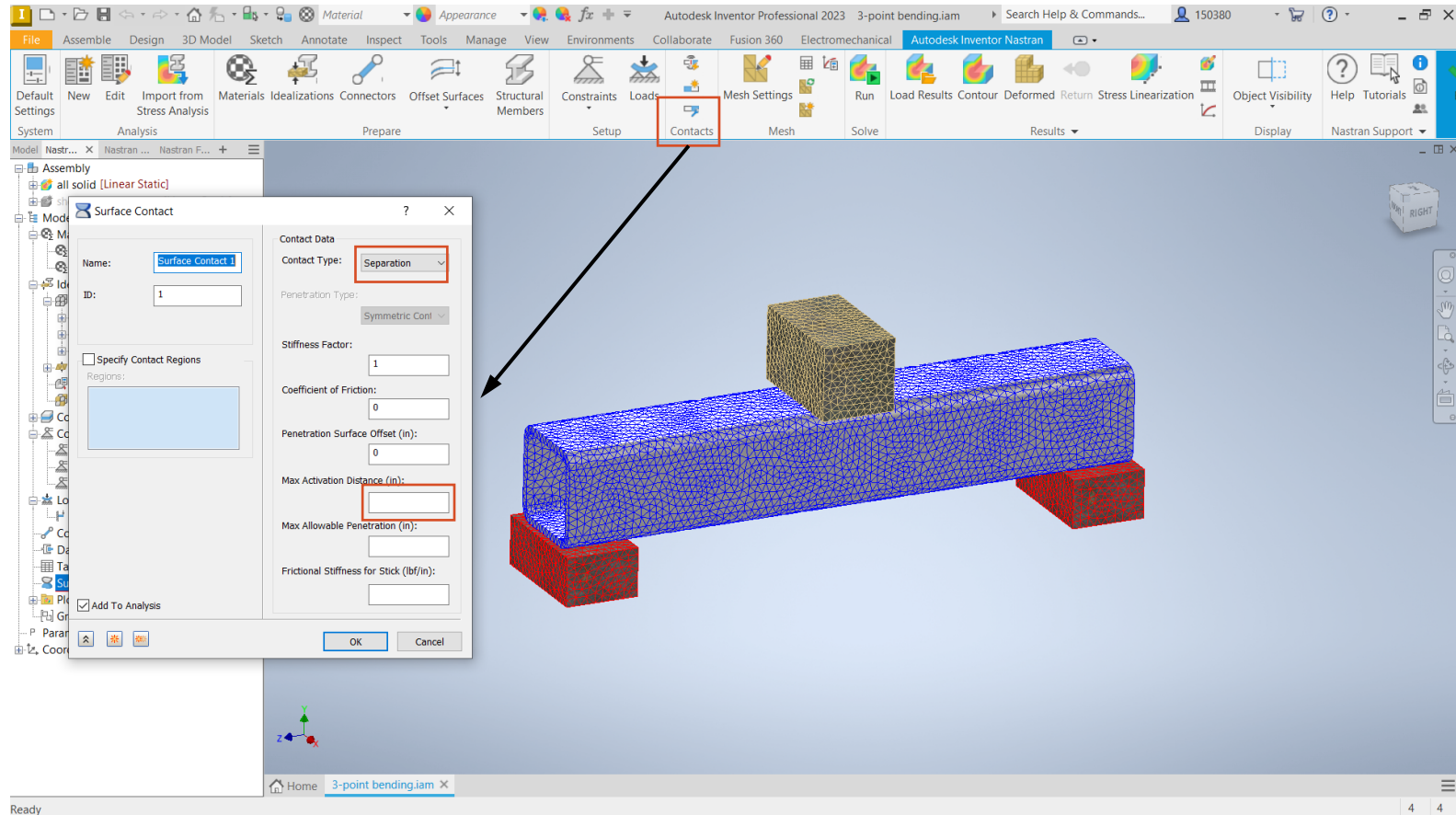


Mode 12 – Flexible

Remember to Zero the RBCHECKMODES Parameter



Setting Up Solver Based Contact



Running with Linear Statics – Linear Contact

RESULTS PROCESSOR MODULE

CALCULATING RESULTS FOR SUBCASE 1
WRITING GRID POINT DISPLACEMENTS FOR SUBCASE 1 TO FILE: ukc7k7jpe.
WRITING GRID POINT FORCES FOR SUBCASE 1 TO FILE: ukc7k7jpe.GPF
CALCULATING TET ELEMENT RESULTS IN VOLUME 0 FOR SUBCASE 1
PERCENT COMPLETE: 100

MAXIMUM TET ELEMENT PRINCIPAL STRESS = 2.801939E+03 AT ELEMENT 7
MINIMUM TET ELEMENT PRINCIPAL STRESS = -4.112149E+03 AT ELEMENT 7
MAXIMUM TET ELEMENT SHEAR STRESS = 1.799363E+03 AT ELEMENT 7
MAXIMUM TET ELEMENT VON MISES STRESS = 3.402773E+03 AT ELEMENT 3

MAXIMUM TET ELEMENT SAFETY FACTOR = 3.557295E+05 AT ELEMENT 4458
MINIMUM TET ELEMENT SAFETY FACTOR = 1.470506E+01 AT ELEMENT 3309

WRITING ELEMENT RESULTS FOR SUBCASE 1 TO FILE: ukc7k7jpe.ELS
CALCULATING ELEMENT RESULTS ON CONTACT SURFACE 1 FOR SUBCASE 1
PERCENT COMPLETE: 100

MAXIMUM CONTACT SURFACE NORMAL STRESS = 4.463511E+03 AT GRID 650
MINIMUM CONTACT SURFACE NORMAL STRESS = -1.730152E+02 AT GRID 618
MAXIMUM CONTACT SURFACE SHEAR STRESS = 4.667125E-06 AT GRID 650
MINIMUM CONTACT SURFACE SHEAR STRESS = 0.000000E+00 AT GRID 922

TOTAL PRIMARY CONTACT SURFACE NORMAL FORCE = 1.999515E+03
TOTAL SECONDARY CONTACT SURFACE NORMAL FORCE = 1.999515E+03
TOTAL PRIMARY CONTACT SURFACE SHEAR FORCE = 2.606635E-06
TOTAL SECONDARY CONTACT SURFACE SHEAR FORCE = 2.504042E-06

CALCULATING STRESS ERROR MEASURES IN VOLUME 0 FOR SUBCASE 1
PERCENT COMPLETE: 100

MAXIMUM SOLID ELEMENT NORMALIZED STRESS ERROR = 5.027688E-02 AT
MINIMUM SOLID ELEMENT NORMALIZED STRESS ERROR = 0.000000E+00 AT
SOLID ELEMENT RELATIVE STRESS ERROR = 5.027688E-02

DELETING FILE: ukc7k7jpe.NDB
GENERATING RESULTS NEUTRAL FILE
PERCENT COMPLETE: 100
DELETING FILE: ukc7k7jpe.ECD
MODEL ANALYSIS TIME SUMMARY

TOTAL CPU TIME = 140.84 SECONDS
WALLCLOCK TIME = 90.12 SECONDS
EXECUTION TERMINATED NORMALLY

TOTAL VARINGS = 125
TOTAL FATAL ERRORS = 0

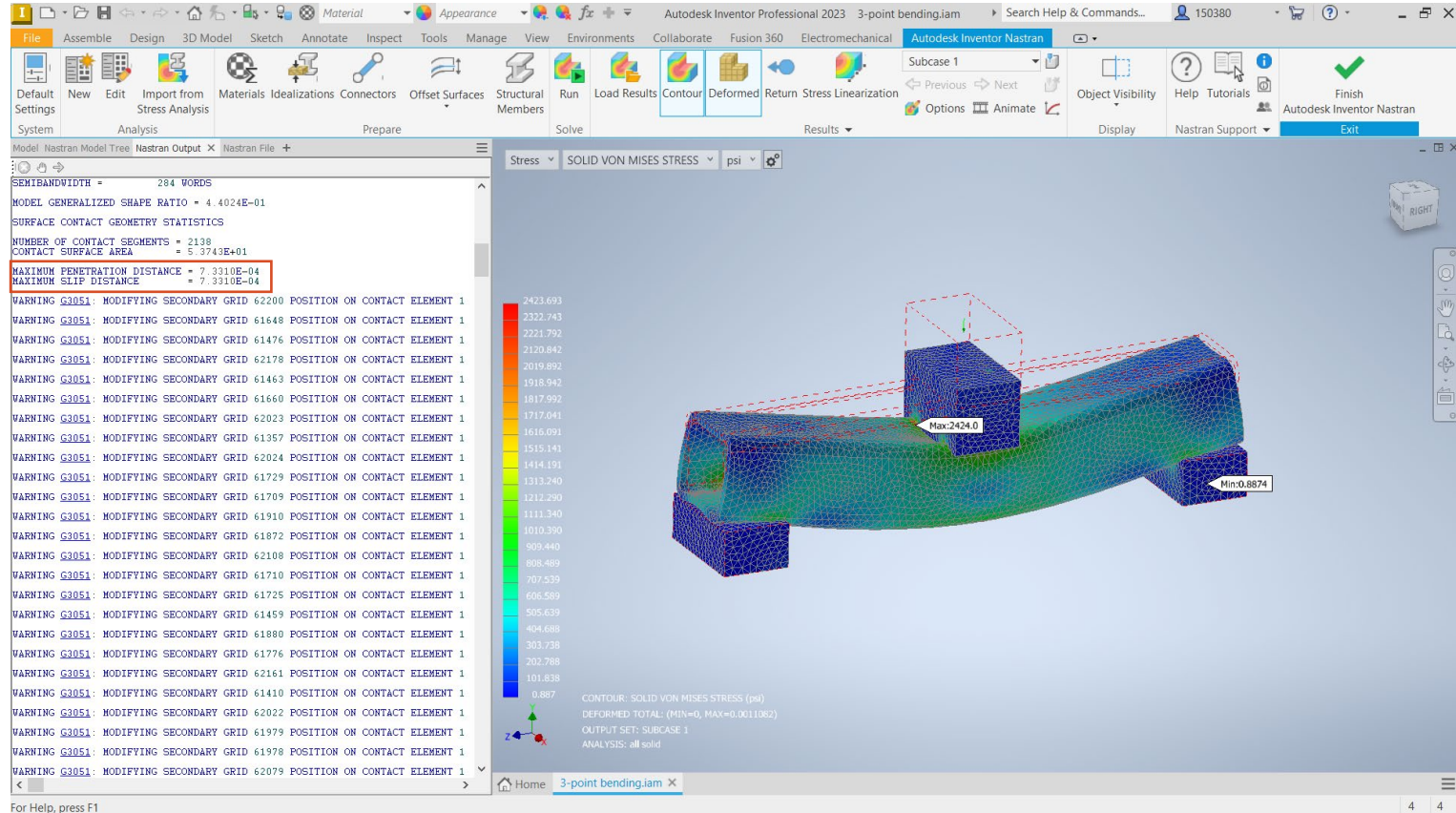
Autodesk Inventor Nastran
Nastran Solution Complete.
OK

3-point bending.iam

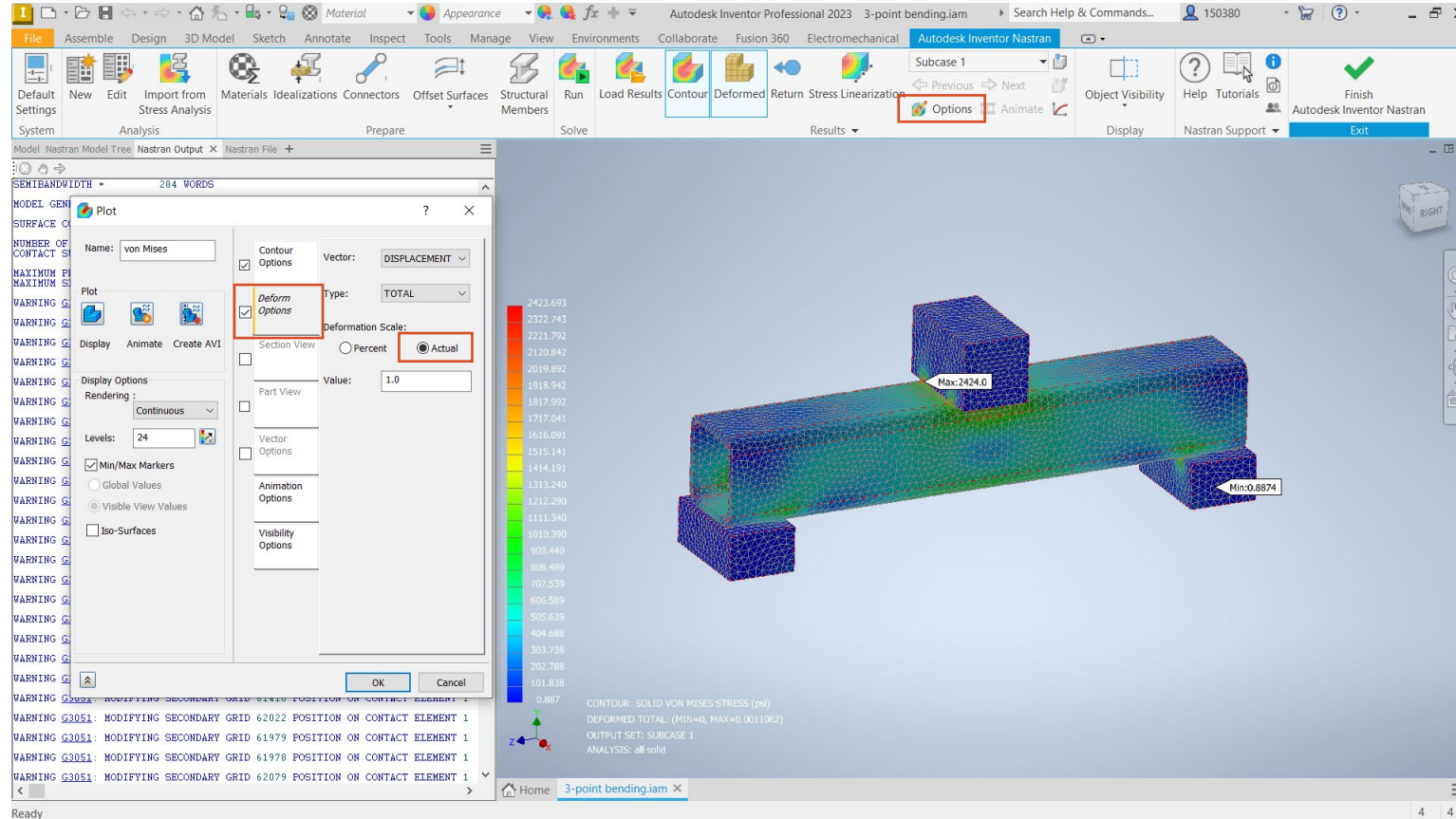
For Help, press F1

4 4

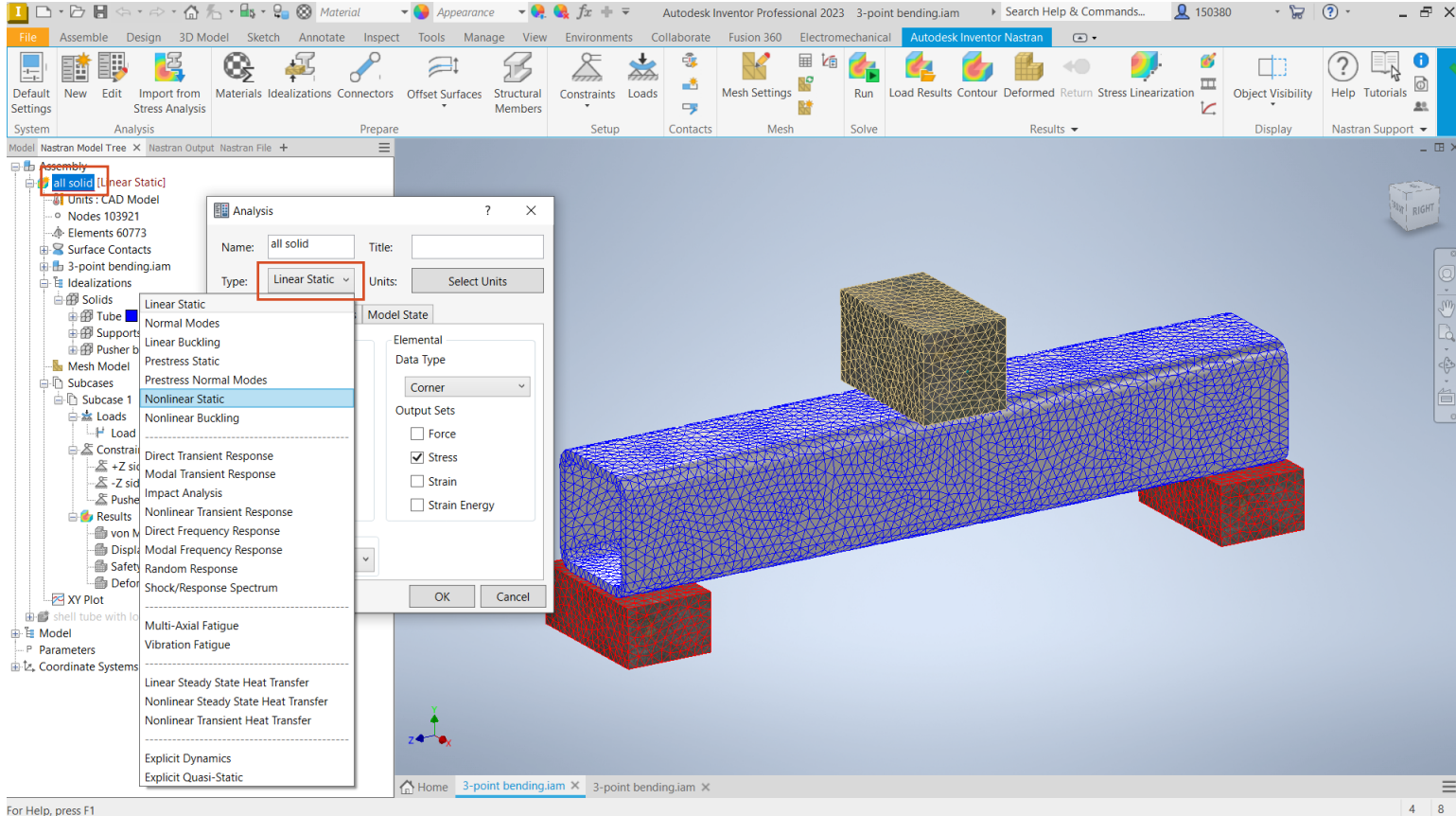
Review Warning Messages



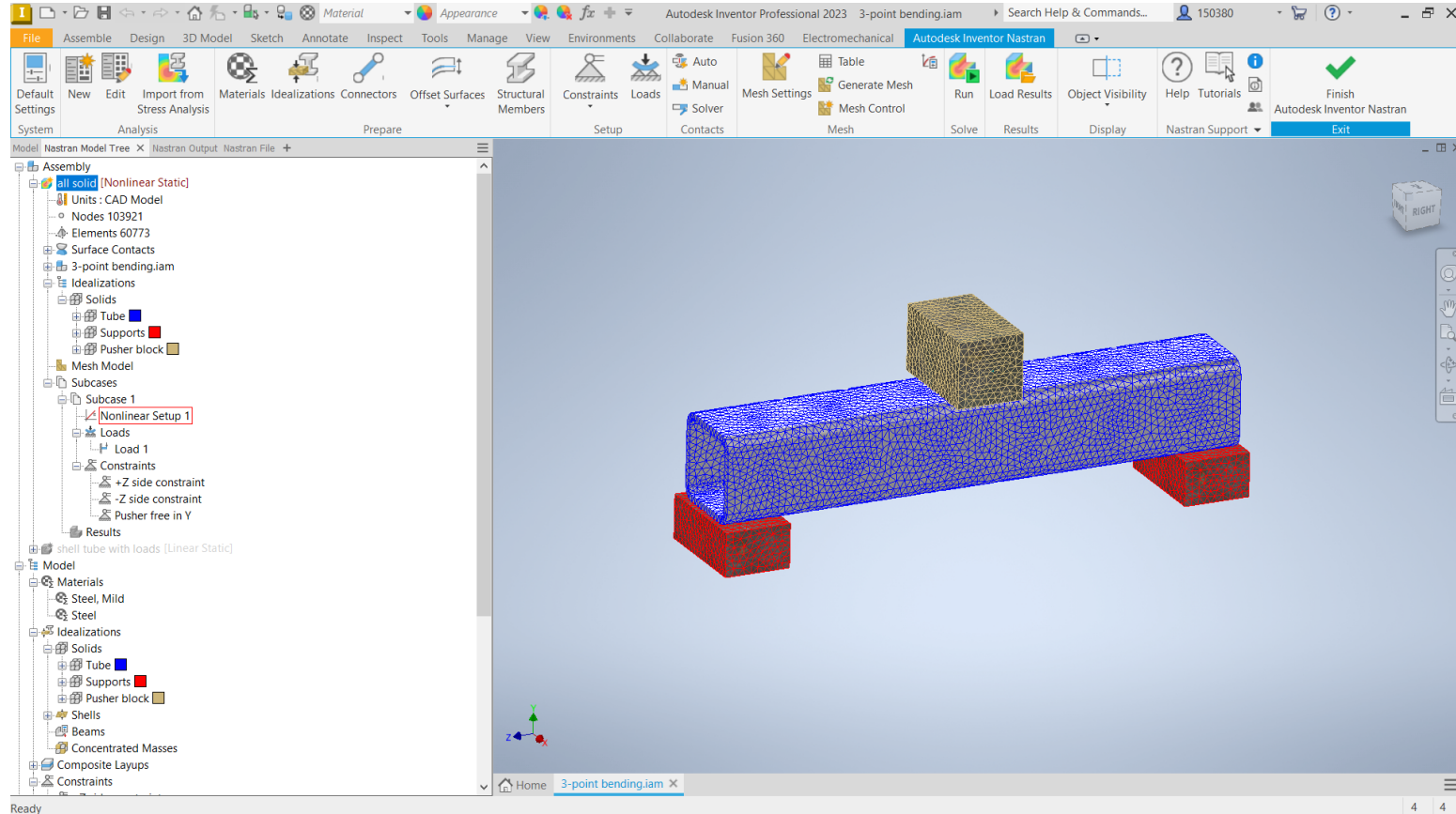
View Deformed Shape Using Actual Deformations



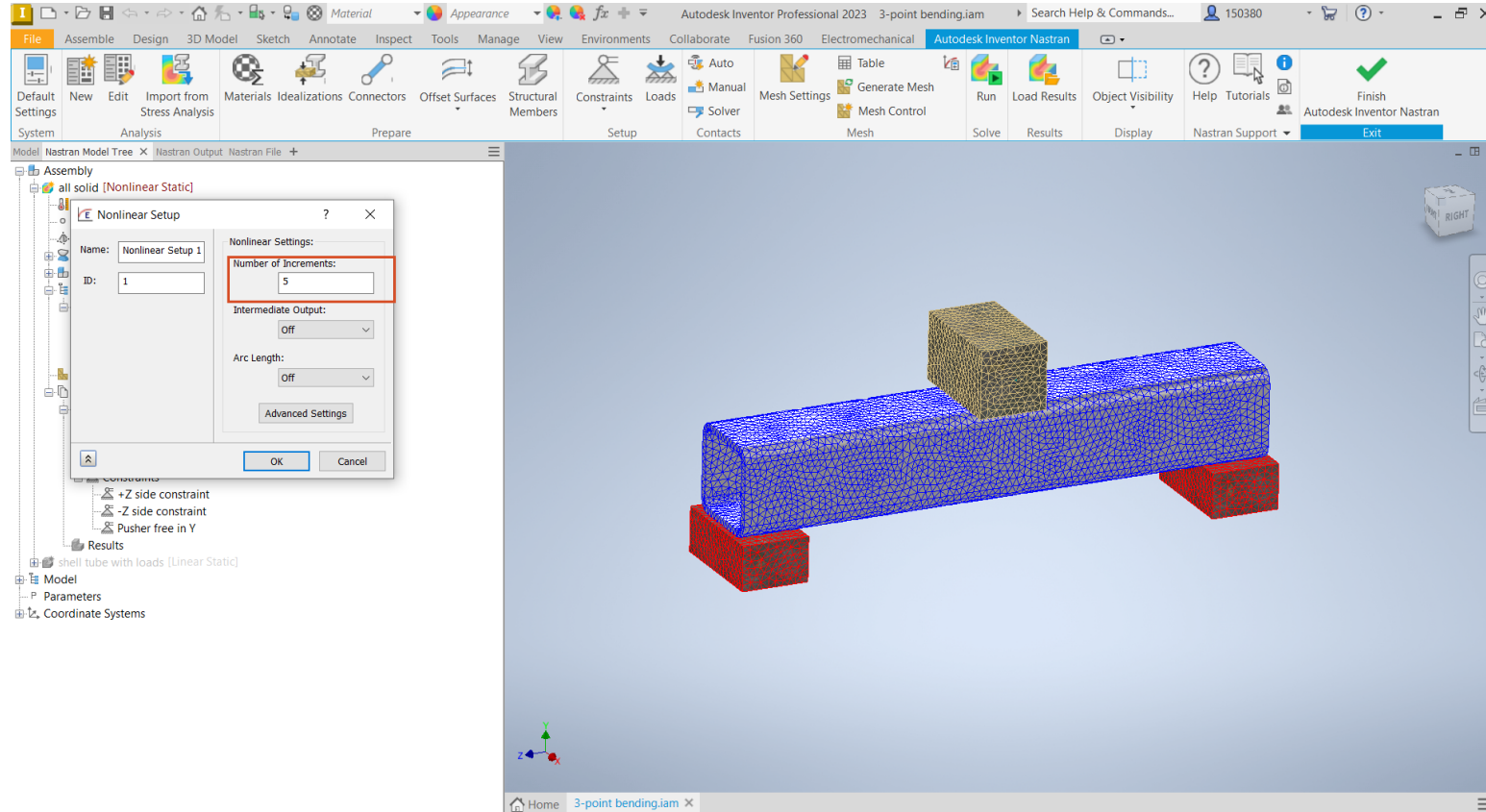
Running Nonlinear Statics



Setting Up Nonlinear Statics



Setting Up Nonlinear Statics



Nonlinear Statics Fails?

Autodesk Inventor Professional 2023 3-point bending.iam

File Assemble Design 3D Model Sketch Annotate Inspect Tools Manage View Environments Collaborate Fusion 360 Electromechanical Autodesk Inventor Nastran

Default Settings New Edit Import from Stress Analysis Materials Idealizations Connectors Offset Surfaces Structural Members Constraints Loads Auto Manual Solver Mesh Settings Generate Mesh Mesh Control Stop Load Results Object Visibility Help Tutorials Nastran Support Finish Autodesk Inventor Nastran Exit

Model Nastran Model ... Nastran Du... X Nastran File +

REORDERING METHOD REQUESTED = AUTO
REORDERING METHOD USED = VRM7
FACTORED SPARSE MATRIX SIZE = 135062856 WC
ADDITIONAL MEMORY ALLOCATED = 200523776 WC
WARNING E5042: SINGULARITY DETECTED AT GRID 186
FACTORIZATION TIME FOR 135062856 WORDS = 4.34 S
WARNING E5022: DIFFERENTIAL STIFFNESS MATRIX IC
ASSEMBLING GLOBAL STIFFNESS MATRIX FOR INCREMENT
PERCENT COMPLETE: 100
GLOBAL STIFFNESS MATRIX ANALYSIS STATISTICS
SPARSE MATRIX SIZE = 11931352 WORDS 91
MEMORY ALLOCATED = 17897028 WORDS 136
MAXIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =
MINIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =
REDUCTION IN GLOBAL STIFFNESS MATRIX SIZE =
ASSEMBLY TIME FOR 62911 ELEMENTS = 3.09 SECONDS
AVAILABLE VIRTUAL MEMORY = 4352390656 WORDS
AVAILABLE PHYSICAL MEMORY = 9154347008 WORDS
DRIVE C: DISK SPACE = 18570971136 WORDS 1
FACTORIZING GLOBAL STIFFNESS MATRIX FOR INCREMENT
PERCENT COMPLETE: 100
GLOBAL STIFFNESS MATRIX FACTORIZATION STATISTIC
NUMBER OF NEGATIVE TERMS ON FACTOR DIAGONAL = C
REORDERING METHOD REQUESTED = AUTO
REORDERING METHOD USED = VRM7
FACTORED SPARSE MATRIX SIZE = 135062856 WC
ADDITIONAL MEMORY ALLOCATED = 200523776 WC
FATAL ERROR E5000: SINGULARITY DETECTED AT GRID 186
FACTORIZATION TIME FOR 135062856 WORDS = 4.46 S
DELETING FILE: uke7k7jpe.ECD
MODEL ANALYSIS TIME SUMMARY
TOTAL CPU TIME = 233.72 SECONDS
WALLCLOCK TIME = 121.21 SECONDS
EXECUTION TERMINATED DUE TO DISPLAYED ERROR(S)
TOTAL WARNINGS = 131
TOTAL FATAL ERRORS = 1

Nonlinear Status

Subcase: 1
Increment: 1
Iteration: 10
Displacement:
Load:
Work:

Autodesk Inventor Nastran

Nastran Solution Failed.

OK

Home 3-point bending.iam X

For Help, press F1

Nonlinear Statics Fails?

Model Nastran Model ... X Nastran File +

```
REORDERING METHOD REQUESTED = AUTO
REORDERING METHOD USED = VRM7
FACTORED SPARSE MATRIX SIZE = 135062856 WC
ADDITIONAL MEMORY ALLOCATED = 200523776 WC
WARNING E5042: SINGULARITY DETECTED AT GRID 186
FACTORIZATION TIME FOR 135062856 WORDS = 4.34 S
WARNING E5022: DIFFERENTIAL STIFFNESS MATRIX IC
ASSEMBLING GLOBAL STIFFNESS MATRIX FOR INCREME
PERCENT COMPLETE: 100
GLOBAL STIFFNESS MATRIX ASSEMBLY STATISTICS
SPARSE MATRIX SIZE = 11931352 WORDS 91.
MEMORY ALLOCATED = 17897028 WORDS 136.
MAXIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =
MINIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =
REDUCTION IN GLOBAL STIFFNESS MATRIX SIZE =
ASSEMBLY TIME FOR 62911 ELEMENTS = 3.09 SECONDS
AVAILABLE VIRTUAL MEMORY = 4352390656 WORDS
AVAILABLE PHYSICAL MEMORY = 9154347008 WORDS
DRIVE C: DISK SPACE = 18570971136 WORDS 1
FACTORIZING GLOBAL STIFFNESS MATRIX FOR INCREME
PERCENT COMPLETE: 100
GLOBAL STIFFNESS MATRIX FACTORIZATION STATISTIC
NUMBER OF NEGATIVE TERMS ON FACTOR DIAGONAL = C
REORDERING METHOD REQUESTED = AUTO
REORDERING METHOD USED = VRM7
FACTORED SPARSE MATRIX SIZE = 135062856 WC
ADDITIONAL MEMORY ALLOCATED = 200523776 WC
FATAL ERROR E5000: SINGULARITY DETECTED AT GRID
FACTORIZATION TIME FOR 135062856 WORDS = 4.46 S
DELETING FILE: uke7k7jpe.ECD
MODEL ANALYSIS TIME SUMMARY
TOTAL CPU TIME = 233.72 SECONDS
WALLCLOCK TIME = 121.21 SECONDS
EXECUTION TERMINATED DUE TO DISPLAYED ERROR(S)
TOTAL WARNINGS = 131
TOTAL FATAL ERRORS = 1
```

E5000 FATAL ERROR: E5000 SINGULARITY DETECTED AT GRID *id* COMPONENT *n*

Cause: A singularity has been detected in the stiffness matrix during decomposition.

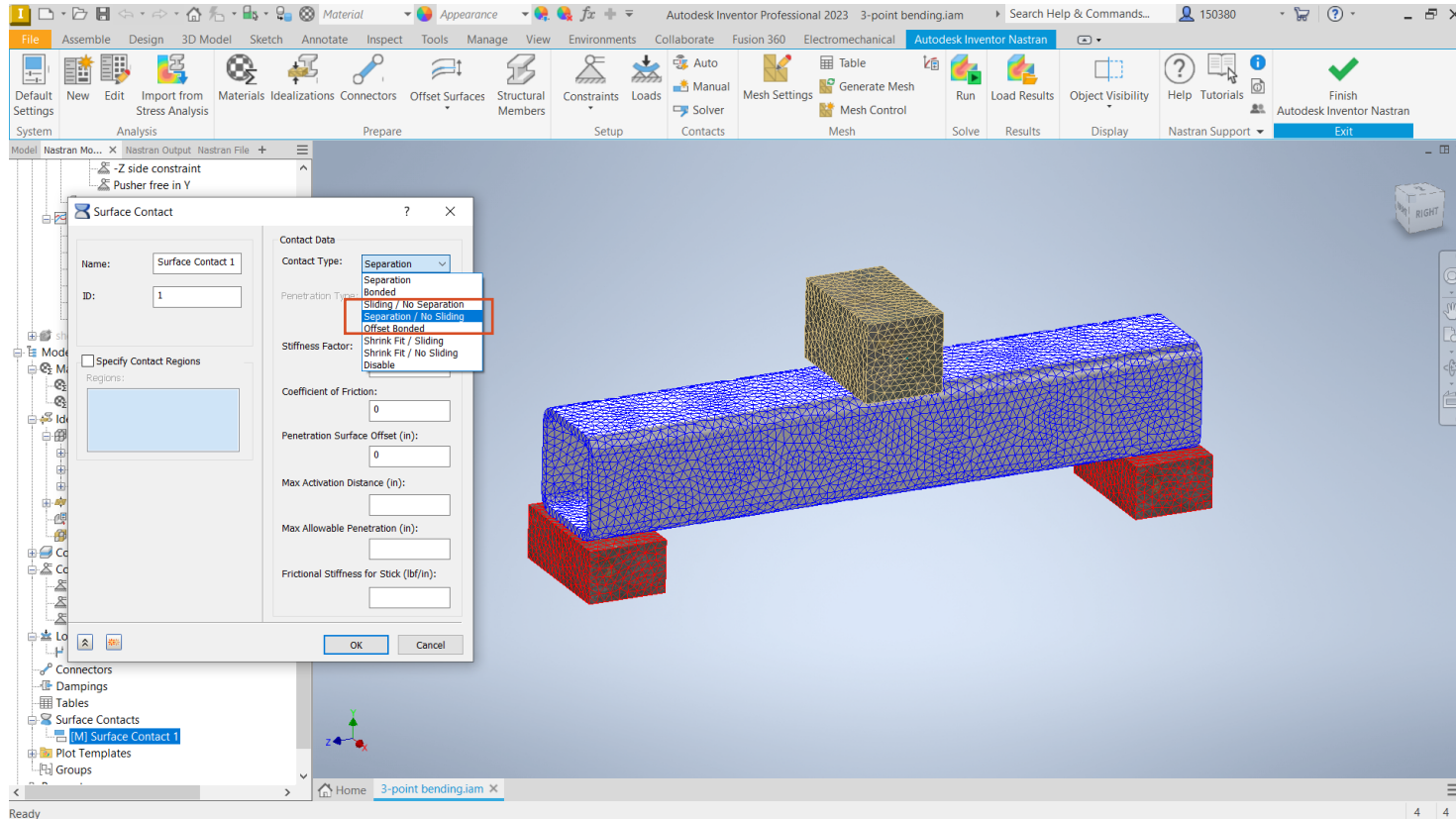
Action: Investigate the degree of freedom at which the singularity occurred for a lack of stiffness. Also, check your model for a lack of constraint. If using shell elements, either set the K6ROT model parameter to a value between 1.0 and 100.0, or set the SHELLRNODE model parameter to ON. If this does not solve the problem or you are unable to locate the source of the singularity, consider setting the SOLUTIONERROR model parameter to ON. Note that solution accuracy may be degraded with this parameter set to ON. Check the EPSILON value in the Model Results Output or Results Summary File. Values greater than 0.001 indicate a possible loss of accuracy.

Remarks: See Autodesk Nastran Reference Guide, Section 5, Parameters, for more information on the K6ROT, SHELLRNODE, and SOLUTIONERROR parameters.

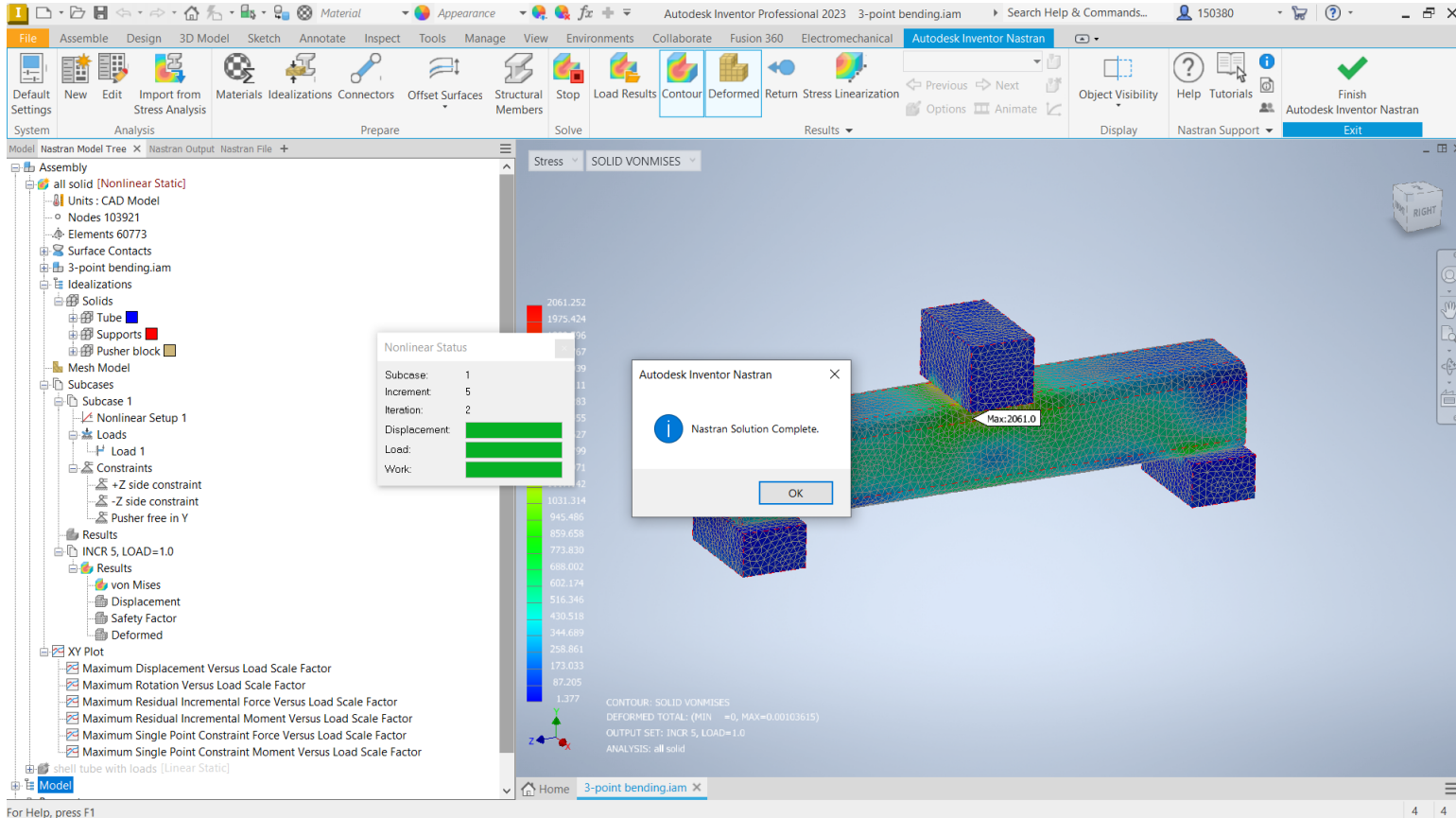
Home 3-point bending.iam X

For Help, press F1

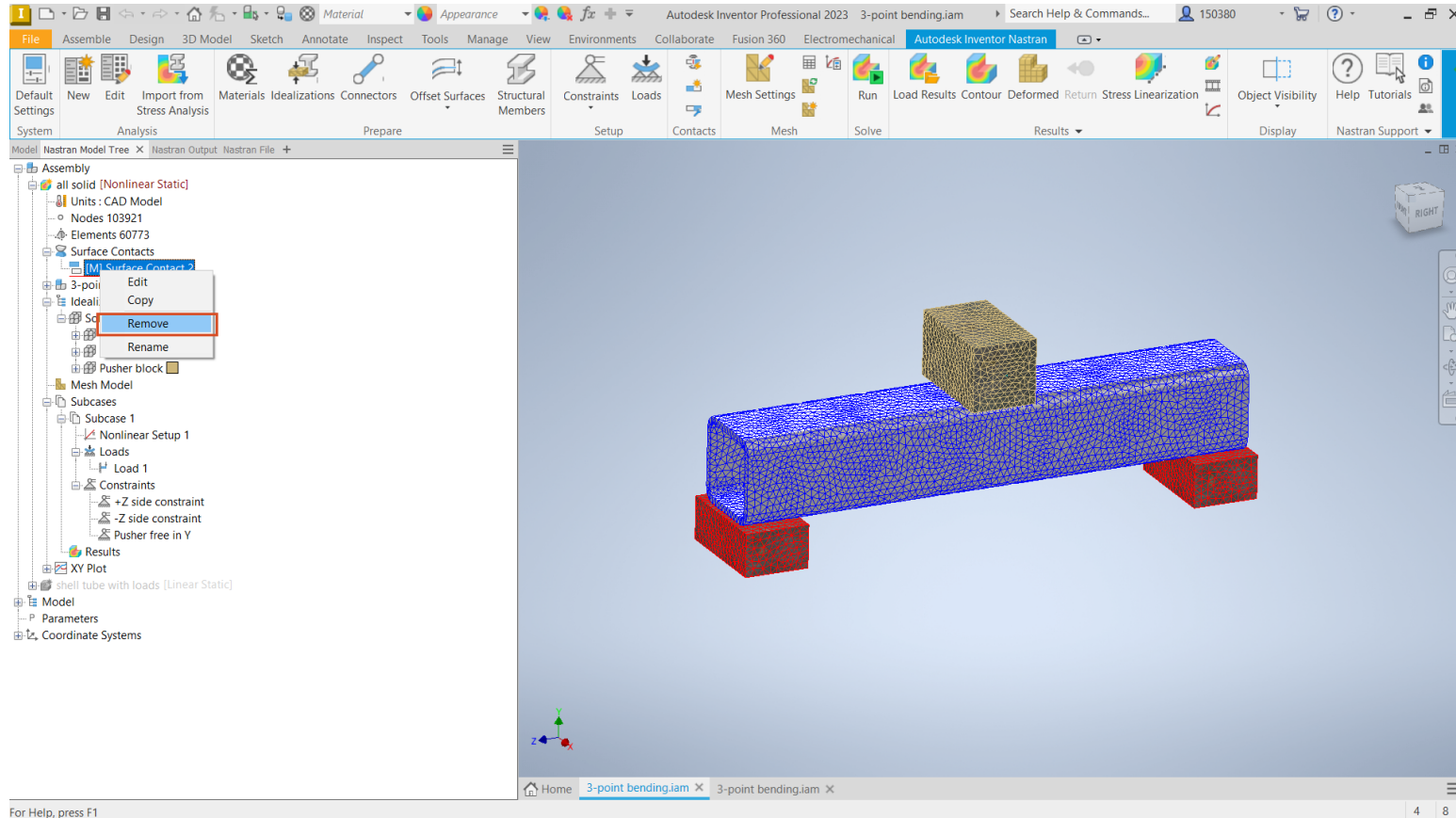
Setting Up Rough Contact to Prevent Sliding



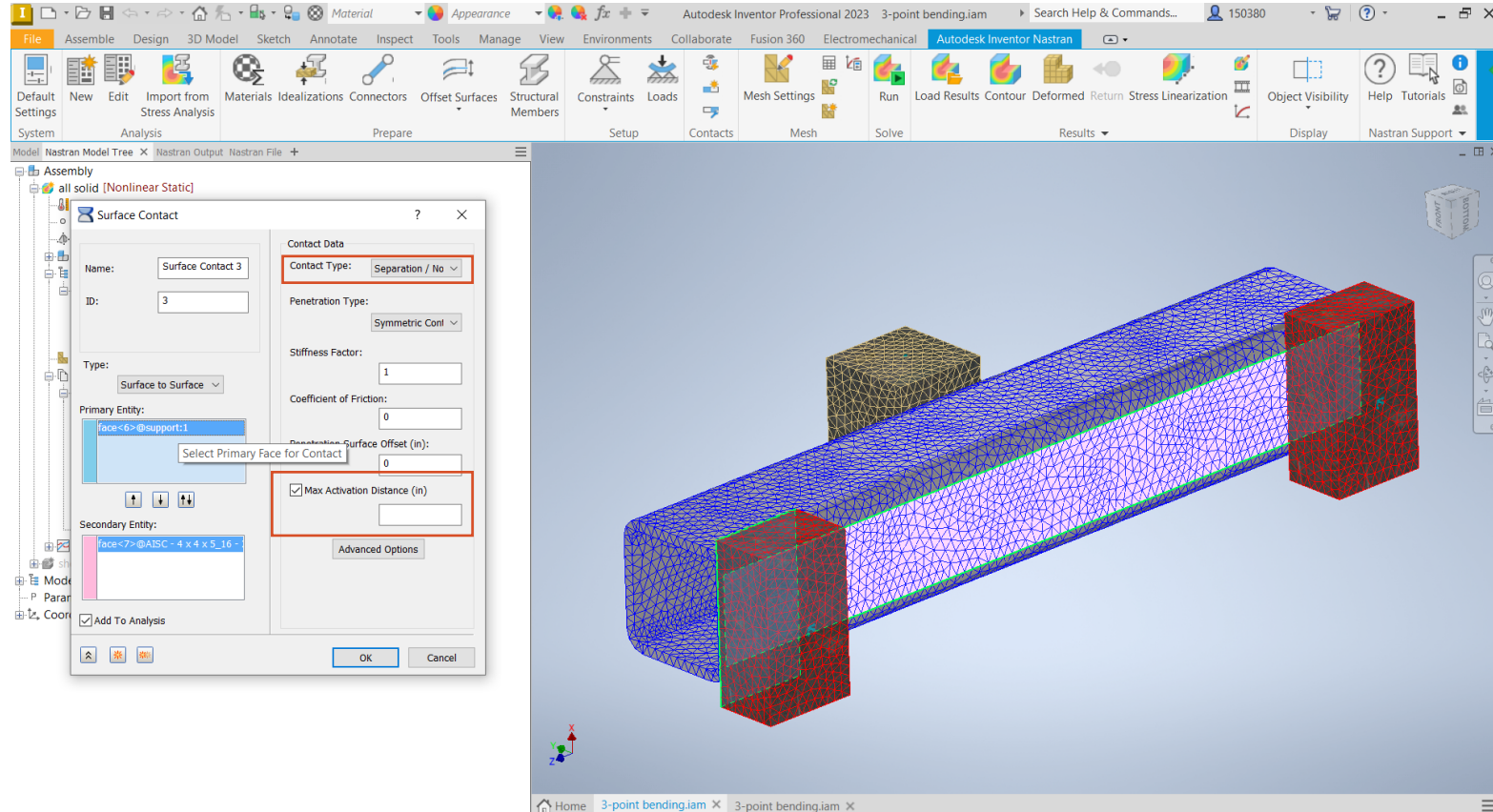
Nonlinear Statics with Rough Contact



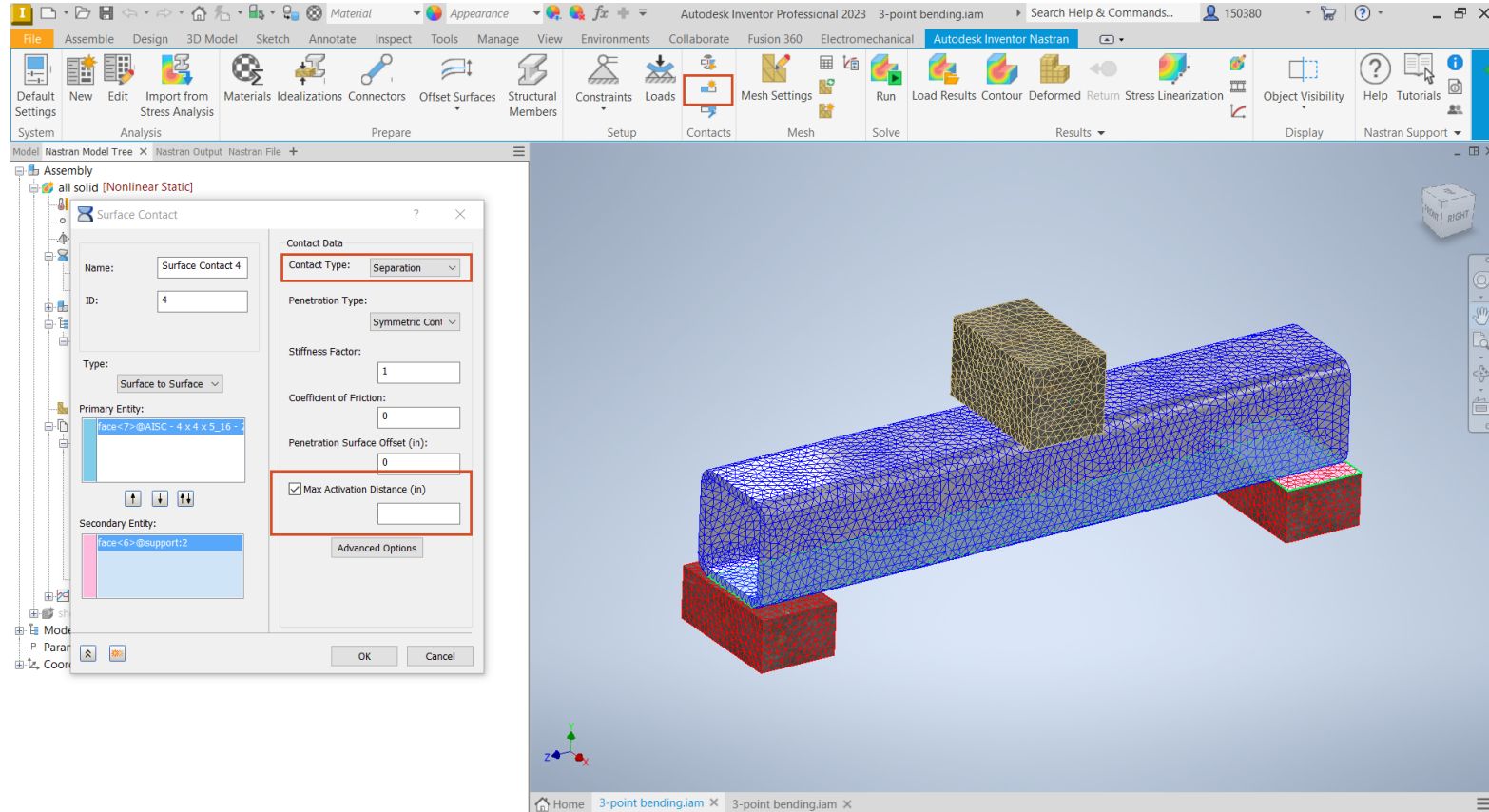
Setting Up Manual Contact – Delete Solver Contact



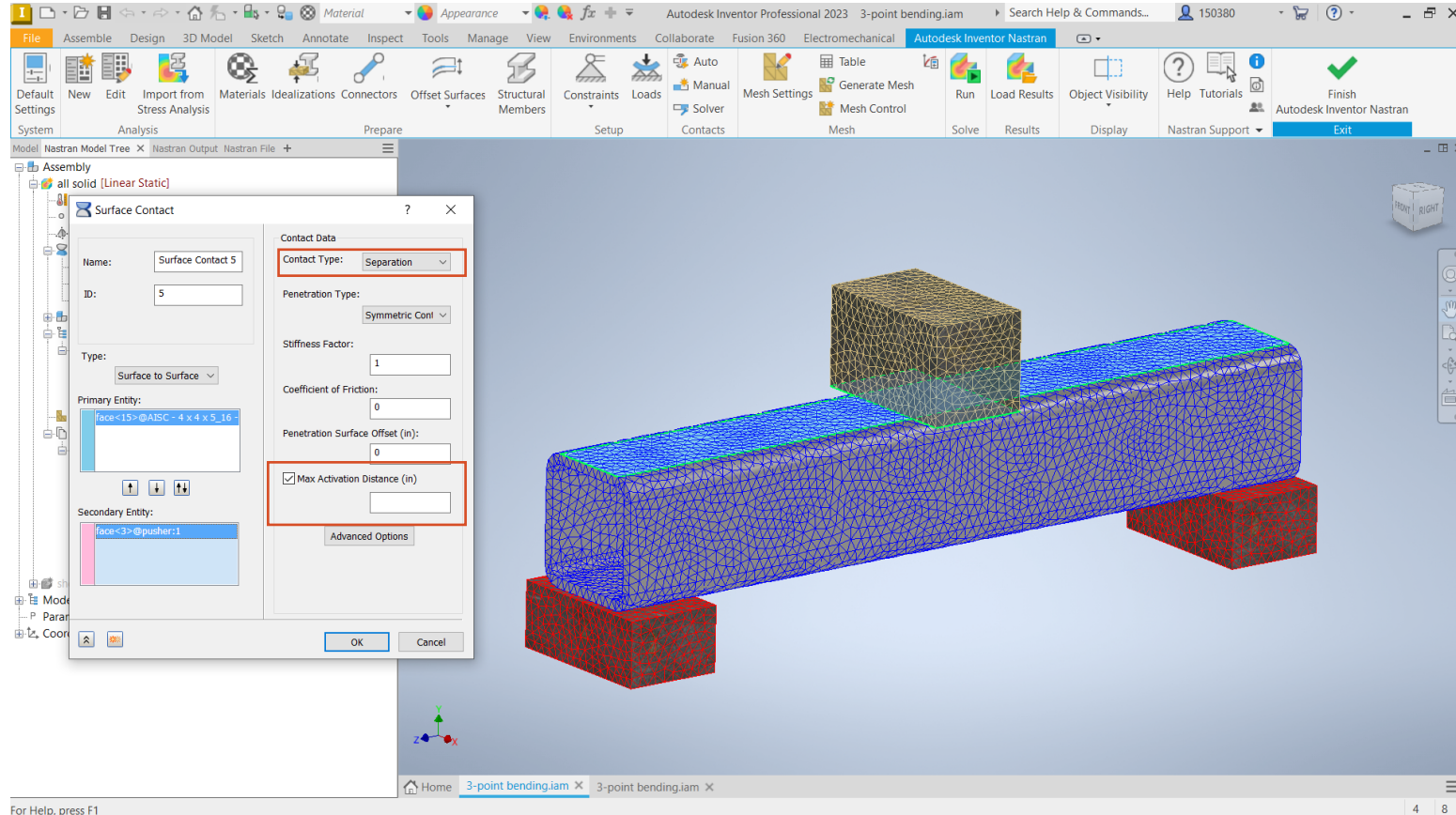
Setting Up +Z Support Manual Contact



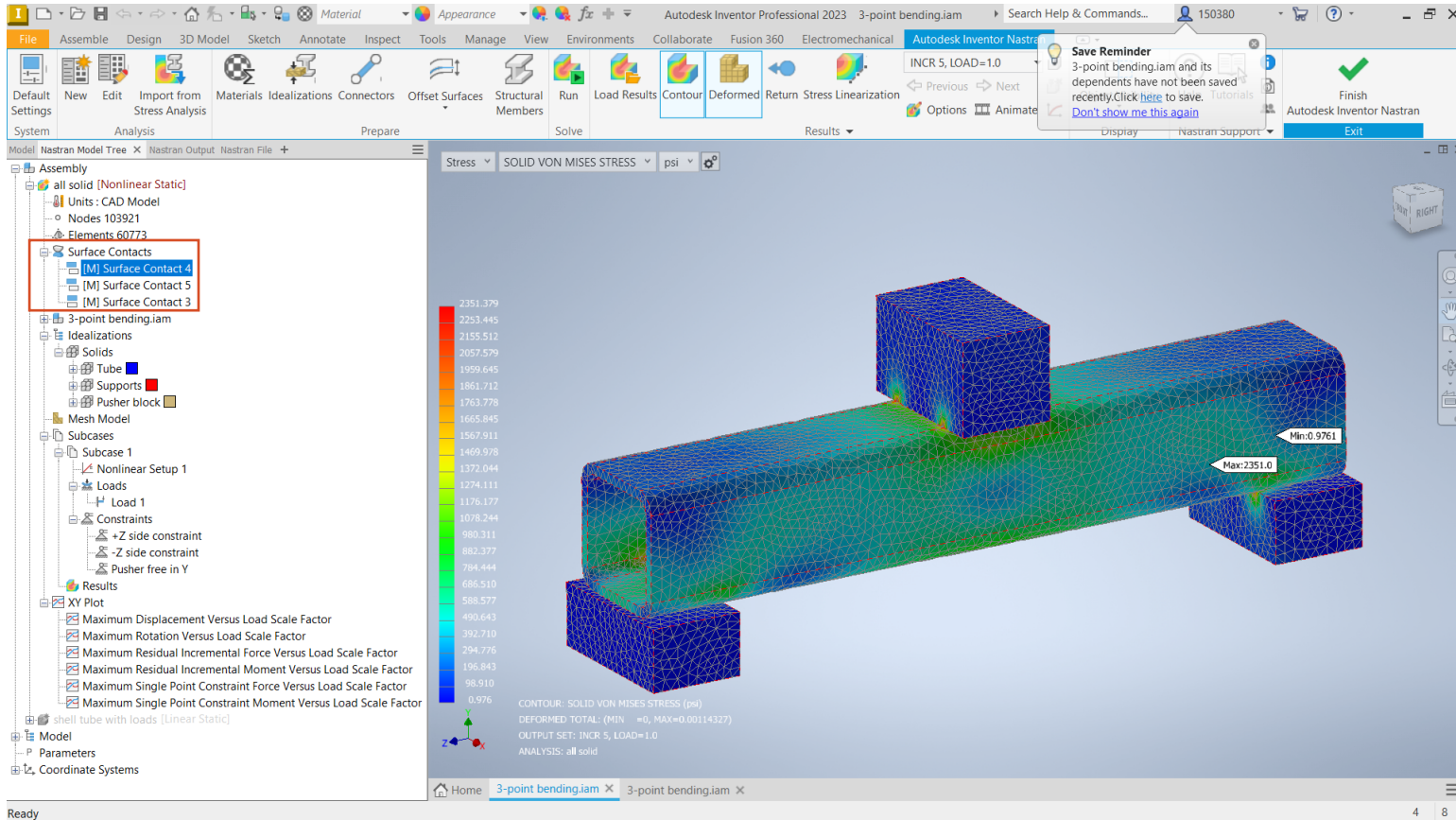
Setting Up -Z Support Manual Contact



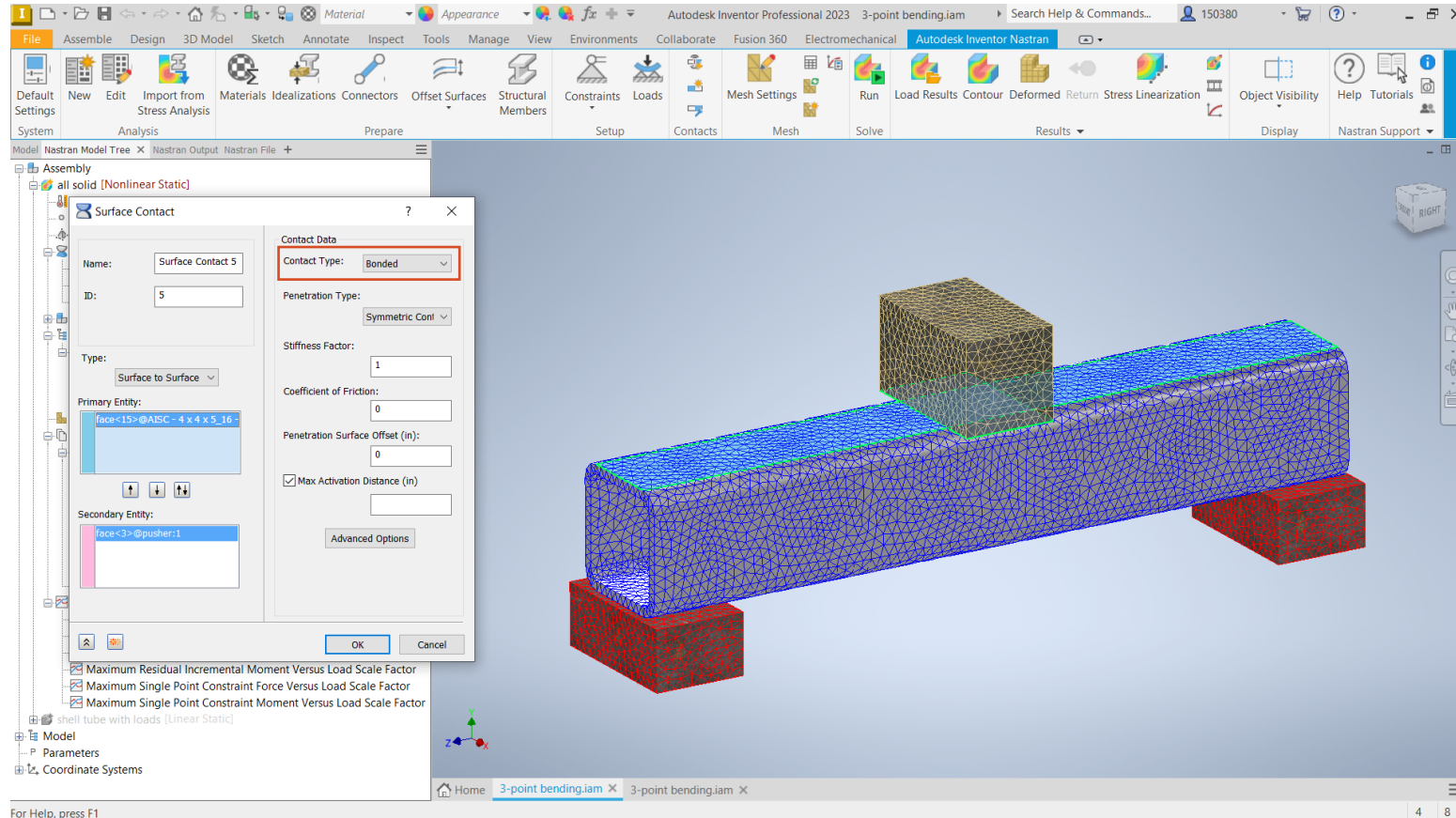
Setting Up Pusher Manual Contact



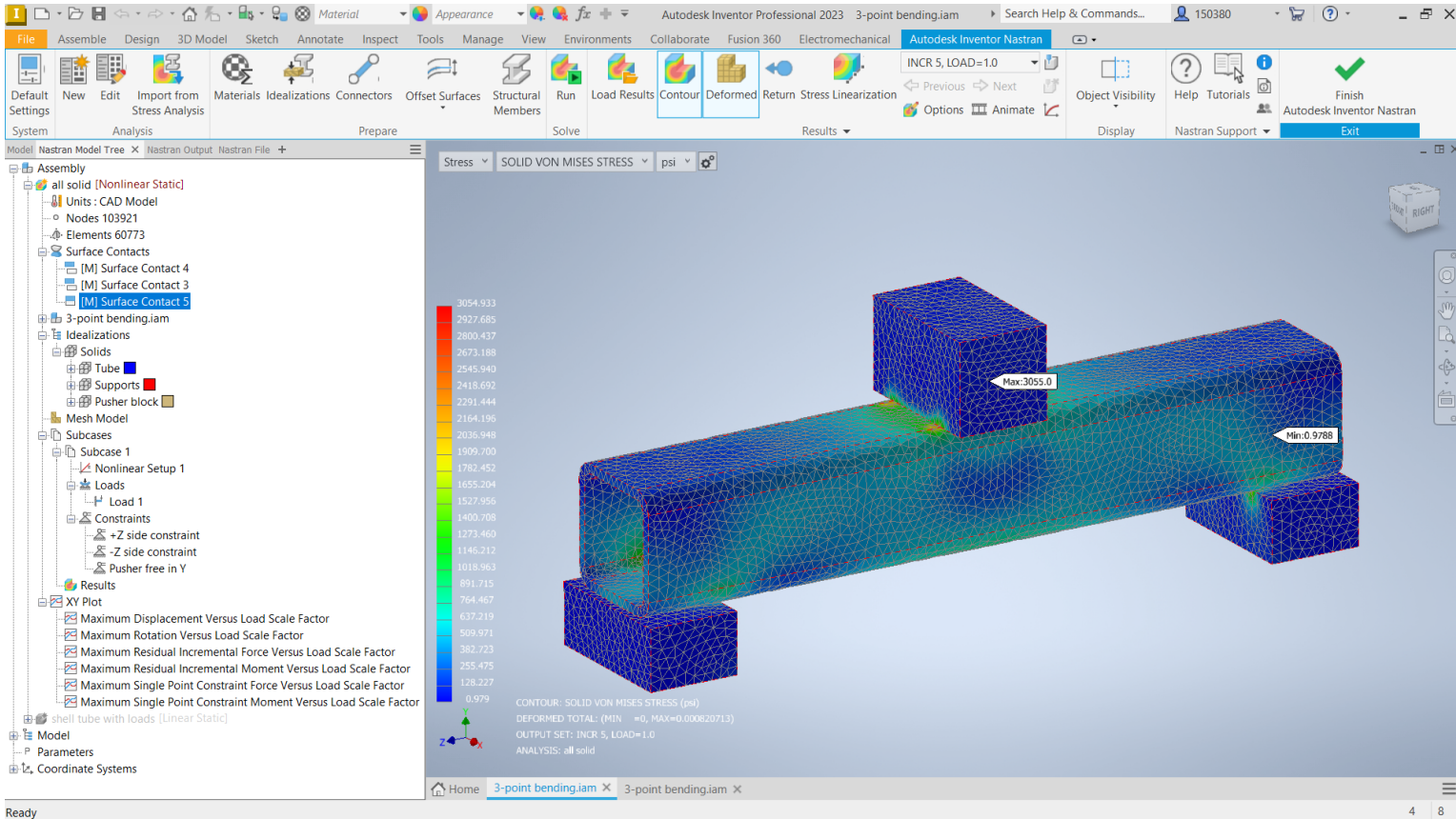
Nonlinear Solution with No Sliding on +Z Support



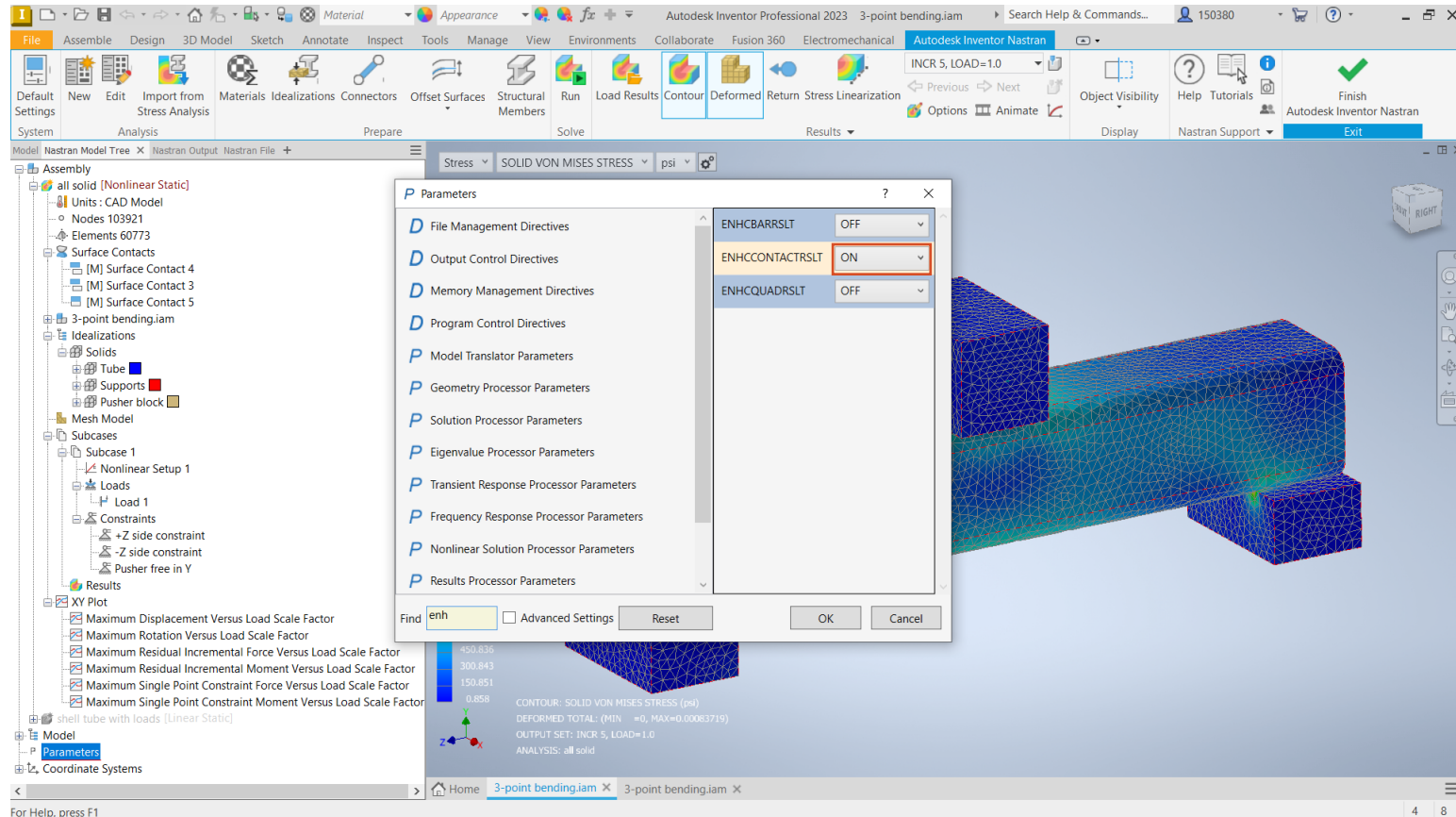
Welding Pusher to Tube



Nonlinear Solution with Welded Pusher

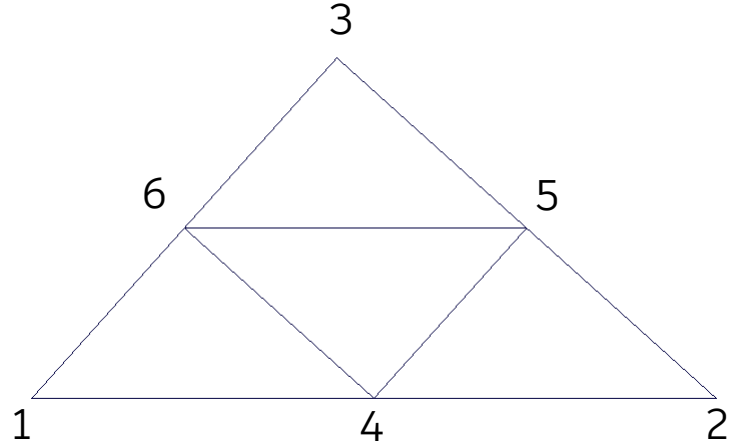


Enhanced Contact



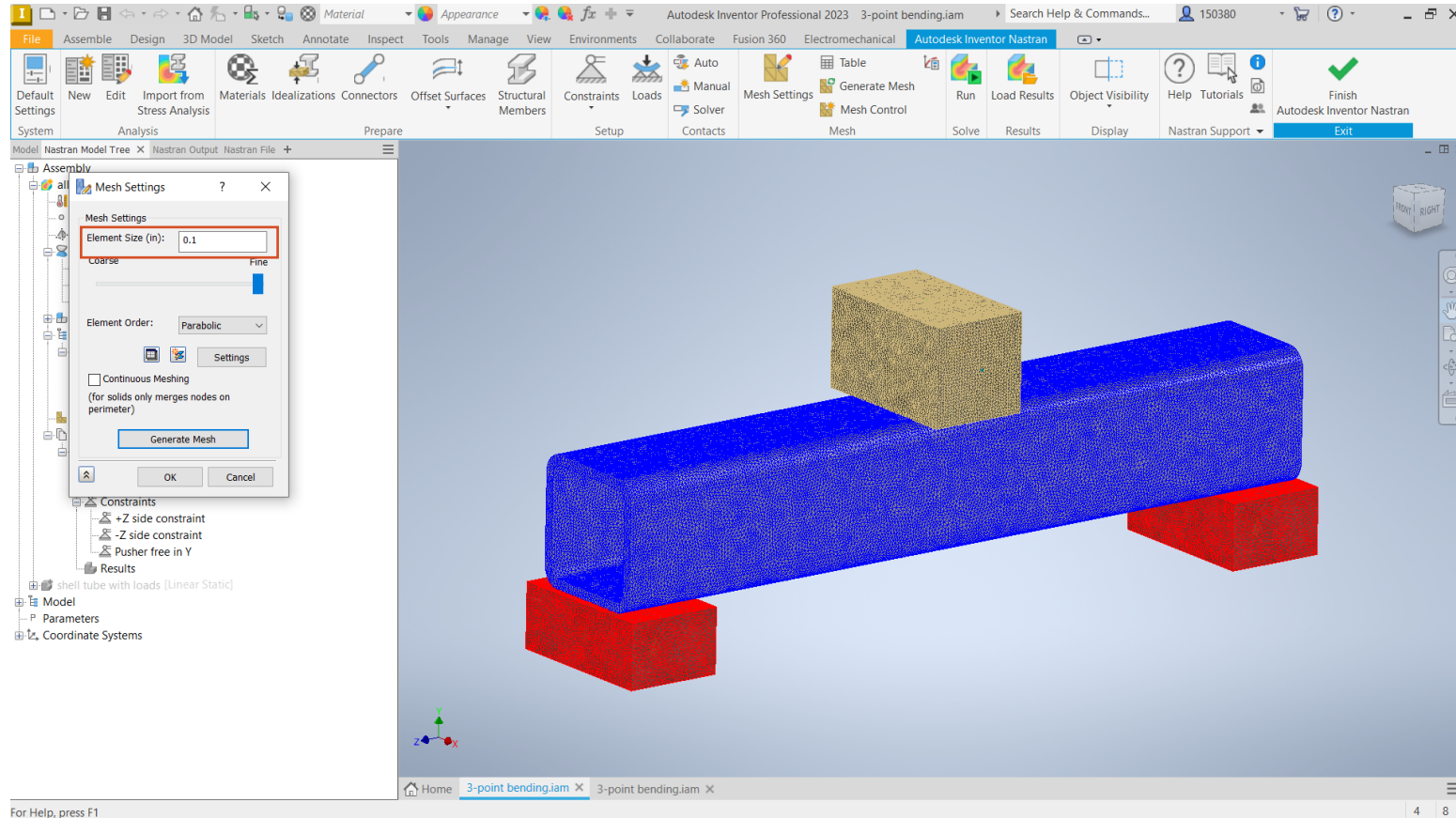
Enhanced Contact

- Enhanced contact is enabled with the ENHCCONTACTRSLT parameter
- Enabling this will improve accuracy for models with thin parts and only one or two TET elements through the thickness
- The big disadvantage is that performance will be much slower for larger models
- We are currently working on an improved enhanced contact which will provide better results and only a moderate degradation in performance





Using a Finer Mesh for Improved Results



Solution Fails Due to Model Size

The screenshot shows the Autodesk Inventor Professional 2023 interface with the Nastran module active. The Nastran Output window is open, displaying the following text:

```
LINES READ: 7626402
DETERMINING MODEL SIZE
PERCENT COMPLETE: 100
PROCESSING CASE CONTROL COMMANDS
PERCENT COMPLETE: 100
13 CASE CONTROL COMMANDS WERE RECOGNIZED OF A TOTAL OF 13 READ IN
PROCESSING BULK DATA ENTRIES
PERCENT COMPLETE: 100
7626388 BULK DATA ENTRIES WERE RECOGNIZED OF A TOTAL OF 7626388
MAXIMUM CONTACT RADIAL ACTIVATION DISTANCE = 1.0000E+00
MAXIMUM CONTACT NORMAL ACTIVATION DISTANCE = 1.0000E+00
MAXIMUM CONTACT ALLOWABLE PENETRATION DISTANCE = 3.9746E-01
WRITING OUT MODEL DATA
PAGES WRITTEN: 51

MODEL DATABASE SIZE
SUBCASES = 1
COORDINATE SYSTEMS = 4
GRID POINTS = 2233432
ELEMENTS = 32641878
ELEMENT PROPERTIES = 3
MATERIAL PROPERTIES = 2
SINGLE POINT CONSTRAINTS = 16251
GRID POINT FORCES = 4622
SETS = 4
SURFACES = 1
VOLUMES = 1

MODULE SEQUENCE FOR SOLUTION: NONLINEAR STATIC

GEOMETRY PROCESSOR MODULE

FATAL ERROR G3004: PROFILE LIMIT EXCEEDED
VIRTUAL MEMORY = 3956073216 WORDS 30188.5 MEGABYTES
REAL MEMORY = 8916679680 WORDS 68028.9 MEGABYTES
DRIVE C: DISK SPACE = 15525443584 WORDS 118449.7 MEGABYTES

MODEL ANALYSIS TIME SUMMARY
TOTAL CPU TIME = 766.22 SECONDS
WALLCLOCK TIME = 771.63 SECONDS

EXECUTION TERMINATED DUE TO DISPLAYED ERROR(S)

TOTAL WARNINGS = 0
TOTAL FATAL ERRORS = 1
```

The 3D model in the background shows a blue beam with a yellow block on top, supported by two red blocks. A dialog box titled "Autodesk Inventor Nastran" with the message "Nastran Solution Failed." and an "OK" button is overlaid on the model.

For Help, press F1

Enhanced Contact/Max Activation Distance/Fine Mesh

Autodesk Inventor Professional 2023 3-point bending.iam

File Assemble Design 3D Model Sketch Annotate Inspect Tools Manage View Environments Collaborate Fusion 360 Electromechanical Autodesk Inventor Nastran

Default Settings New Edit Import from Stress Analysis Materials Idealizations Connectors Offset Surfaces Structural Members Constraints Loads Auto Manual Solver Mesh Settings Generate Mesh Mesh Control Stop Load Results Object Visibility Help Tutorials Finish Autodesk Inventor Nastran Exit

Model Nastran Model Tree Nastran Output Nastran File

LINES READ: 7626402
DETERMINING MODEL SIZE
PERCENT COMPLETE: 100
PROCESSING CASE CONTROL COMMANDS
PERCENT COMPLETE: 100
13 CASE CONTROL COMMANDS WERE RECOGNIZED OF A TOTAL OF 13 READ IN
PROCESSING BULK DATA ENTRIES
PERCENT COMPLETE: 100
7626388 BULK DATA ENTRIES WERE RECOGNIZED OF A TOTAL OF 7626388
MAXIMUM CONTACT RADIAL ACTIVATION DISTANCE = 1.0000E+00
MAXIMUM CONTACT NORMAL ACTIVATION DISTANCE = 1.0000E+00
MAXIMUM CONTACT ALLOWABLE PENETRATION DISTANCE = 3.9746E-01
WRITING OUT MODEL DATA
PAGES WRITTEN: 51
MODEL DATABASE SIZE
SUBCASES = 1
COORDINATE SYSTEMS = 4
GRID POINTS = 2233432
ELEMENTS = 32641878
ELEMENT PROPERTIES = 3
MATERIAL PROPERTIES = 2
SINGLE POINT CONSTRAINTS = 16251
GRID POINT FORCES = 46
SETS = 4
SURFACES = 1
VOLUMES = 1
MODULE SEQUENCE FOR SOLUTION: NONLINEAR STATIC
GEOMETRY PROCESSOR MODULE
FATAL ERROR G3004: PROFILE LIMIT EXCEEDED
VIRTUAL MEMORY = 3956073216 WORDS 30188.5 MEGABYTES
REAL MEMORY = 8916679680 WORDS 68028.9 MEGABYTES
DRIVE C: DISK SPACE = 15525443584 WORDS 118449.7 MEGABYTES
MODEL ANALYSIS TIME SUMMARY
TOTAL CPU TIME = 766.22 SECONDS
WALLCLOCK TIME = 771.63 SECONDS
EXECUTION TERMINATED DUE TO DISPLAYED ERROR(S)
TOTAL WARNINGS = 0
TOTAL FATAL ERRORS = 1

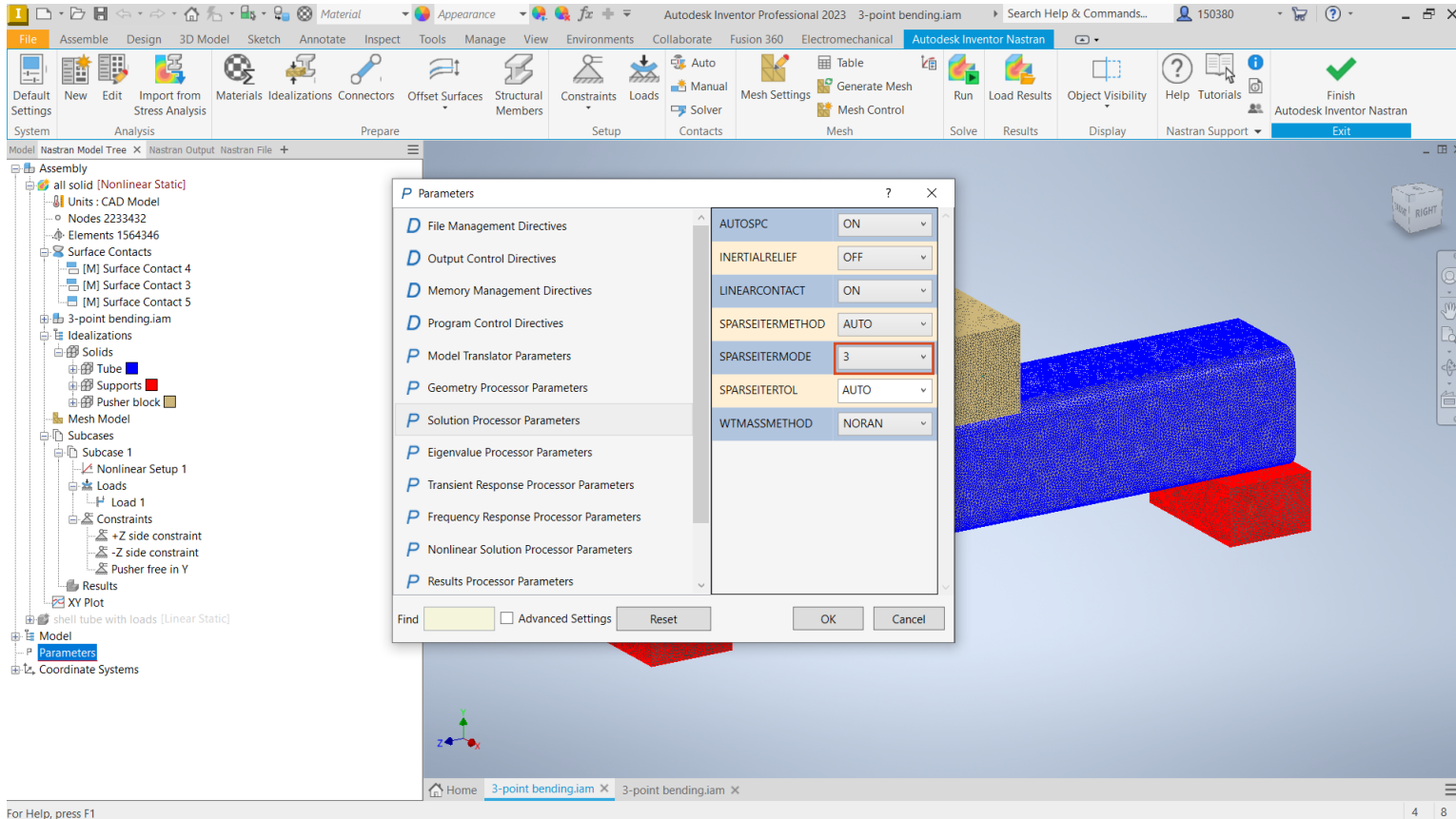
G3004	FATAL ERROR: G3004	PROFILE LIMIT EXCEEDED
Cause:	The stiffness matrix profile (non-zero terms) exceeded 2,147,483,647 words.	
Action:	Set the DECOMPMETHOD Model Initialization directive to PCGLSS and the SPARSEITERMODE model parameter to 3. If this does not correct the issue you must reduce the number of grid points in your model.	
Remarks:	None.	

For Help, press F1

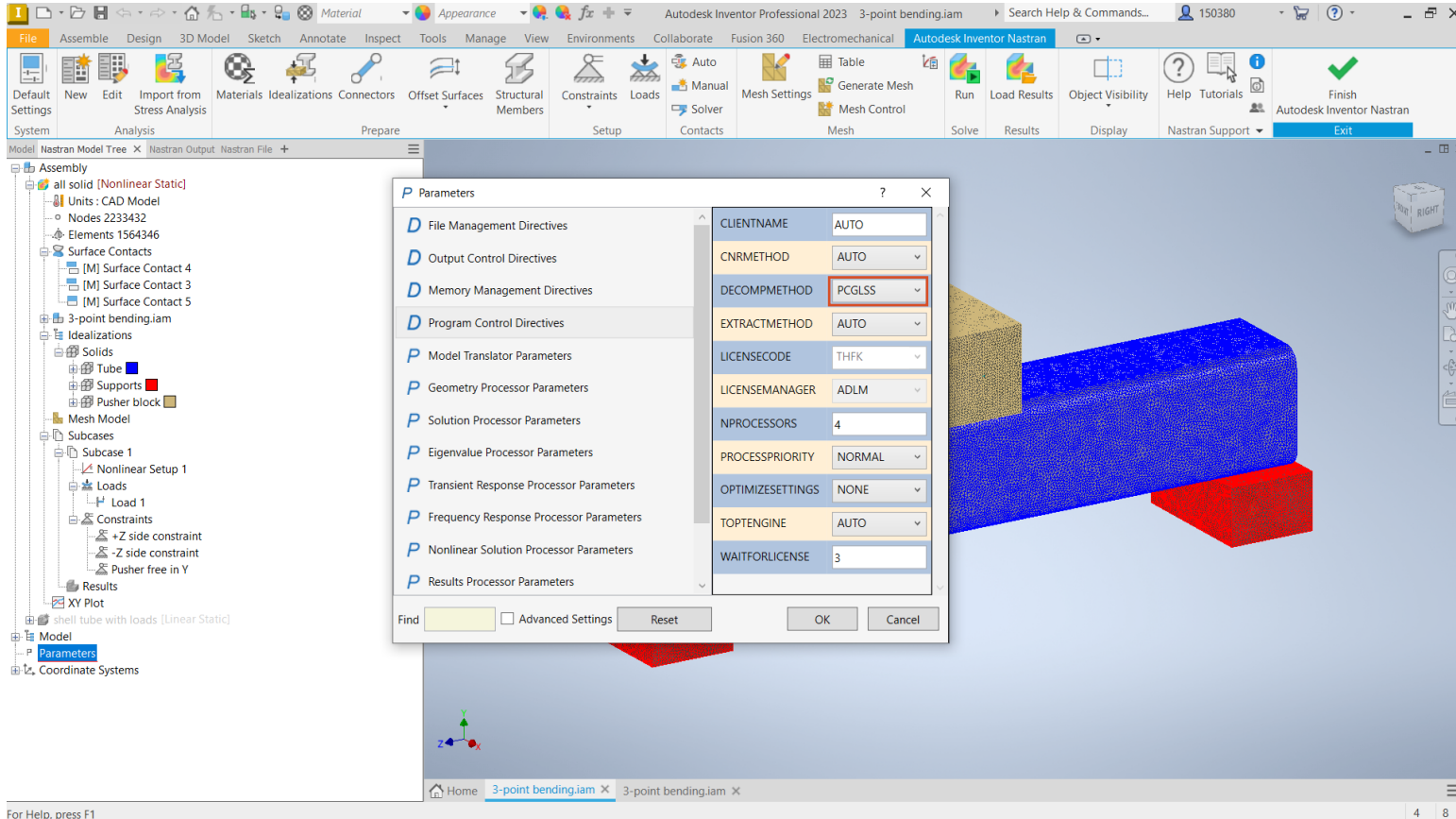
Home 3-point bending.iam 3-point bending.iam

4 8

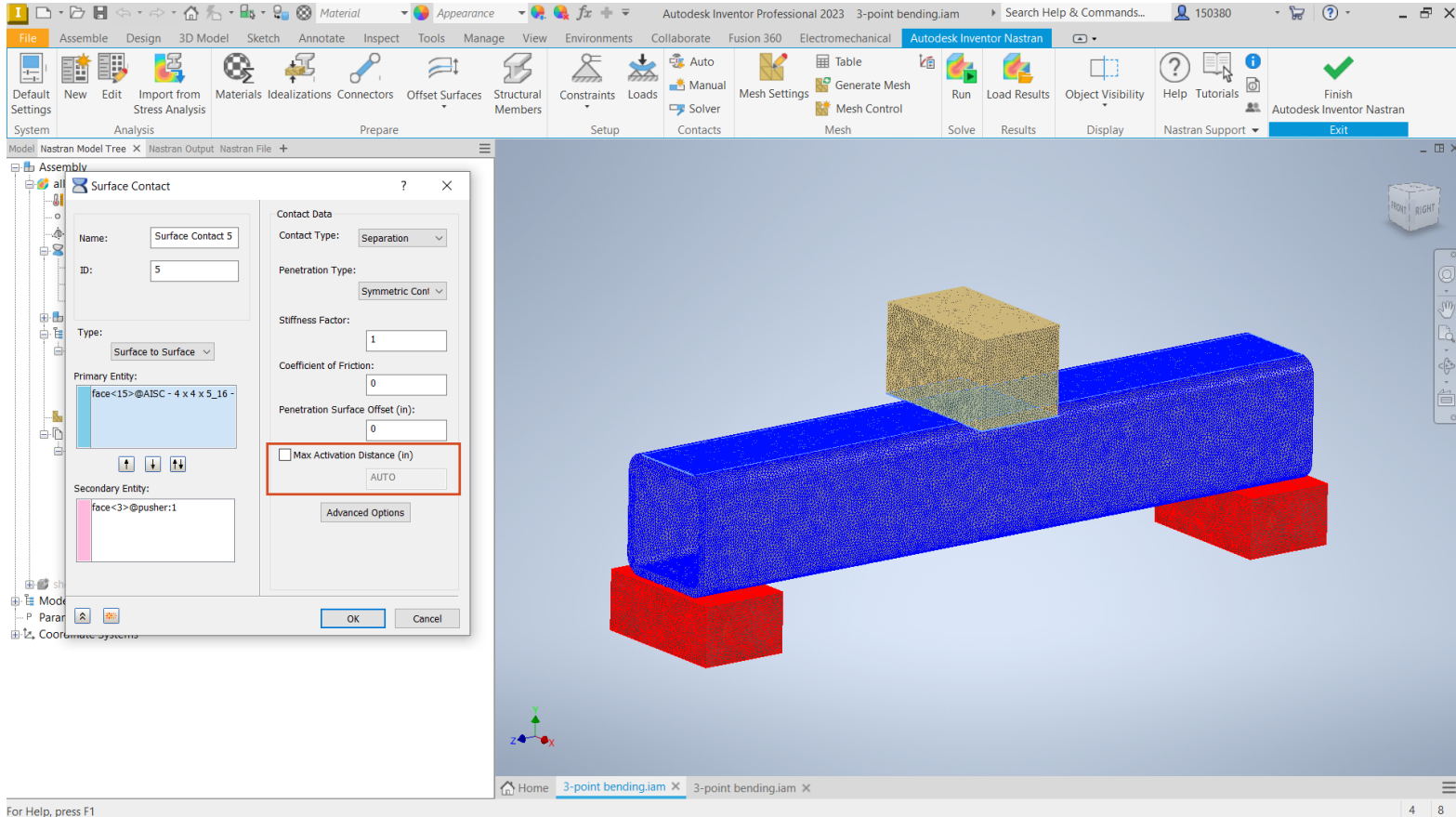
Fixing a G3004 Error Message



Fixing a G3004 Error Message



Reduce the Maximum Activation Distance





Another Option... 1/4 Symmetric Model

- If the loading and boundary conditions are symmetric this option makes the most sense
- The model will be cut down by 1/4 improving performance and allowing a much finer mesh
- The boundary conditions will allow all surfaces to slide with no friction required
- Topology optimization can easily be performed
- Other more complicated nonlinear analyses like material nonlinear can be performed much faster

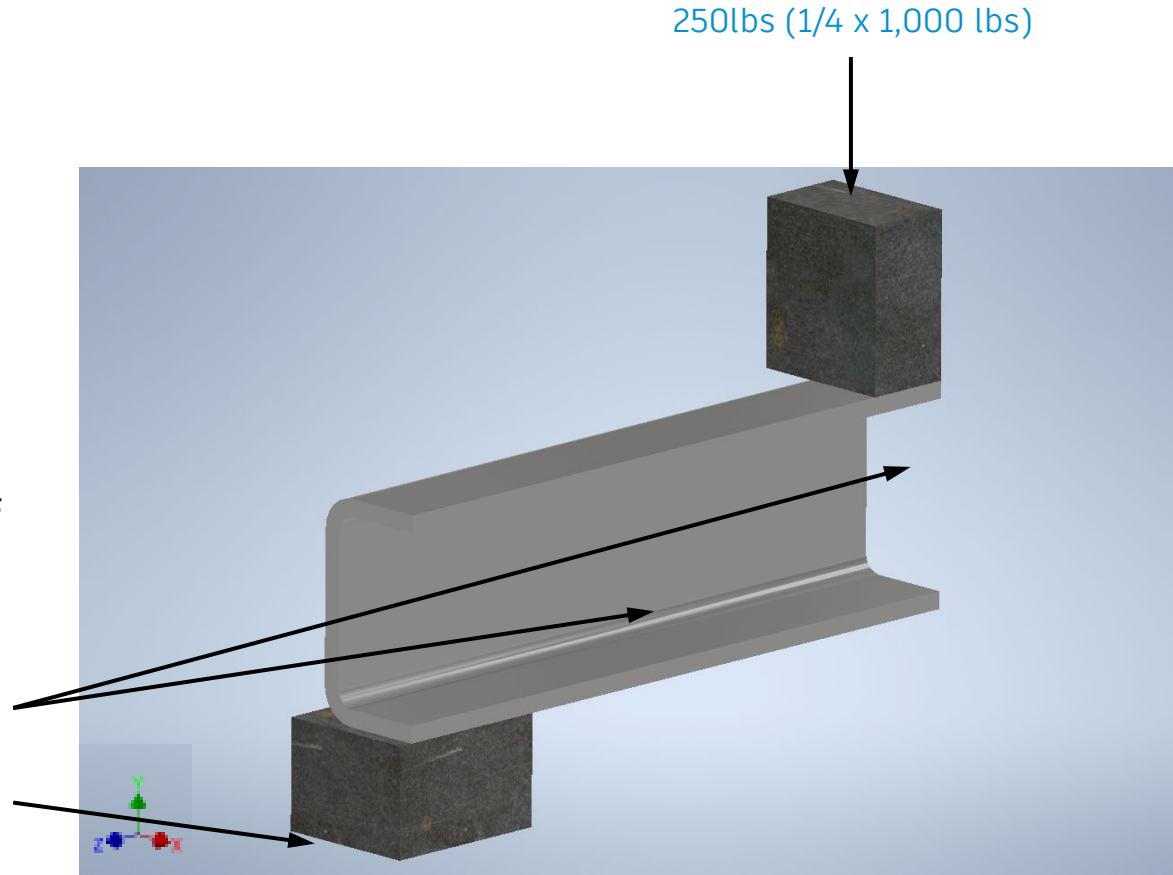


1/4 Symmetric Steel Tube Model

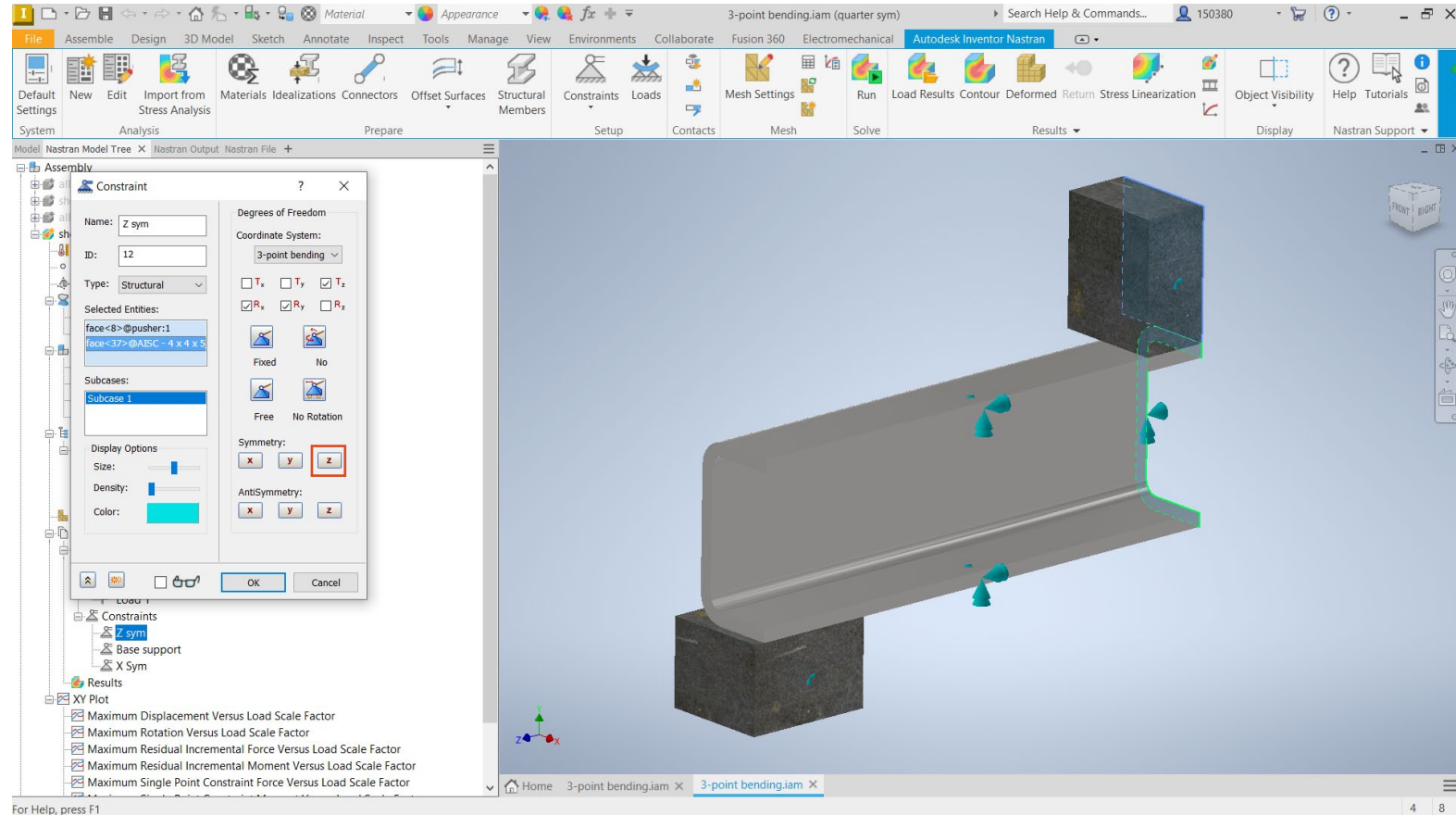
- Tube: AISC Steel:
 - 4 x 4 x 24 inches
 - 5/16 inches thick
 - Yield point 30ksi
- Blocks: Steel
- Constrained at the base of pads and side of pusher block

Symmetry boundary conditions on x and z planes

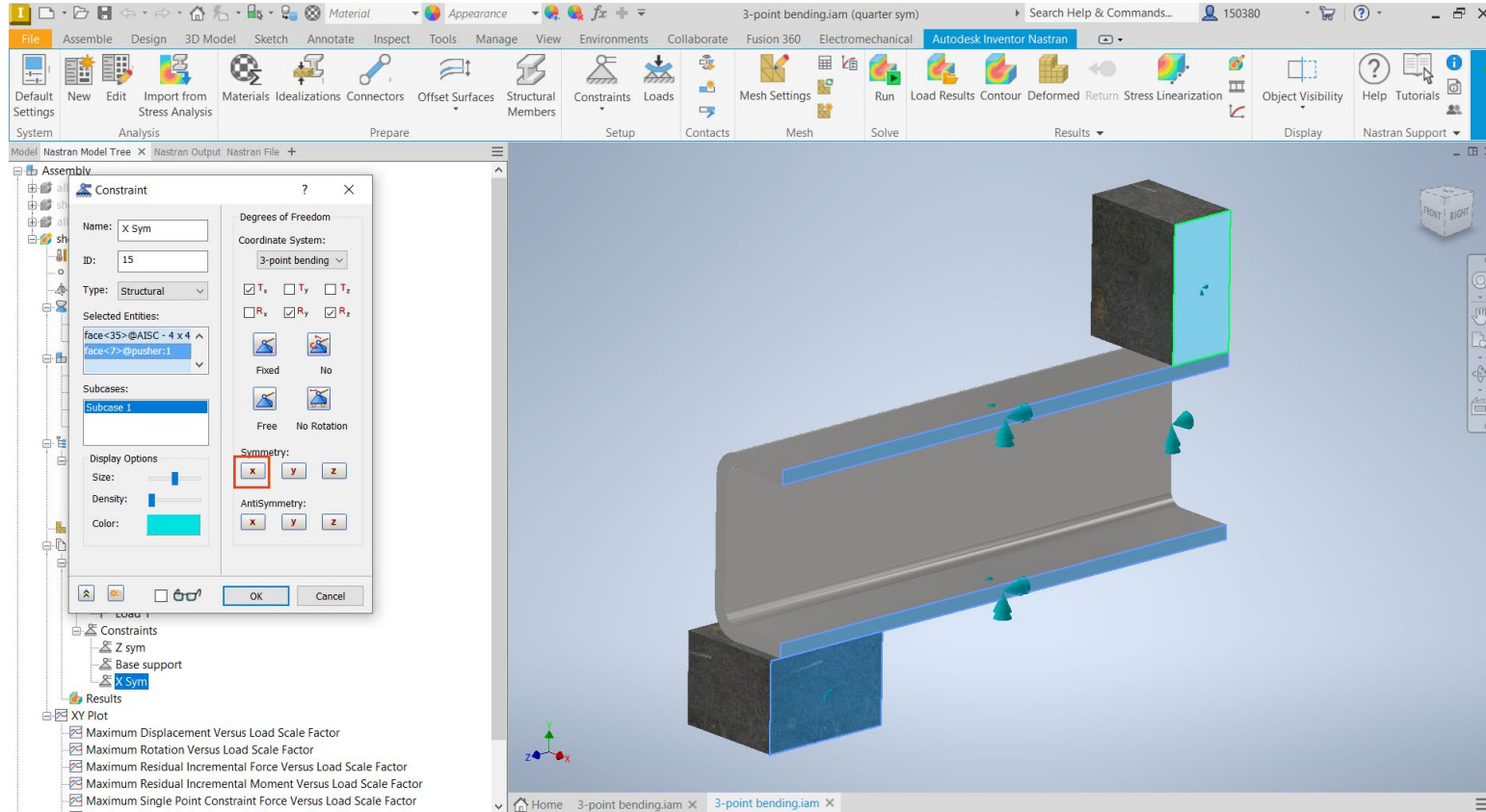
Fully constrained at base of pad



Z-Symmetry Boundary Condition



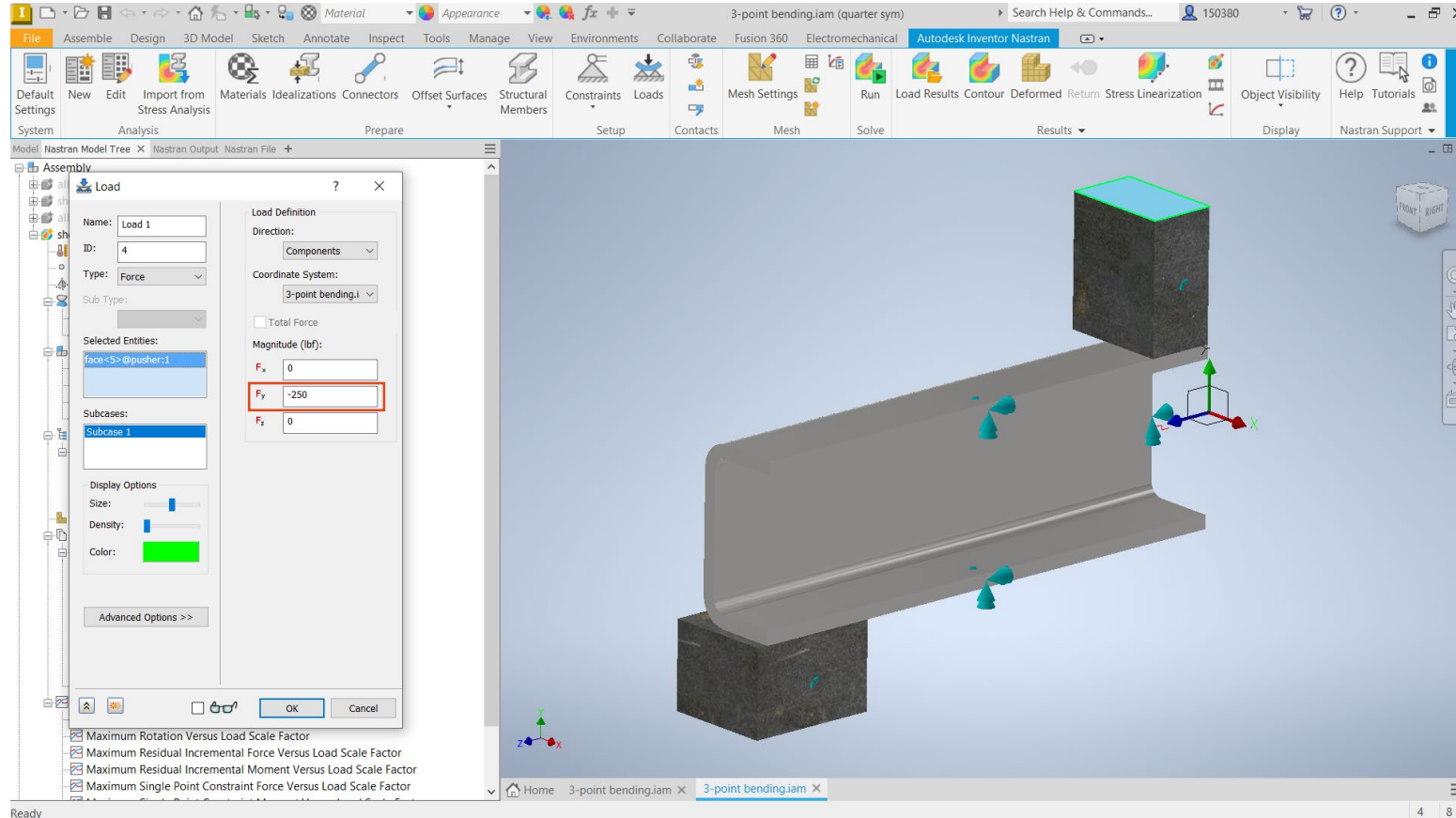
X-Symmetry Boundary Condition



For Help, press F1

4 8

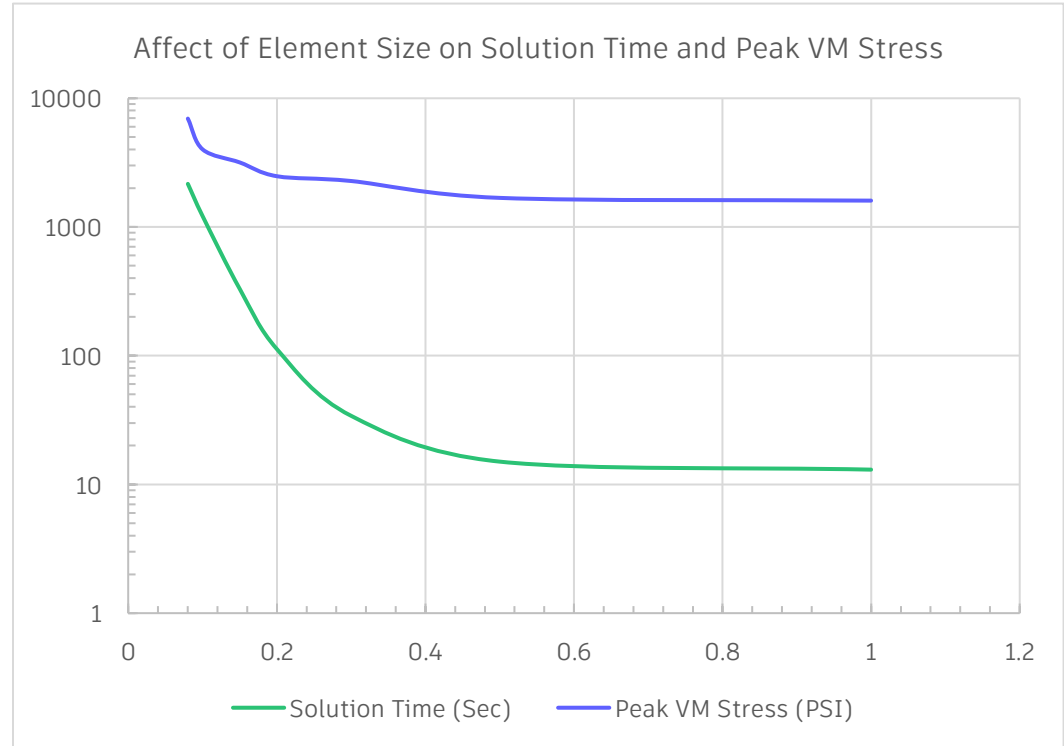
1/4-Symmetry Load Definition



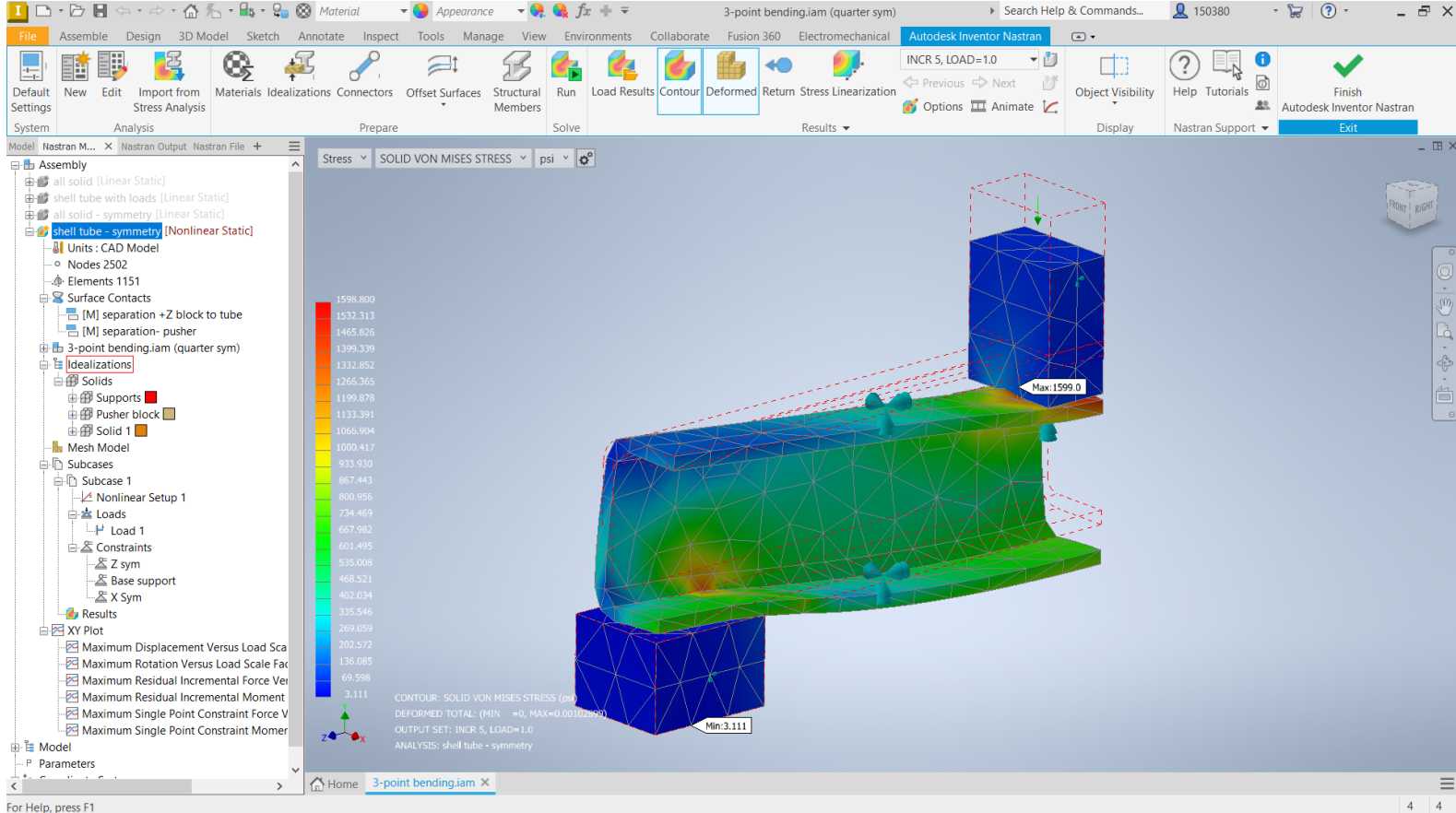


Solid Mesh Convergence Study

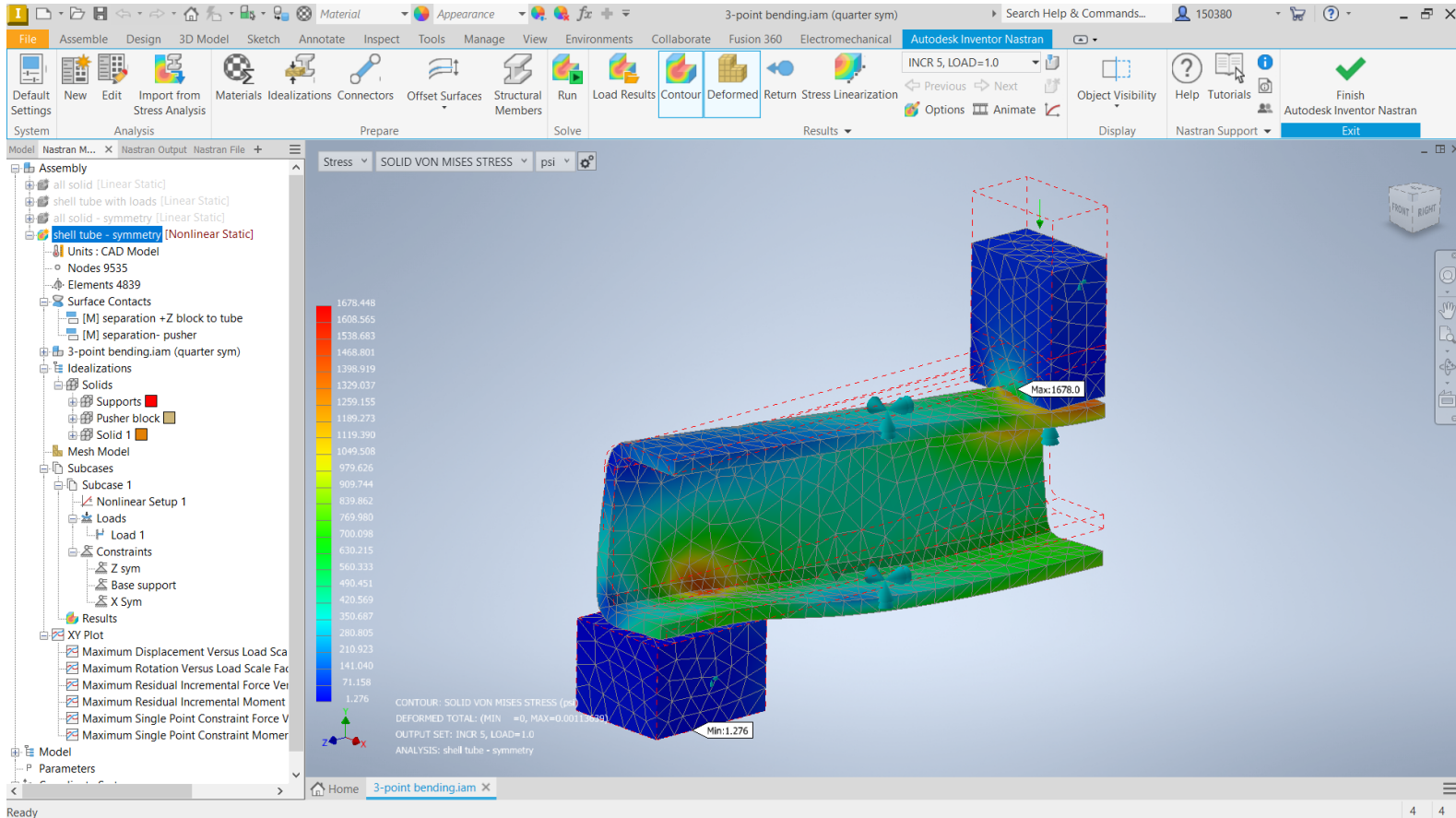
- Enhanced contact is OFF
- Mesh element size is varied from 1.0 to 0.08
- The peak stress will continue to increase due to a singularity at the edge of the support block
- Solution time goes up exponentially with model size or with the inverse of element size



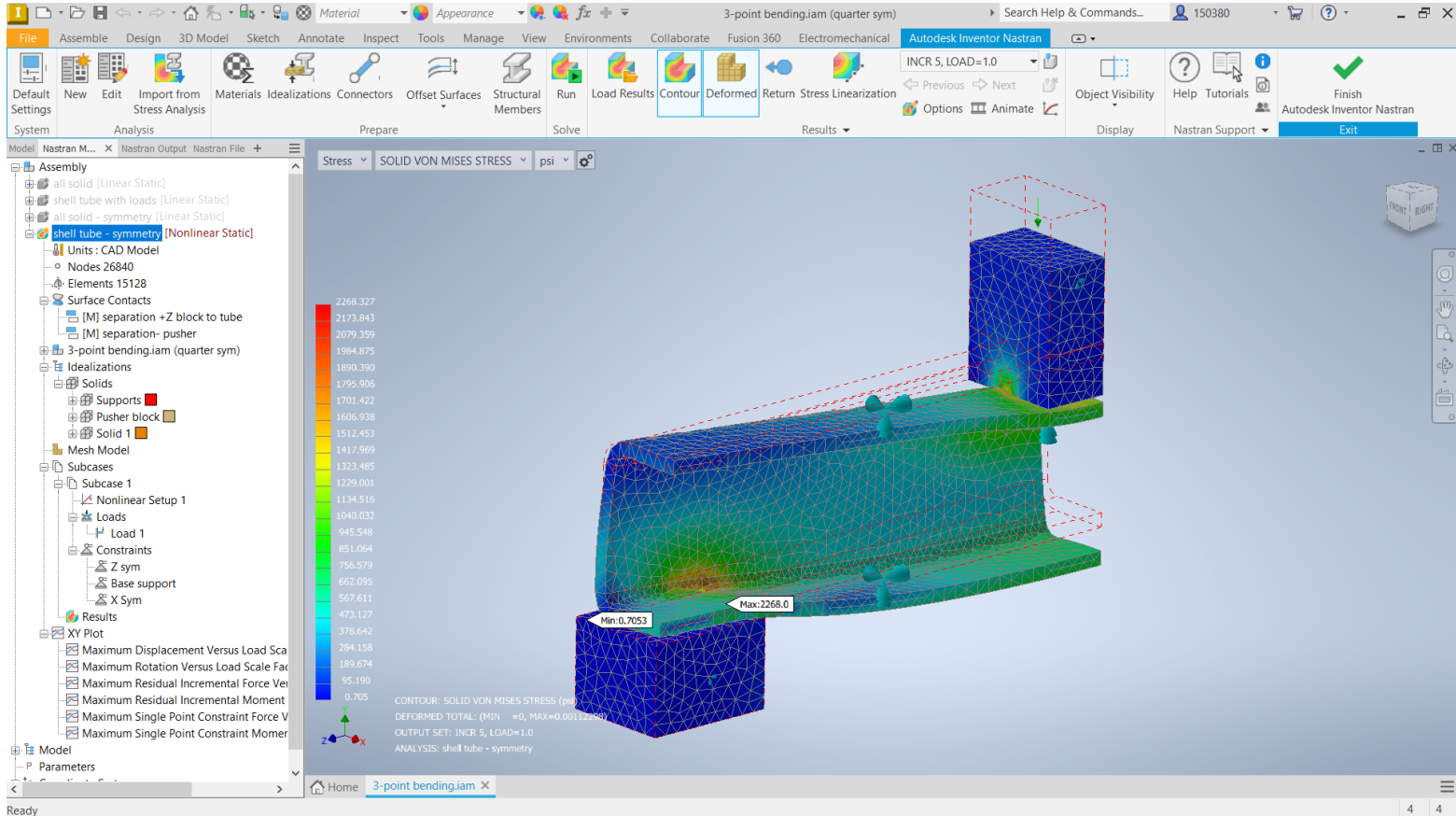
Mesh Convergence – Element Size = 1.0in – 13Sec



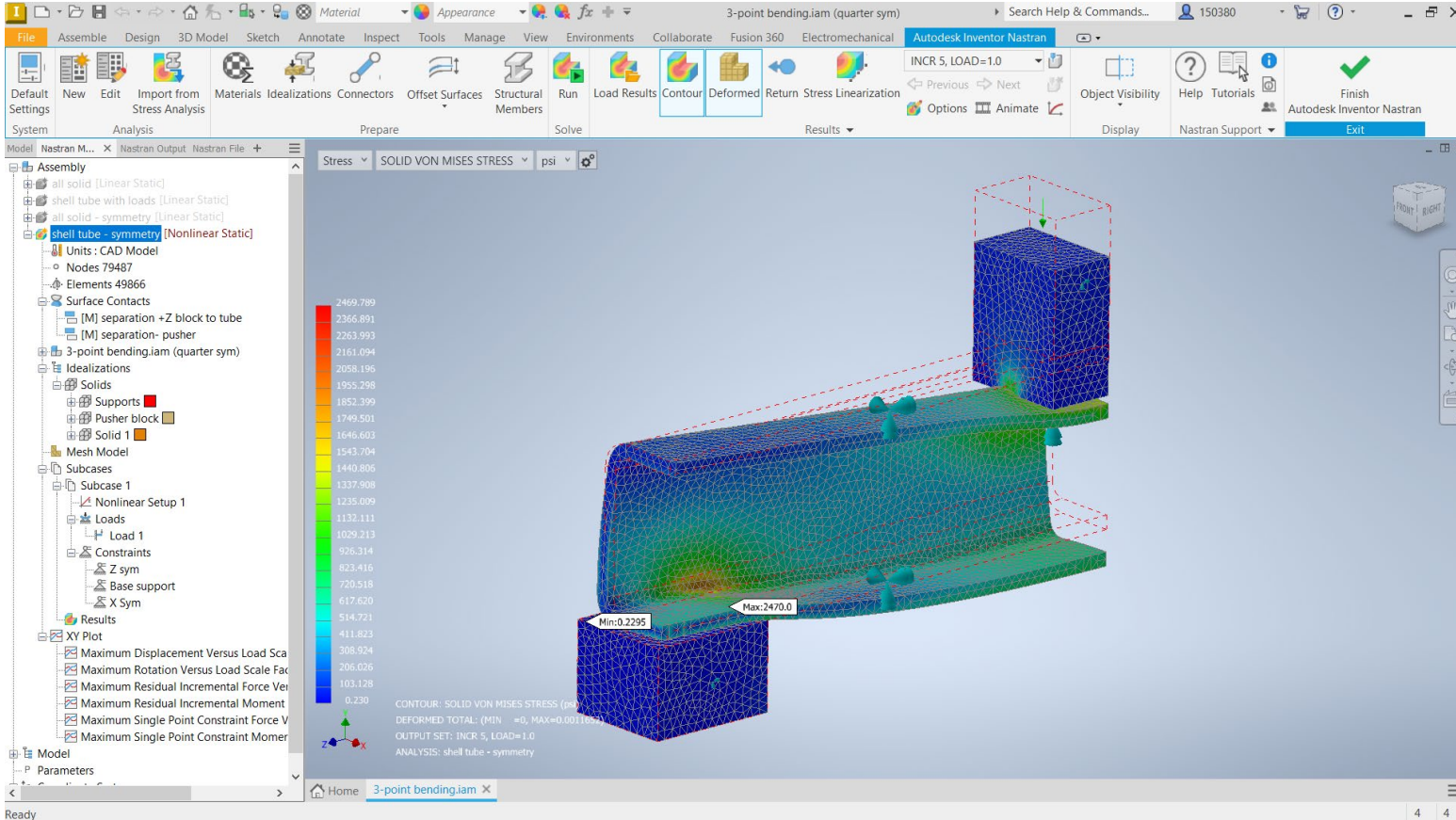
Mesh Convergence – Element Size = 0.5in – 15Sec



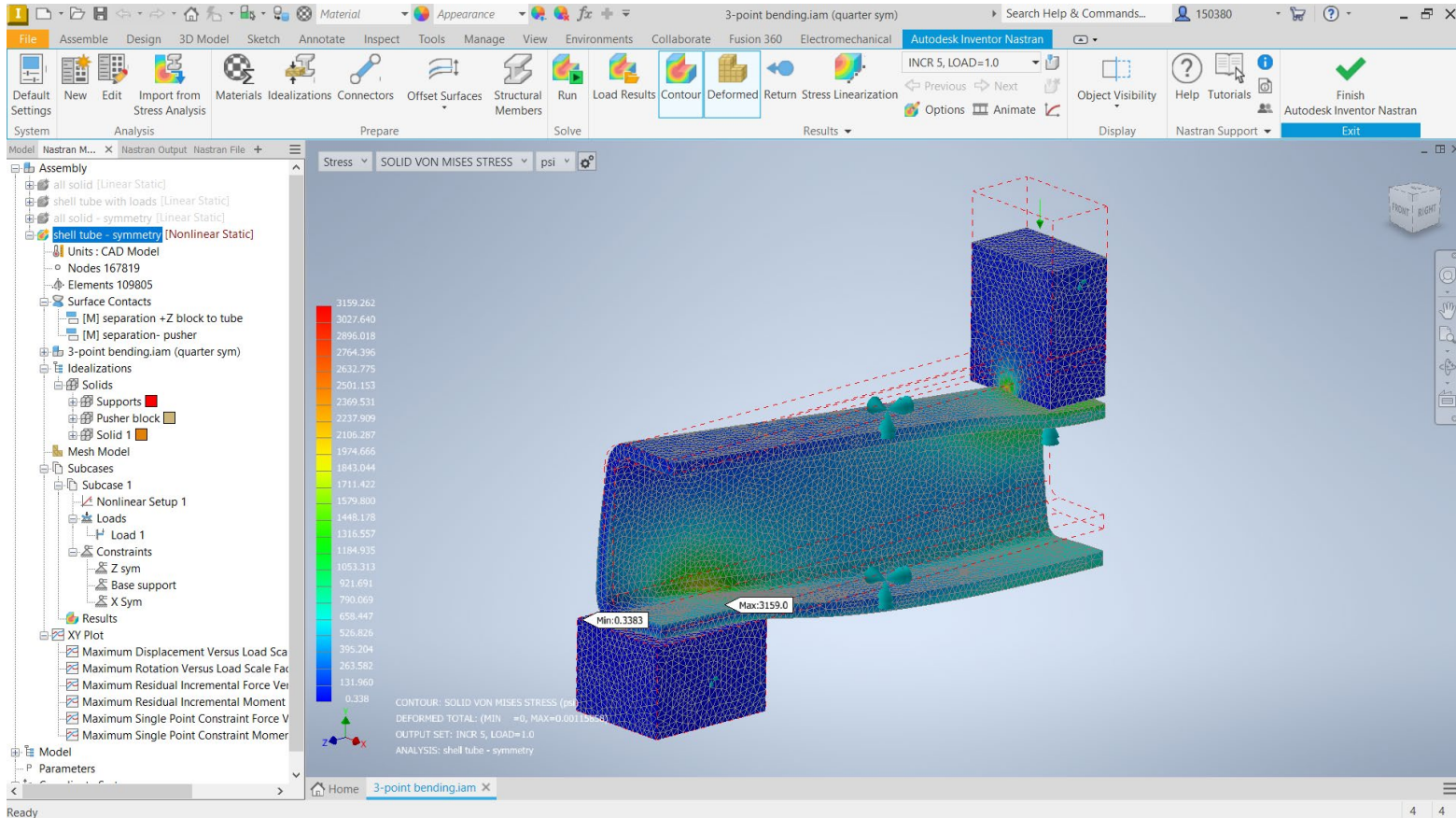
Mesh Convergence – Element Size = 0.3in – 34Sec



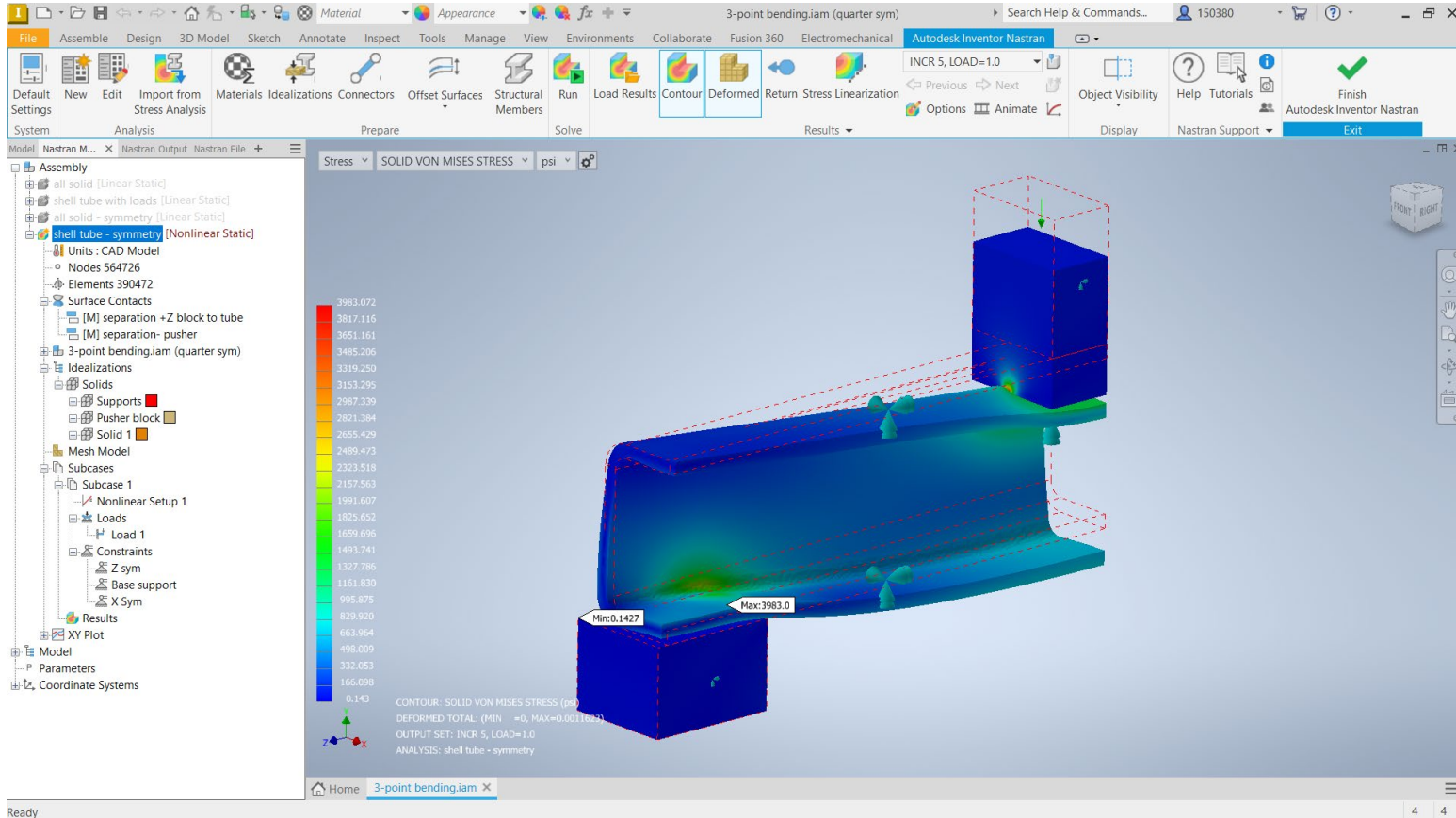
Mesh Convergence – Element Size = 0.2in – 112Sec



Mesh Convergence – Element Size = 0.15in – 328Sec

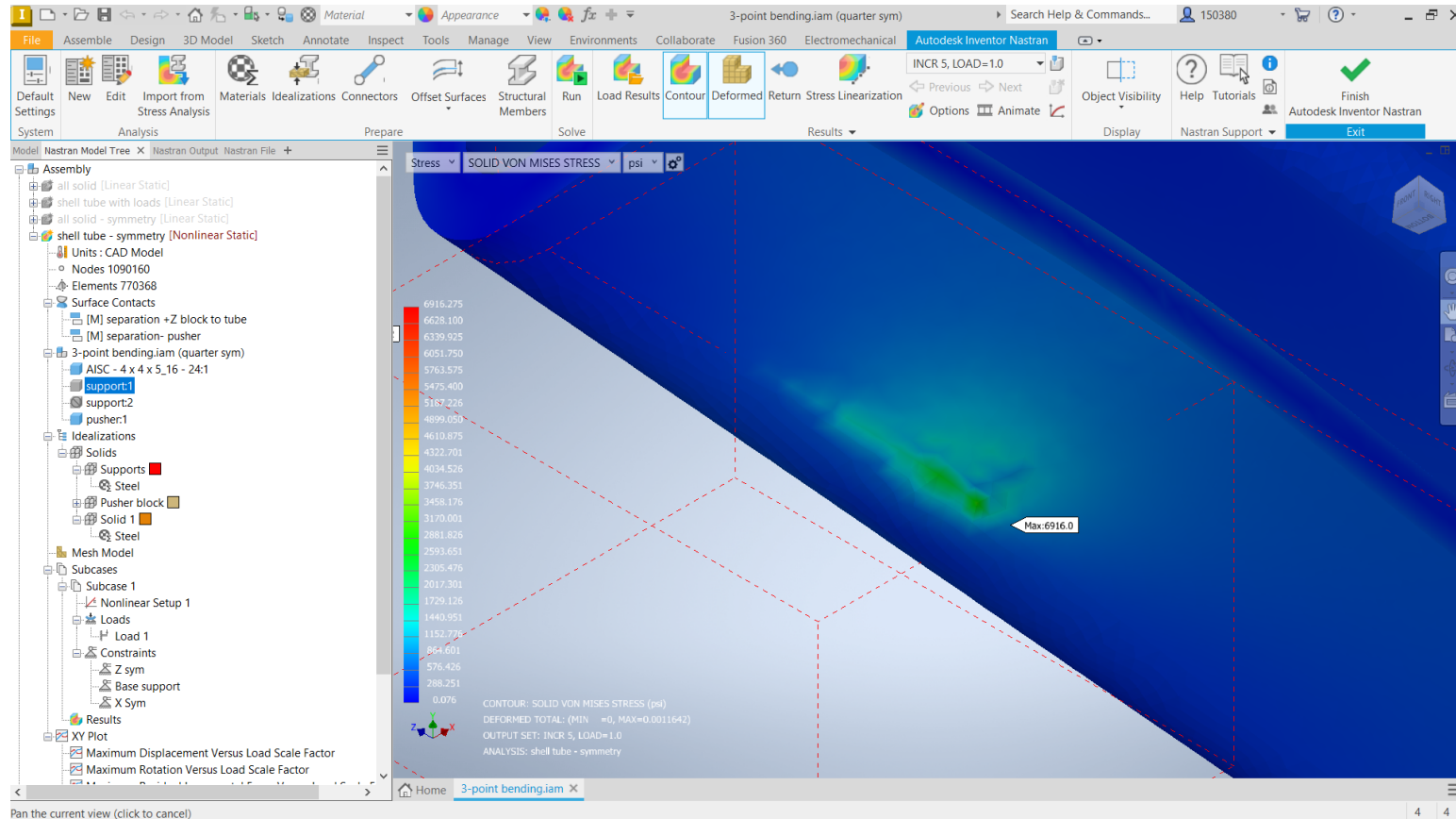


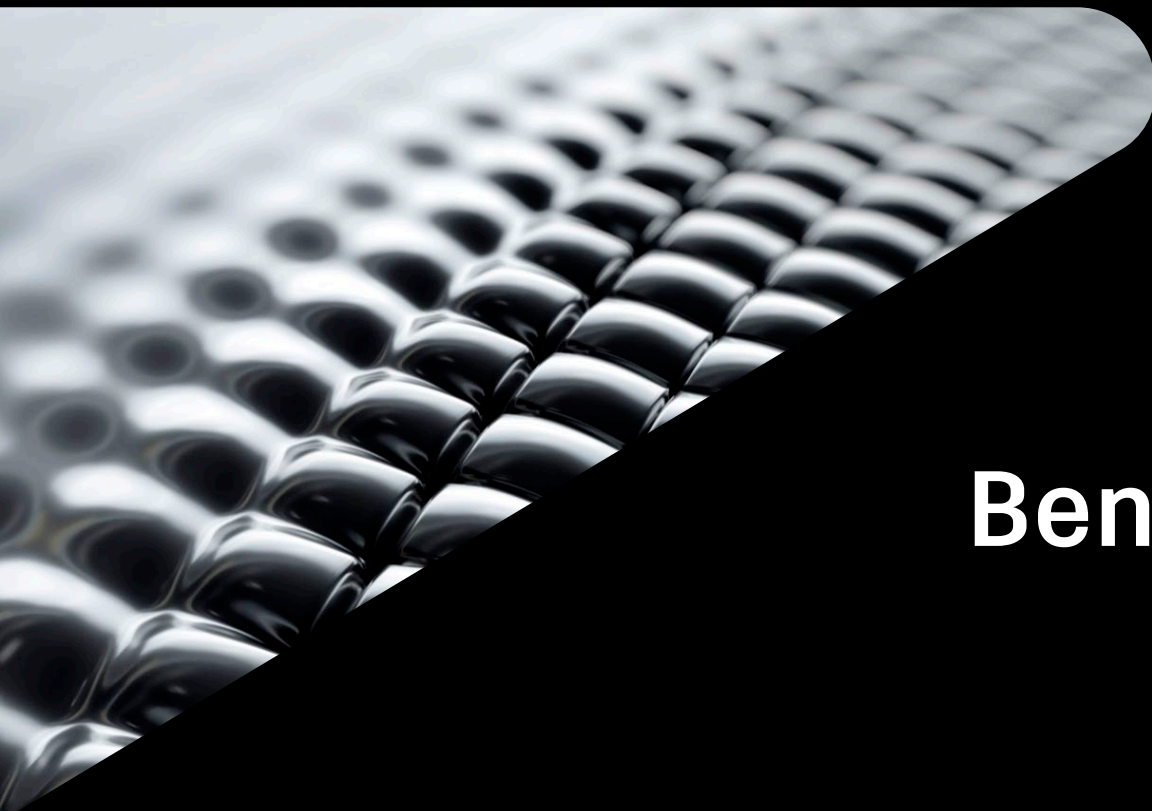
Mesh Convergence – Element Size = 0.1in – 1205Sec



The screenshot displays the Autodesk Inventor Nastran software interface. The top ribbon includes tabs for File, Assemble, Design, 3D Model, Sketch, Annotate, Inspect, Tools, Manage, View, Environments, Collaborate, Fusion 360, Electromechanical, and Autodesk Inventor Nastran. The Nastran ribbon is active, showing options for INCR 5, LOAD=1.0, Previous, Next, Options, Animate, Object Visibility, Help, Tutorials, Finish, and Exit. The left-hand 'Model' tree shows the 'Assembly' structure, including 'all solid (Linear Static)', 'shell tube with loads (Linear Static)', 'all solid - symmetry (Linear Static)', 'shell tube - symmetry (Nonlinear Static)', 'Units: CAD Model', 'Nodes 1090160', 'Elements 770368', 'Surface Contacts', 'Idealizations', 'Solids', 'Mesh Model', 'Subcases', 'Subcase 1', 'Nonlinear Setup 1', 'Loads', 'Load 1', 'Constraints', 'Z sym', 'Base support', 'X Sym', 'Results', and 'XY Plot'. The main 3D view shows a mechanical assembly under stress analysis. The stress results are displayed as a color-coded contour plot, with a legend on the left indicating stress values from 0.076 to 6916.275 psi. The text 'SOLID VON MISES STRESS (psi)' is visible. The bottom status bar shows the active file '3-point bending.iam'.

Stress Will Continue to Increase as Mesh is Refined

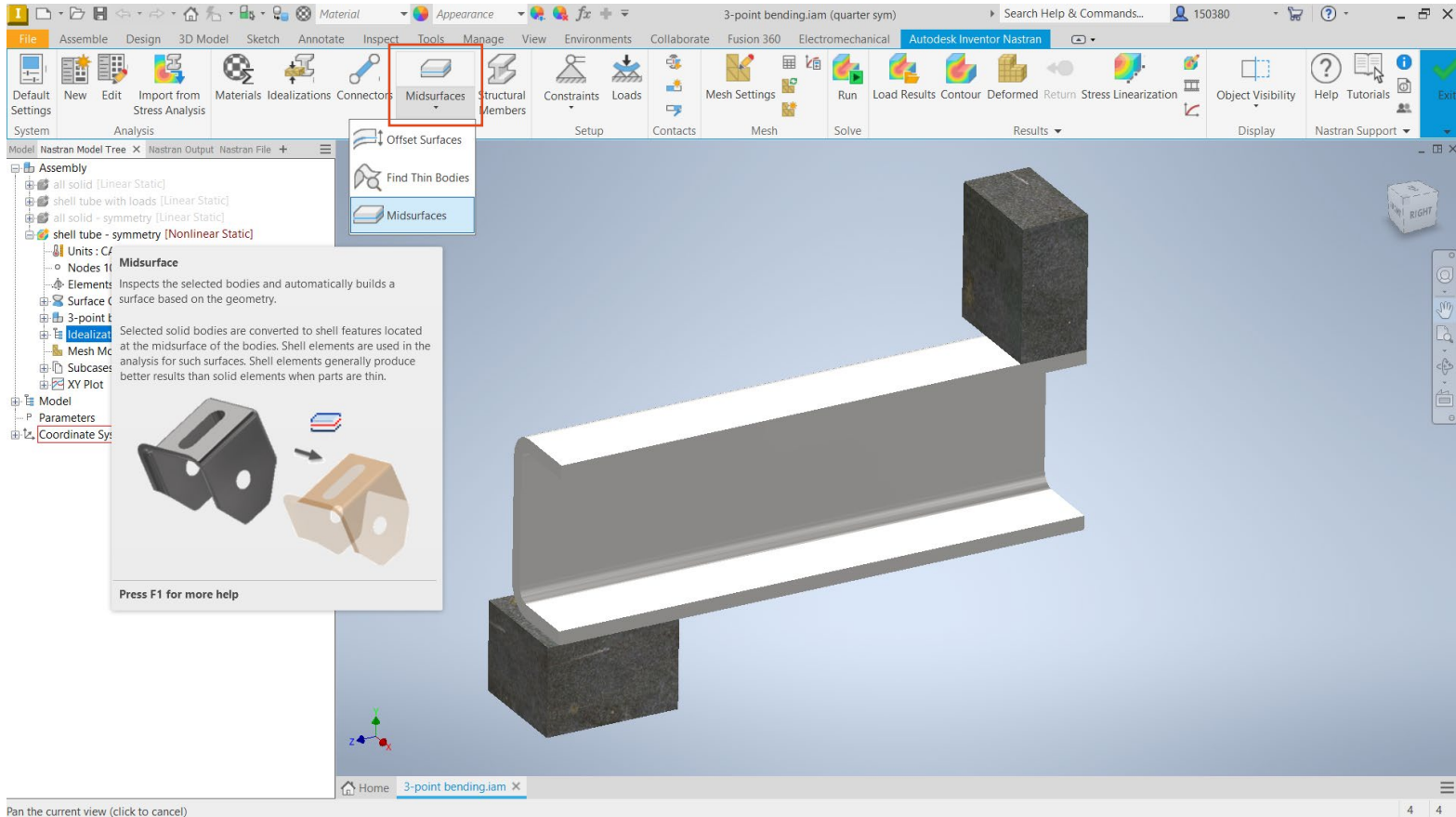




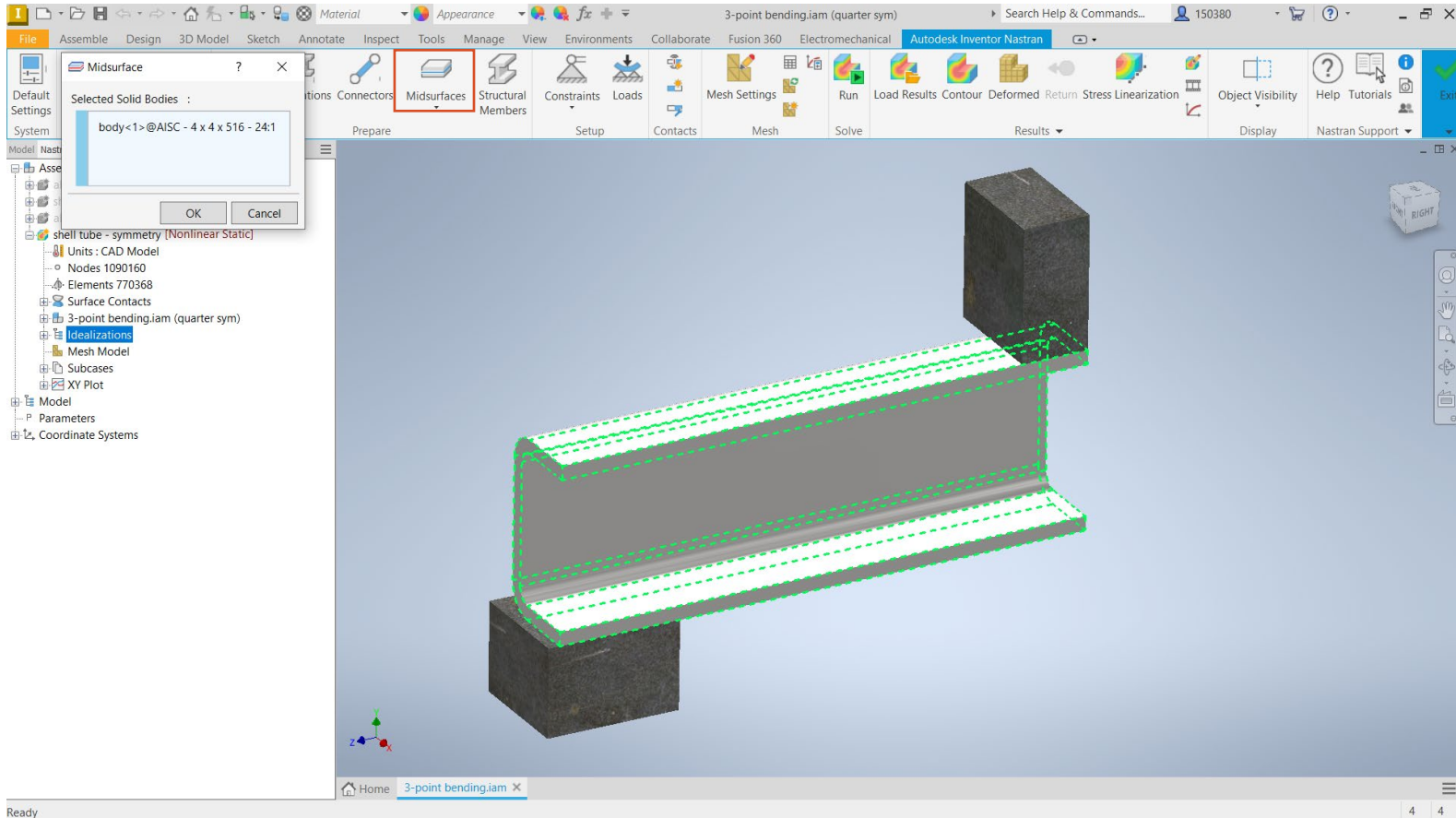
3-Point Tube Bending Example

Shell Elements

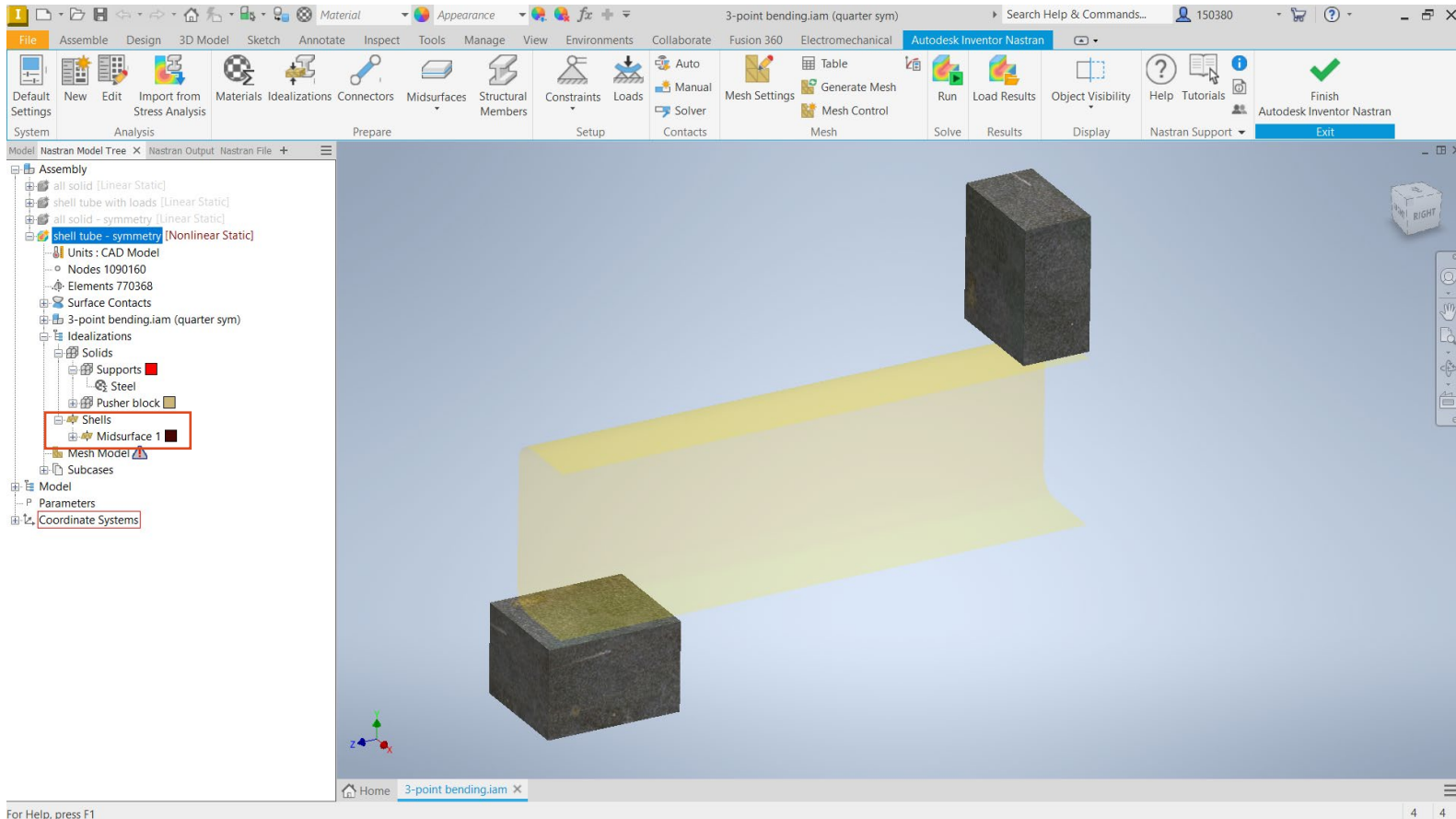
Midsurfacing the Steel Tube



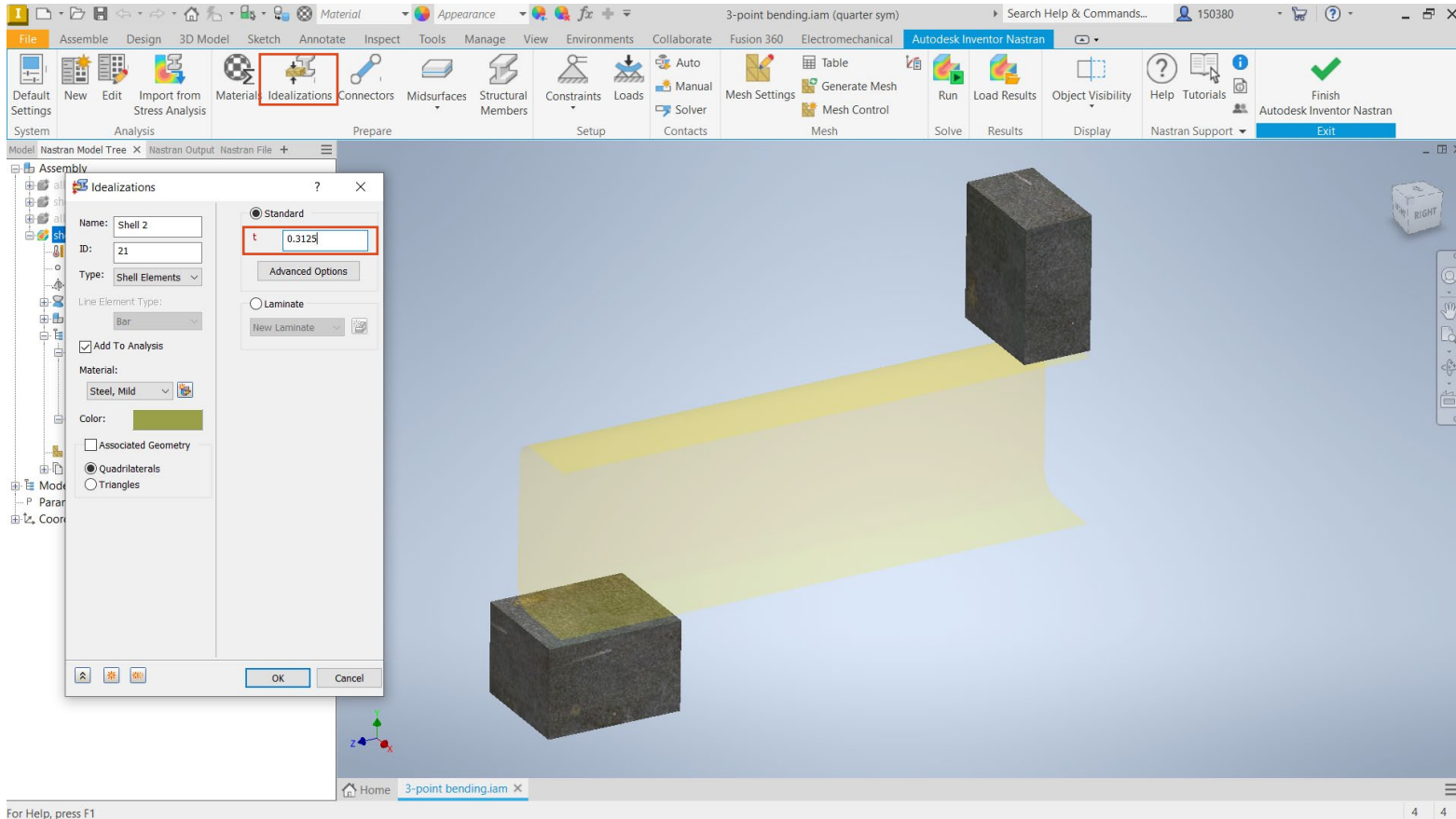
Midsurfacing the Steel Tube



Midsurfacing the Steel Tube – Shell Idealization



Midsurfacing the Steel Tube – Specify Thickness



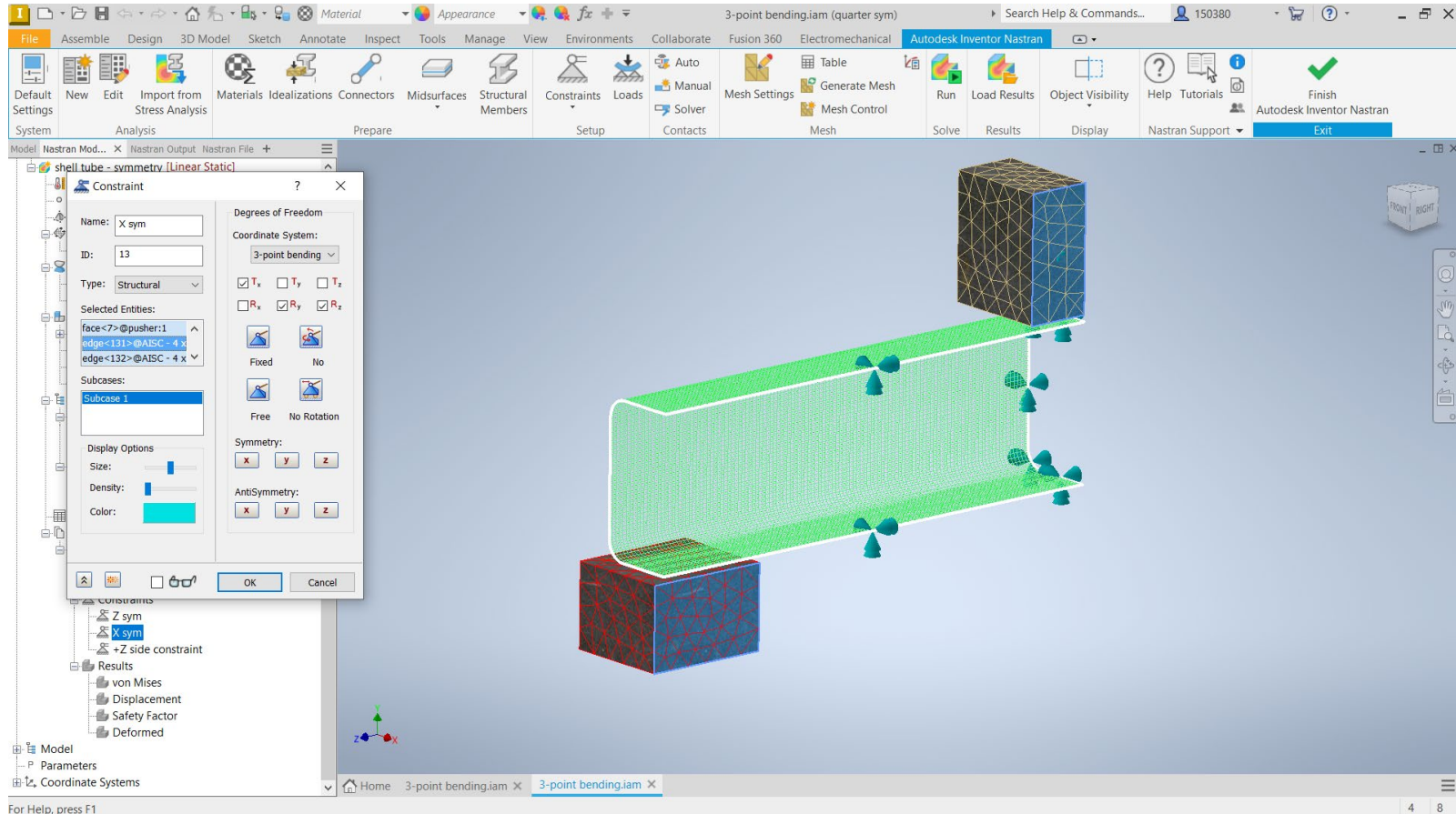
Using the Mesh Table to Define Different Meshes

The screenshot displays the Autodesk Inventor Nastran interface. The **Mesh Table** dialog is open, showing a table of mesh definitions for different parts of the model. The table includes columns for Part Name, Visibility, Color, Size (in), Tolerance (in), Element Order, Settings, Nodes, and Elements. The parts listed are 3-point bend..., AISC - 4 x 4 x 5..., pusher:1, support:1, and support:2. The AISC part is defined with a Parabolic element order and a size of 0.1 inches. The pusher and supports are defined with Linear element orders and a size of 0.5 inches. The 3-point bend part is defined with a Parabolic element order and a size of 0.1 inches. The dialog also includes buttons for Generate Mesh, Delete, and Continuous Meshing (for solids only merges nodes on perimeter).

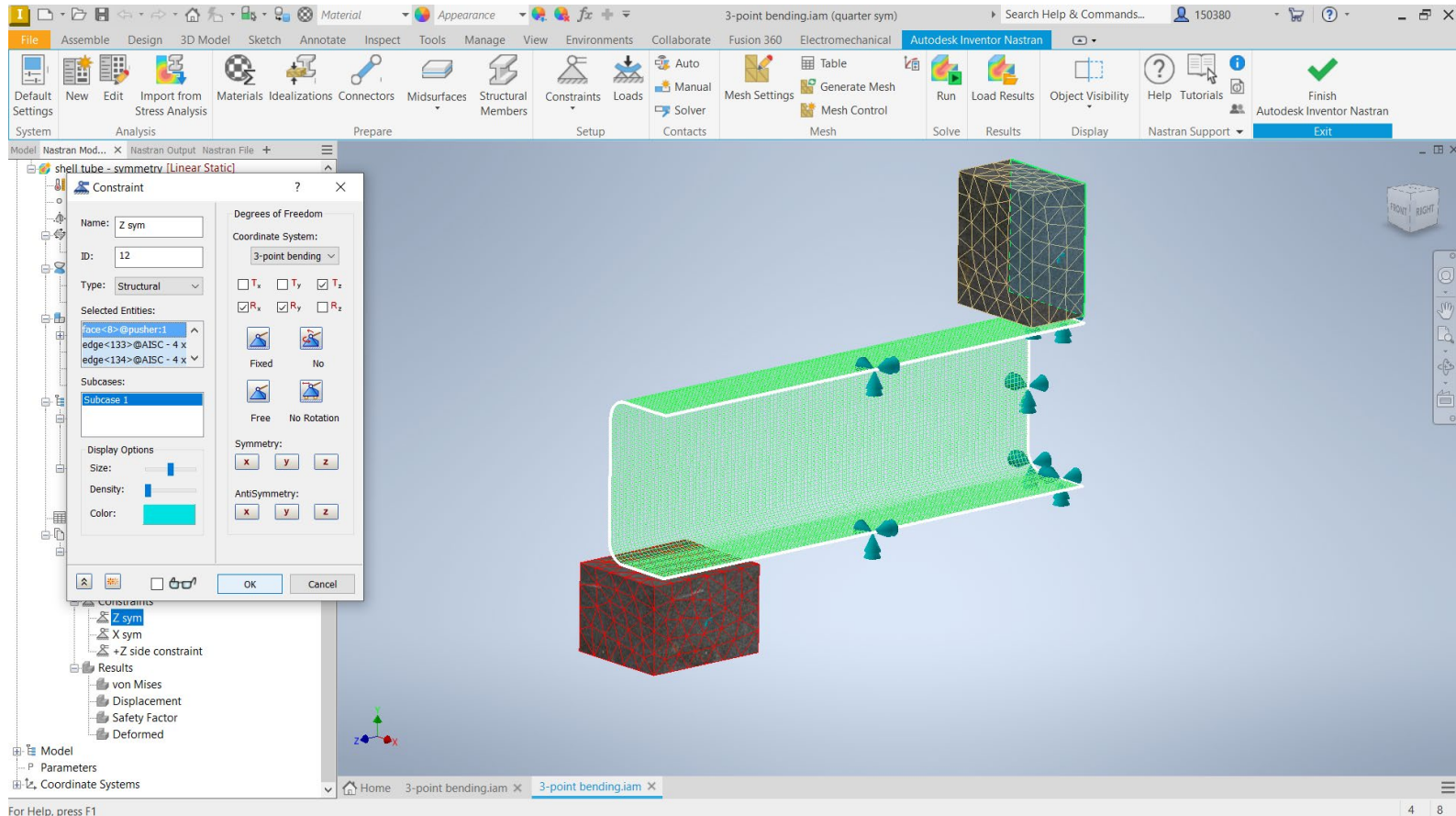
Part Name	Visibility	Color	Size (in)	Tolerance (in)	Element Order	Settings	Nodes	Elements
3-point bend...	<input checked="" type="checkbox"/>		0.1	1.15537e-05	Parabolic	Settings	0	0
aisc - 4 x 4 x 5...	<input checked="" type="checkbox"/>		0.1	1.84932e-05	Parabolic	Settings	25220	8281
pusher:1	<input checked="" type="checkbox"/>		0.5	4.91808e-06	Linear	Settings	211	718
support:1	<input checked="" type="checkbox"/>		0.5	4.62331e-06	Linear	Settings	248	902
support:2	<input checked="" type="checkbox"/>		0.5	4.62331e-06	Linear	Settings	0	0

The 3D model shows a 3-point bending test setup. A central beam (AISC) is supported by two blocks (pusher:1 and support:2) and loaded by a third block (support:1). The mesh is applied to the beam and supports, with different element orders and sizes defined in the Mesh Table. The interface also shows the Autodesk Inventor Nastran ribbon with options like Run, Load Results, Object Visibility, Help, Tutorials, and Finish. The bottom status bar indicates the current file is 3-point bending.iam.

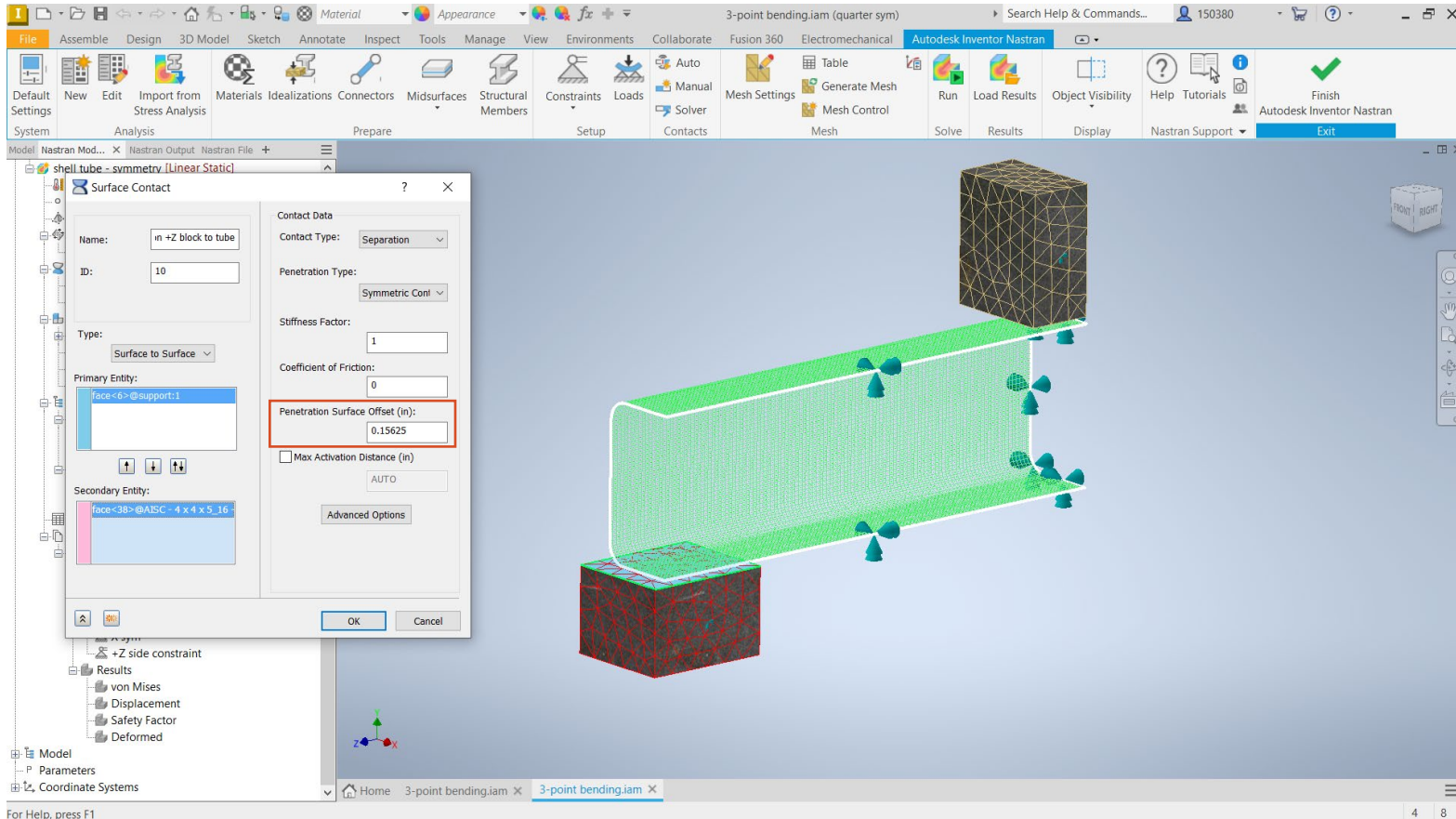
Add Shell Edge Constraints – X-Sym



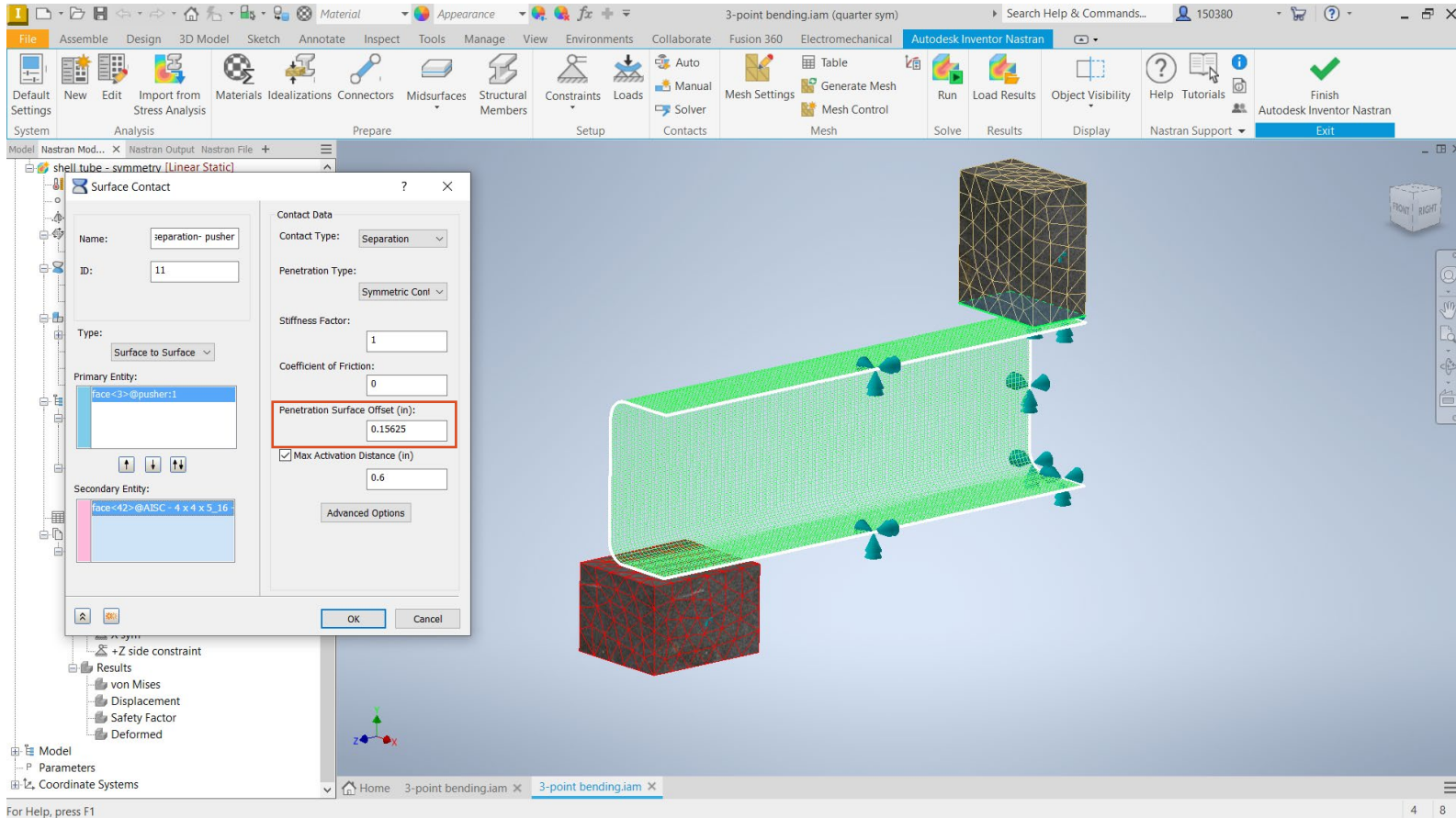
Add Shell Edge Constraints – Z-Sym



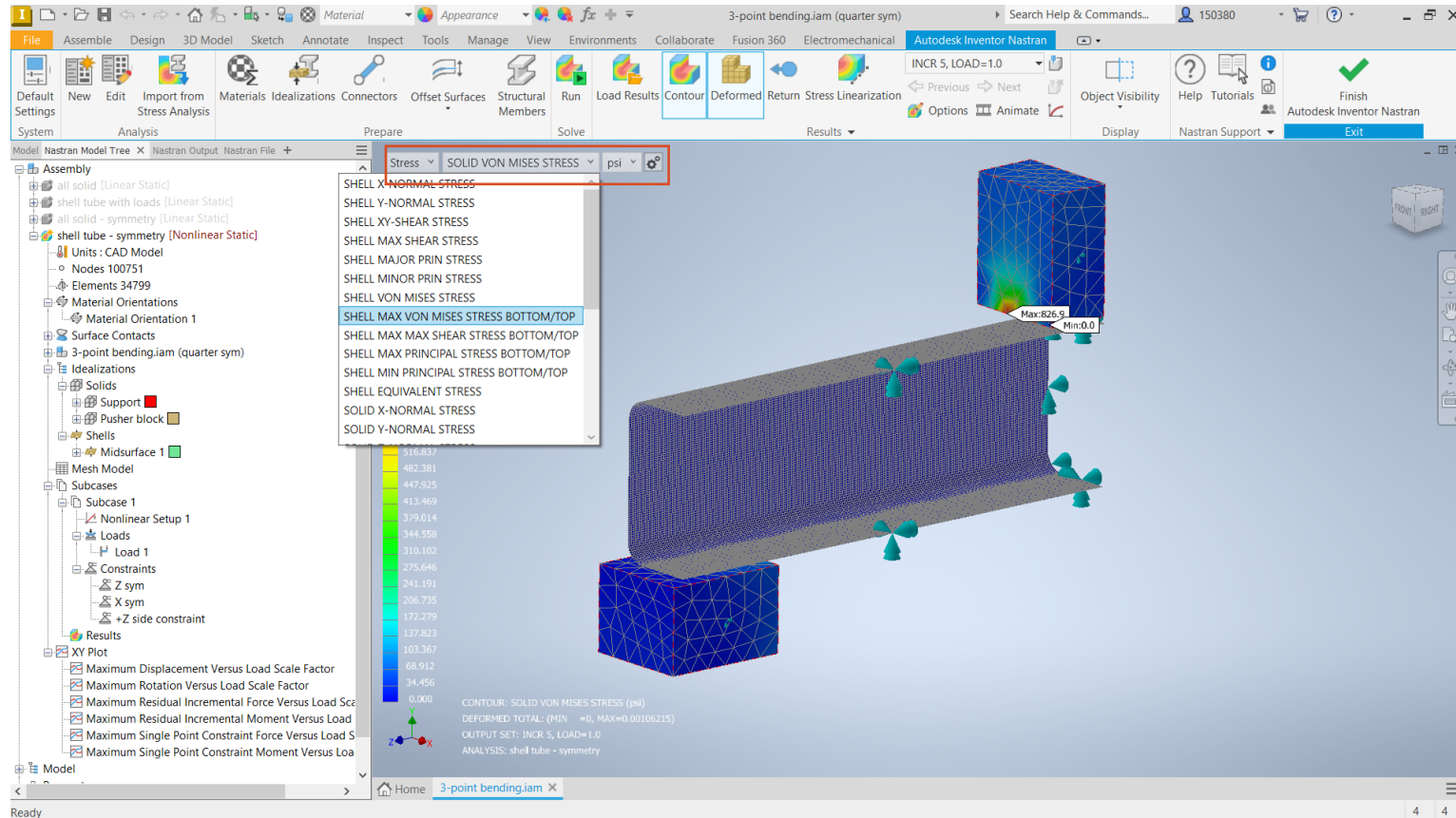
Modify Contact Definition to Account for Shell Offset



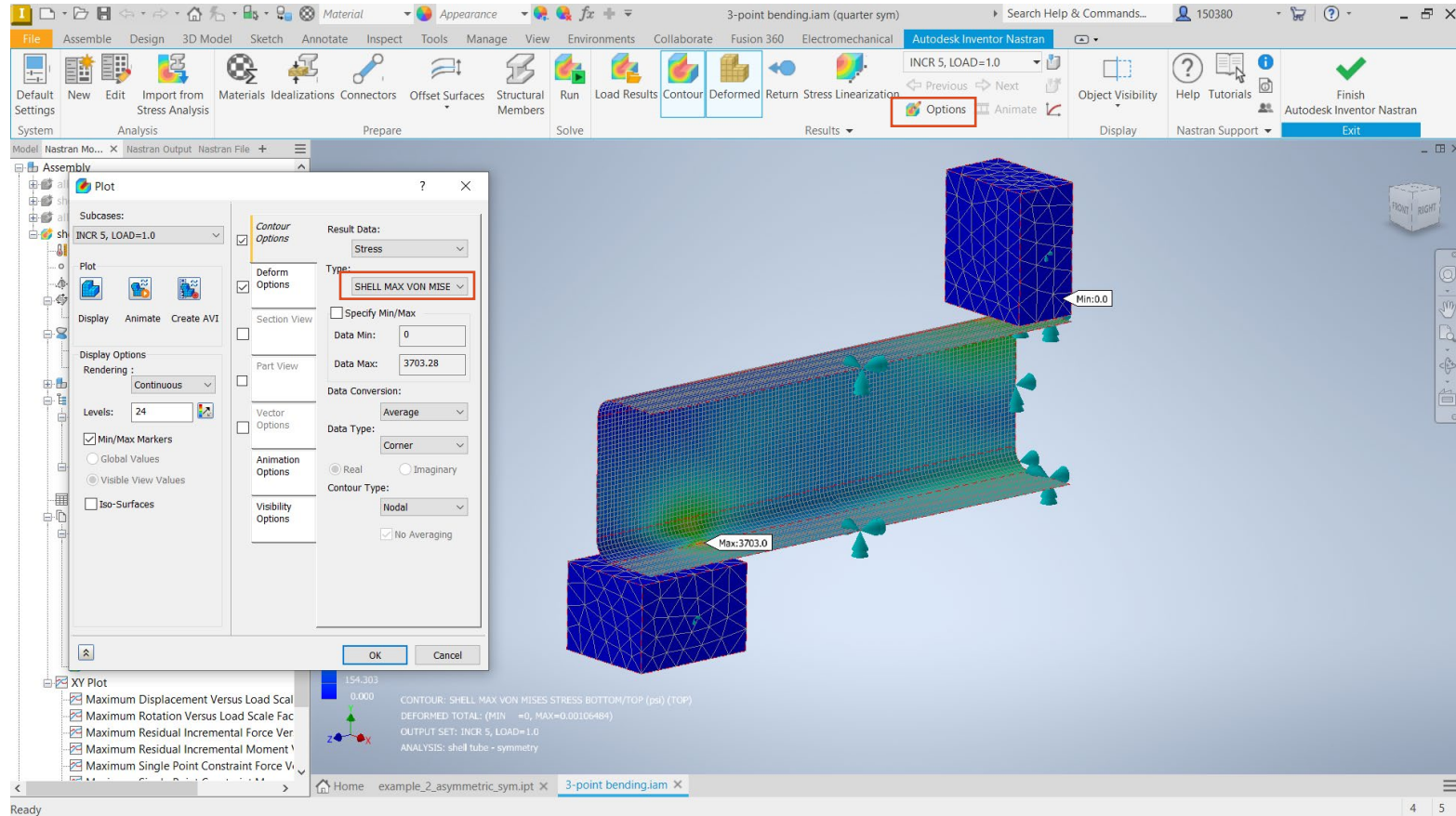
Modify Contact Definition to Account for Shell Offset



Switch Default Solid VM Stress to Shell Max VM Top/Bottom



Switch Default Solid VM Stress to Shell Max VM Top/Bottom

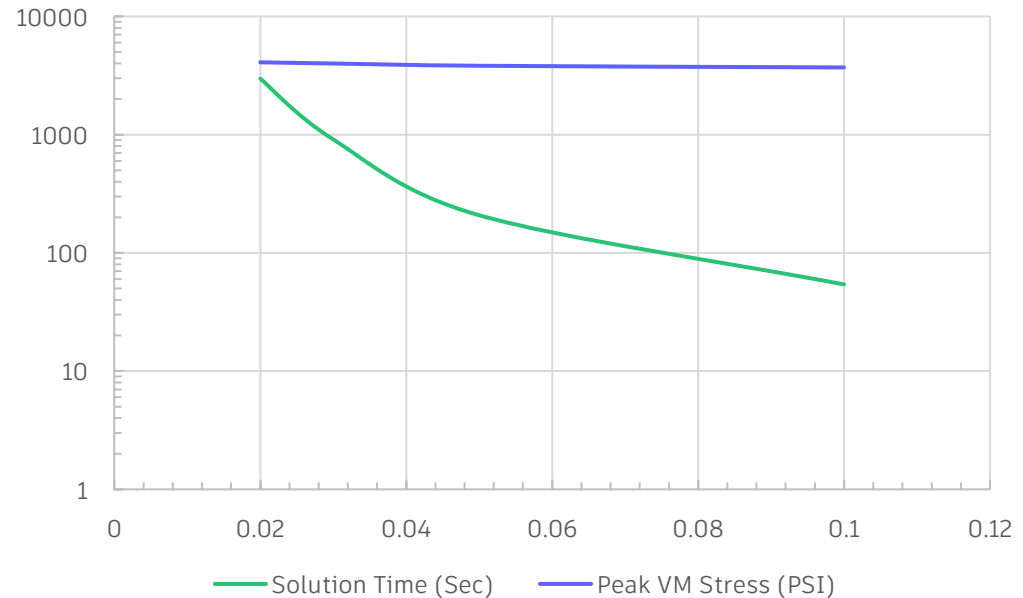




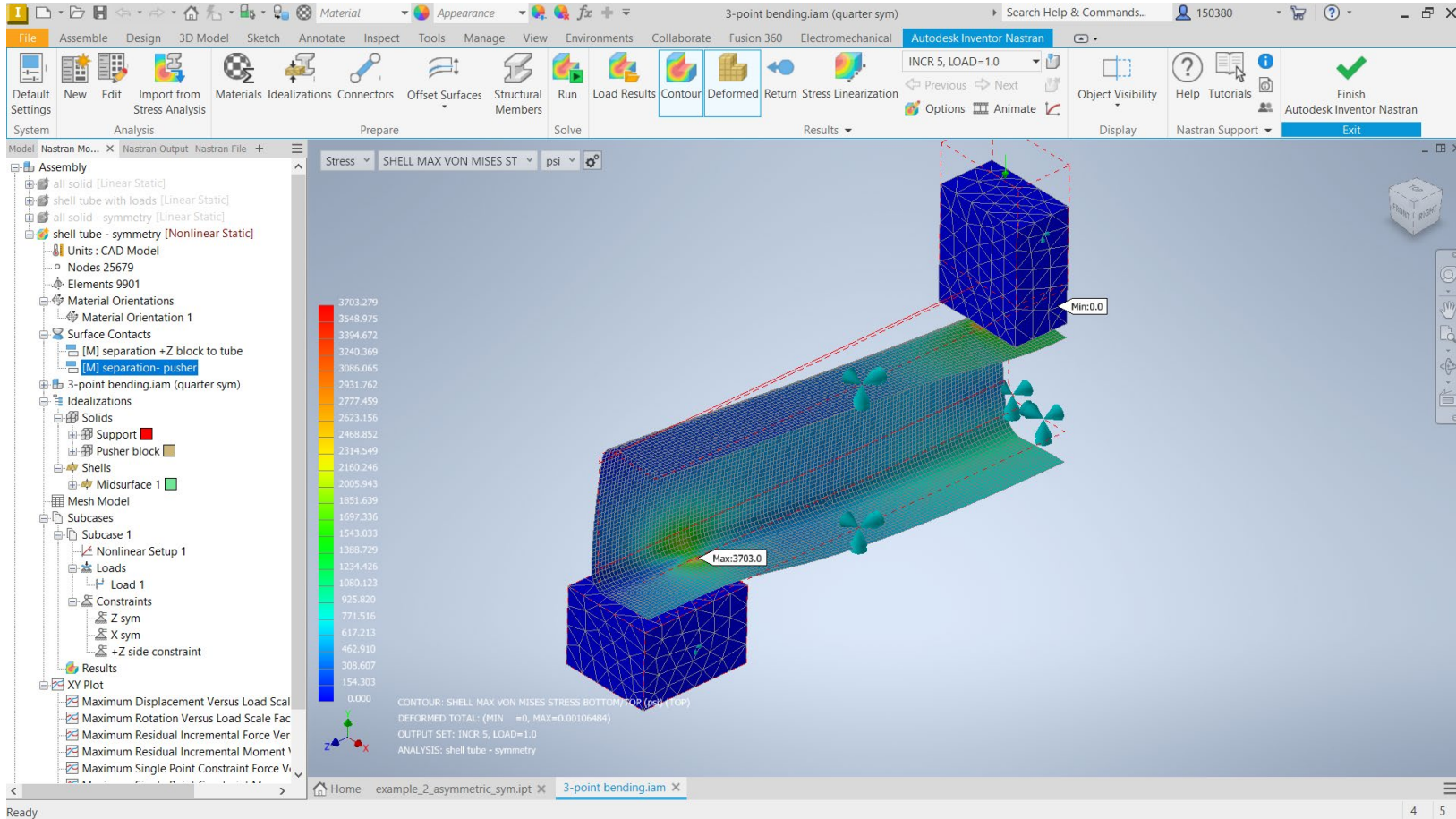
Shell Mesh Convergence Study

- Enhanced contact is OFF
- Mesh element size is varied from 0.1 to 0.02
- The peak stress does not increase with the same rate as it did with solids
- Solution time still goes up exponentially with model size or with the inverse of element size

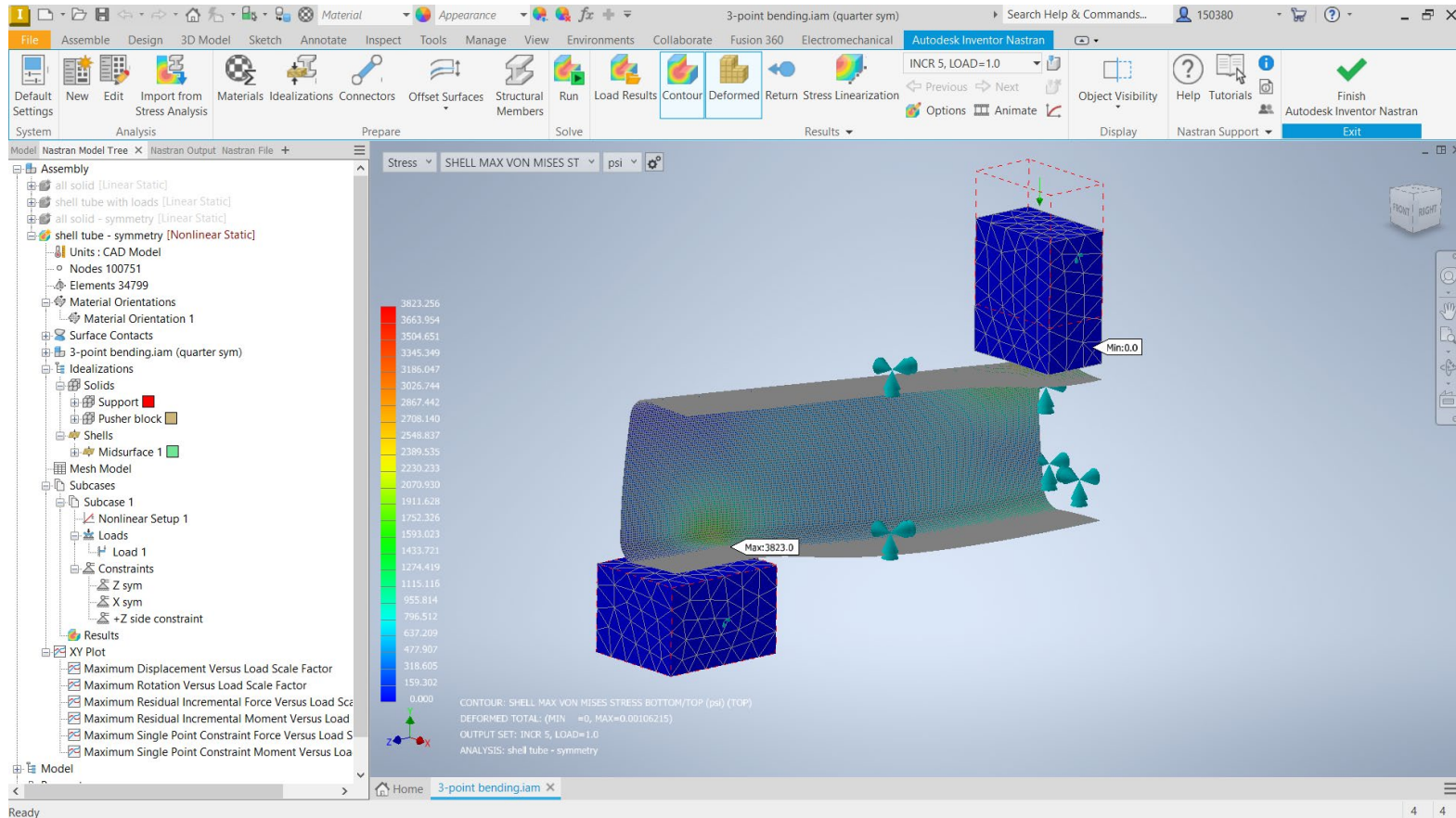
Affect of Element Size on Solution Time and Peak VM Stress



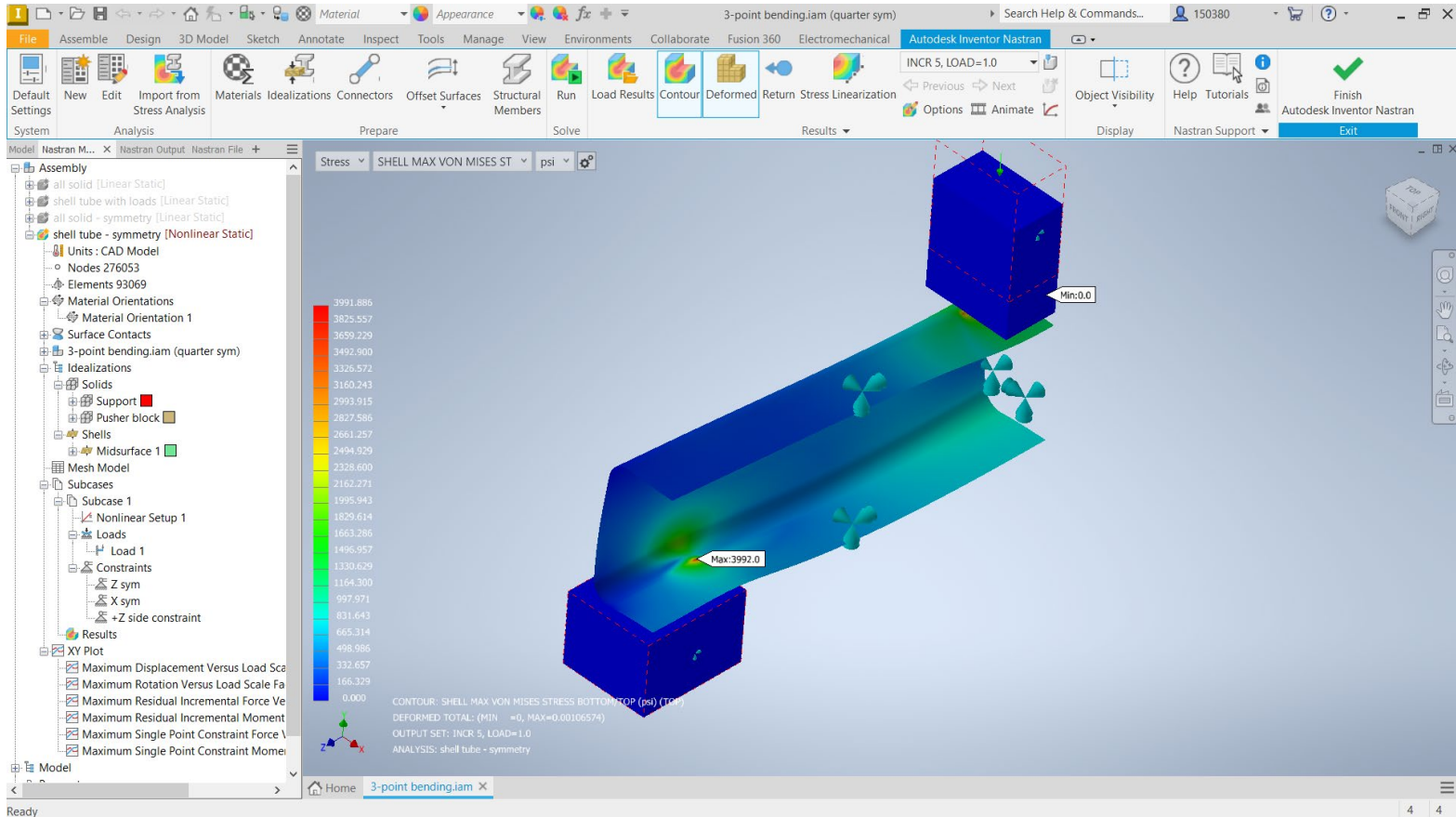
Mesh Convergence – Element Size = 0.1in – 54Sec



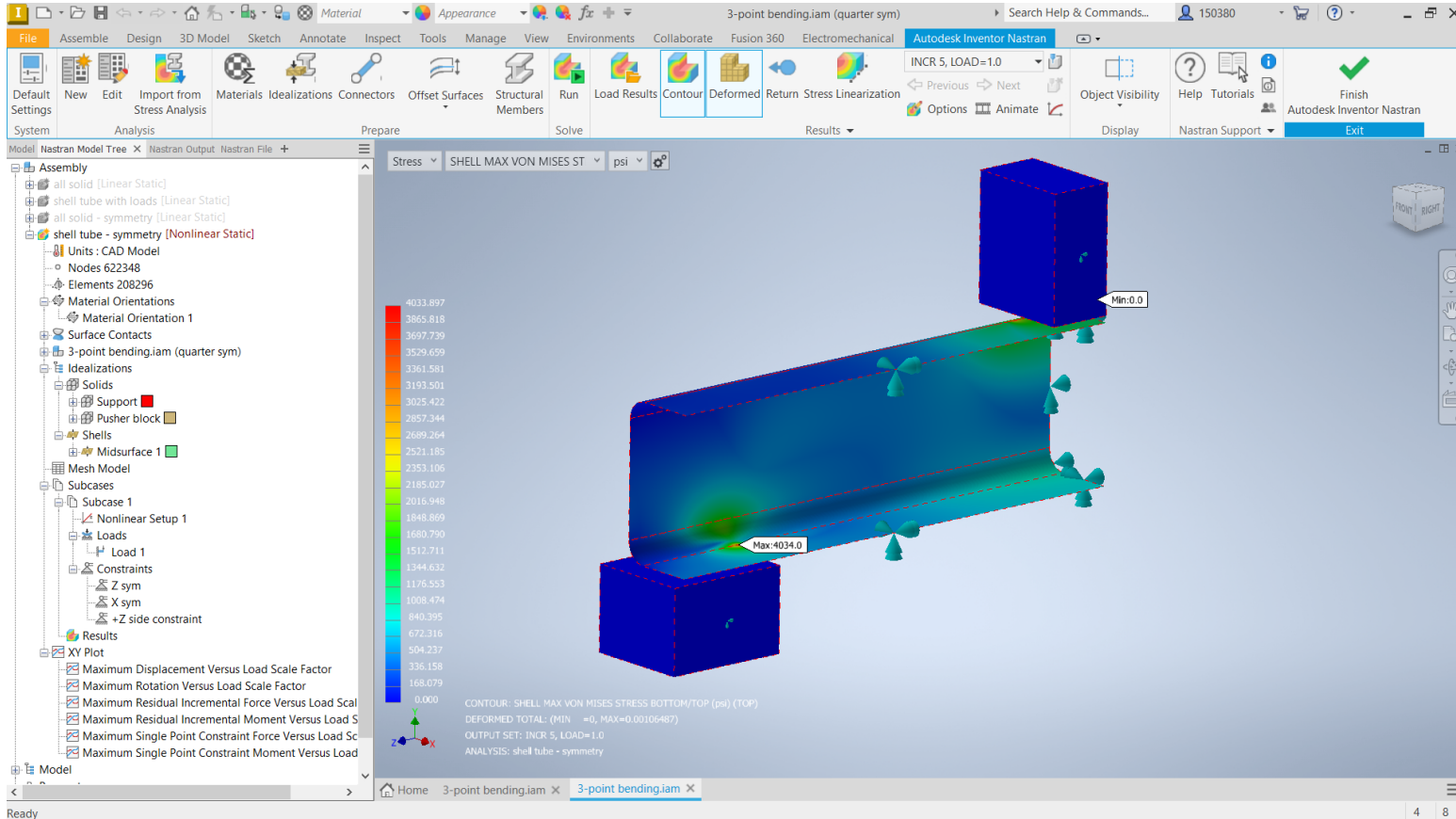
Mesh Convergence – Element Size = 0.05in – 208Sec

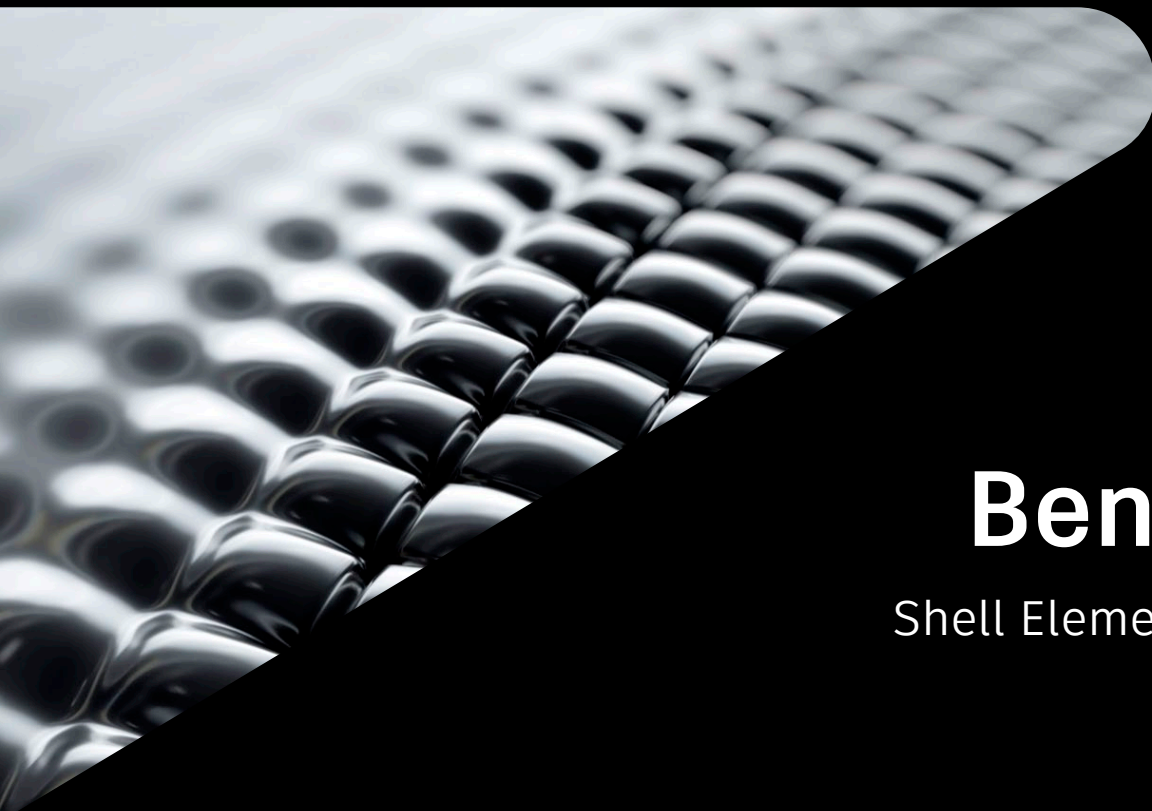


Mesh Convergence – Element Size = 0.03in – 912Sec



Mesh Convergence – Element Size = 0.02in – 2983Sec

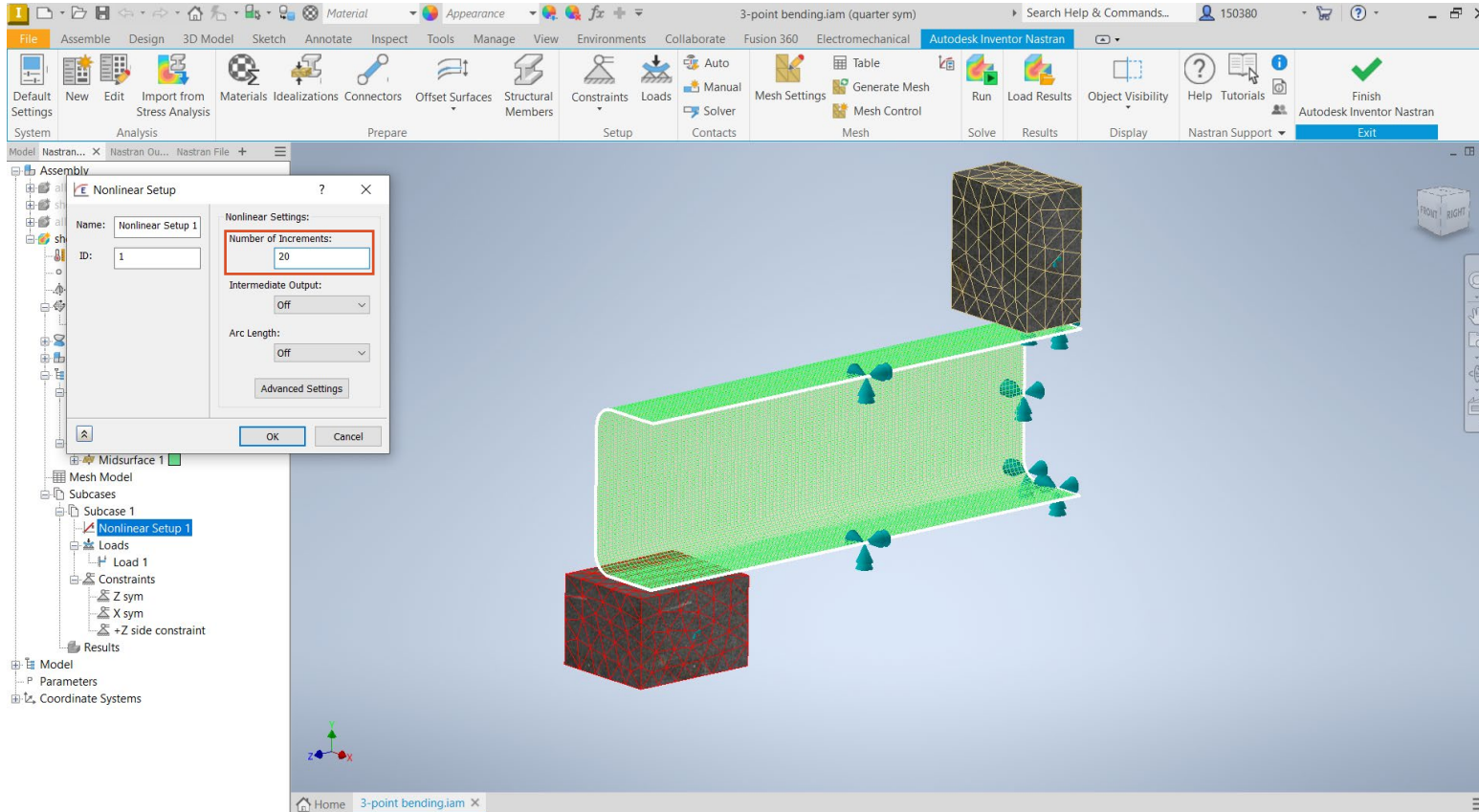




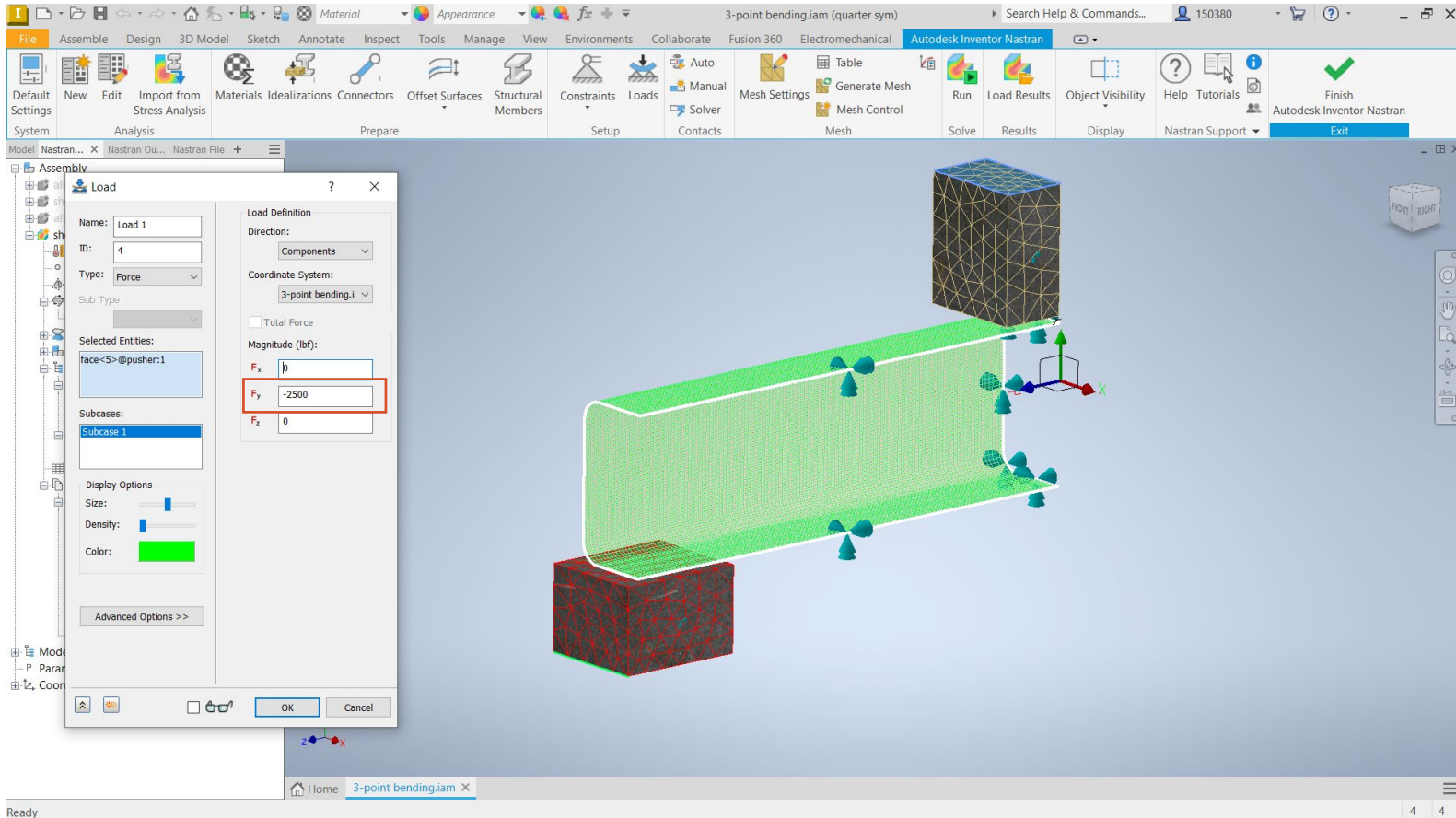
3-Point Tube Bending Example

Shell Element Material Nonlinear Analysis

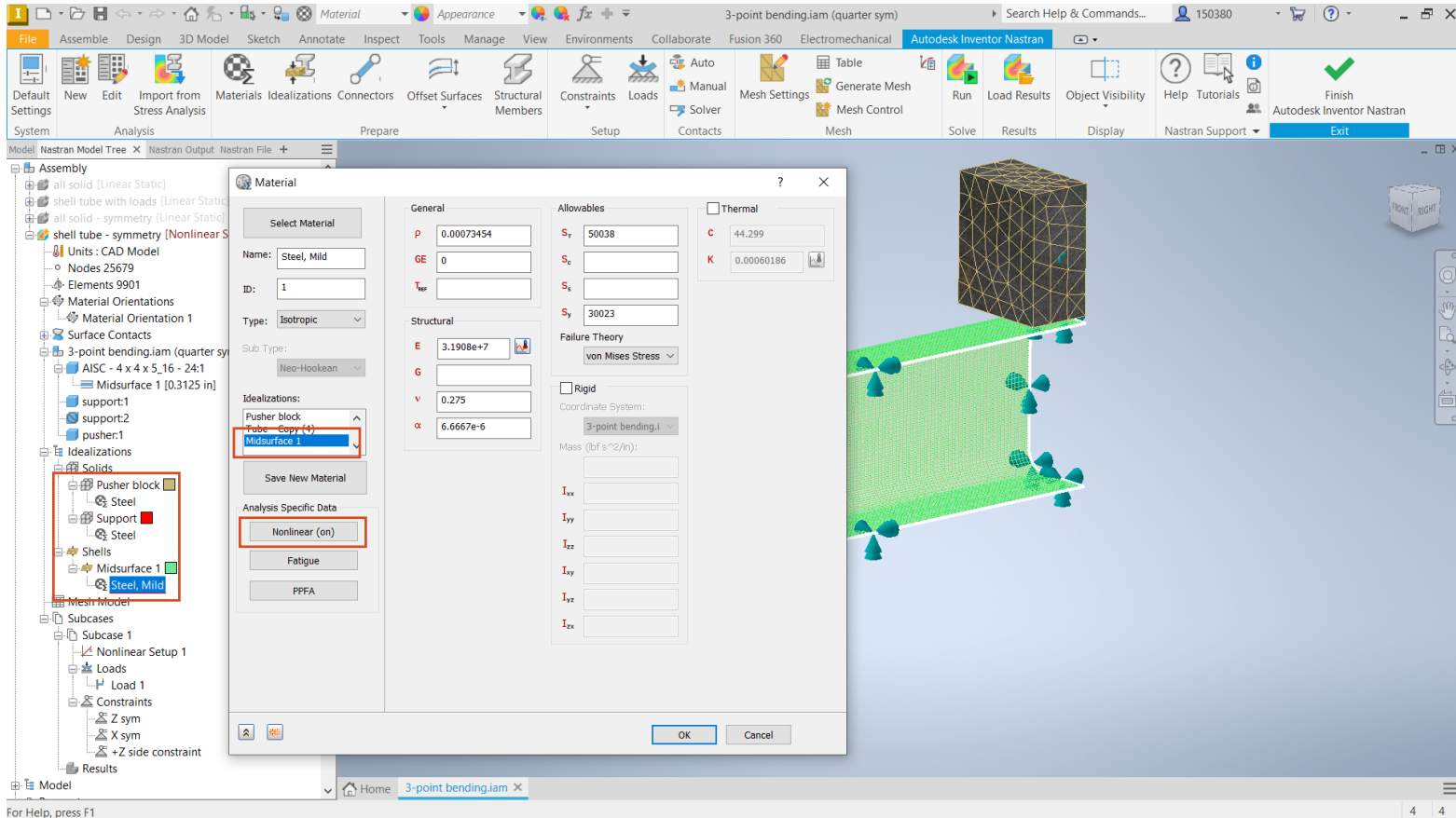
For Material Nonlinear Use 20-40 Increments



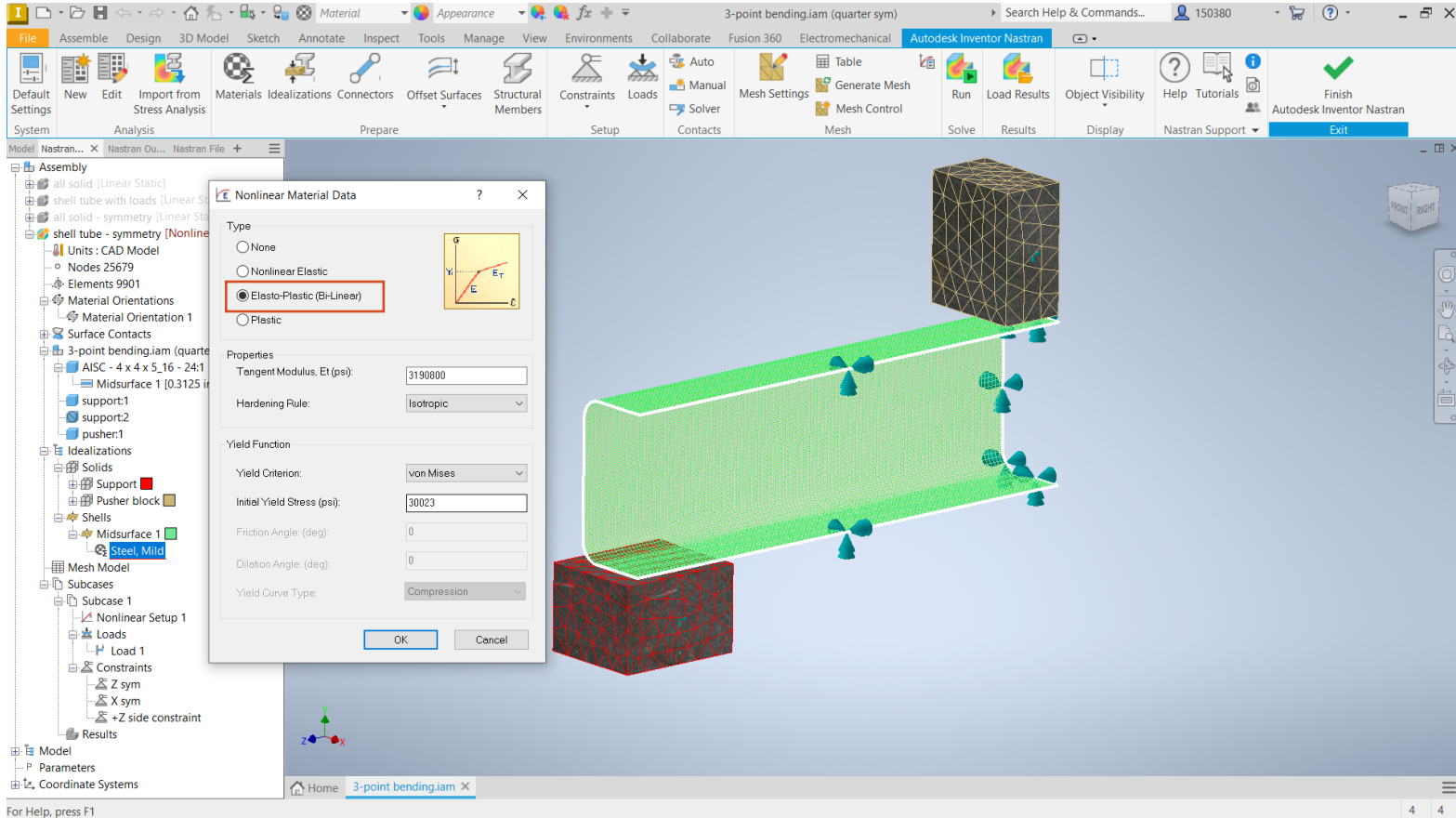
Increase the Load 10x So the Tube Will Yield



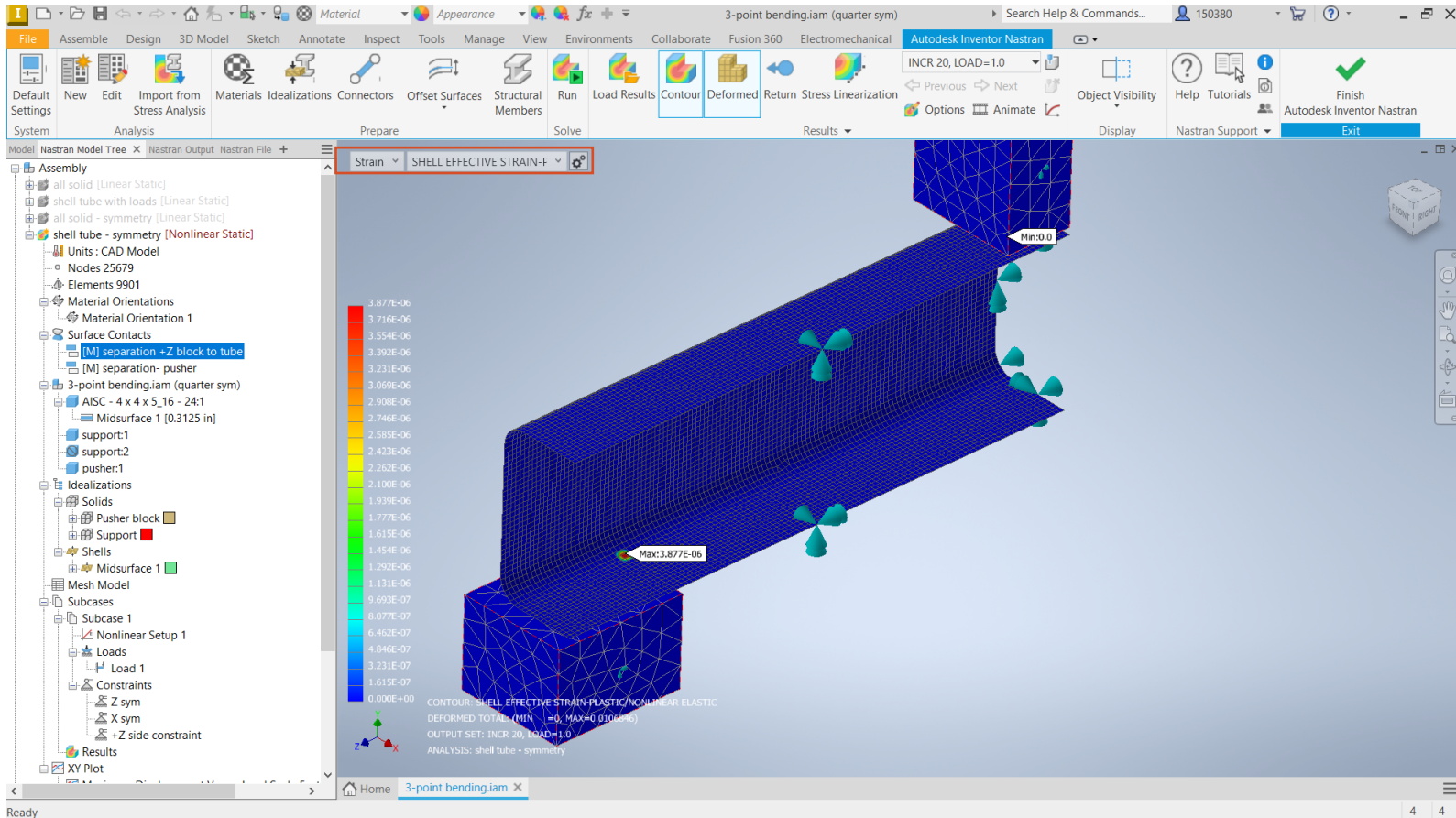
Enable a Nonlinear Material for Tube Only



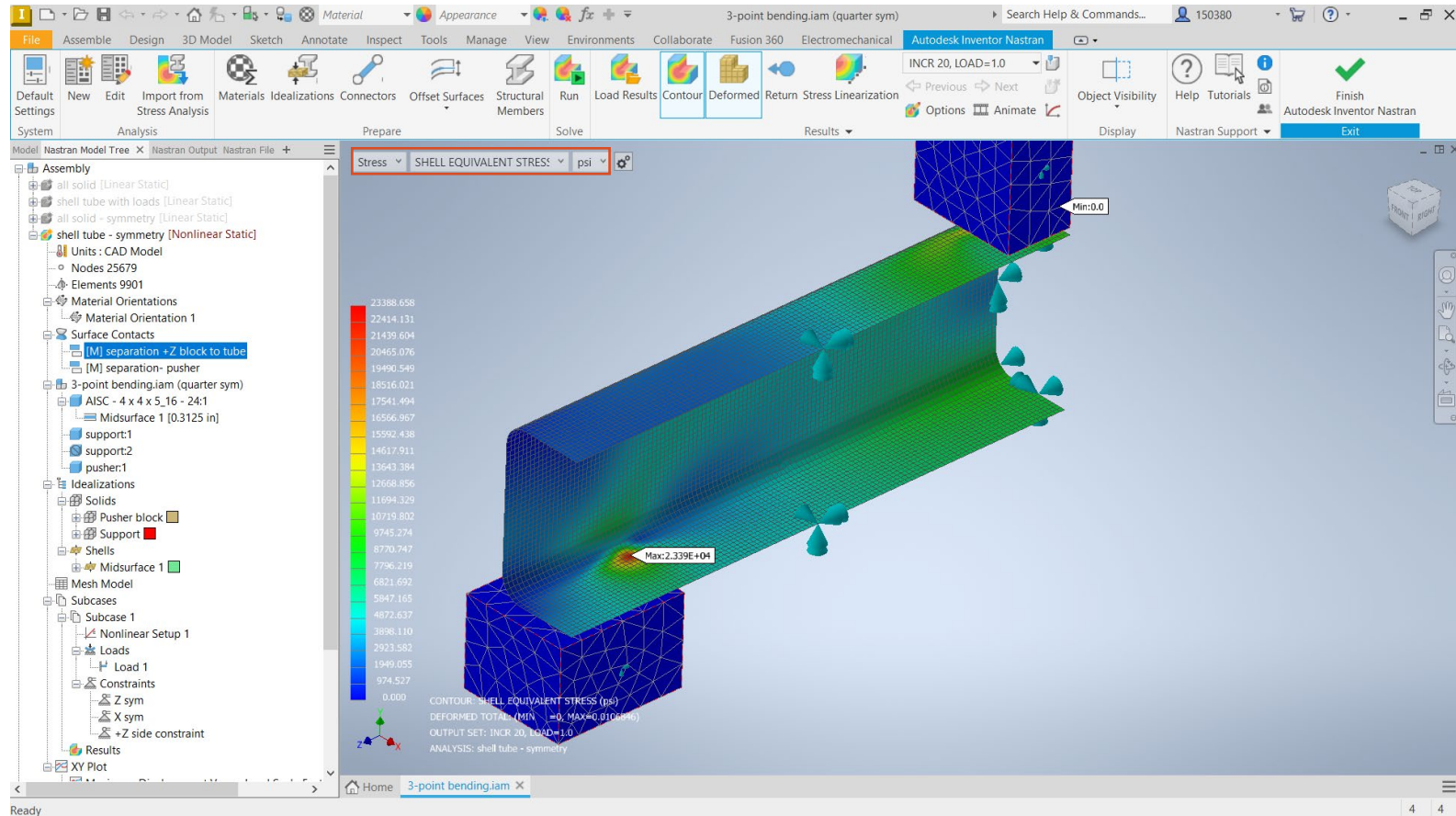
Enable a Nonlinear Material



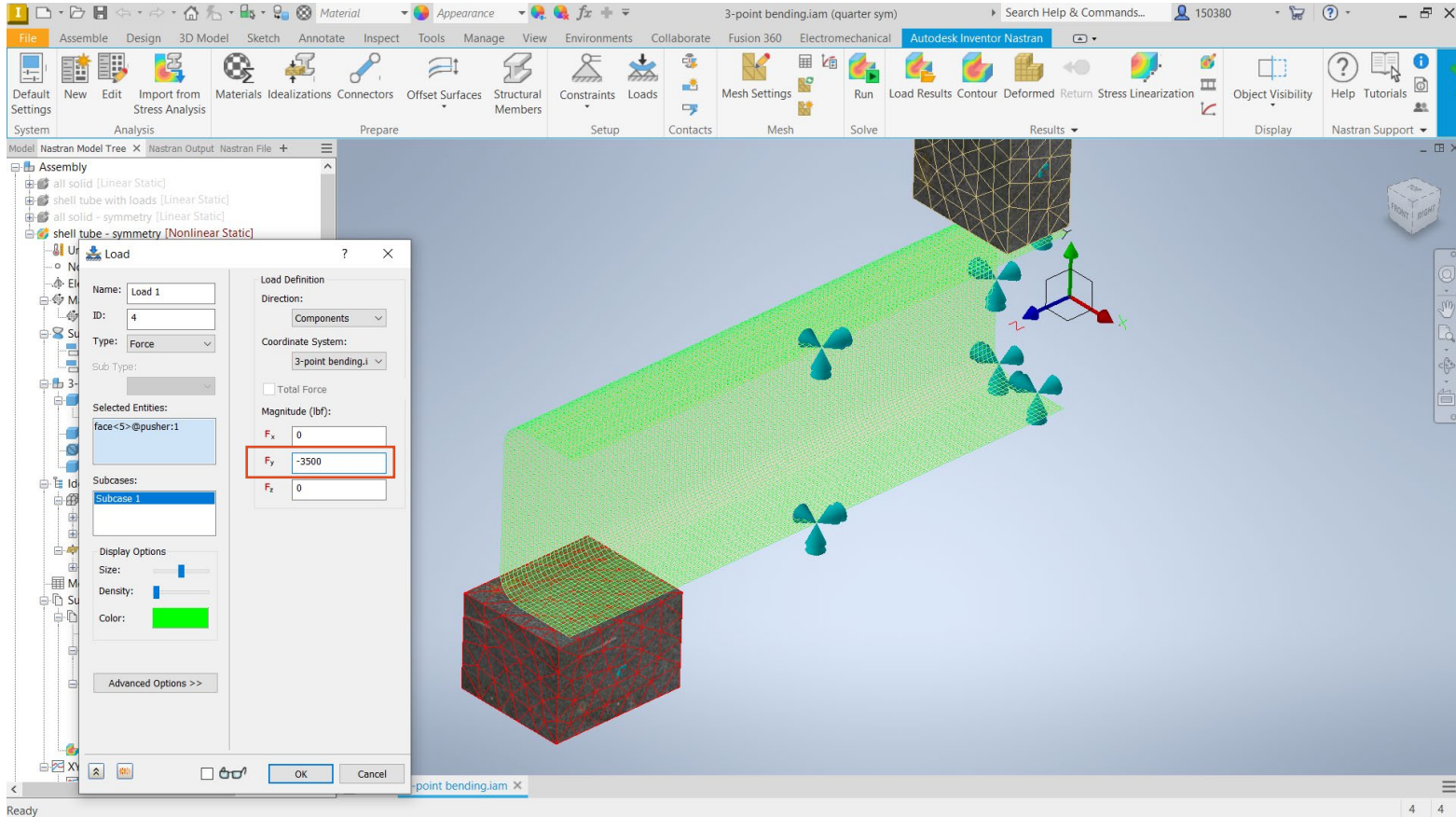
Plastic Strain Due to Yielding



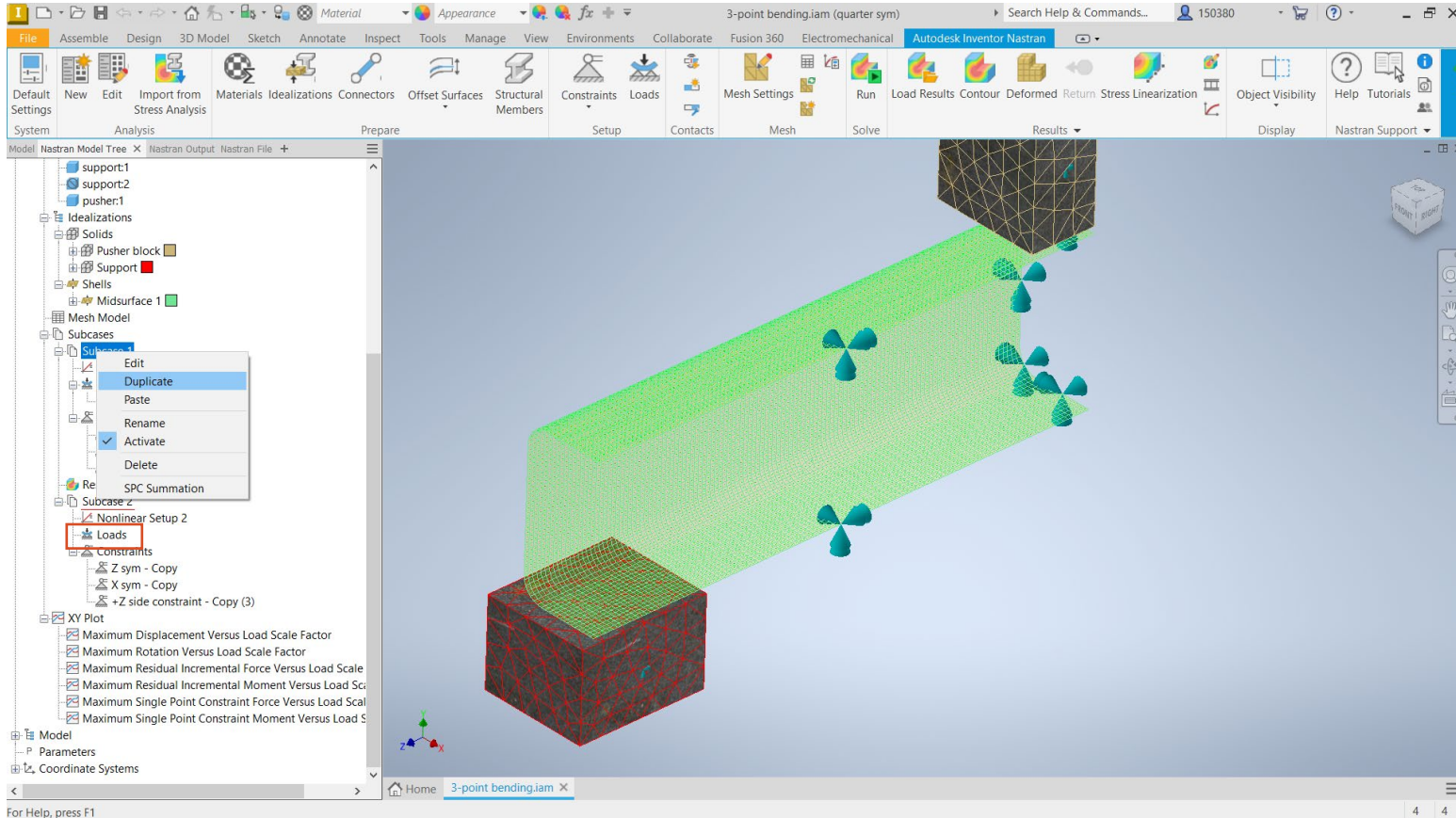
10x Stress is Effectively Less Due to Plastic Strain



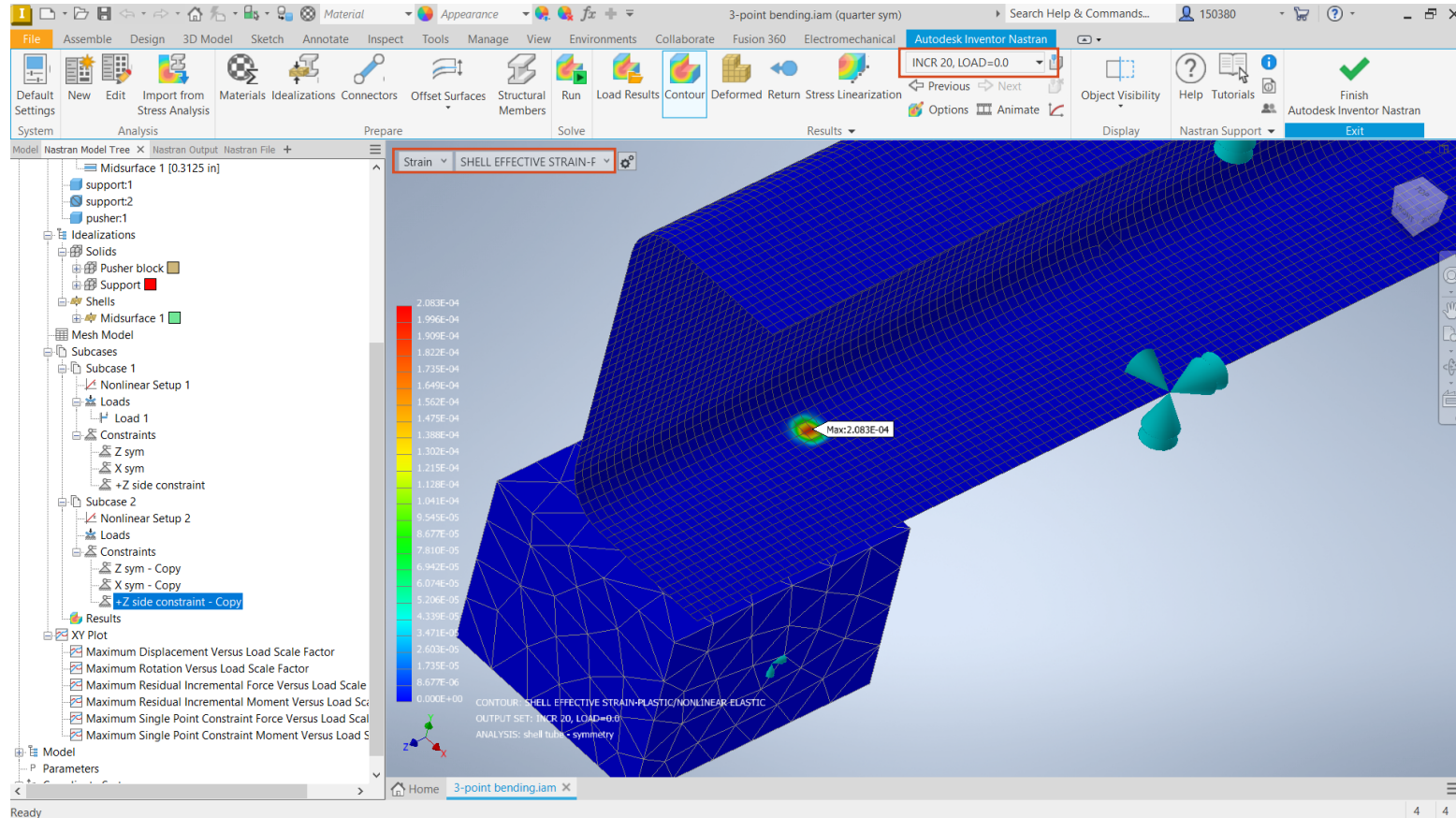
Increase Load Further to See More Plastic Strain



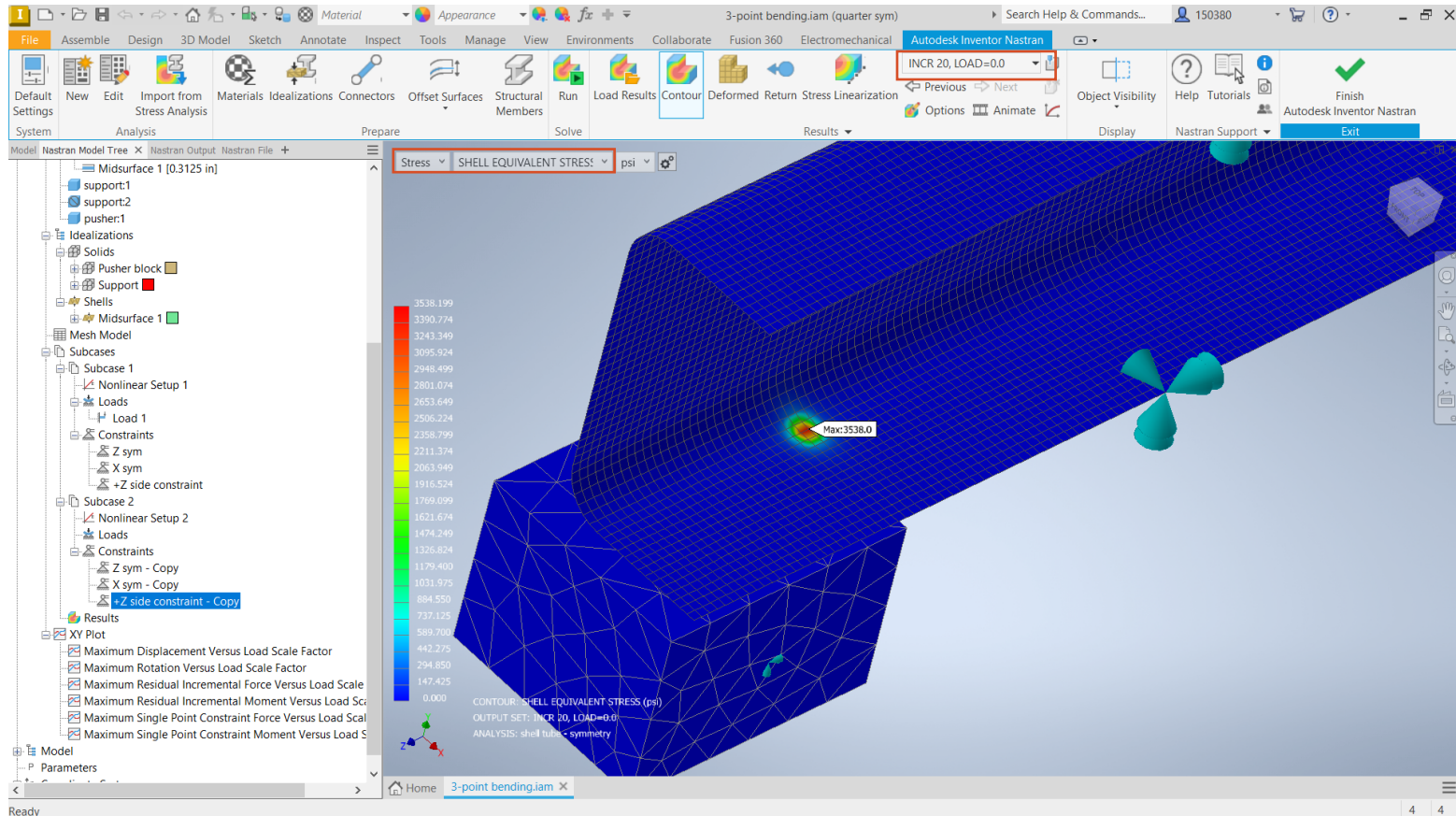
Duplicate Subcase 1 and Delete Load to See Unload

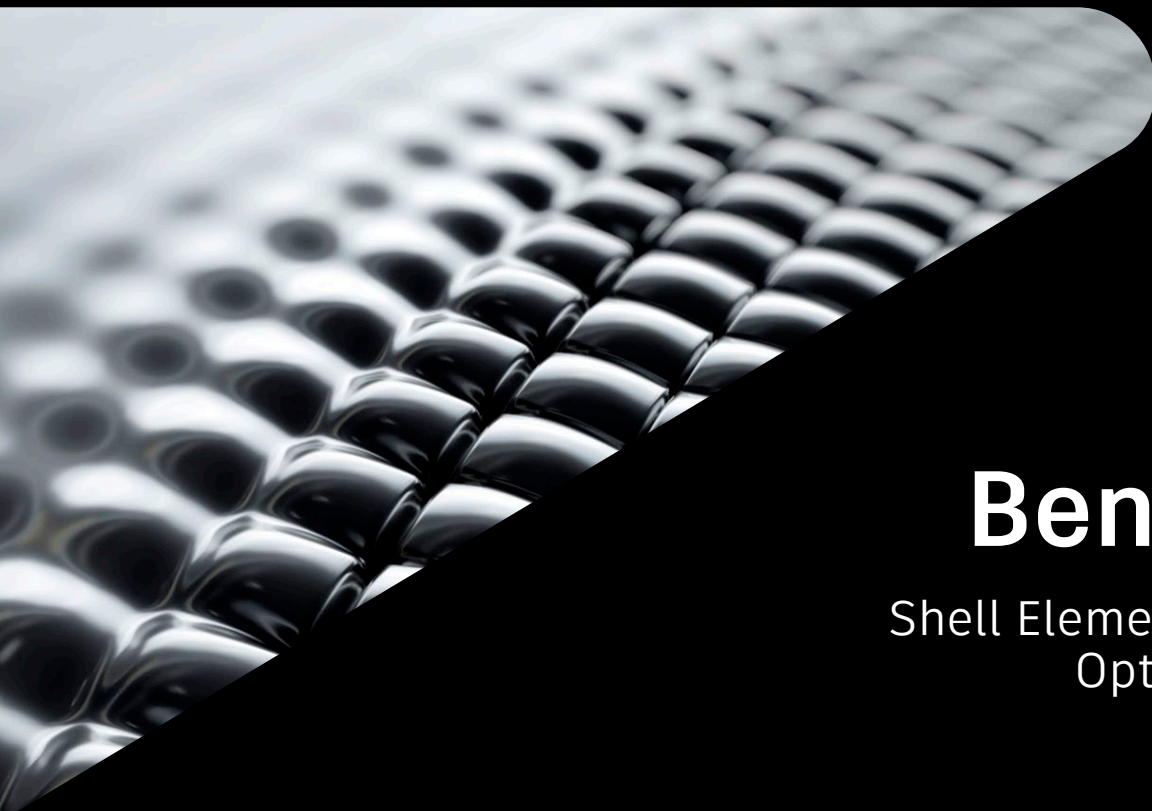


Select the Last Load Case to See Residual Effective Strain



Select the Last Load Case to See Residual Effective Stress





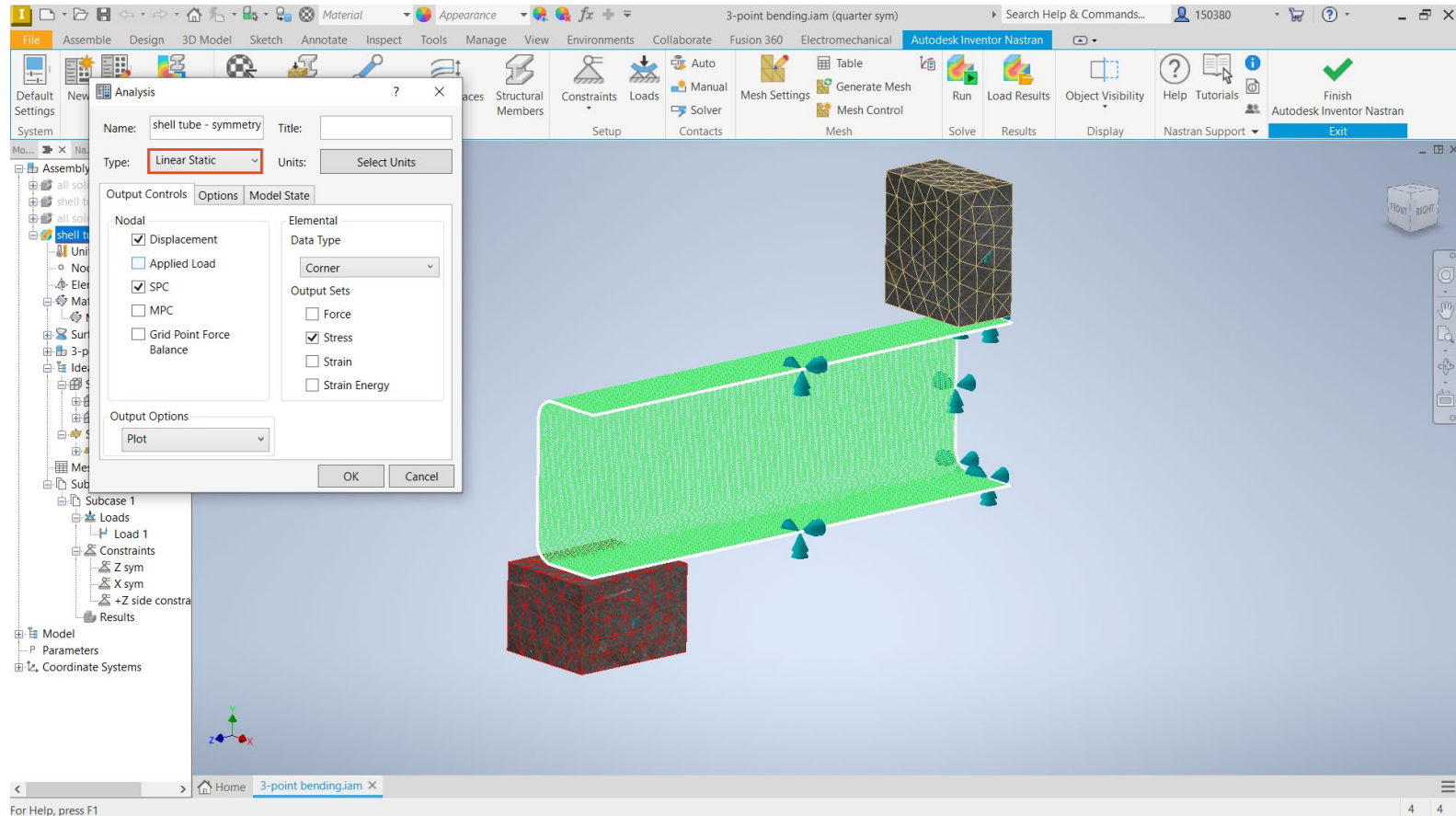
3-Point Tube Bending Example

Shell Element and Solid Element Topology
Optimization and Light-Weighting

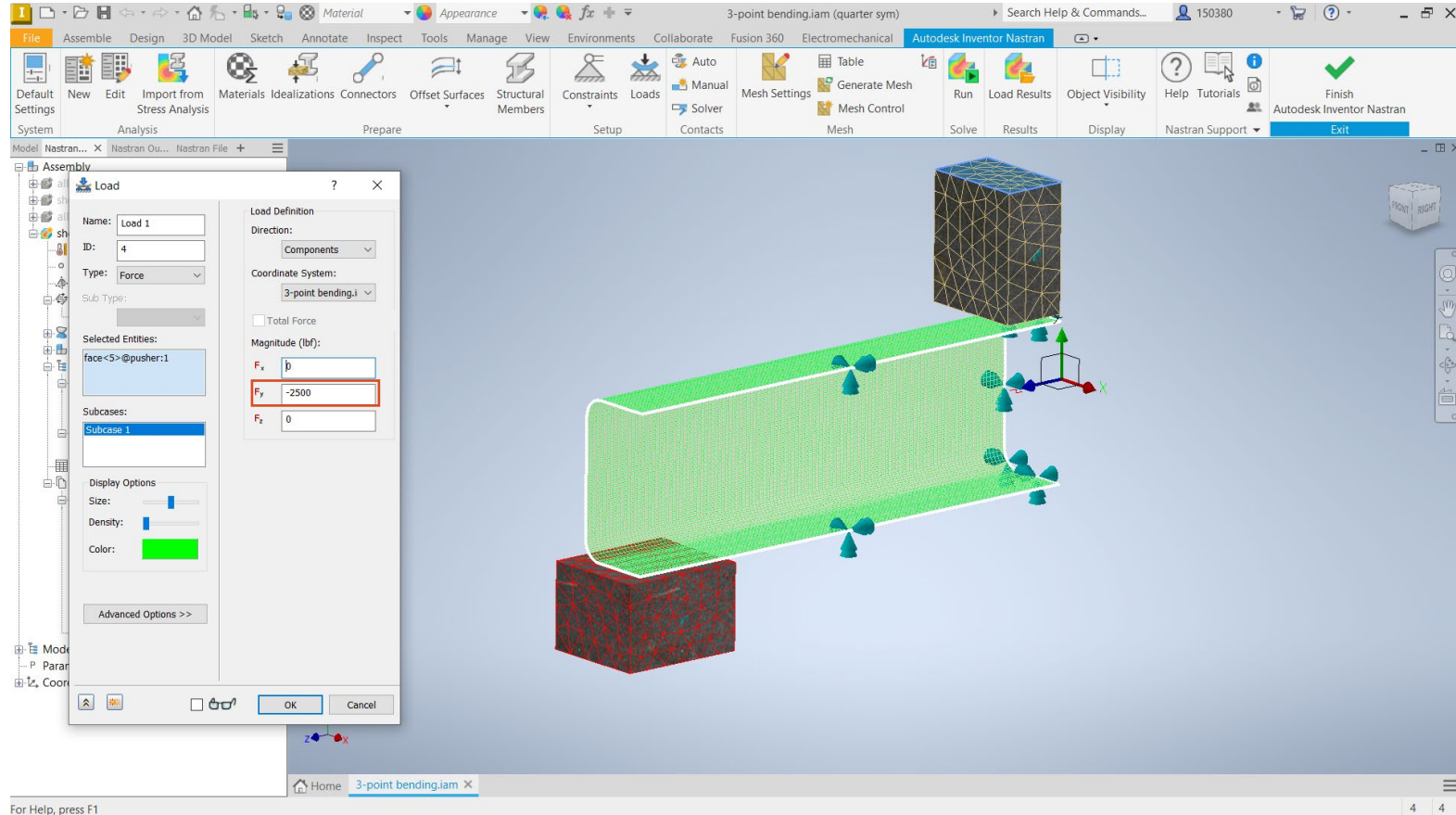
Definitions

- **Objective** – The goal of the design analysis
- **Design Constraint** – Specific limits on results such as displacement at point, temperature, stress, etc.
- **Manufacturing Constraint** – Specifies how a design region will be manufactured such as extruded along an axis or symmetric about a plane
- **Compliance** – The inverse of stiffness
- **Volume Fraction** – The ratio of full volume to reduced volume (effectively the same as mass fraction when density is constant in a design region)
- **Design Sensitivity** - The gradient (change) of the objective (or constraint) with respect to the design variable (element density)

Change Solution Type Back to Linear Statics



Change Solution Type Back to Linear Statics



Enable Topology Optimization Using the TOPGEN Parameter

The screenshot displays the Autodesk Inventor Nastran software interface. The main window shows a 3D model of a 3-point bending test setup. The left-hand tree view lists the assembly structure, including the model, constraints, and results. The top ribbon contains various toolbars for design, analysis, and simulation. The 'Parameters' dialog box is open, showing the 'Design Optimization Processor Parameters' section. The 'TOPGEN' parameter is set to 'COMPVF'. The 'Output Control Directives' section is also visible, showing various output options.

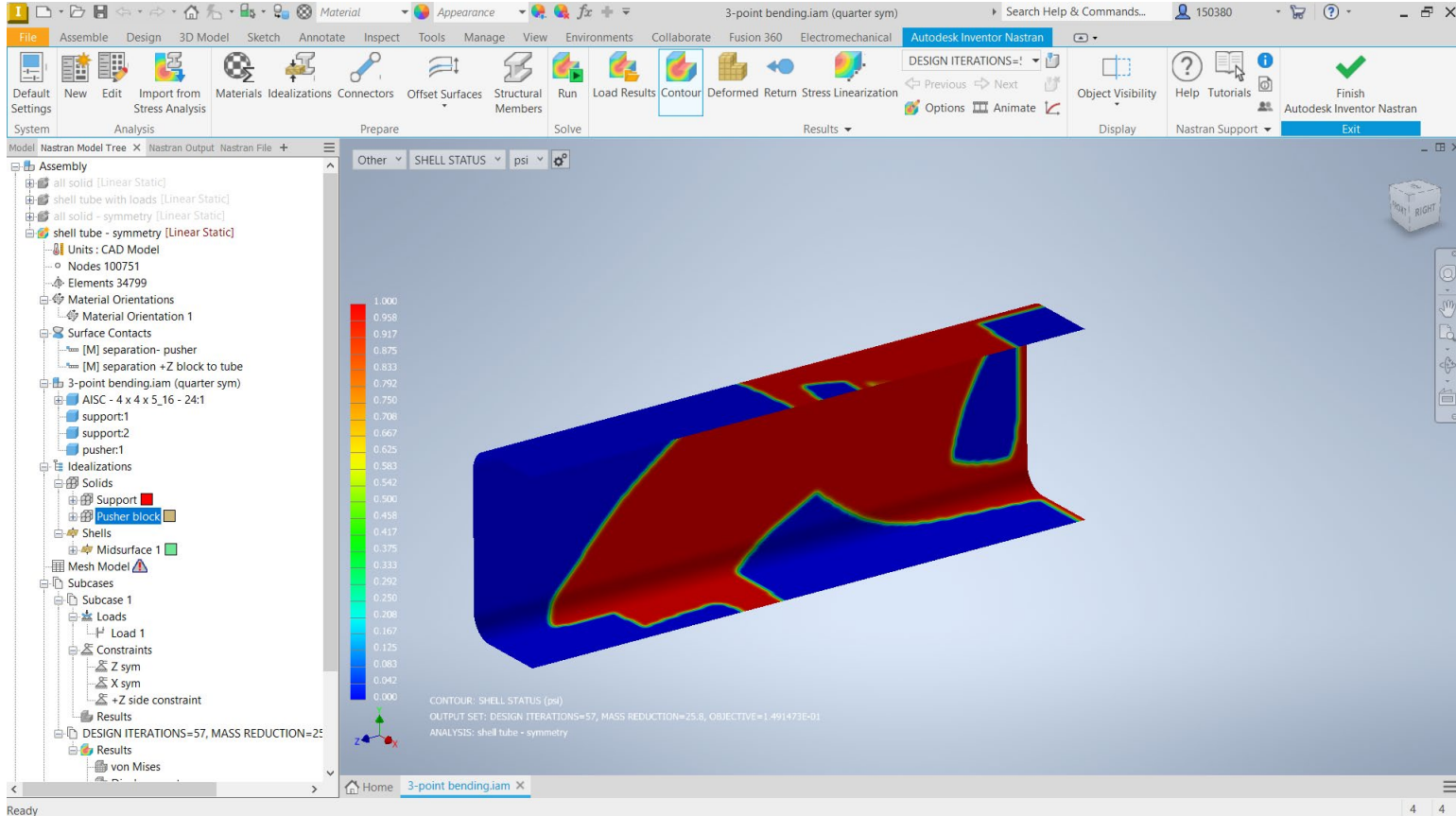
Parameters Dialog Box - Design Optimization Processor Parameters

Parameter	Value
TOPTCOMPINDEX	1.0E+10
TOPTDATABASE	DELETE
TOPTDEPXITER	AUTO
TOPTDESIGNCONSTR	0.5
TOPTDESIGNMODE	ACCURACY
TOPTDESIGNREGION	18
TOPTDESIGNTOL	AUTO
TOPTDTHRESHOLD	1.0E-3
TOPTTELEMEXTTOL	1.0E-2
TOPTTELEMSYMTOL	1.0E-2
TOPTGEN	COMPVF

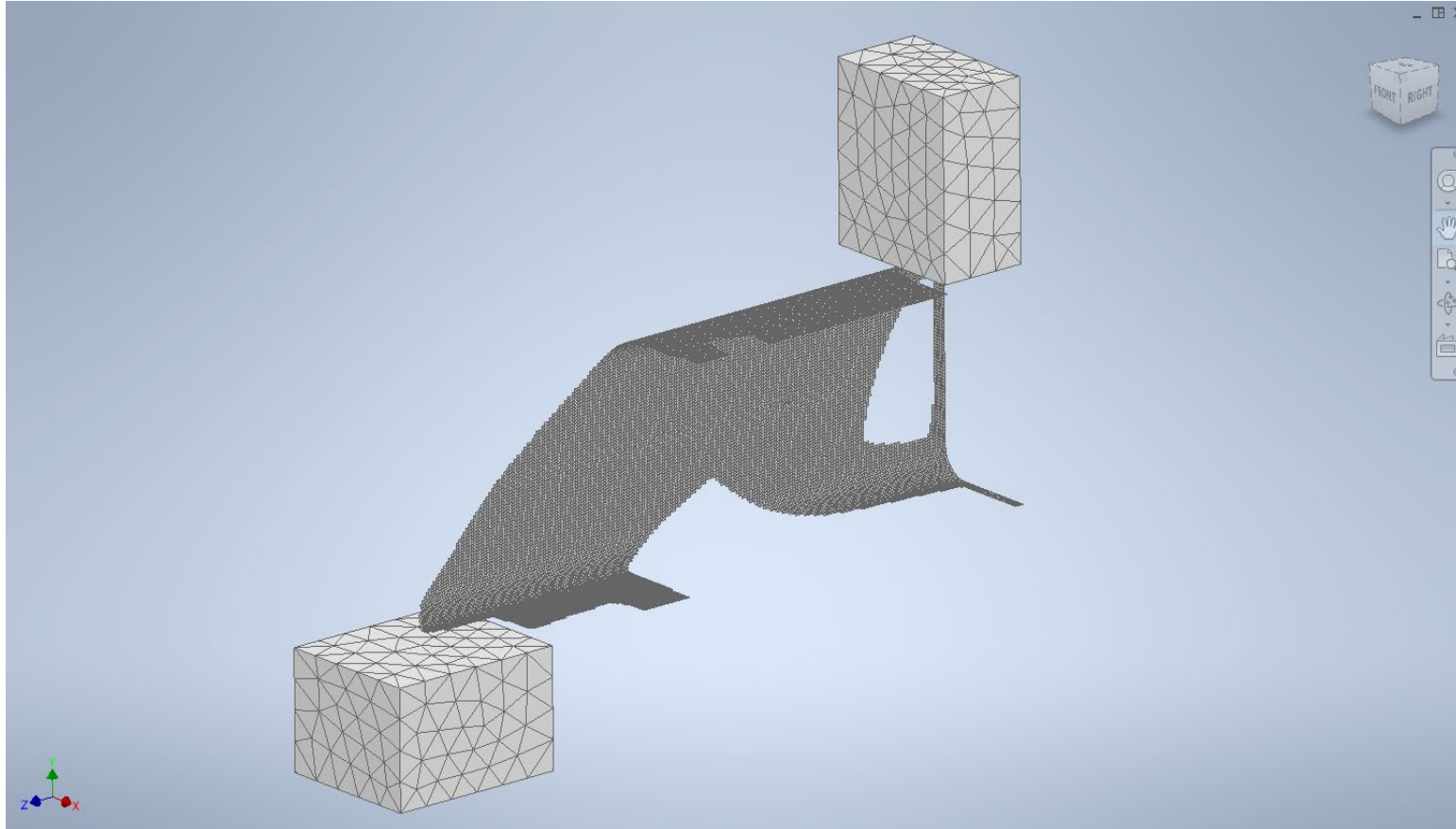
Parameters Dialog Box - Output Control Directives

Parameter	Value
TRSLDISPDATA	OFF
TRSLDMIDATA	OFF
TRSLLOADDATA	OFF
TRSLMODLDATA	OFF
TRSLPRESDATA	OFF
TRSLRBSADATA	OFF
TRSLSPCADATA	OFF
TRSLSTRNDATA	OFF
TRSLTEMPDATA	OFF
TRSLTOPTDATA	ON
TRSLTOQEDATA	OFF
XYPLOTCSVOUT	OFF

Min Compliance with a Fixed VF Constraint



Generated .STL File



Min Mass with a Stress and Compliance Index Constraint

The screenshot displays the Autodesk Inventor Nastran interface for a 3-point bending analysis. The Nastran Model Tree on the left shows the model structure, including supports, loads, and constraints. The Parameters dialog box is open, showing various Nastran parameters. The background shows a 3D model of a tube under stress, with a color-coded stress distribution.

Nastran Model Tree:

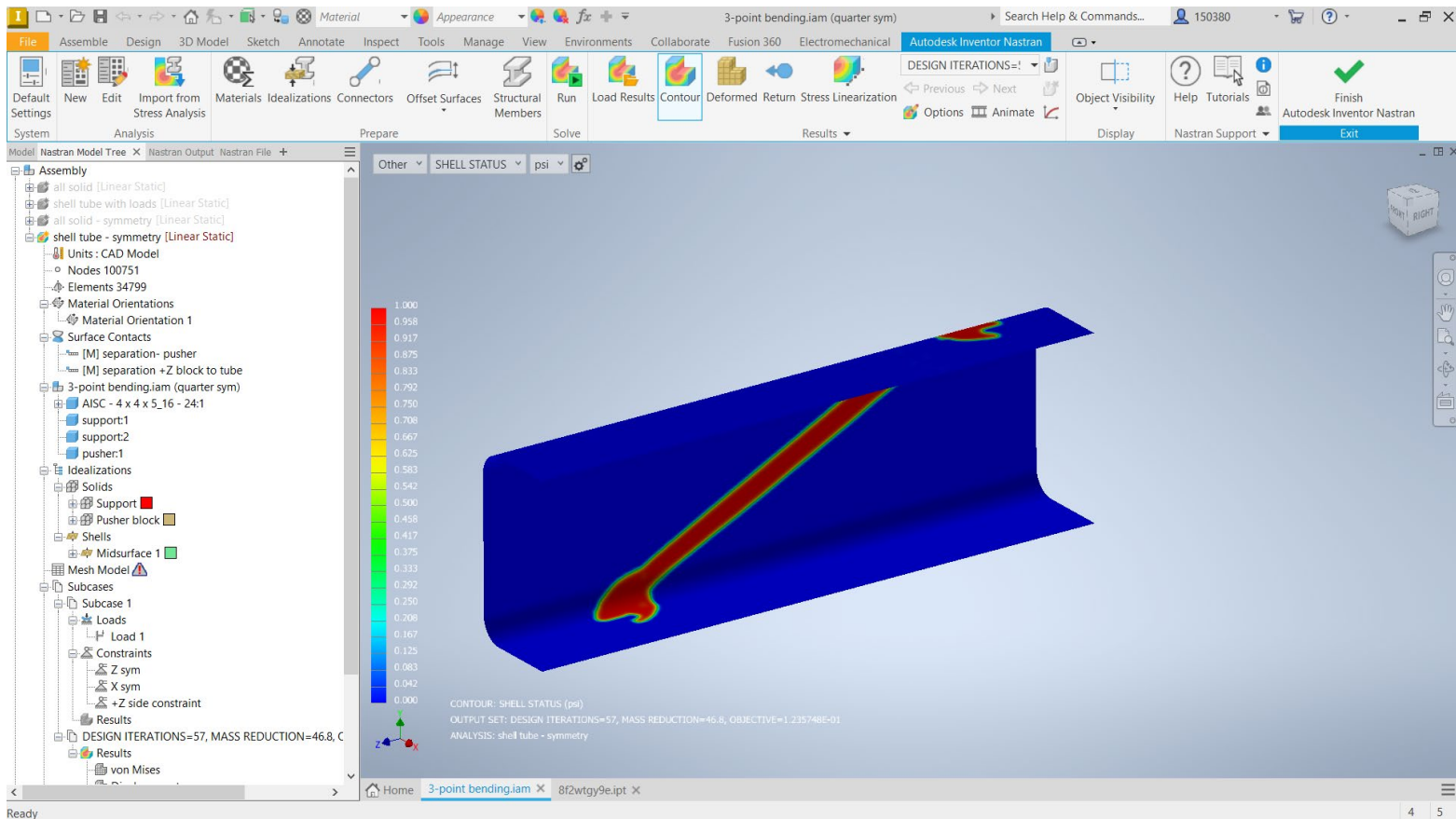
- Model
- Elements 34799
- Material Orientations
- Surface Contacts
 - [M] separation- pusher
 - [M] separation + Z block to tube
- 3-point bending.iam (quarter sym)
- AISC - 4 x 4 x 5.16 - 24:1
 - support:1
 - support:2
 - pusher:1
- Idealizations
 - Solids
 - Support
 - Pusher block
 - Shells
 - Midsurface 1
- Mesh Model
 - Subcases
 - Subcase 1
 - Loads
 - Load 1
 - Constraints
 - Z sym
 - X sym
 - + Z side constraint
 - Results
 - DESIGN ITERATIONS=57, MASS REDUCTION=25.8, C
- Results
 - von Mises
 - Displacement
 - Safety Factor
 - Deformed
- XY Plot

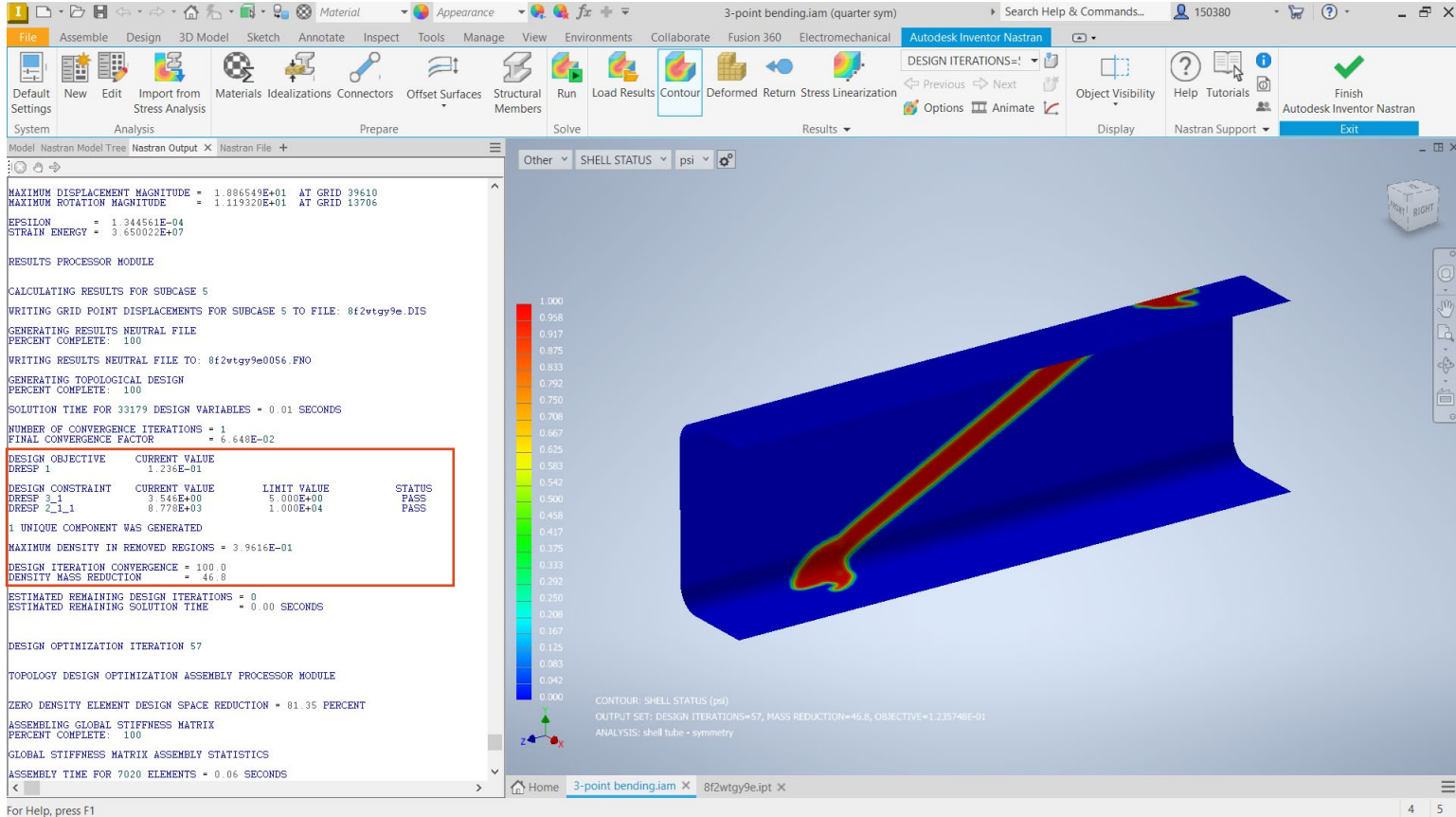
Parameters Dialog Box:

Category	Parameter	Value
Memory Management Directives	NTOPTSTRESSDIV	AUTO
	TOPTCOMPINDEX	5
Program Control Directives	TOPTDESIGNCONSTR	10000.0
	TOPTDESIGNREGION	18
Model Translator Parameters	TOPTDTHRESHOLD	1.0E-3
	TOPTGEN	VFSTRESS
Geometry Processor Parameters	TOPTMANCONSTR	DISABLE
	TOPTMANCORD	VFSTRESS
Solution Processor Parameters	VFDISP	VFSPCF
	VFFREQ	VFFREQ
Eigenvalue Processor Parameters	TOPTMANDIR	
Transient Response Processor Parameters		
Frequency Response Processor Parameters		
Nonlinear Solution Processor Parameters		
Results Processor Parameters		
Design Optimization Processor Parameters		

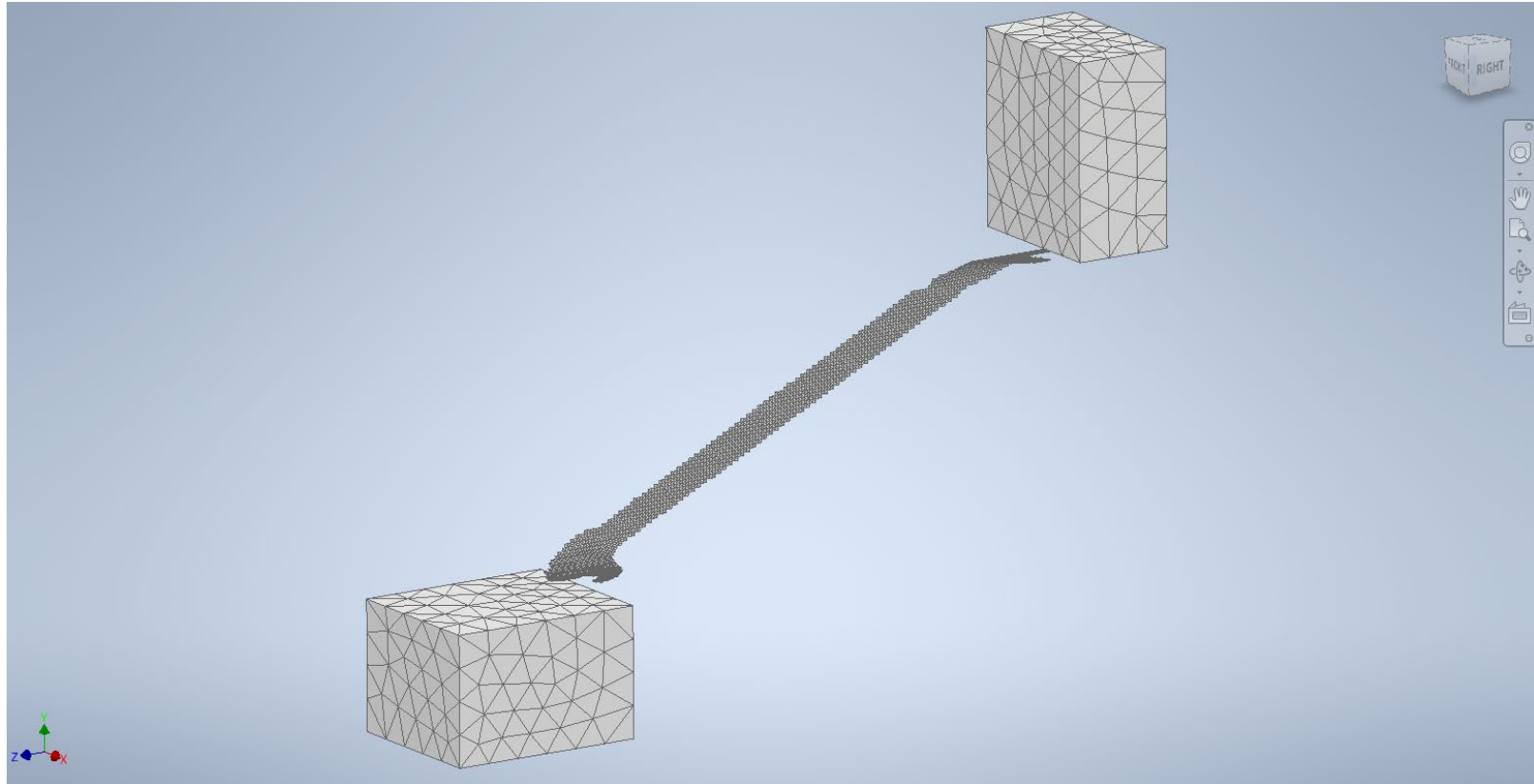
Output:

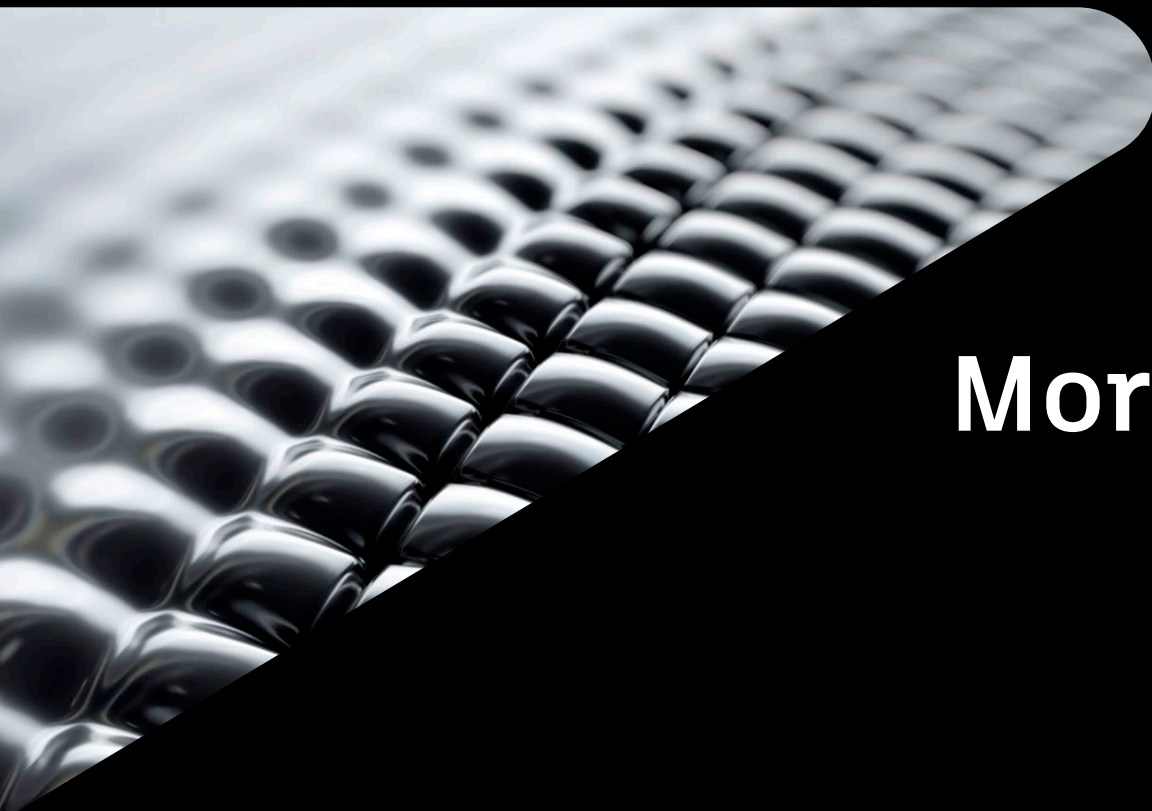
CONTOUR: SHELL STATUS (psi)
OUTPUT SET: DESIGN ITERATIONS=57, MASS REDUCTION=25.8, OBJECTIVE=1.491473E-01
ANALYSIS: shell tube - symmetry





Generated .STL File

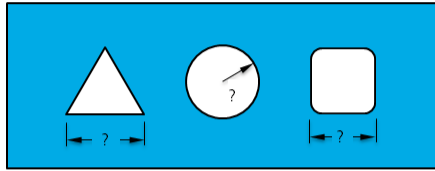




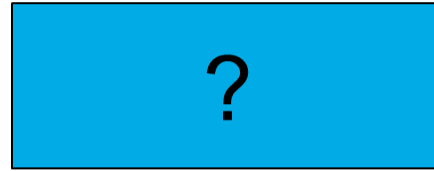
More On Inventor Optimization

Topology Optimization

- Shape optimization: Maintain the topology, change dimensions
- Topology optimization: Determine layouts



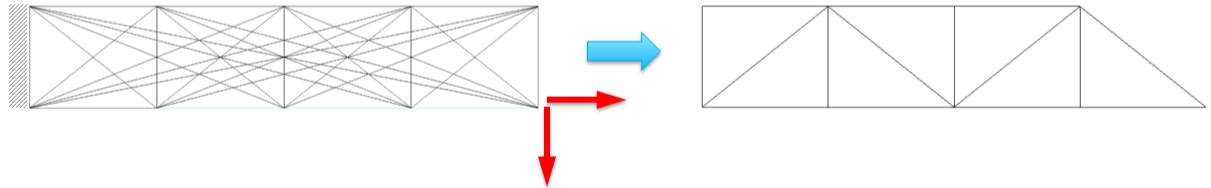
Shape Optimization



Topology Optimization

- Method with Finite Element Analysis

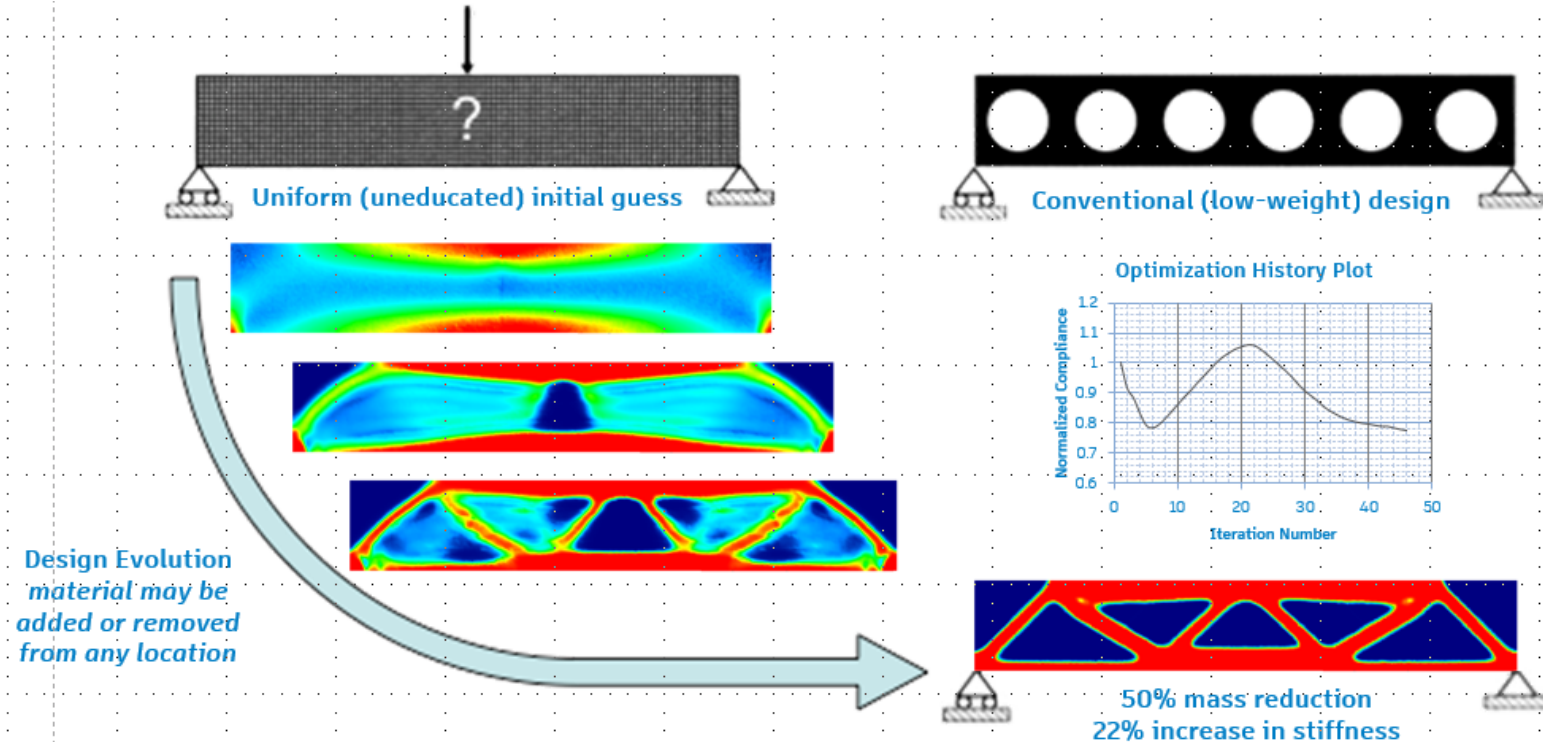
- Initial method...truss



- Change member area and remove when area goes to zero
 - Discrete variables, predetermined nodal locations

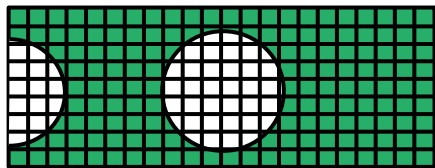
Topology Optimization with FEA

- Determination of optimal principal material distribution for a given problem
- A powerful tool for concept design stage



Topology Optimization Using SIMP - Nastran

- For fixed mesh, determine density (x_e) of each element



$x_e = 0$: void
 $x_e = 1$: material

} Design variable

- Structural volume

$$V(\mathbf{x}) = \sum_{e=1}^{NE} x_e v_0$$

v_0 : volume of an element

- Element stiffness

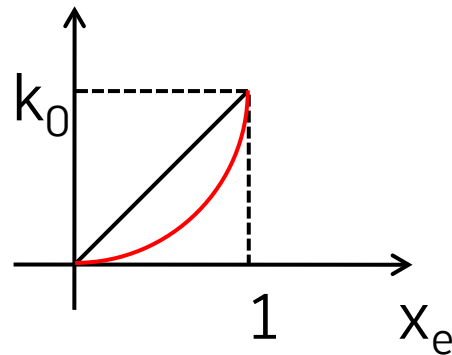
$$[\mathbf{k}_e] = (x_e)^p [\mathbf{k}_0]$$

- SIMP = Solid Isotropic Material Penalization

(Not limited to isotropic materials)

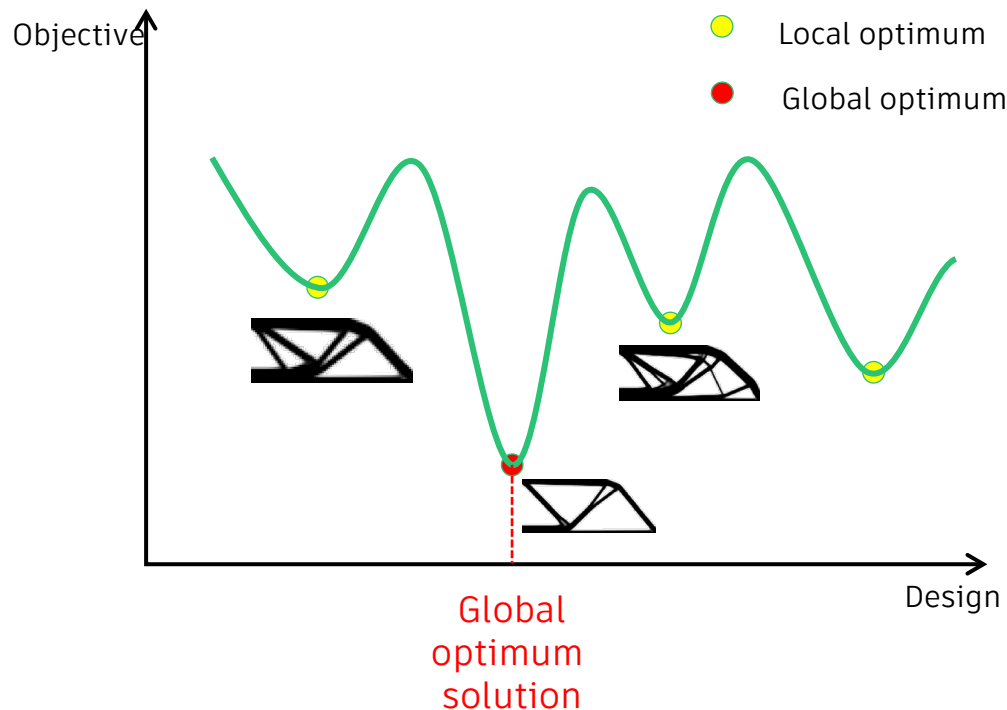
- Exponent p :

- Reduce grey area, force zero or one
- Typically, $p = 3$



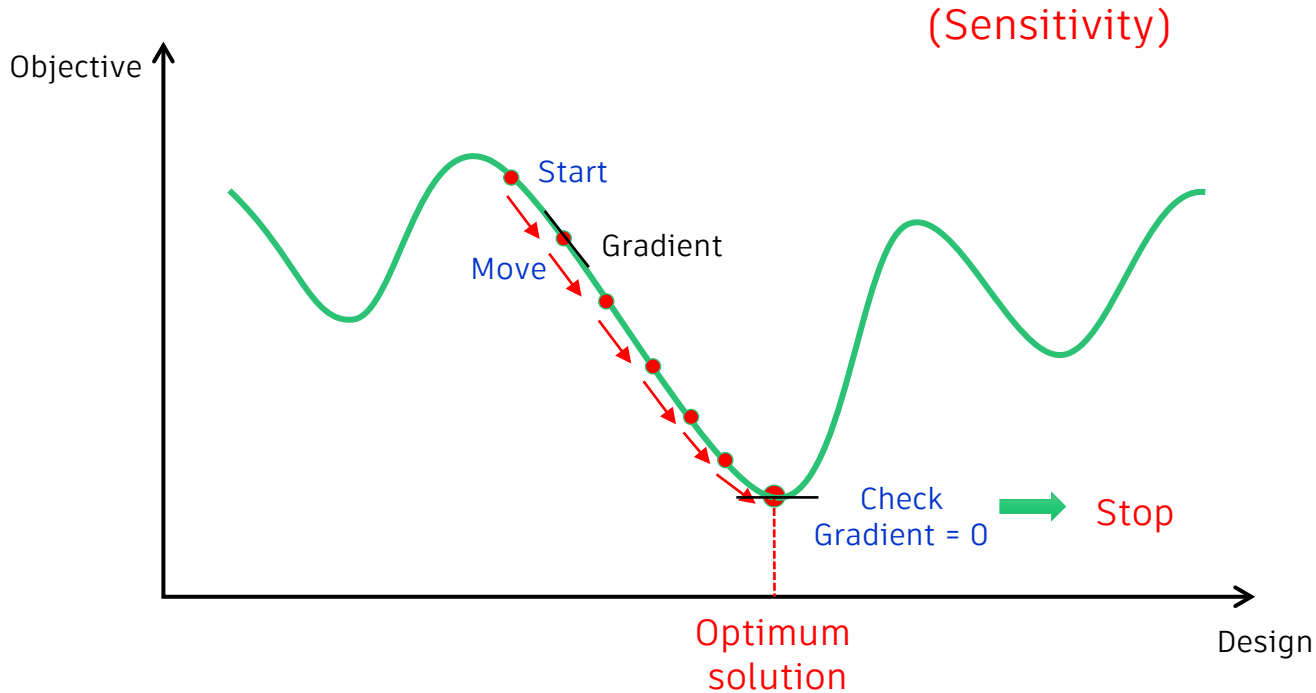
Global Versus Local Minimum

- Optimization algorithm searches for local minimum...global minimum is not guaranteed
- Starting with different initial volume fractions and different mesh densities will result in different designs



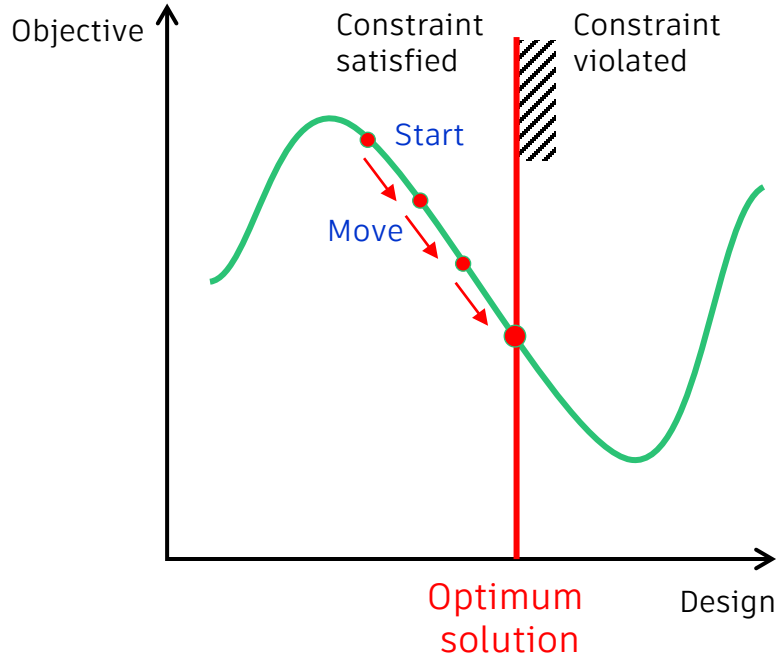
Gradient-based Methods

- We do not know the function before optimization
- We can only evaluate the function and gradient at a given design

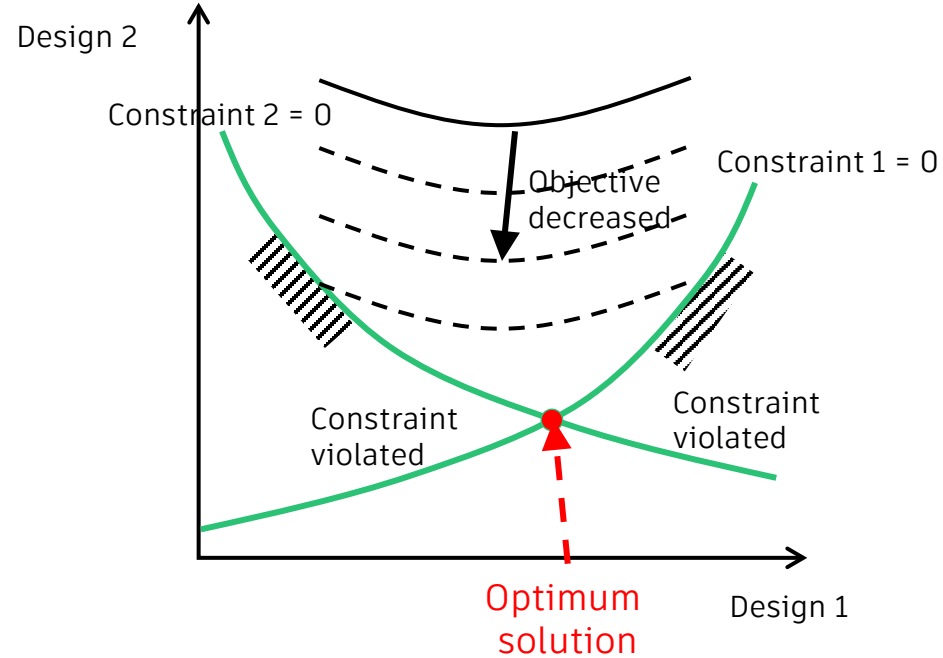


How Constraints Play in Optimization?

Most cases, constraints determine optimal design



Single constraint example



Two constraints example

Nastran Topology Optimization Objectives

Objective	Min/Max/Neither	Multiple Load Cases	Solution Sequence
Compliance	Min	Yes	LS
Compliance Index	Min	Yes	LS
Max Displacement Component in Model	Min	No	LS
Specific Grid Point Displacement Component	Min	No	LS
Max Constraint Force Component in Model	Min	No	LS
Specific Constraint Force Component	Min	No	LS
Stress of a Specific TOPVAR Region	Min	No	LS
Stress of all TOPVAR Regions	Min	No	LS
Volume Fraction (Mass Fraction) of a specific TOPVAR Region	Min	Yes	LS
Volume Fraction (Mass Fraction) of all TOPVAR Regions	Min	Yes	LS
Thermal Energy of a Specific TOPVAR Region (Compliance)	Min	Yes	LSSHT
Thermal Energy of all TOPVAR Regions (Compliance)	Min	Yes	LSSHT
Average Temperature of a Specific Set of Nodes	Either	No	LSSHT
Delta Temperature of a Specific Set of Nodes	Either	No	LSSHT
Global Temperature of a Specific Set of Nodes	Either	No	LSSHT
Normal Modes Frequency	Max	Yes	NM
Normal Modes Eigenvalue	Max	Yes	NM
Buckling Modes Eigenvalue (load factor)	Max	No	LB

LS = Linear Statics, LSSHT = Linear Steady-State Heat Transfer, NM= Normal Modes, LB = Linear Buckling

Nastran Topology Optimization Design Constraints

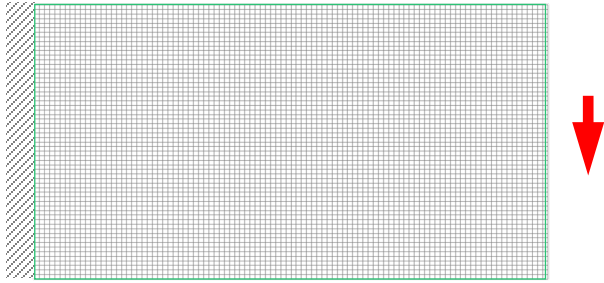
Design Constraints	Range	Multiple Load Cases	Individual Load Cases	Solution Sequence
Compliance	Range	Yes	Yes	LS
Compliance Index	< Upper	Yes	Yes	LS
Max Displacement Component in Model	< Upper	Yes	Yes	LS
Specific Grid Point Displacement Component	Range	Yes	Yes	LS
Max Constraint Force Component in Model	< Upper	Yes	Yes	LS
Specific Constraint Force Component	Range	Yes	Yes	LS
Stress of a Specific TOPVAR Region	< Upper	Yes	Yes	LS
Stress of all TOPVAR Regions	< Upper	Yes	Yes	LS
Volume Fraction (Mass Fraction) of a specific TOPVAR Region	< Upper	Yes	Yes	LS
Volume Fraction (Mass Fraction) of all TOPVAR Regions	< Upper	Yes	Yes	LS
Thermal Energy of a Specific TOPVAR Region	Range	Yes	Yes	LSSHT
Thermal Energy of all TOPVAR Regions	Range	Yes	Yes	LSSHT
Average Temperature of a Specific Set of Nodes	Range	Yes	Yes	LSSHT
Delta Temperature of a Specific Set of Nodes	Range	Yes	Yes	LSSHT
Global Temperature of a Specific Set of Nodes	Range	Yes	Yes	LSSHT
Normal Modes Frequency	> Lower	No	No	NM
Normal Modes Eigenvalue	Range	No	No	NM
Buckling Modes Eigenvalue (load factor)	Range	No	No	LB

LS = Linear Statics, LSSHT = Linear Steady-State Heat Transfer, NM= Normal Modes, LB = Linear Buckling

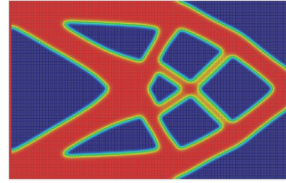
Nastran Topology Optimization Manufacturing Constraints

Manufacturing Constraints	Combinable With
Non-Design Regions	All
Minimum Member Size	All
Symmetry	Min Member Size
Design for Extrusion	Min Member Size
Design for Milling	Min Member Size
Design for AM	Min Member Size

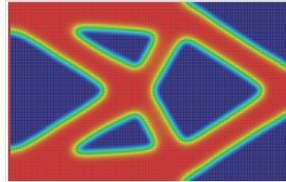
Minimum Member Size Manufacturing Constraint



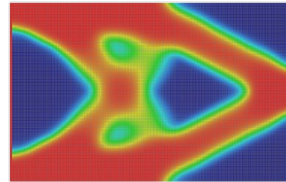
- Fixed at one end and edge loaded at the other end
- Objective is minimize mass
- Constraint is maximum vertical displacement at loaded edge
- Manufacturing constraint: minimum member size (prevents non-designable feature generation)



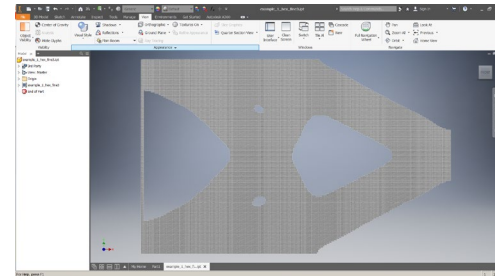
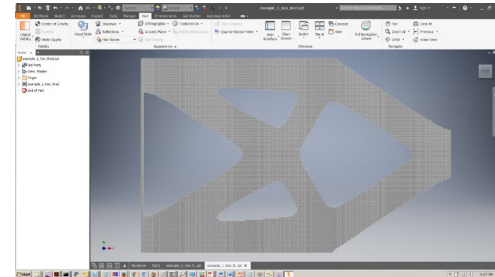
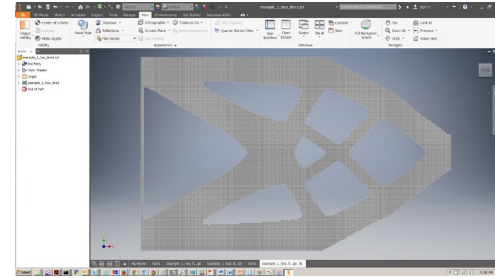
Min member size = 2.0
50.6% mass reduction



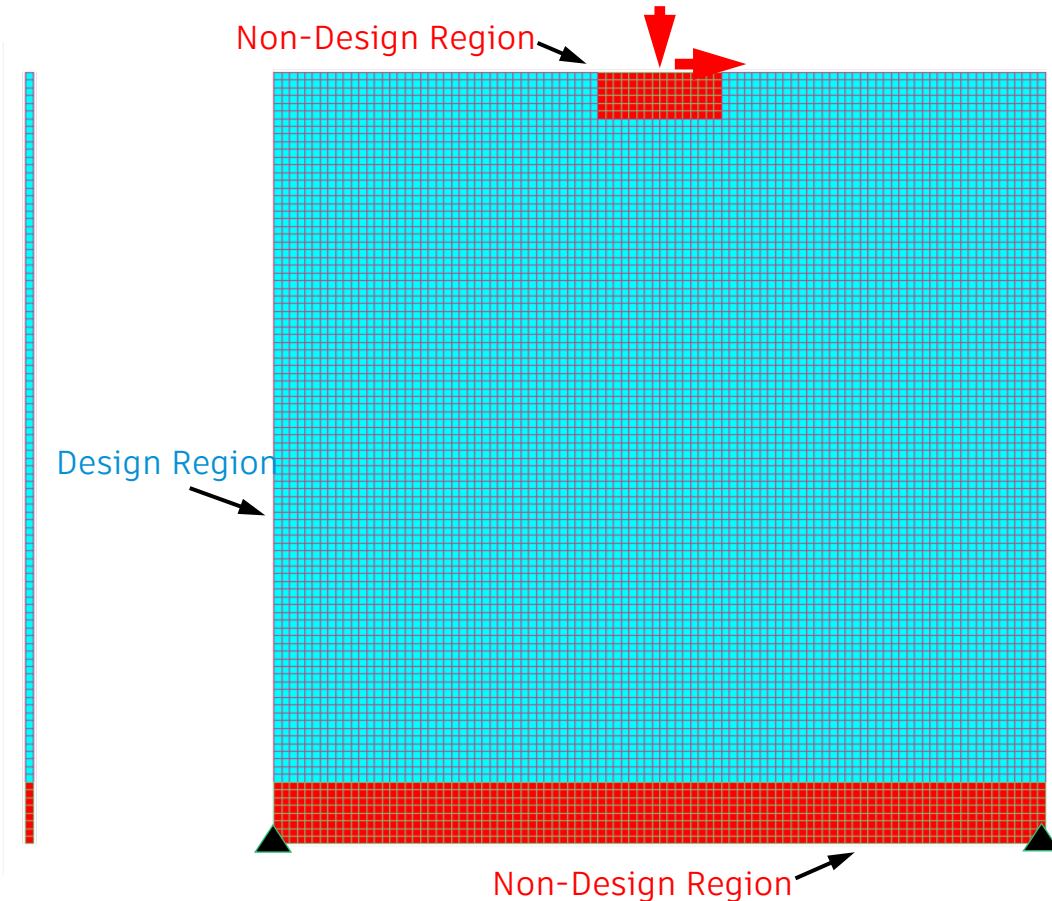
Min member size = 4.0
47.2% mass reduction



Min member size = 6.0
40.4% mass reduction

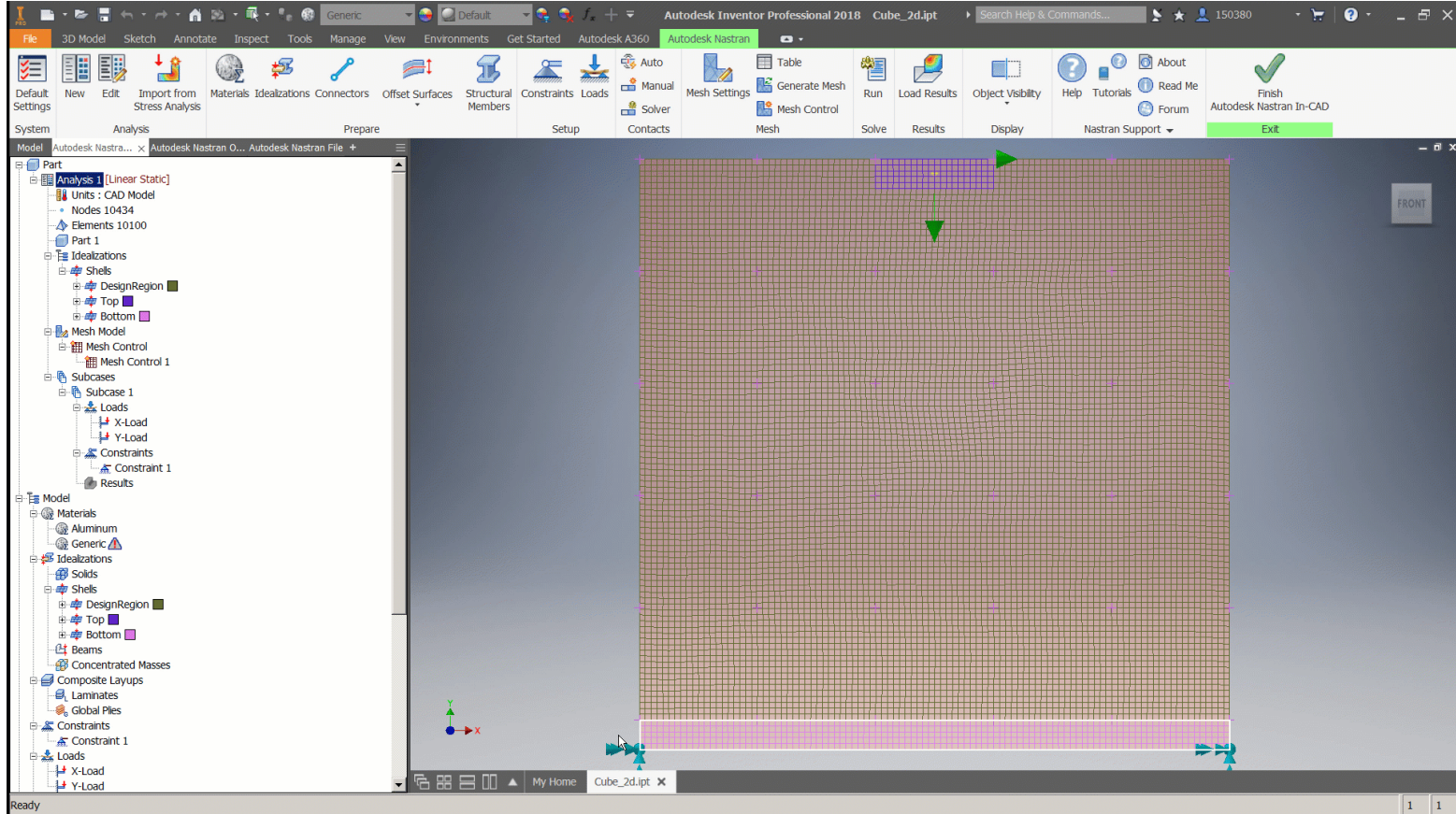


Example #1 Model Definition

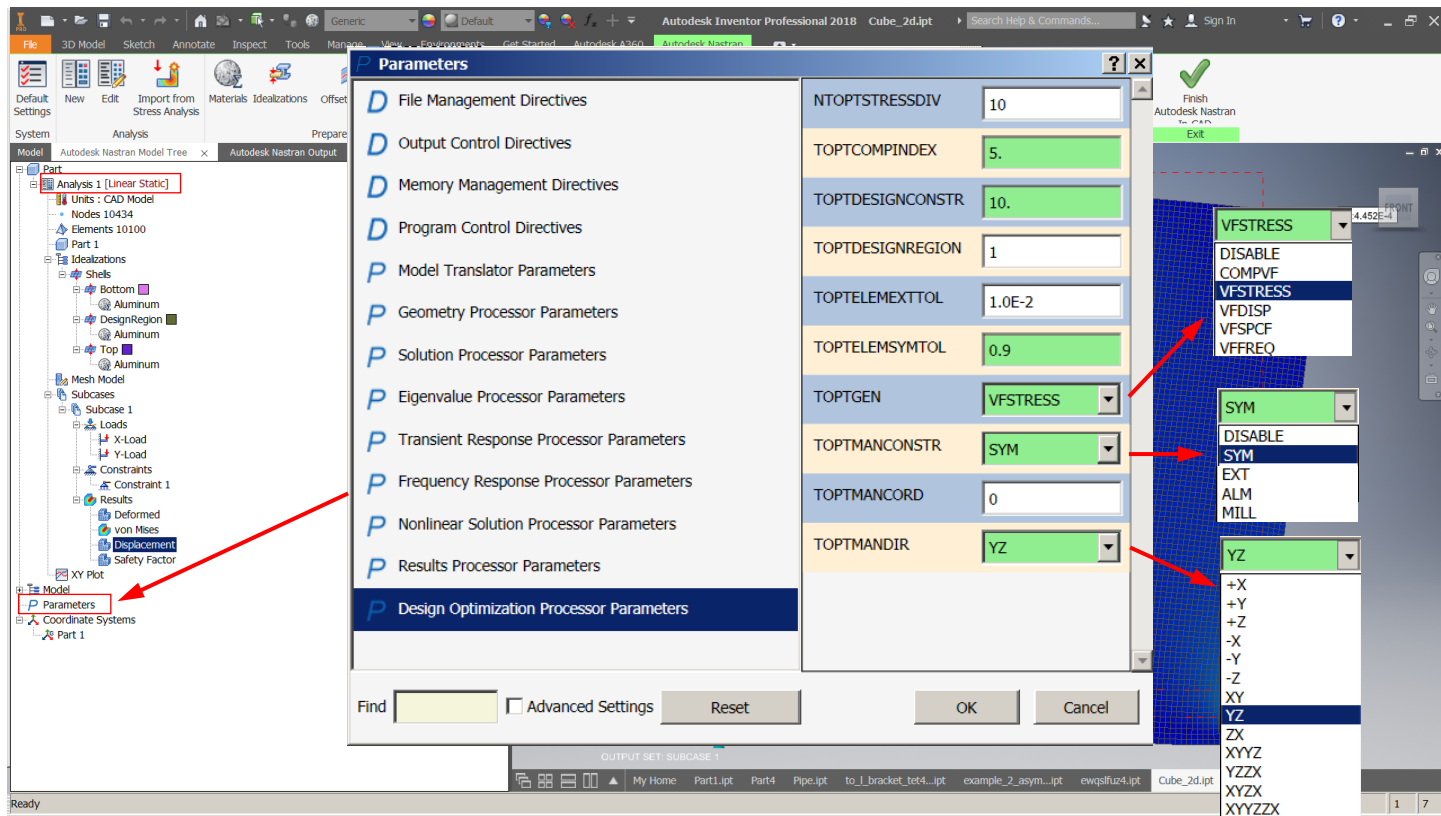


- **Boundary Condition:** Fixed at bottom corners
- **Loading:**
 - Point load in vertical and shear directions
- **Design constraints:**
 - Desired volume fraction
 - Stress limit
 - Displacement limit
 - Lowest frequency
- **Objectives:**
 - Minimize compliance
 - Minimize mass/volume
- **Manufacturing constraints:**
 - No symmetry
 - With symmetry

Topology Optimization Example #1



Topology Optimization Example #1



Inventor Nastran Optimization Parameters - **TOPTGEN**

The screenshot shows the 'Parameters' dialog box in Inventor Nastran. The 'Design Optimization Processor Parameters' section is expanded. The 'TOPTGEN' dropdown menu is highlighted with a red box, showing 'VFSTRESS' as the selected option. Other parameters visible include:

- MAXTOPTITER: 200
- NTOPTSTRESSDIV: 10
- TOPTBTHRESHOLD: 0.5
- TOPTCOMPINDEX: 1.0E+10
- TOPTDATABASE: DELETE
- TOPTDESIGNCONSTR: 5
- TOPTDESIGNREGION: 3
- TOPTDESIGNTOL: 1.0E-13
- TOPTTELEMXTOL: 1.0E-2
- TOPTTELEMSYMTOL: 1.0E-2
- TOPTITERTOL: DISABLE
- TOPTMANCONSTR: COMPVF
- TOPTMANCORDER: VFSTRESS
- TOPTMANDIR: XY
- TOPTMAXACTDIST: AUTO
- TOPTMAXBETA: AUTO

Keyword	Objective	Design Constraint(s)	Solution Type
DISABLE	N/A	Topology optimization is disabled	N/A
COMPVF	Minimize compliance	Mass/volume fraction below	Linear Statics
VFSTRESS	Minimize mass	Max stress and compliance index in design region below a specified value	Linear Statics
VFDISP	Minimize mass	Max displacement and compliance index in model below a specified value	Linear Statics
VFSPCF	Minimize mass	Max reaction force and compliance index in model below a specified value	Linear Statics
VFFREQ	Minimize mass	Frequency above a specified value	Normal Modes

Inventor Nastran Optimization Parameters - TOPTDESIGNCONSTR

The screenshot shows the 'Parameters' dialog box in Inventor Nastran. The left pane lists various parameter categories, with 'Design Optimization Processor Parameters' selected. The right pane displays the following parameters:

- MAXTOPTITER: 200
- NTOPTSTRESSDIV: 10
- TOPTBTHRESHOLD: 0.5
- TOPTCOMPINDEX: 1.0E+10
- TOPTDATABASE: DELETE
- TOPTDESIGNCONSTR: 5** (highlighted with a red box)
- TOPTDESIGNREGION: 3
- TOPTDESIGNTOL: 1.0E-13
- TOPTTELEMEXTOL: 1.0E-2
- TOPTTELEMSYMTOL: 1.0E-2
- TOPTGEN: VFSTRESS
- TOPTITERTOL: 5.0E-3
- TOPTMANCONSTR: DISABLE
- TOPTMANCORD: 0
- TOPTMANDIR: XY
- TOPTMAXACTDIST: AUTO
- TOPTMAXBETA: AUTO

At the bottom, there is a 'Find' field, a checked 'Advanced Settings' checkbox, and 'Reset', 'OK', and 'Cancel' buttons.

Description	Type	Default
Topology design optimization design constraint value based on the TOPTGEN setting. See TOPTGEN.	Real > 0.0	1.0E+10

TOPTGEN Setting	TOPTDESIGNCONSTR Description
COMPVF	Volume fraction upper limit between 0.05 and 1.0
VFSTRESS	Stress upper limit
VFDISP	Displacement upper limit
VFSPCF	Reaction force upper limit
VFFREQ	Frequency lower limit

Inventor Nastran Optimization Parameters - TOPTCOMPINDEX

Parameters

File Management Directives

Output Control Directives

Memory Management Directives

Program Control Directives

Model Translator Parameters

Geometry Processor Parameters

Solution Processor Parameters

Eigenvalue Processor Parameters

Transient Response Processor Parameters

Frequency Response Processor Parameters

Nonlinear Solution Processor Parameters

Results Processor Parameters

Design Optimization Processor Parameters

Eigenvalue Processor Parameters

Transient Response Processor Parameters

Frequency Response Processor Parameters

Nonlinear Solution Processor Parameters

Results Processor Parameters

Design Optimization Processor Parameters

MAXTOPTITER: 200

NTOPTSTRESSDIV: 10

TOPTBTHRESHOLD: 0.5

TOPTCOMPINDEX: 1.0E+10

TOPTDATABASE: DELETE

TOPTDESIGNCONSTR: 5

TOPTDESIGNREGION: 3

TOPTDESIGNTOL: 1.0E-13

TOPTTELEXTTOL: 1.0E-2

TOPTTELEMSYMTOL: 1.0E-2

TOPTGEN: VFSTRESS

TOPTITERTOL: 5.0E-3

TOPTMANCONSTR: DISABLE

TOPTMANCORD: 0

TOPTMANDIR: XY

TOPTMAXACTDIST: AUTO

TOPTMAXBETA: AUTO

Find ☒ Advanced Settings

Description	Type	Default
Topology design optimization compliance index design constraint value. Applicable only when TOPTGEN is set to 2, 3, or 4.	Real > 1.0	1.0E+10

Model Autodesk Nastran Model Tr... Autodesk Nastran Output x Autodesk Nastran File +

DESIGN OBJECTIVE CURRENT VALUE
DRESP 1 3.706E-01

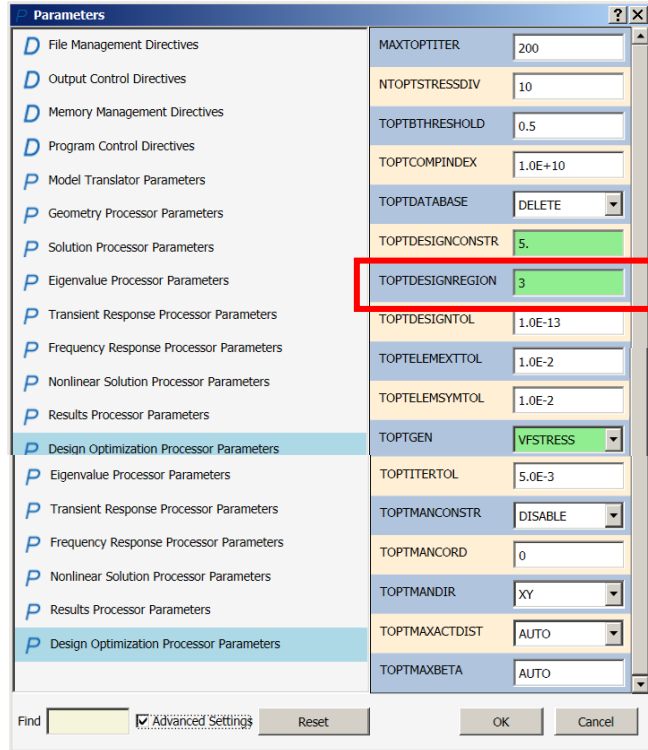
DESIGN CONSTRAINT	CURRENT VALUE	LIMIT VALUE	STATUS
DRESP 3_1	1.081E+00	1.000E+10	PASS
DRESP 2_1_1	4.443E+00	5.000E+00	PASS
DRESP 2_1_2	4.030E+00	5.000E+00	PASS
DRESP 2_1_3	4.890E+00	5.000E+00	PASS
DRESP 2_1_4	5.003E+00	5.000E+00	PASS
DRESP 2_1_5	5.034E+00	5.000E+00	PASS
DRESP 2_1_6	4.835E+00	5.000E+00	PASS
DRESP 2_1_7	4.595E+00	5.000E+00	PASS
DRESP 2_1_8	4.247E+00	5.000E+00	PASS
DRESP 2_1_9	4.121E+00	5.000E+00	PASS
DRESP 2_1_10	2.583E+00	5.000E+00	PASS

DESIGN ITERATION CONVERGENCE = 100.0
DENSITY MASS REDUCTION = 58.8

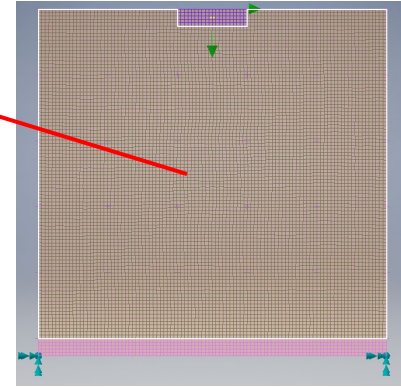
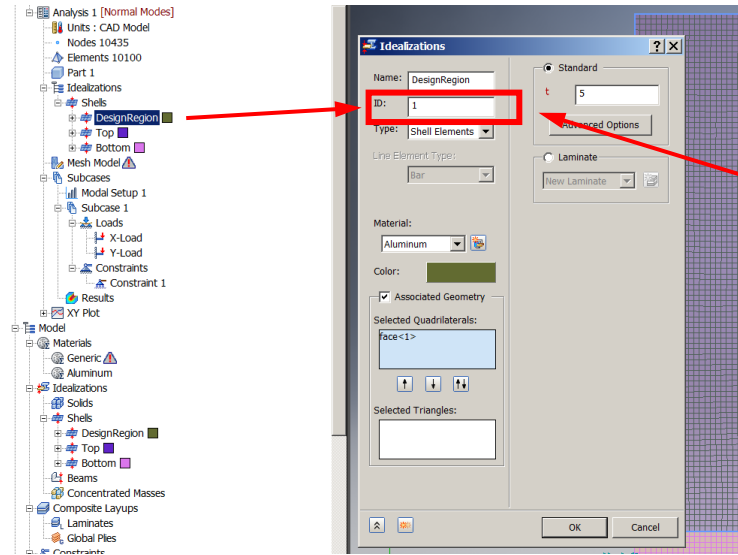
ESTIMATED REMAINING DESIGN ITERATIONS = 0
ESTIMATED REMAINING SOLUTION TIME = 0.0 SECONDS

DESIGN OPTIMIZATION ITERATION 87

Inventor Nastran Optimization Parameters - TOPTDESIGNREGION

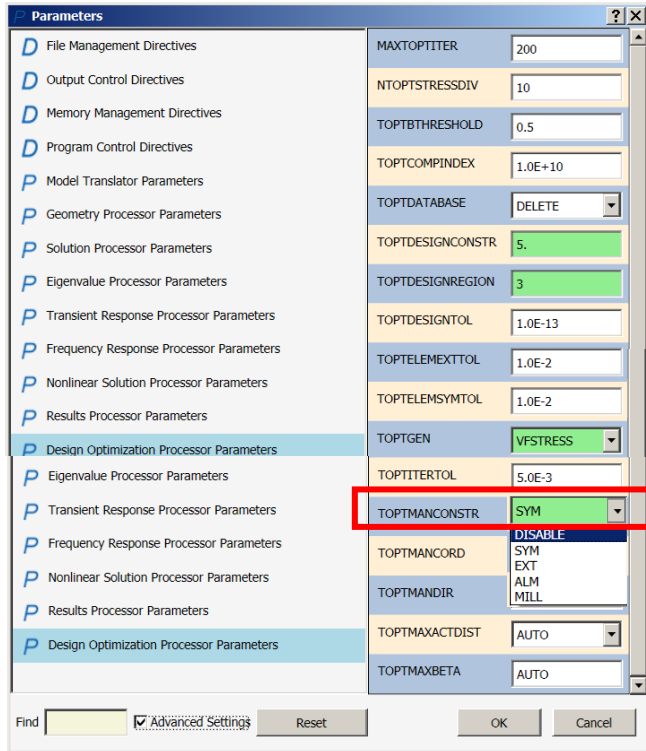


Description	Type	Default
Topology design optimization design region property identification number.	Integer > 0	1



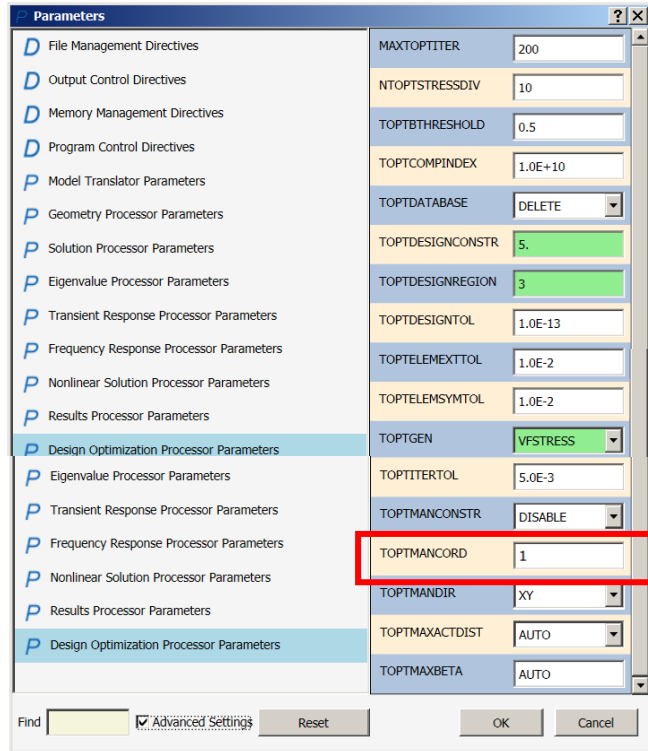
- Note: Specifying the wrong ID may result in an 2299 or 5125 fatal error

Inventor Nastran Optimization Parameters - TOPTMANCONSTR

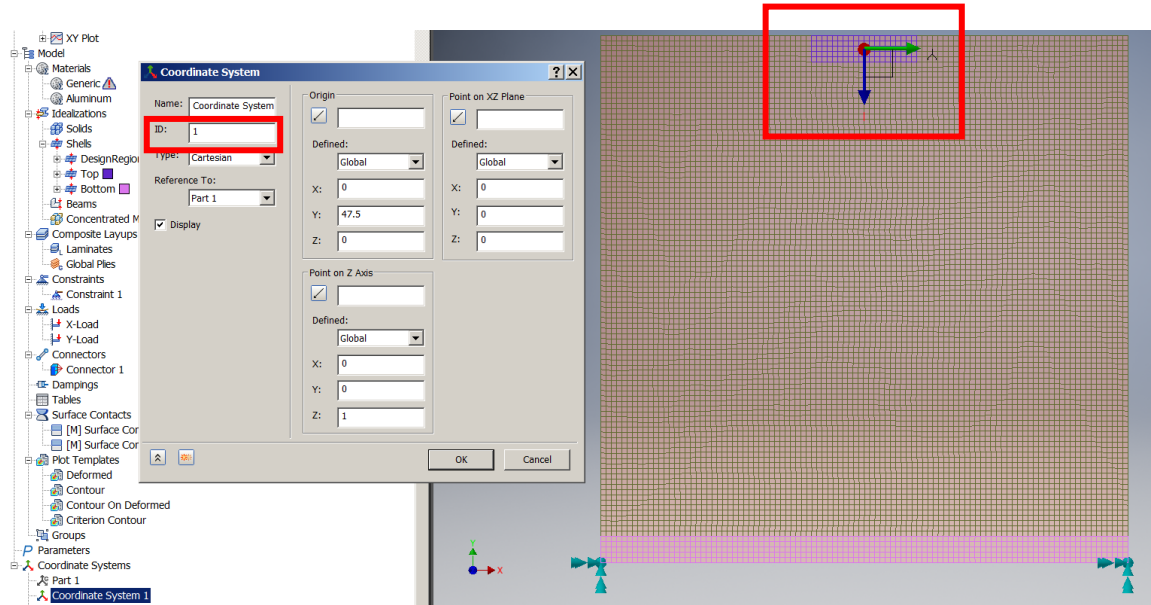


Description	Type	Default
<p>Defines the type of manufacturing constraint to be used in Automated Topology Optimization Generation (ATOG). There are three options:</p> <p>0 - No manufacturing constraint is specified.</p> <p>1 - Symmetry using either 1, 2, or 3 planes of symmetry is specified.</p> <p>2 - Extrude design constraint is specified.</p> <p>3 - Additive layer manufacturing design constraint is specified.</p> <p>4 - 3-axis milling manufacturing design constraint is specified.</p> <p>The character variables: DISABLE, SYM, EXT, and ALM may be used in place of the numerical options 0 through 3. See also TOPTMANDIR and TOPTMANCORD.</p>	<p>$0 \leq$ Integer \leq 3</p> <p>DISABLE/ SYM/ EXT/ ALM/ MILL</p>	0

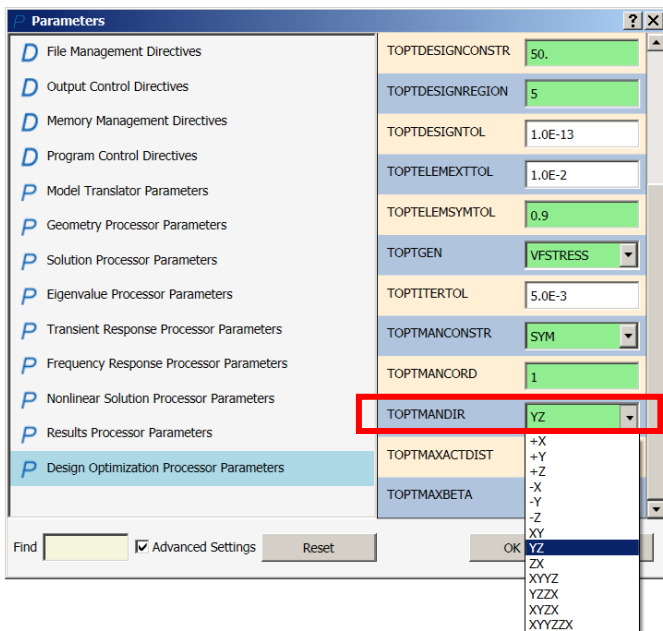
Inventor Nastran Optimization Parameters - TOPTMANCORD



Description	Type	Default
Specifies the topology design optimization manufacturing constraint coordinate system corresponding to TOPTMANCONSTR . See also TOPTMANDIR .	Integer > 0	0



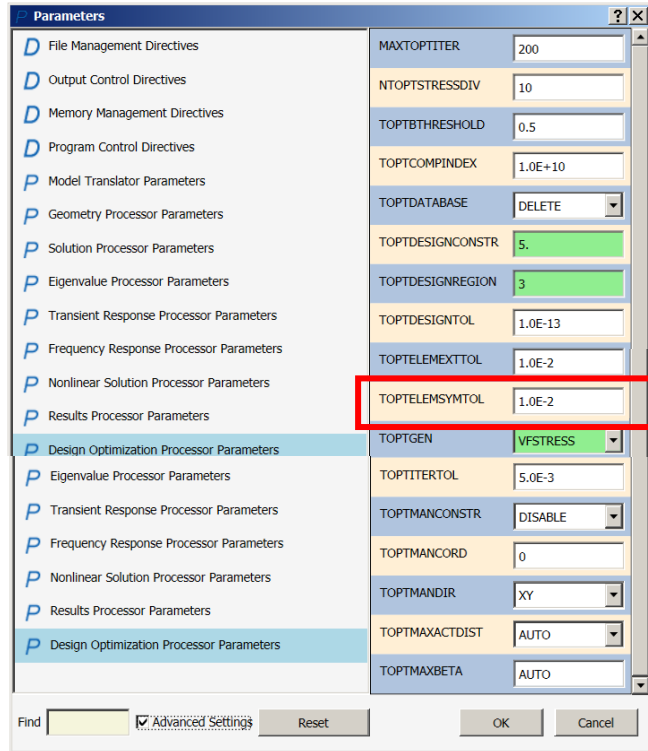
Inventor Nastran Optimization Parameters - TOPTMANDIR



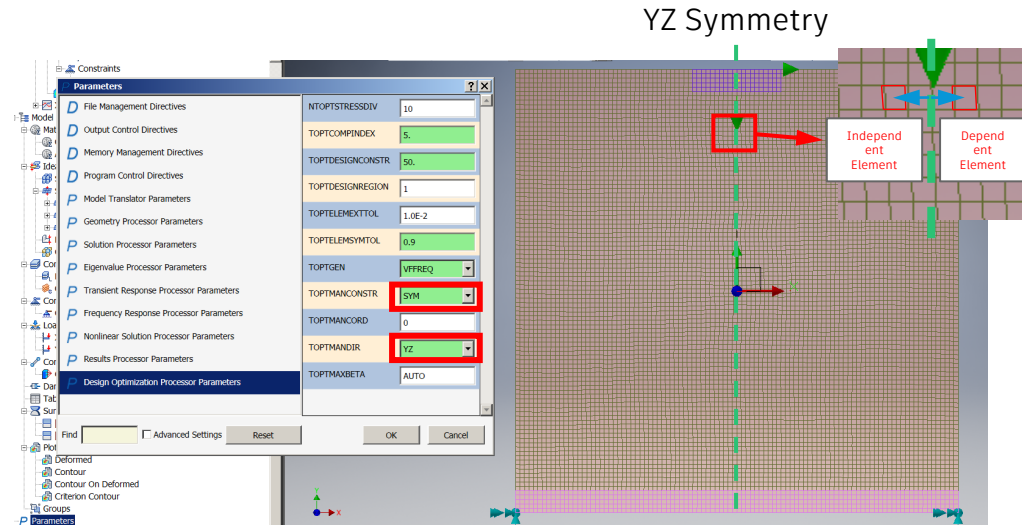
Description	Type	Default
Specifies the topology design optimization manufacturing constraint symmetry plane(s), extrude direction, or print direction depending on the TOPTMANCONSTR value specified.	+X/+Y/+Z/ -X/-Y/-Z/ XYZ/ZY/ZX/ XYZZYZZX/ XYZX/ XYZZX	XY

Keyword	TOPTMANDIR Definition
DISABLE	No manufacturing constraints specified
SYM	Symmetry plane or planes specified in the TOPTMANCORDER system
EXT	Extrude direction axis specified in the TOPTMANCORDER system
ALM	Print direction axis specified in the TOPTMANCORDER system
MILL	Mill direction axis specified in the TOPTMANCORDER system

Inventor Nastran Optimization Parameters - TOPTTELEMSYMTOL



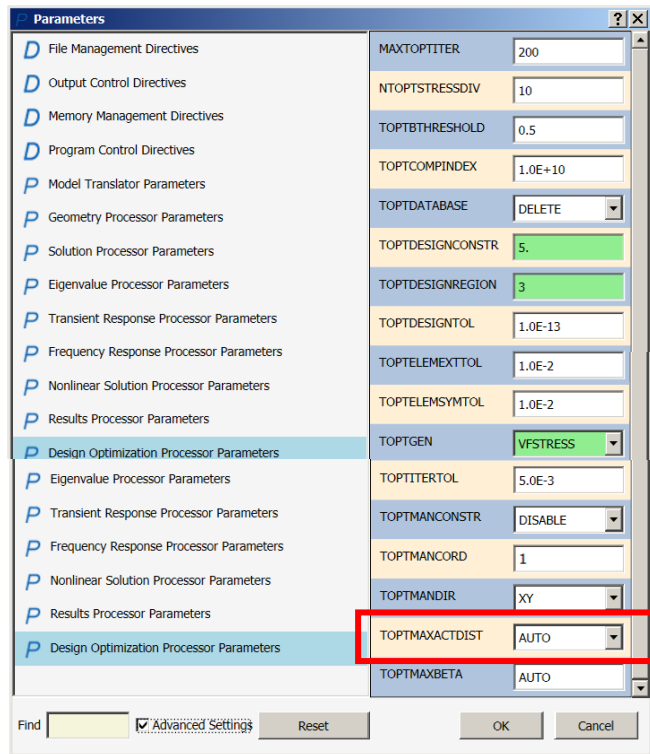
Description	Type	Default
Near tolerance used to identify elements which are symmetric with respect to the specified TOPVAR Bulk Data entry mirror symmetry plane. The actual tolerance is derived using TOPTTELEMSYMTOL and an element reference dimension.	Real	1.0E-2



Inventor Nastran Optimization Parameters -

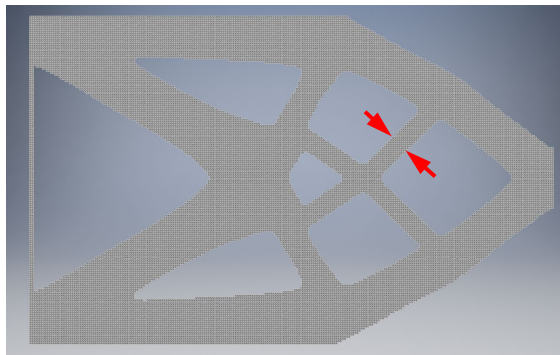
TOPTMAXACTDIST

Use this to specify a minimum member size



Description	Type	Default
Topology design optimization maximum distance for identifying adjacent elements. Elements within distance TOPTMAXACTDIST are used for sensitivity filtering. The default AUTO setting is recommended since large values may result in slower performance and undesired results.	Real AUTO	AUTO

- TOPTMAXACTDIST is 1/2 minimum member size
- To specify a minimum member size of 1 use 0.5.



Inventor Nastran Optimization Parameters - TOPTMAXBETA

The screenshot shows the 'Parameters' dialog box in Inventor Nastran. The left pane lists various parameter categories, with 'Design Optimization Processor Parameters' selected. The right pane displays a list of parameters for this category. The 'TOPTMAXBETA' parameter is highlighted with a red rectangular box. Below the parameter list, there are buttons for 'Find', 'Advanced Settings' (checked), 'Reset', 'OK', and 'Cancel'.

Parameter	Value
MAXTOPTITER	200
NTOPTSTRESSDIV	10
TOPTBTHRESHOLD	0.5
TOPTCOMPINDEX	1.0E+10
TOPTDATABASE	DELETE
TOPTDESIGNCONSTR	5.
TOPTDESIGNREGION	3
TOPTDESIGNTOL	1.0E-13
TOPTTELEMTXTOL	1.0E-2
TOPTTELEMSYMTOL	1.0E-2
TOPTGEN	VFSTRESS
TOPTITERTOL	5.0E-3
TOPTMANCONSTR	DISABLE
TOPTMANCORD	1
TOPTMANDIR	XY
TOPTMAXACTDIST	AUTO
TOPTMAXBETA	AUTO

Description	Type	Default
Specifies the penalty value for enforcing minimum member size manufacturing constraints. A value between 1.0 and 16.0 is recommended. The default AUTO value selects the best value depending on what other constraints are specified.	Real AUTO	AUTO

Inventor Nastran Optimization Parameters - MAXOPTITER

Parameters

File Management Directives

Output Control Directives

Memory Management Directives

Program Control Directives

Model Translator Parameters

Geometry Processor Parameters

Solution Processor Parameters

Eigenvalue Processor Parameters

Transient Response Processor Parameters

Frequency Response Processor Parameters

Nonlinear Solution Processor Parameters

Results Processor Parameters

Design Optimization Processor Parameters

Eigenvalue Processor Parameters

Transient Response Processor Parameters

Frequency Response Processor Parameters

Nonlinear Solution Processor Parameters

Results Processor Parameters

Design Optimization Processor Parameters

MAXOPTITER 200

TOPTSTRESSDIV 10

TOPTBTHRESHOLD 0.5

TOPTCOMPINDEX 1.0E+10

TOPTDATABASE DELETE

TOPTDESIGNCONSTR 5

TOPTDESIGNREGION 3

TOPTDESIGNTOL 1.0E-13

TOPTTELEXTTOL 1.0E-2

TOPTTELEMSYMTOL 1.0E-2

TOPTGEN VFSTRESS

TOPTTITERTOL 5.0E-3

TOPTMANCONSTR DISABLE

TOPTMANCORD 0

TOPTMANDIR XY

TOPTMAXACTDIST AUTO

TOPTMAXBETA AUTO

Find ☒ Advanced Settings

Description	Type	Default
Topology design optimization maximum number of convergence iterations permitted. The solver will iterate until the convergence factor set by TOPTTERTOL is reached or MAXOPTITER iterations have been performed. A zero setting will result in iteration until convergence is reached.	Integer ≥ 0	200

Model Autodesk Nastran Model Tr... Autodesk Nastran Output Autodesk Nastran File

DESIGN OBJECTIVE CURRENT VALUE
DRESP 1 3.706E-01

DESIGN CONSTRAINT	CURRENT VALUE	LIMIT VALUE	STATUS
DRESP 3_1	1.081E+00	1.000E+10	PASS
DRESP 2_1_1	4.443E+00	5.000E+00	PASS
DRESP 2_1_2	4.030E+00	5.000E+00	PASS
DRESP 2_1_3	4.890E+00	5.000E+00	PASS
DRESP 2_1_4	5.003E+00	5.000E+00	PASS
DRESP 2_1_5	5.034E+00	5.000E+00	PASS
DRESP 2_1_6	4.835E+00	5.000E+00	PASS
DRESP 2_1_7	4.595E+00	5.000E+00	PASS
DRESP 2_1_8	4.247E+00	5.000E+00	PASS
DRESP 2_1_9	4.121E+00	5.000E+00	PASS
DRESP 2_1_10	2.583E+00	5.000E+00	PASS

DESIGN ITERATION CONVERGENCE = 100.0
DENSITY MASS REDUCTION = 58.8

ESTIMATED REMAINING DESIGN ITERATIONS = 0
ESTIMATED REMAINING SOLUTION TIME = 0.0 SECONDS

DESIGN OPTIMIZATION ITERATION 87

Inventor Nastran Optimization Parameters - TOPTITERTOL

Parameters

- File Management Directives
- Output Control Directives
- Memory Management Directives
- Program Control Directives
- Model Translator Parameters
- Geometry Processor Parameters
- Solution Processor Parameters
- Eigenvalue Processor Parameters
- Transient Response Processor Parameters
- Frequency Response Processor Parameters
- Nonlinear Solution Processor Parameters
- Results Processor Parameters
- Design Optimization Processor Parameters**
- Eigenvalue Processor Parameters
- Transient Response Processor Parameters
- Frequency Response Processor Parameters
- Nonlinear Solution Processor Parameters
- Results Processor Parameters
- Design Optimization Processor Parameters

MAXTOPTITER: 200

NTOPTSTRESSDIV: 10

TOPTBTHRESHOLD: 0.5

TOPTCOMPINDEX: 1.0E+10

TOPTDATABASE: DELETE

TOPTDESIGNCONSTR: 5

TOPTDESIGNREGION: 3

TOPTDESIGNTOL: 1.0E-13

TOPTTELEXTTOL: 1.0E-2

TOPTTELEMSYMTOL: 1.0E-2

TOPTGEN: VFSTRESS

TOPTITERTOL: 5.0E-3

TOPTMANCONSTR: DISABLE

TOPTMANCORD: 0

TOPTMANDIR: XY

TOPTMAXACTDIST: AUTO

TOPTMAXBETA: AUTO

Find: ☒ Advanced Settings

Description	Type	Default
Topology design optimization Iterative solver convergence factor. The topology optimization solver will iterate until the convergence factor set by TOPTITERTOL is reached or MAXTOPTITER iterations have been performed.	Real	5.0E-3

```
Model Autodesk Nastran Model Tr... Autodesk Nastran Output x Autodesk Nastran File +
DESIGN OBJECTIVE CURRENT VALUE
DRESP 1 3.706E-01
DESIGN CONSTRAINT CURRENT VALUE LIMIT VALUE STATUS
DRESP 3_1 1.081E+00 1.000E+10 PASS
DRESP 2_1_1 4.443E+00 5.000E+00 PASS
DRESP 2_1_2 4.030E+00 5.000E+00 PASS
DRESP 2_1_3 4.890E+00 5.000E+00 PASS
DRESP 2_1_4 5.003E+00 5.000E+00 PASS
DRESP 2_1_5 5.034E+00 5.000E+00 PASS
DRESP 2_1_6 4.835E+00 5.000E+00 PASS
DRESP 2_1_7 4.595E+00 5.000E+00 PASS
DRESP 2_1_8 4.247E+00 5.000E+00 PASS
DRESP 2_1_9 4.121E+00 5.000E+00 PASS
DRESP 2_1_10 2.583E+00 5.000E+00 PASS
DESIGN ITERATION CONVERGENCE = 100.0
ESTIMATED MASS REDUCTION = 30.0
ESTIMATED REMAINING DESIGN ITERATIONS = 0
ESTIMATED REMAINING SOLUTION TIME = 0.0 SECONDS
DESIGN OPTIMIZATION ITERATION 87
```

Inventor Nastran Optimization Parameters - TOPTDATABASE

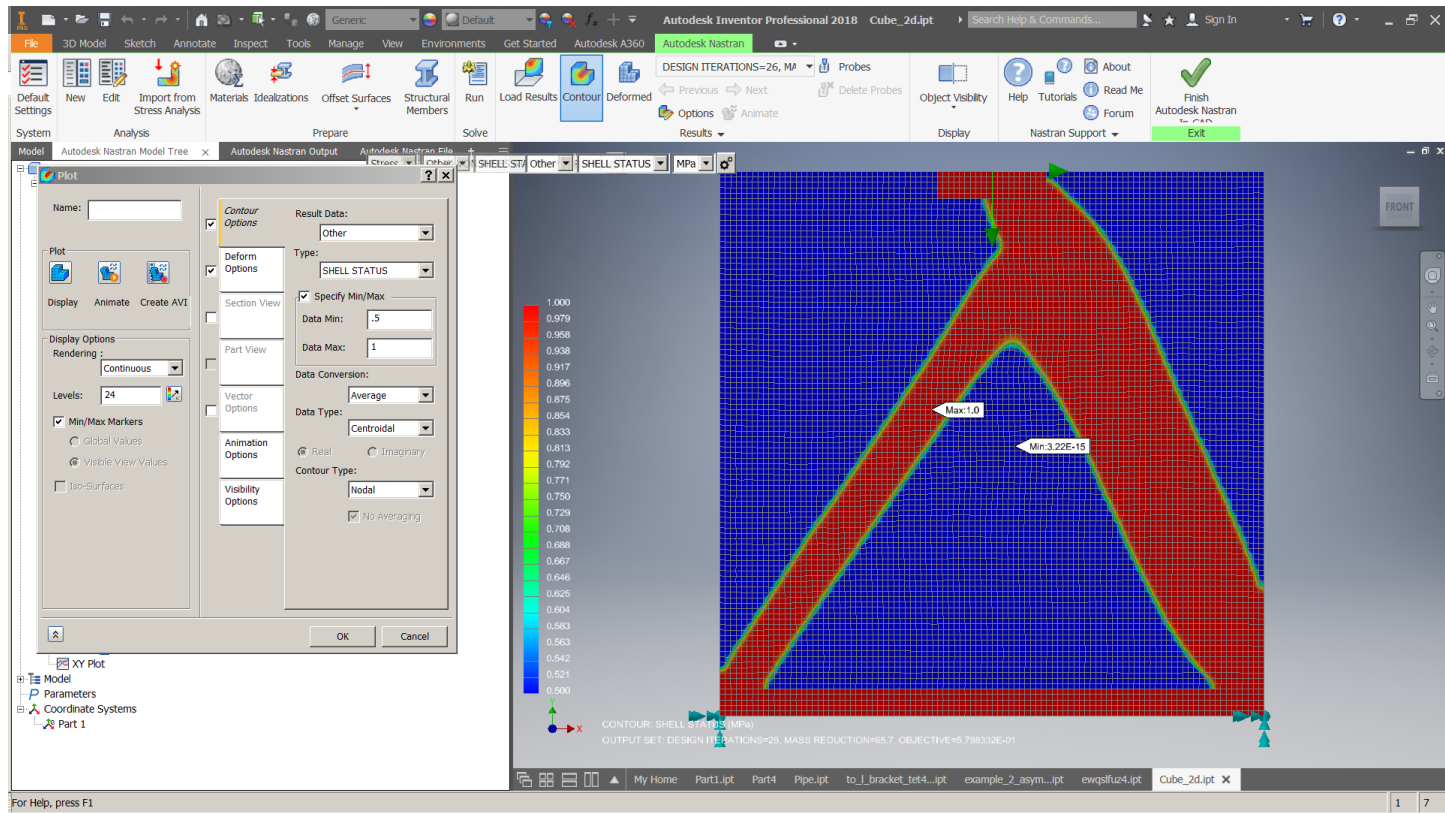
The screenshot shows the 'Parameters' dialog box in Inventor Nastran. The left sidebar lists various parameter categories, with 'Design Optimization Processor Parameters' selected. The right pane displays a list of parameters for this category. The 'TOPTDATABASE' parameter is highlighted with a red rectangular box. It is a dropdown menu currently set to 'DELETE'. Other parameters include MAXTOPTITER (200), NTOPTSTRESSDIV (10), TOPTBTHRESHOLD (0.5), TOPTCOMPINDEX (1.0E+10), TOPTDESIGNCONSTR (5), TOPTDESIGNREGION (3), TOPTDESIGNTOL (1.0E-13), TOPTTELEMEXTOL (1.0E-2), TOPTTELEMSYMTOL (1.0E-2), TOPTGEN (VFSTRESS), TOPTITERTOL (5.0E-3), TOPTMANCONSTR (DISABLE), TOPTMANCORD (0), TOPTMANDIR (XY), TOPTMAXACTDIST (AUTO), and TOPTMAXBETA (AUTO). At the bottom, there are buttons for 'Find', 'Advanced Settings' (checked), 'Reset', 'OK', and 'Cancel'.

Parameter	Value
MAXTOPTITER	200
NTOPTSTRESSDIV	10
TOPTBTHRESHOLD	0.5
TOPTCOMPINDEX	1.0E+10
TOPTDATABASE	DELETE
TOPTDESIGNCONSTR	5
TOPTDESIGNREGION	3
TOPTDESIGNTOL	1.0E-13
TOPTTELEMEXTOL	1.0E-2
TOPTTELEMSYMTOL	1.0E-2
TOPTGEN	VFSTRESS
TOPTITERTOL	5.0E-3
TOPTMANCONSTR	DISABLE
TOPTMANCORD	0
TOPTMANDIR	XY
TOPTMAXACTDIST	AUTO
TOPTMAXBETA	AUTO

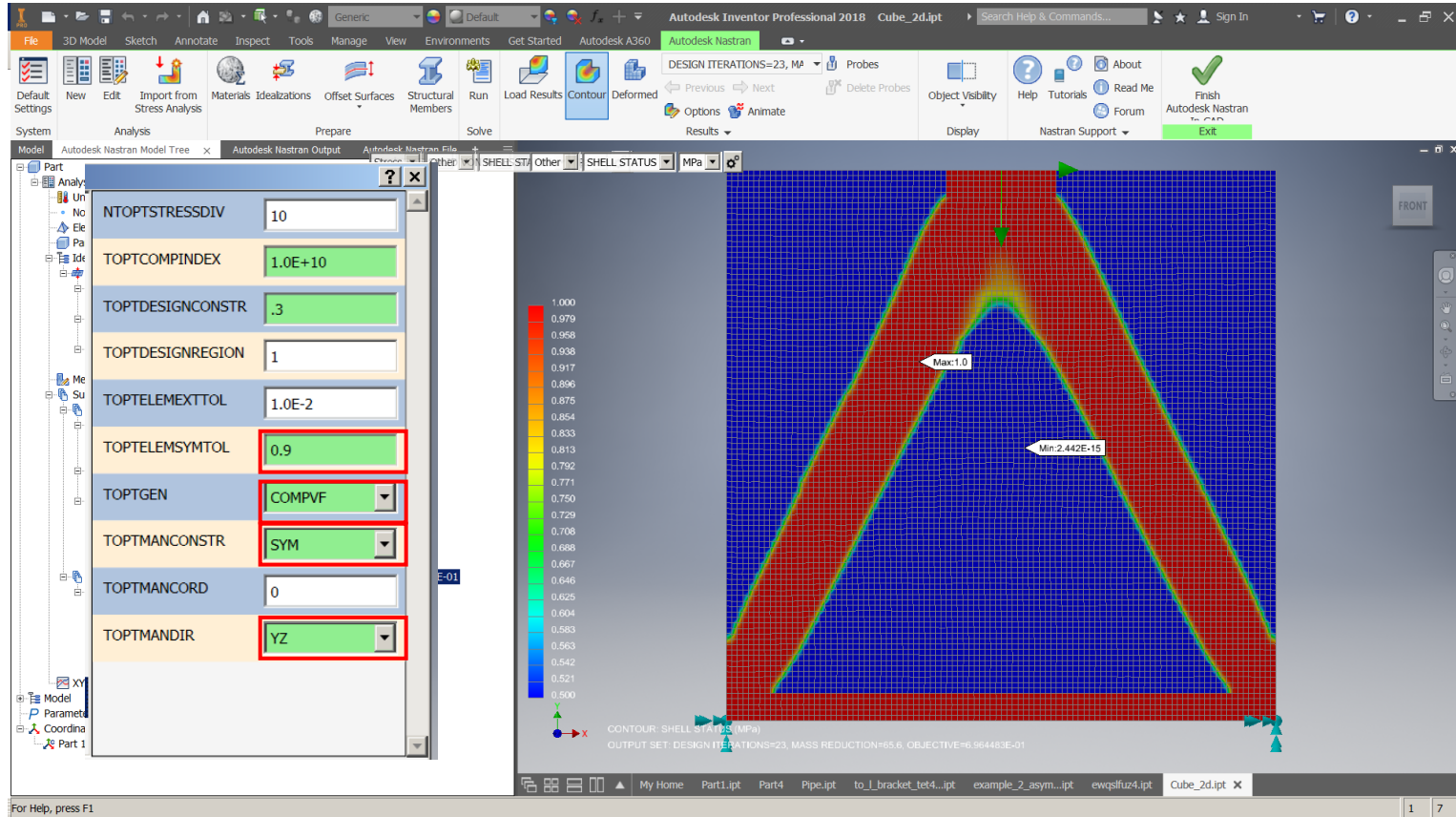
Description	Type	Default
Controls the storage and retrieval of topology design optimization density data. The default value DELETE purges all element density data when the program terminates normally. When set to STORE, the converged optimized design is stored in a single file with the same base name as the Model Results Output File and a .ODB file extension. When set to FETCH, the optimized design specified by the TOPTDATFILE directive is retrieved and used as the starting point for the subsequent topology design optimization solution sequence. When set to UPDATE, the optimized design data will be retrieved and stored.	DELETE FETCH STORE UPDATE	DELETE

Obj: Min. Compliance, Constraint: Desired VF, no Sym

NTOPTSTRESSDIV	10
TOPTCOMPINDEX	1.0E+10
TOPTDESIGNCONSTR	.3
TOPTDESIGNREGION	1
TOPTTELEMXTTOL	1.0E-2
TOPTTELEMSYMTOL	1.0E-2
TOPTGEN	COMPVF
TOPTMANCONSTR	DISABLE
TOPTMANCORD	0
TOPTMANDIR	XY

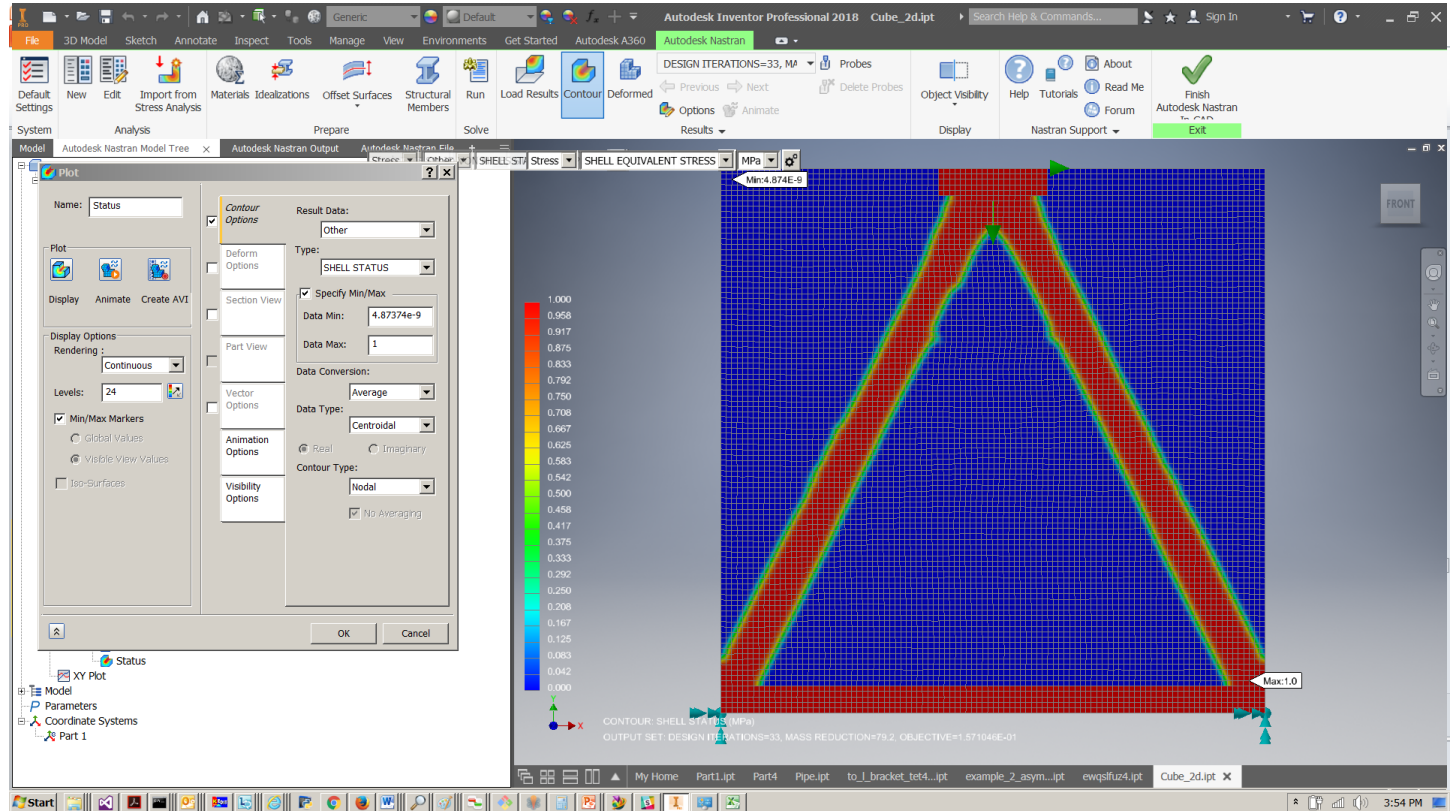


Obj: Min. Compliance, Constraint: Desired VF, with Sym

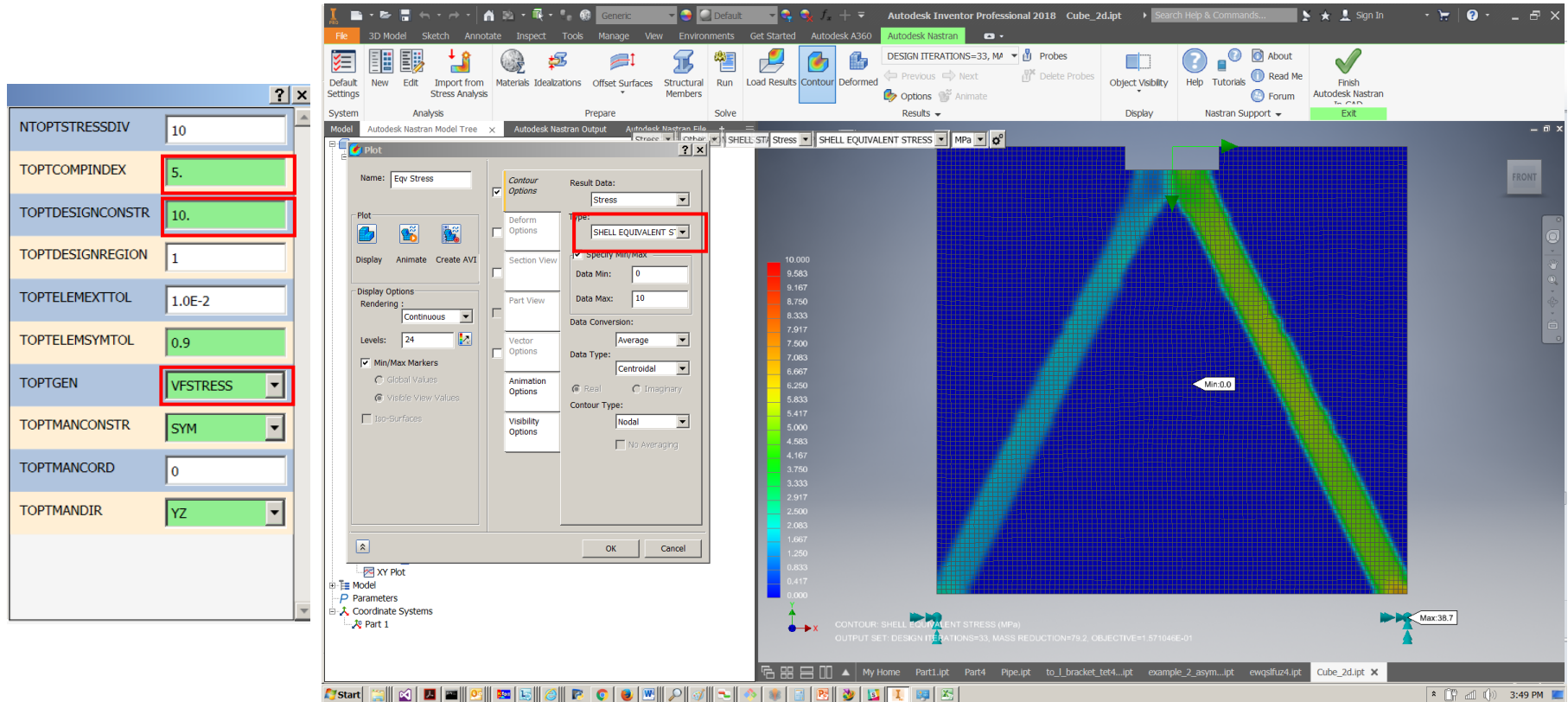


Obj: Min. VF (mass), Constraint: Stress & Comp. Index

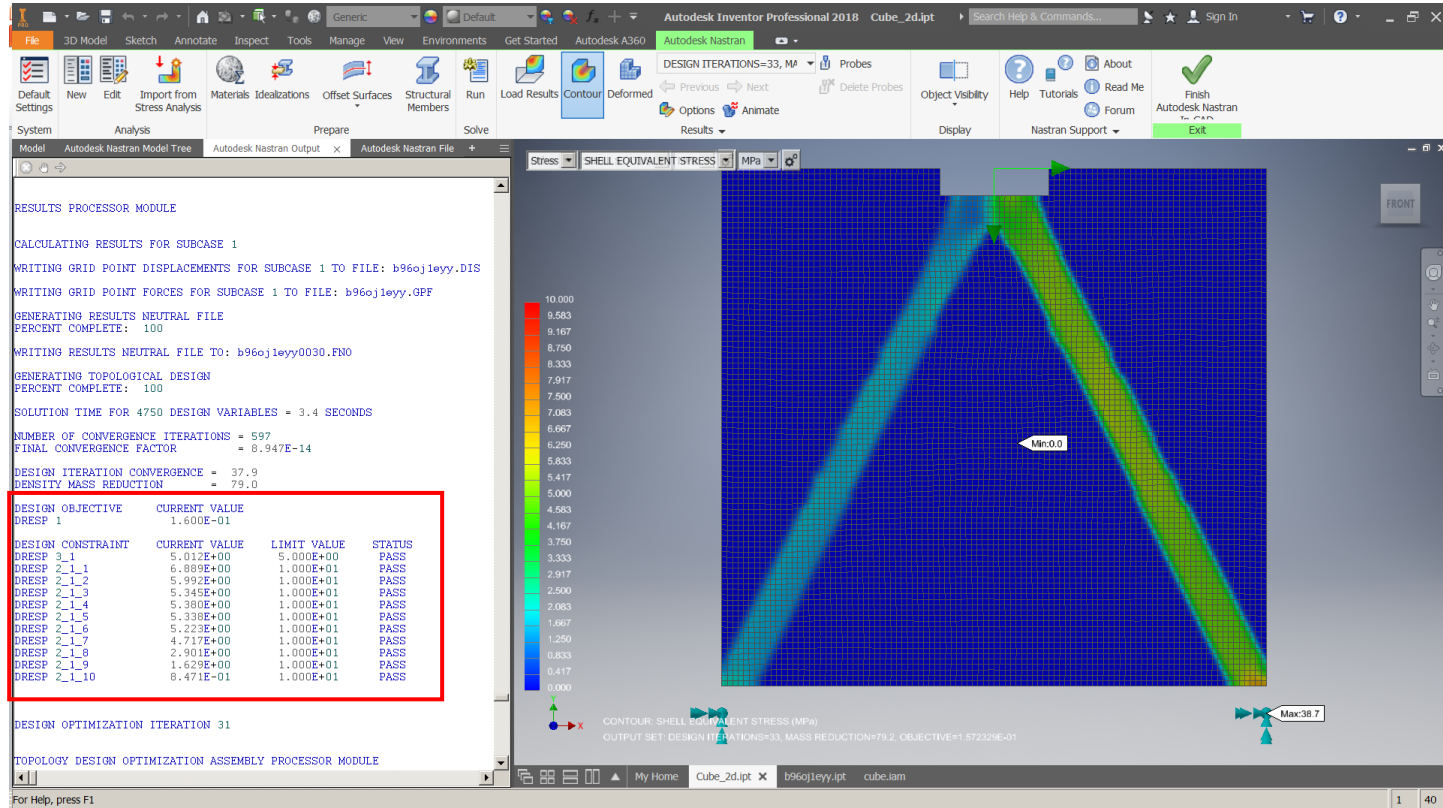
NTOPTSTRESSDIV	10
TOPTCOMPINDEX	5.
TOPTDESIGNCONSTR	10.
TOPTDESIGNREGION	1
TOPTTELEMEXTTOL	1.0E-2
TOPTTELEMSYMTOL	0.9
TOPTGEN	VFSTRESS
TOPTMANCONSTR	SYM
TOPTMANCORD	0
TOPTMANDIR	YZ



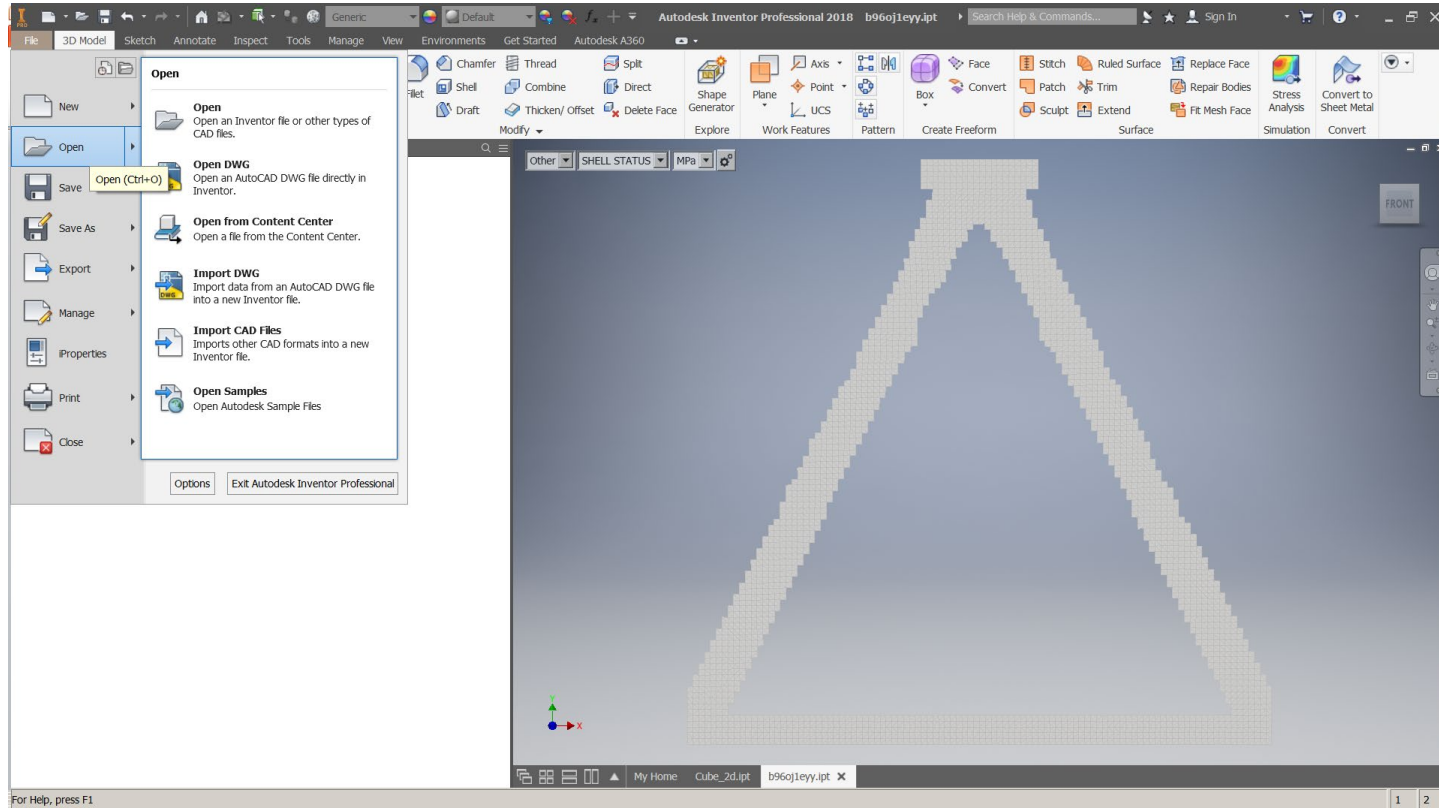
Verify Stress Constraint Using Equivalent Stress



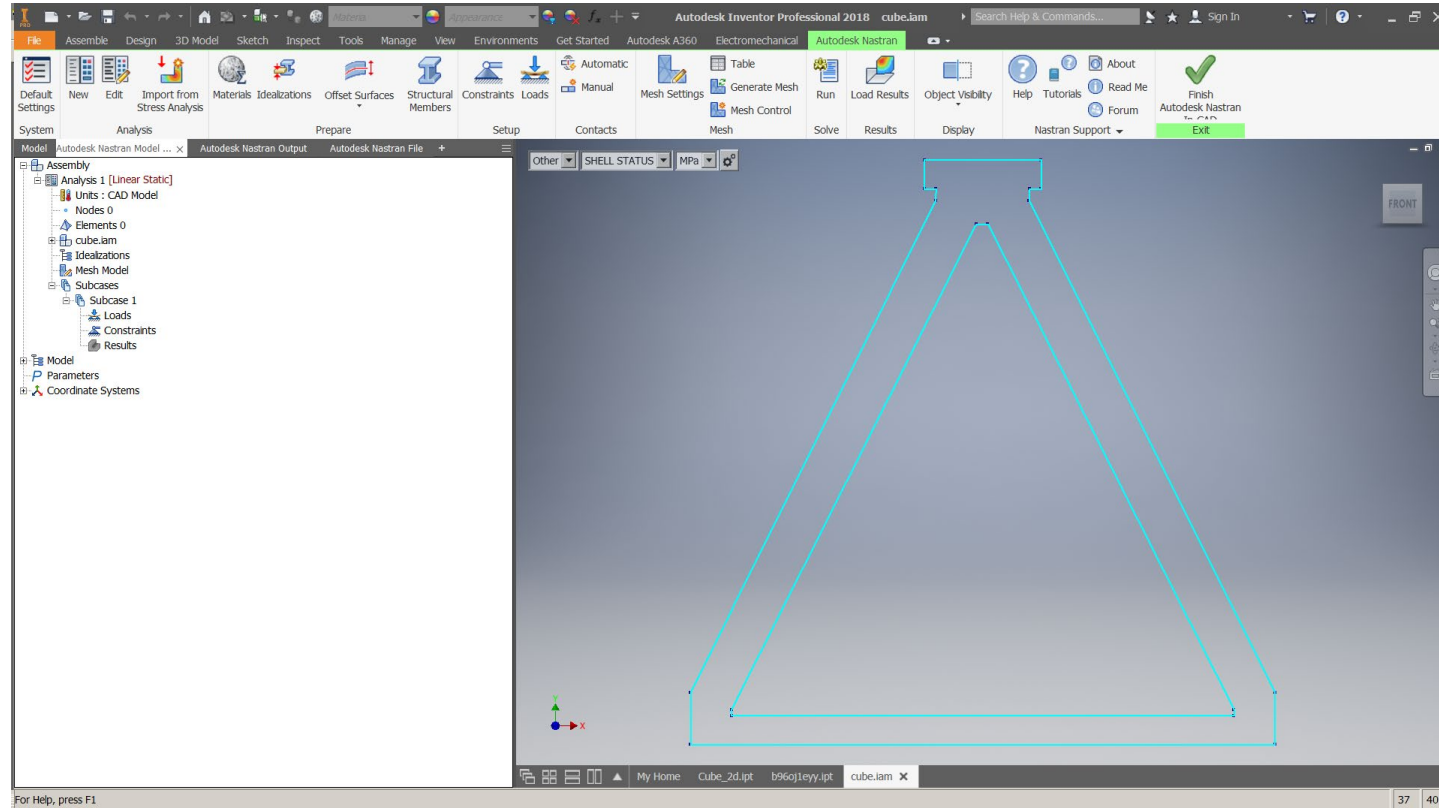
Verify Stress Constraint Using Equivalent Stress



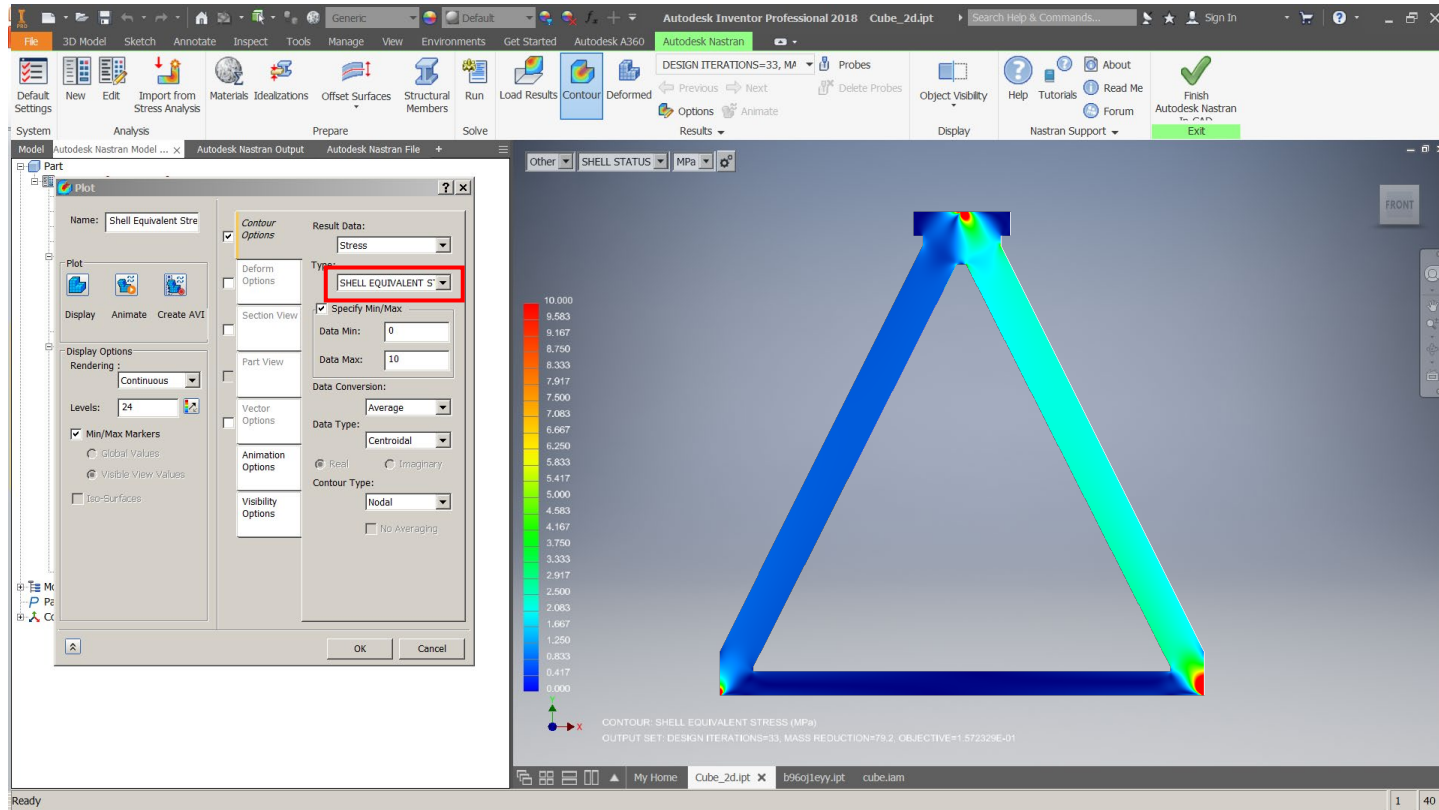
Import Generated Geometry to Build New Design



Import Generated Geometry to Build New Design

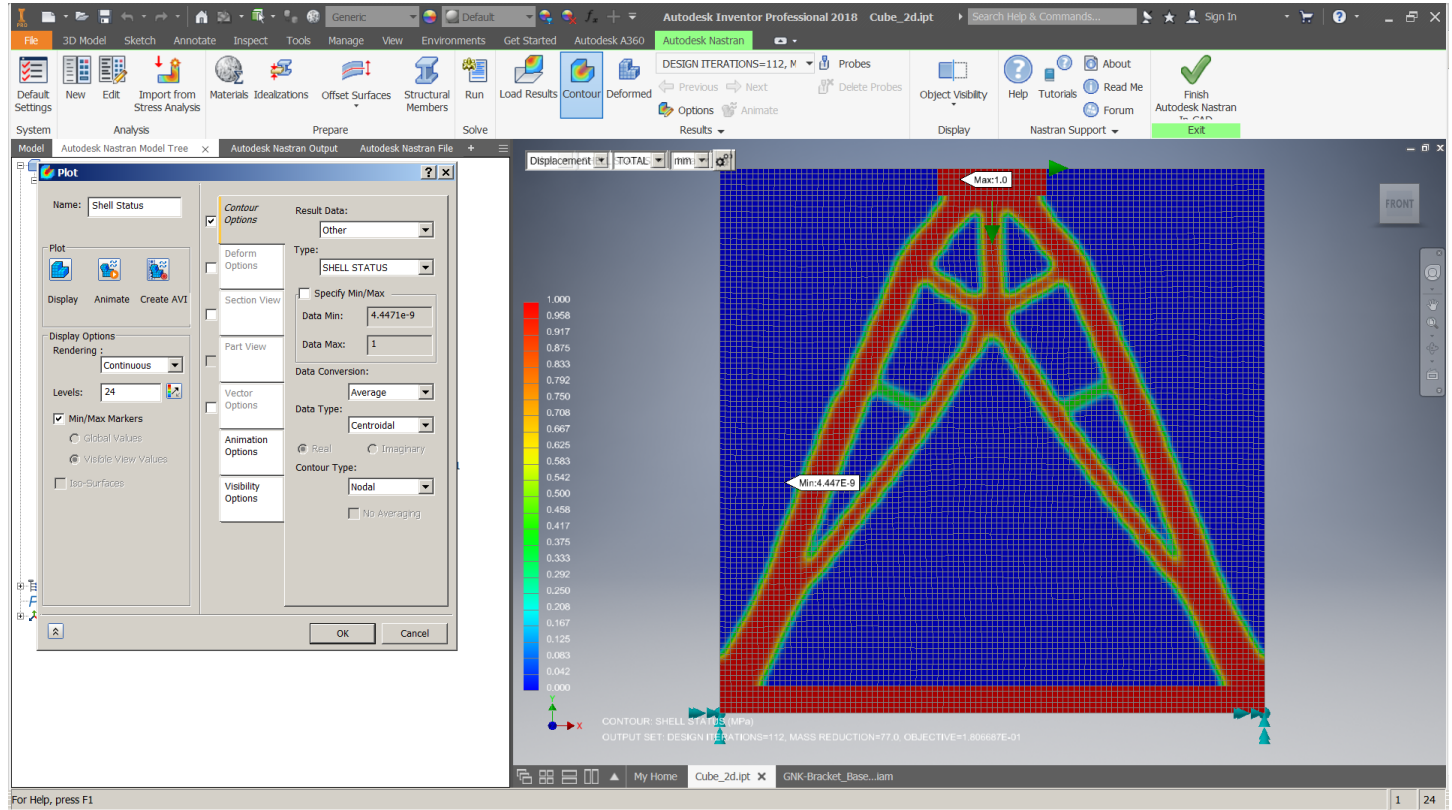


Import Generated Geometry to Build New Design

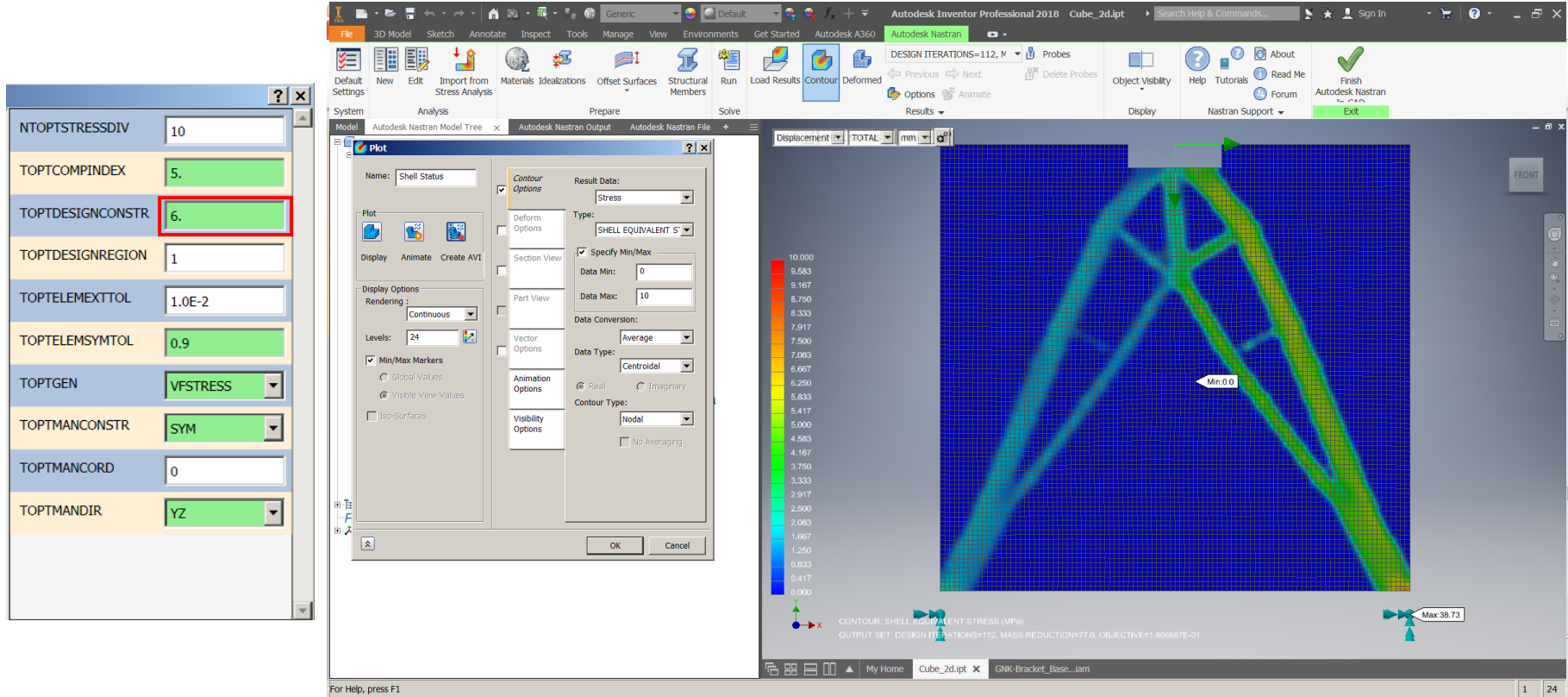


Obj: Min. VF (mass), Constraint: Stress & Comp. Index

NTOPTSTRESSDIV	10
TOPTCOMPINDEX	5.
TOPTDESIGNCONSTR	6.
TOPTDESIGNREGION	1
TOPTTELEMEXTTOL	1.0E-2
TOPTTELEMSYMTOL	0.9
TOPTGEN	VFSTRESS
TOPTMANCONSTR	SYM
TOPTMANCORD	0
TOPTMANDIR	YZ

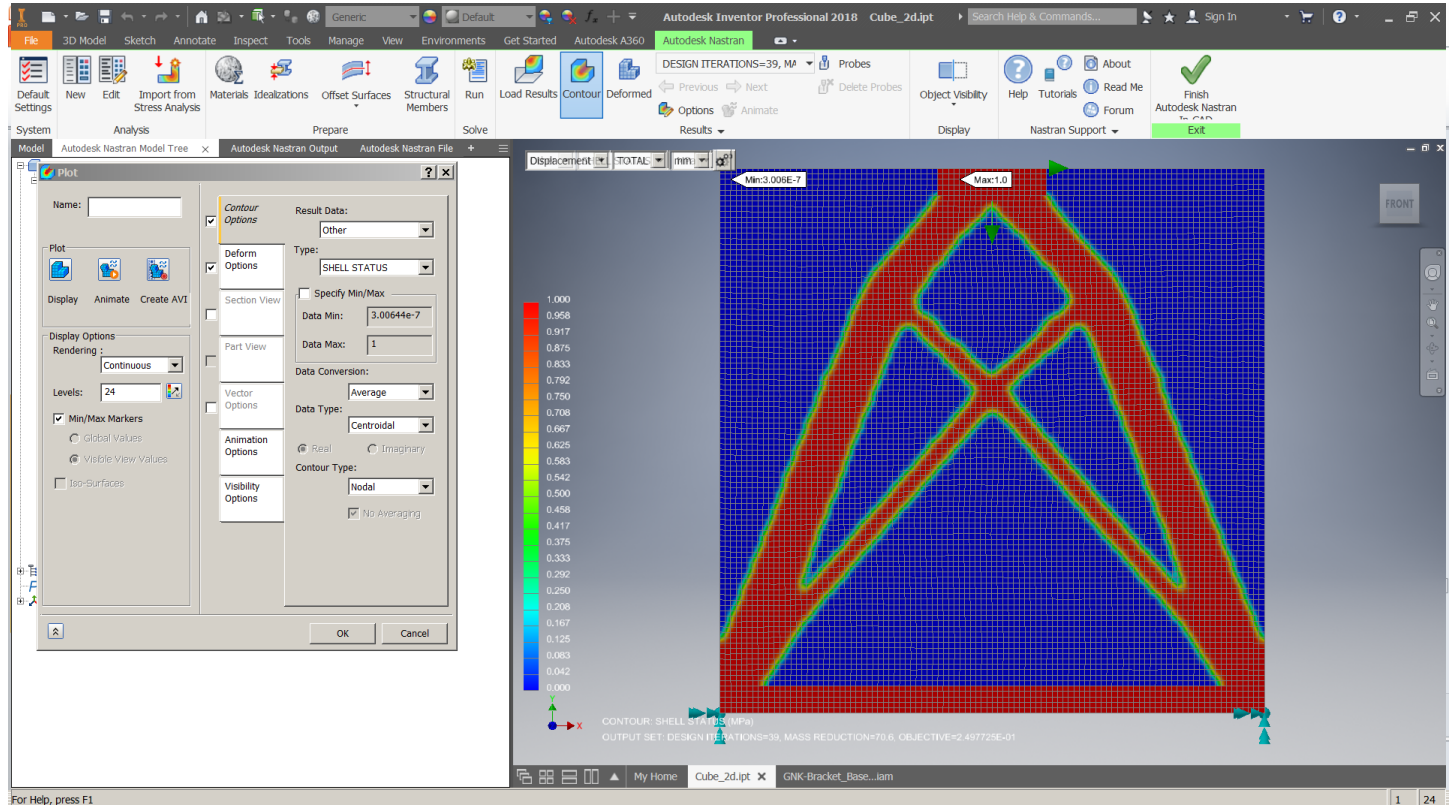


Obj: Min. VF (mass), Constraint: Stress & Comp. Index



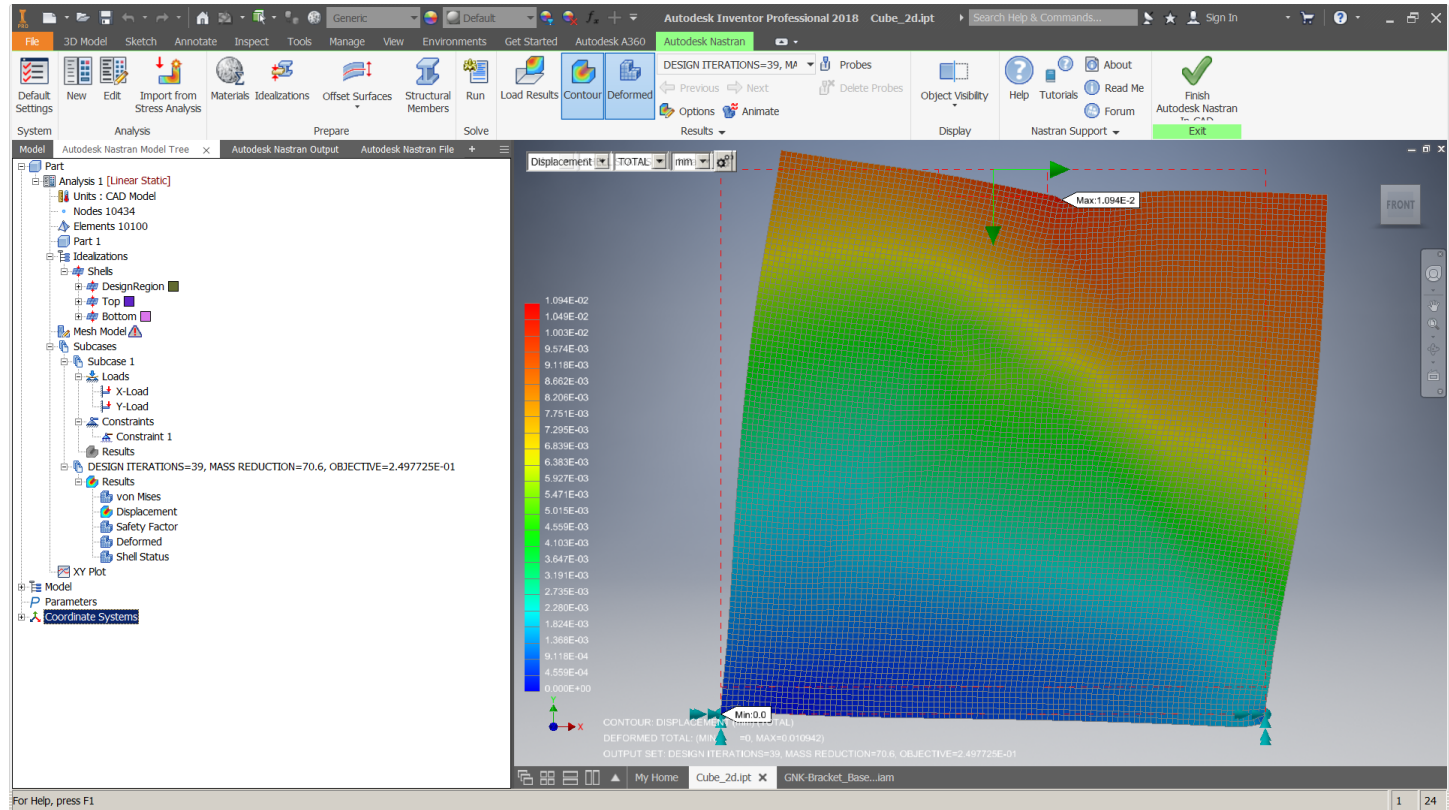
Obj: Min. VF (mass), Constraint: Disp. & Comp. Index

NTOPTSTRESSDIV	10
TOPTCOMPINDEX	5
TOPTDESIGNCONSTR	0.01
TOPTDESIGNREGION	1
TOPTTELEMEXTTOL	1.0E-2
TOPTTELEMSYMTOL	0.9
TOPTGEN	VFDISP
TOPTMANCONSTR	SYM
TOPTMANCORD	0
TOPTMANDIR	YZ

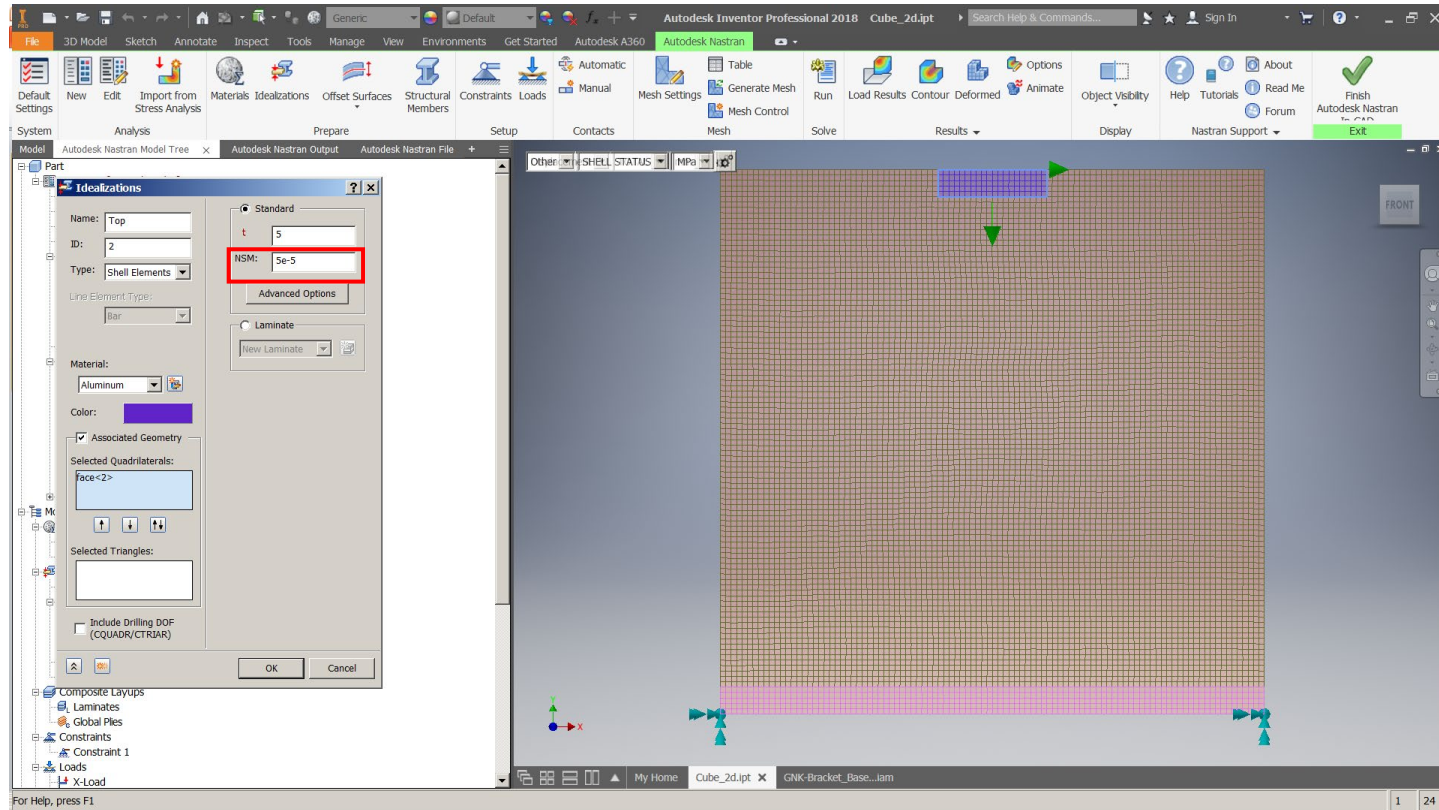


Obj: Min. VF (mass), Constraint: Disp. & Comp. Index

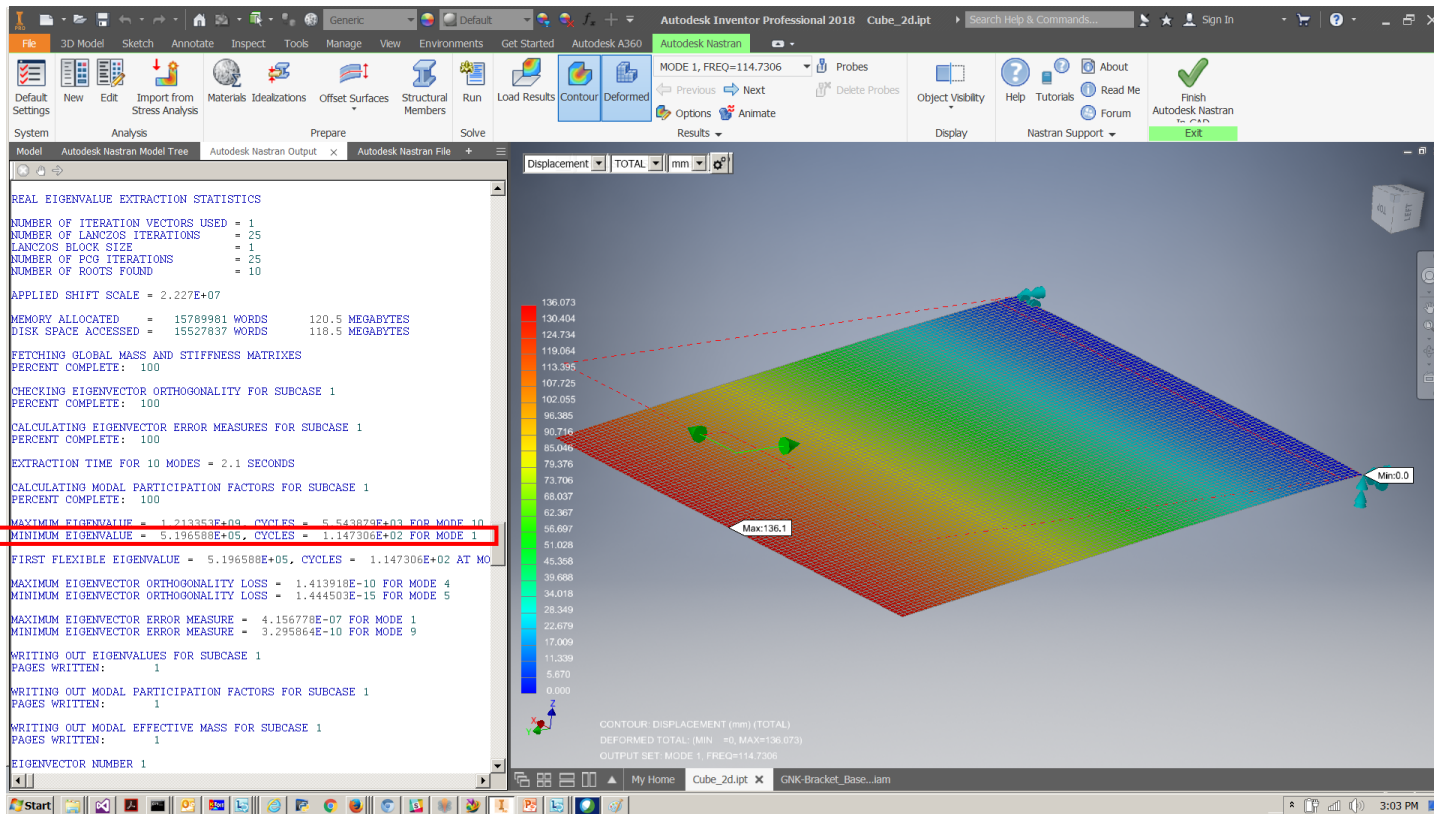
NTOPTSTRESSDIV	10
TOPTCOMPINDEX	5.
TOPTDESIGNCONSTR	0.01
TOPTDESIGNREGION	1
TOPTTELEMEXTTOL	1.0E-2
TOPTTELEMSYMTOL	0.9
TOPTGEN	VFDISP
TOPTMANCONSTR	SYM
TOPTMANCORD	0
TOPTMANDIR	YZ



Obj: Min. VF (mass), Constraint: Frequency

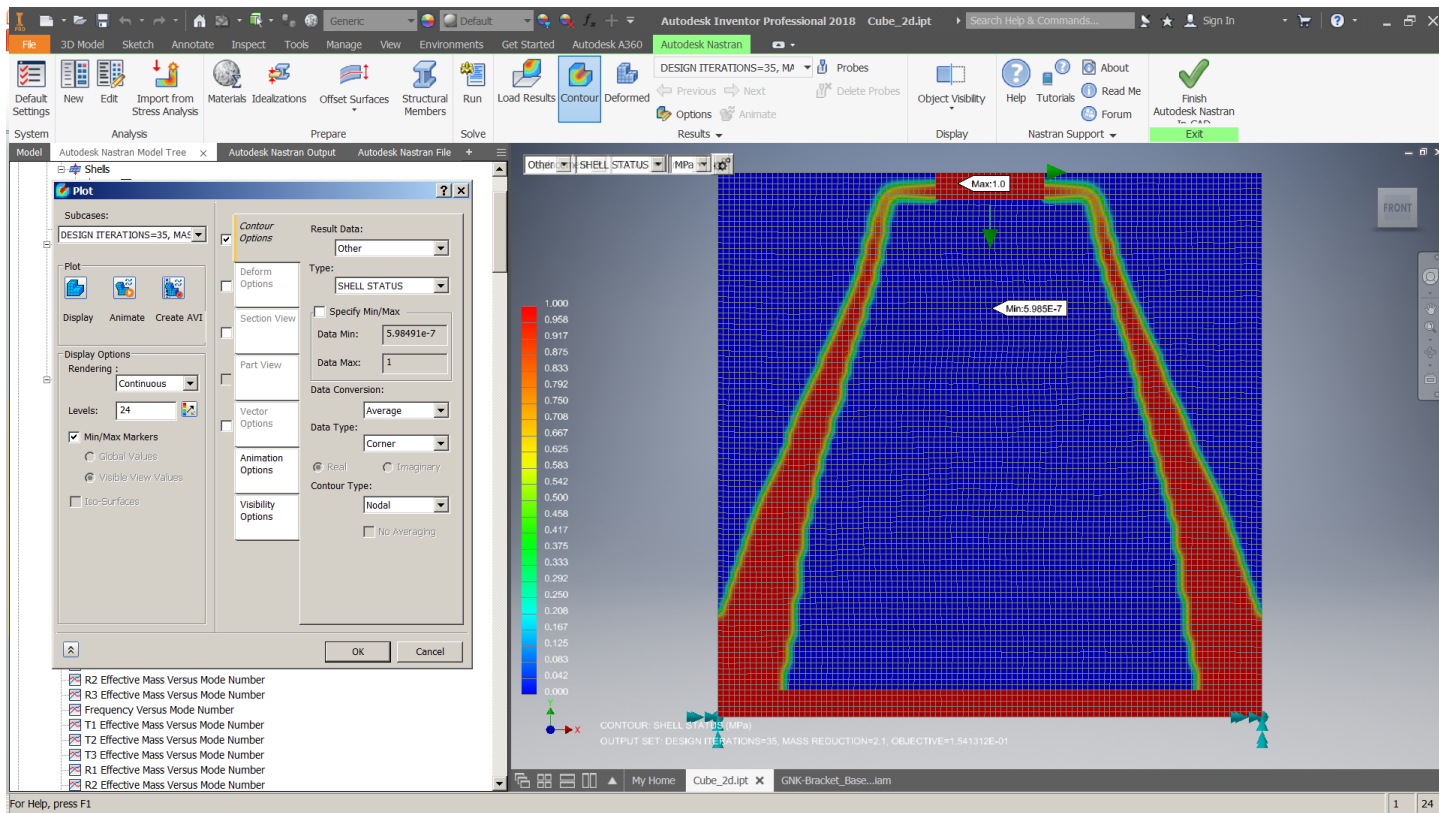


Obj: Min. VF (mass), Constraint: Frequency

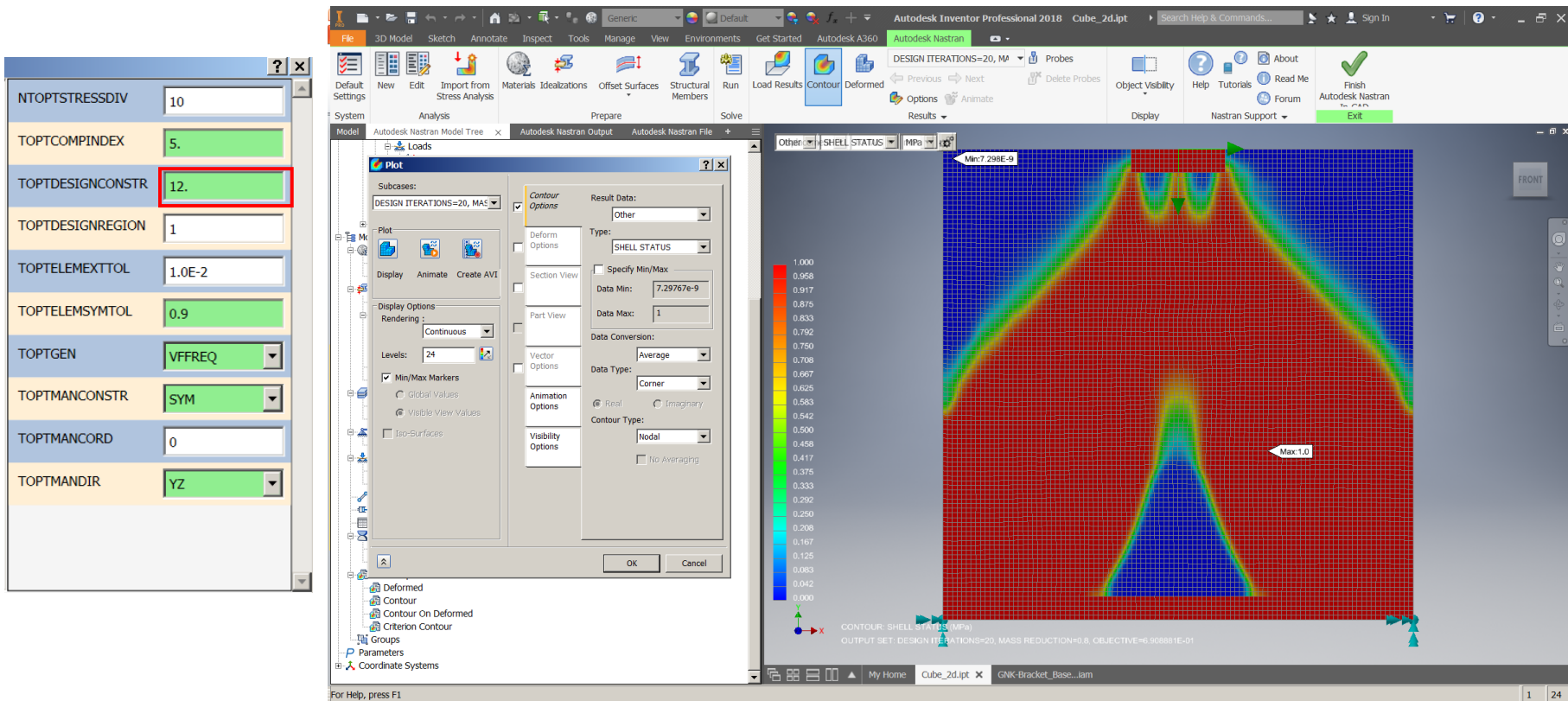


Obj: Min. VF (mass), Constraint: Frequency > 10Hz

NTOPTSTRESSDIV	10
TOPTCOMPINDEX	5.
TOPTDESIGNCONSTR	20.
TOPTDESIGNREGION	1
TOPTTELEMXTTOL	1.0E-2
TOPTTELEMSYMTOL	0.9
TOPTGEN	VFFREQ
TOPTMANCONSTR	SYM
TOPTMANCORD	0
TOPTMANDIR	YZ



Obj: Min. VF (mass), Constraint: Frequency > 12Hz



Commonly Used Topology Optimization Parameters

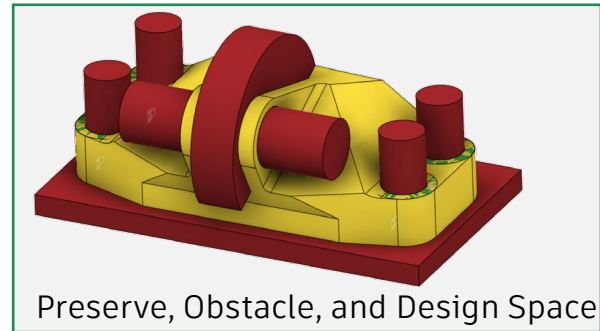
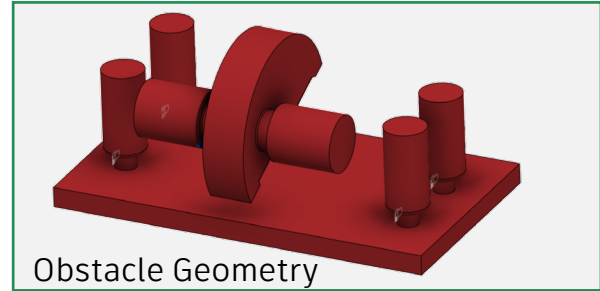
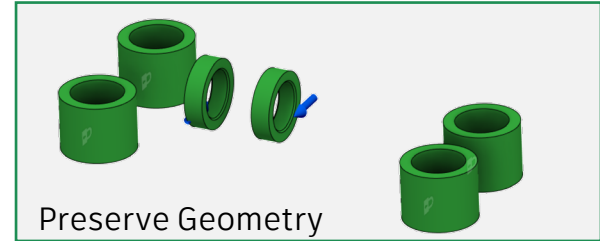
Parameter	Description	Default	Suggested Range	Remarks
MAXOPTITER	Limits the number of design iterations	200	100 - 300	Increase when iteration limit exceeded
TOPTLEMSYMTOL	Tolerance for symmetry manufacturing constraint	1.0E-02	< 1.0	Increase if elements are not linked
TOPTITERTOL	Tolerance for overall design iteration tolerance	5.0E-03	< 1.0E-02	Reduce for better accuracy/increase for better performance

Generative Design Versus Topology Optimization

- Topology Optimization (TO) is a subset of Generative Design
- TO is a tool based on B-Rep geometries and the Finite Element Method that is used primarily **to remove unnecessary material** from an engineer's best guess at one initial design shape of a product
- Generative Design (GD) is a **design exploratory process** that is based on specified constraints such as design space, materials, cost, operational requirements, strength, and proposed manufacturing methods and is based on generative AI algorithms that perform DOE analyses to synthesize combinations of discrete values of the input variables within the user-specified ranges of possible values
- GD does not require an initial design but **establishes a design domain for each set of design variables** and **creates the geometry** associated with the specified boundary conditions such as locations of applied loads, supports, and interfaces with other parts and selected regions within a design domain can be restricted from being modified
- Benefits of using GD:
 - Reduced weight and amount of materials
 - Reduced product development time
 - Reduced costs
 - Affords very complex geometrical designs amenable to AM processes
 - Creates an array of novel, yet feasible, design concepts for consideration by the design engineer

Autodesk Fusion 360 Generative Design

- The user defines preserve geometry and obstacles
 - Preserve geometry is any geometry not in the design space (e.g., attachment and load points)
 - Obstacles are empty spaces where material is not placed (e.g., allows access for attachments and tools)
- A design space is created either using starting geometry or a bounding box if no starting geometry is provided
- The user defines loads and boundary conditions on the preserve geometry
- Better support for manufacturing constraints and the ability to use multiple design constraints such as stress, displacements, frequency, and buckling all in one compliant design
- Multiple outcomes with advanced ways to sort designs based on weight and cost
- Seamless design verification
- Currently solid models only



Fusion 360 Generative Design Outcomes



