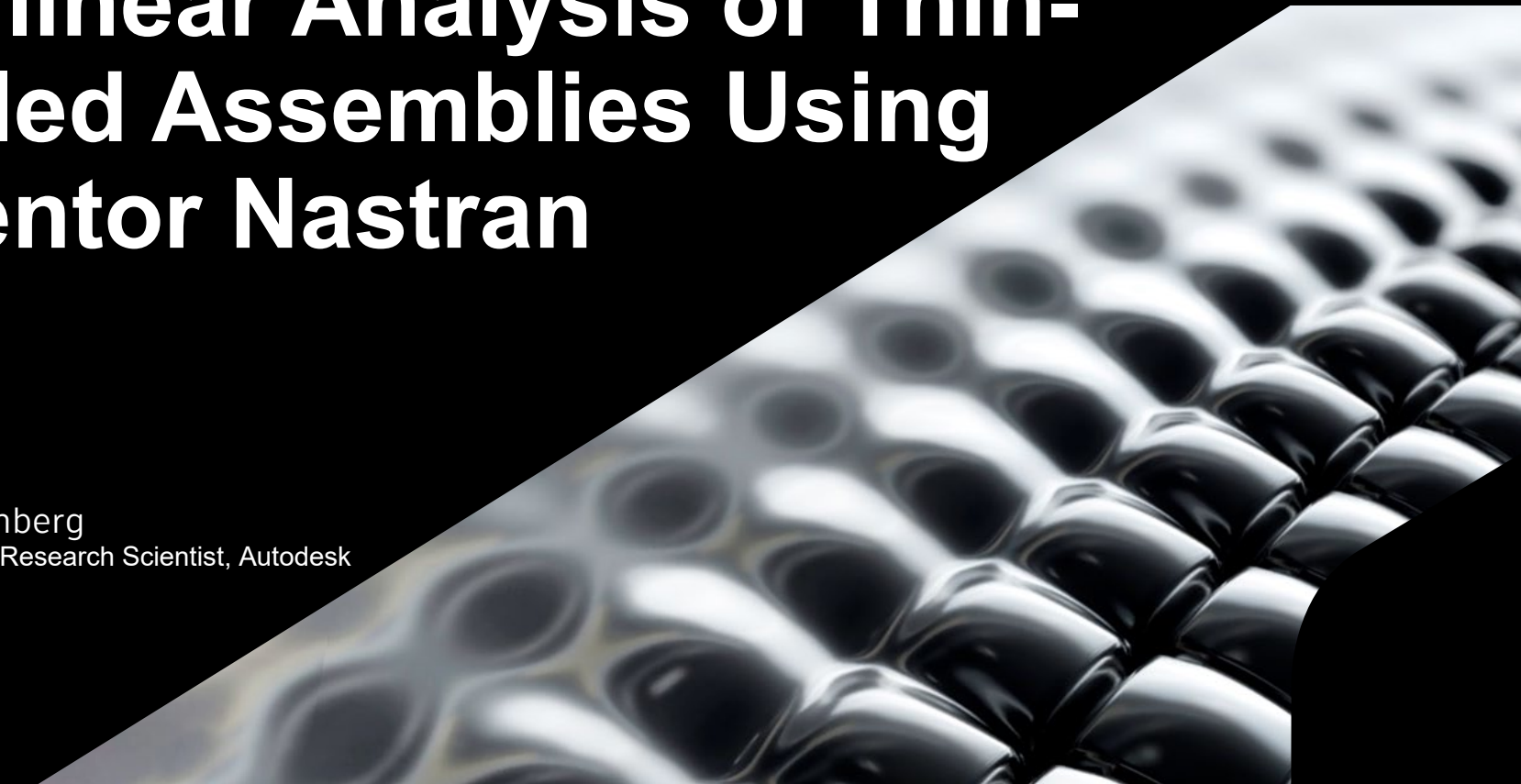
AUTODESK UNIVERSITY

# 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.

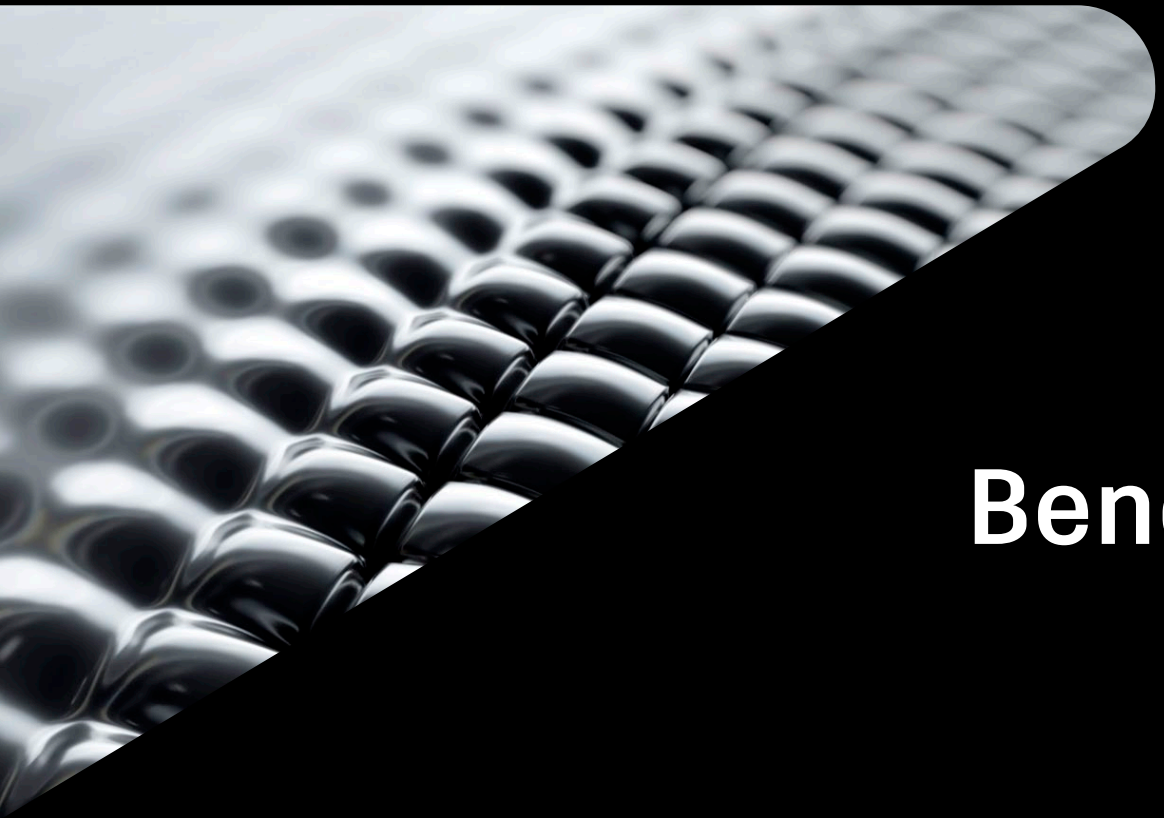
A discussion of factors that may affect future results is contained in our most recent Form 10-K and Form 10-Q filings available at www.sec.gov, including descriptions of the risk factors that may impact us and the forward-looking statements made in these presentations. Autodesk assumes no obligation 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.

# 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.

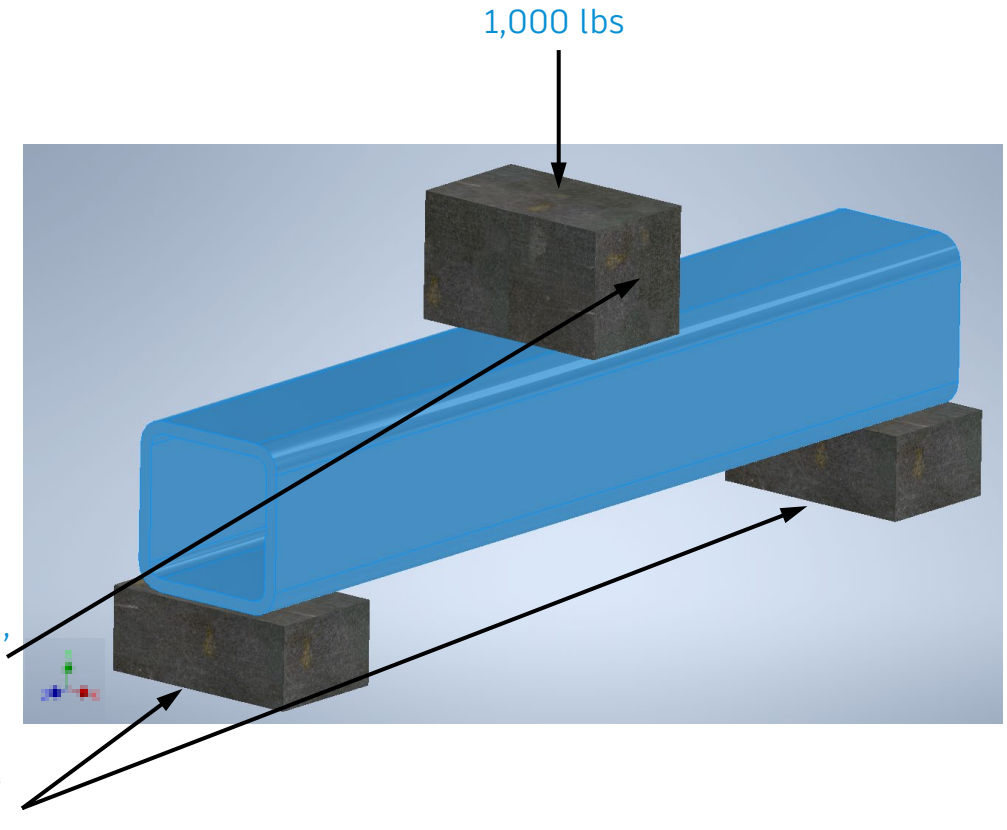


# 3-Point Tube Bending Example

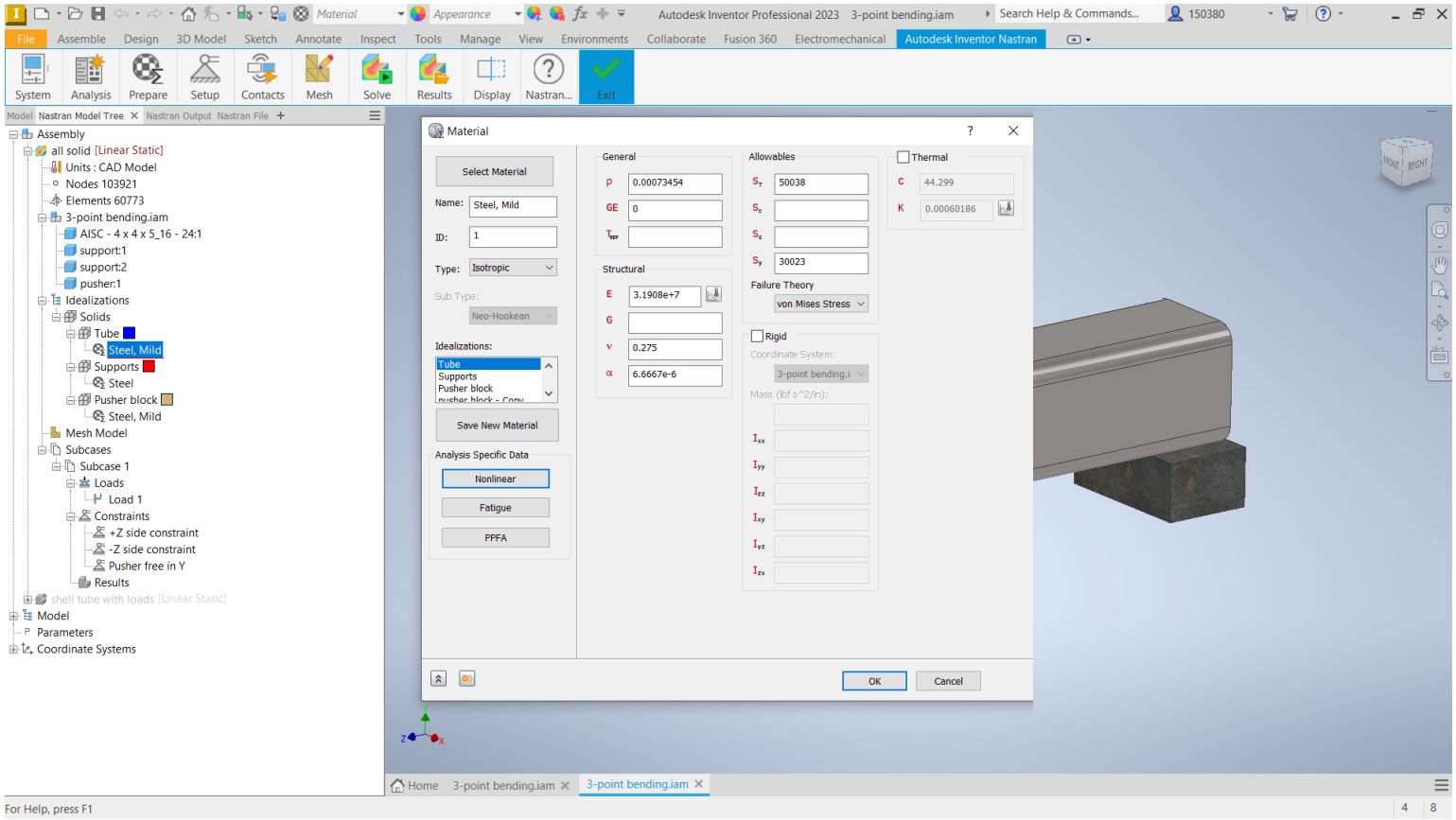
Solid Elements

# Steel Tube Model

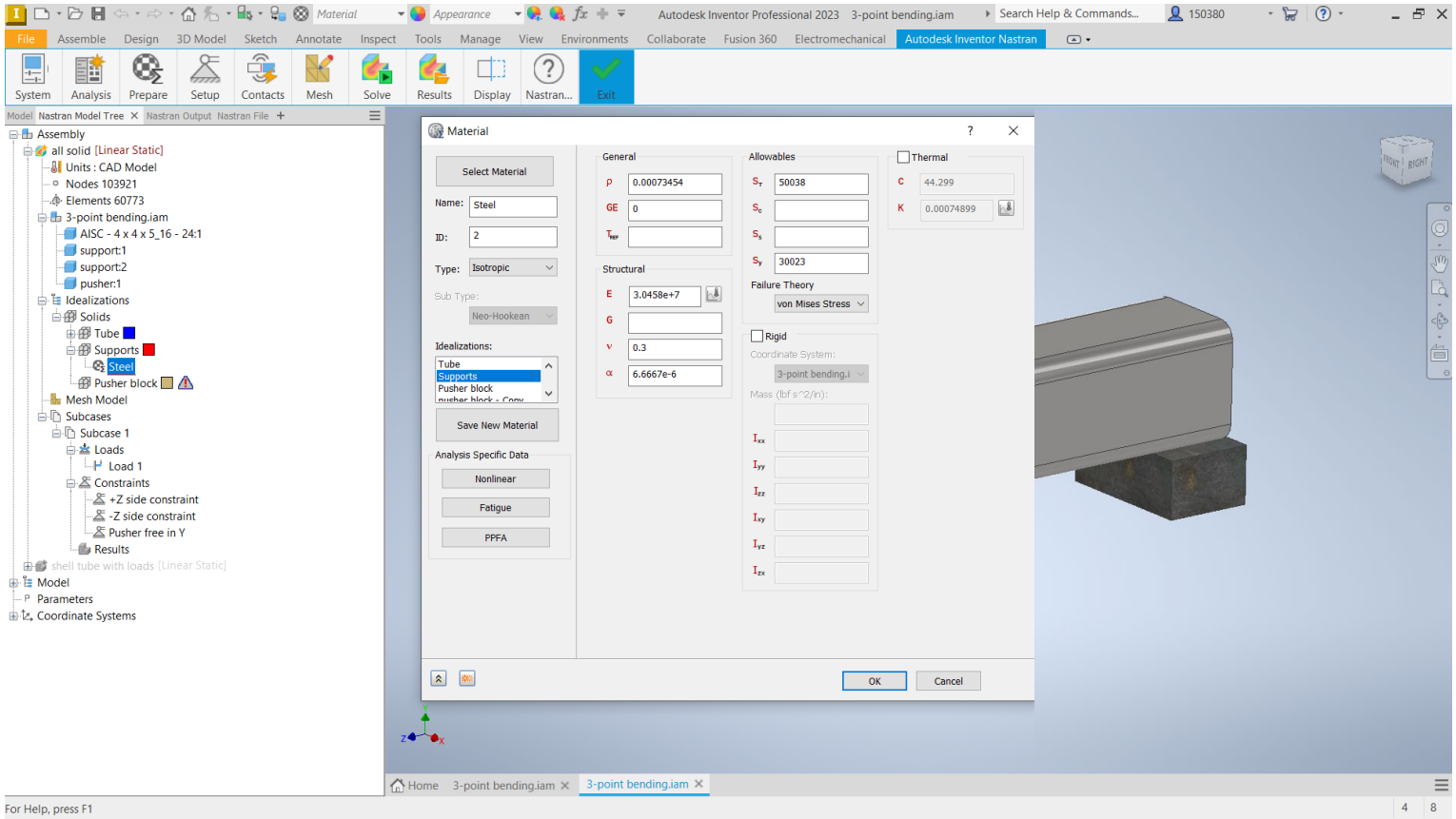
- Tube: AISC Steel:
  - 4 x 4 x 24 inches
  - 5/16 inches thick
  - Yield point 30ksi
- Blocks: Steel
- Constrained at the base of pads and side of pusher block



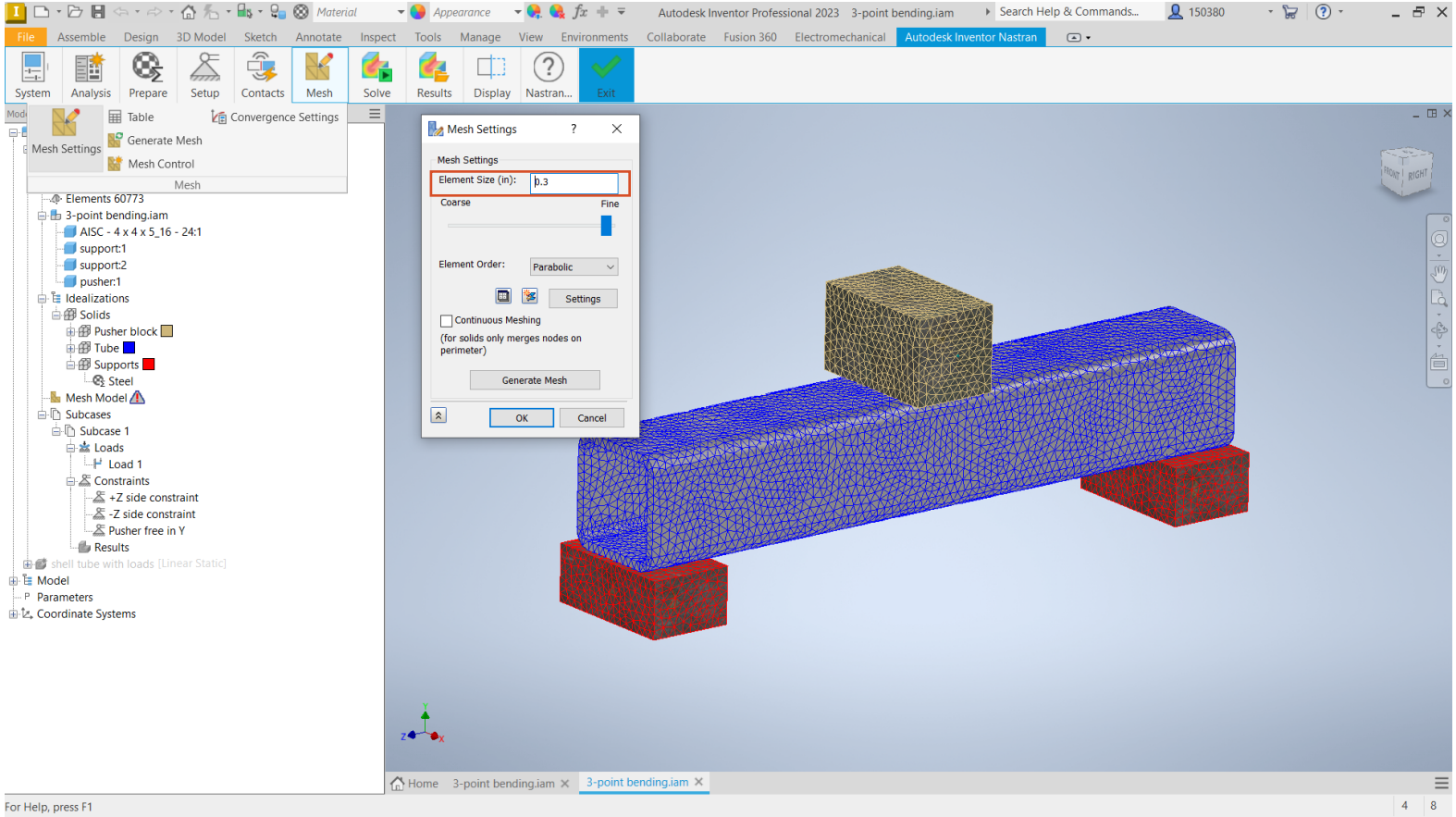
# Steel Tube Material Definition



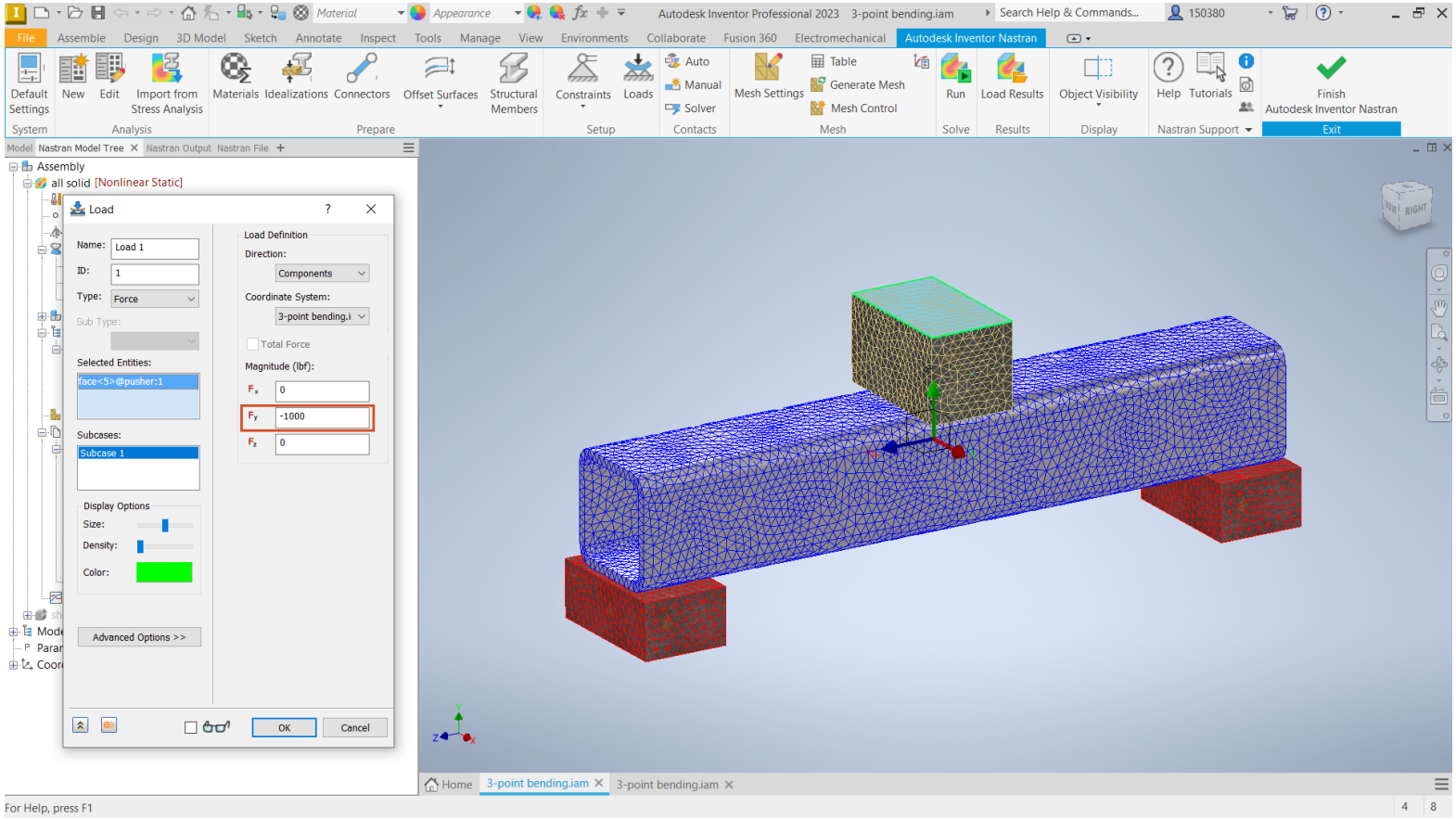
# Steel Supports and Pusher Material Definition



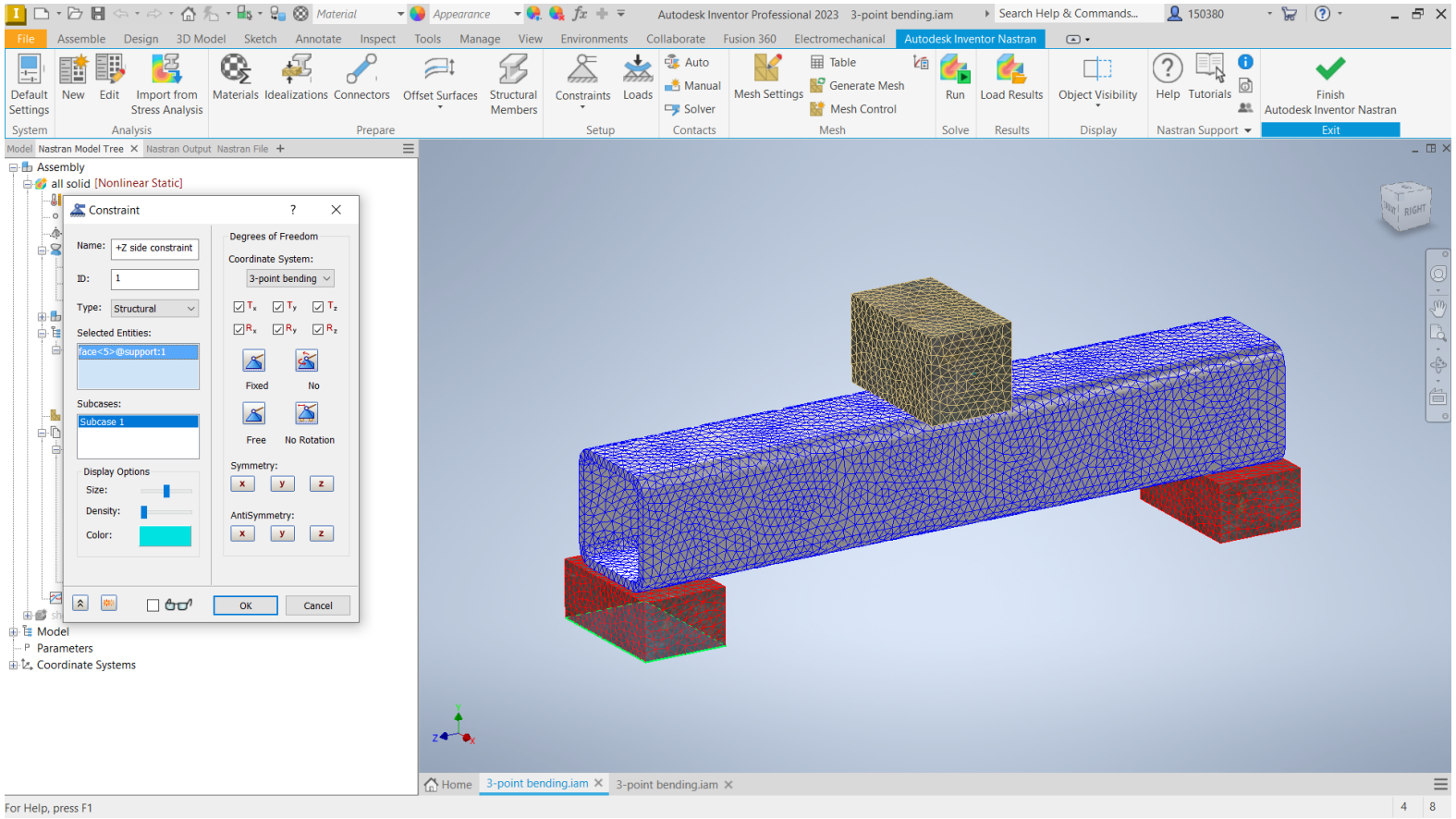
# Model Mesh Size Definition



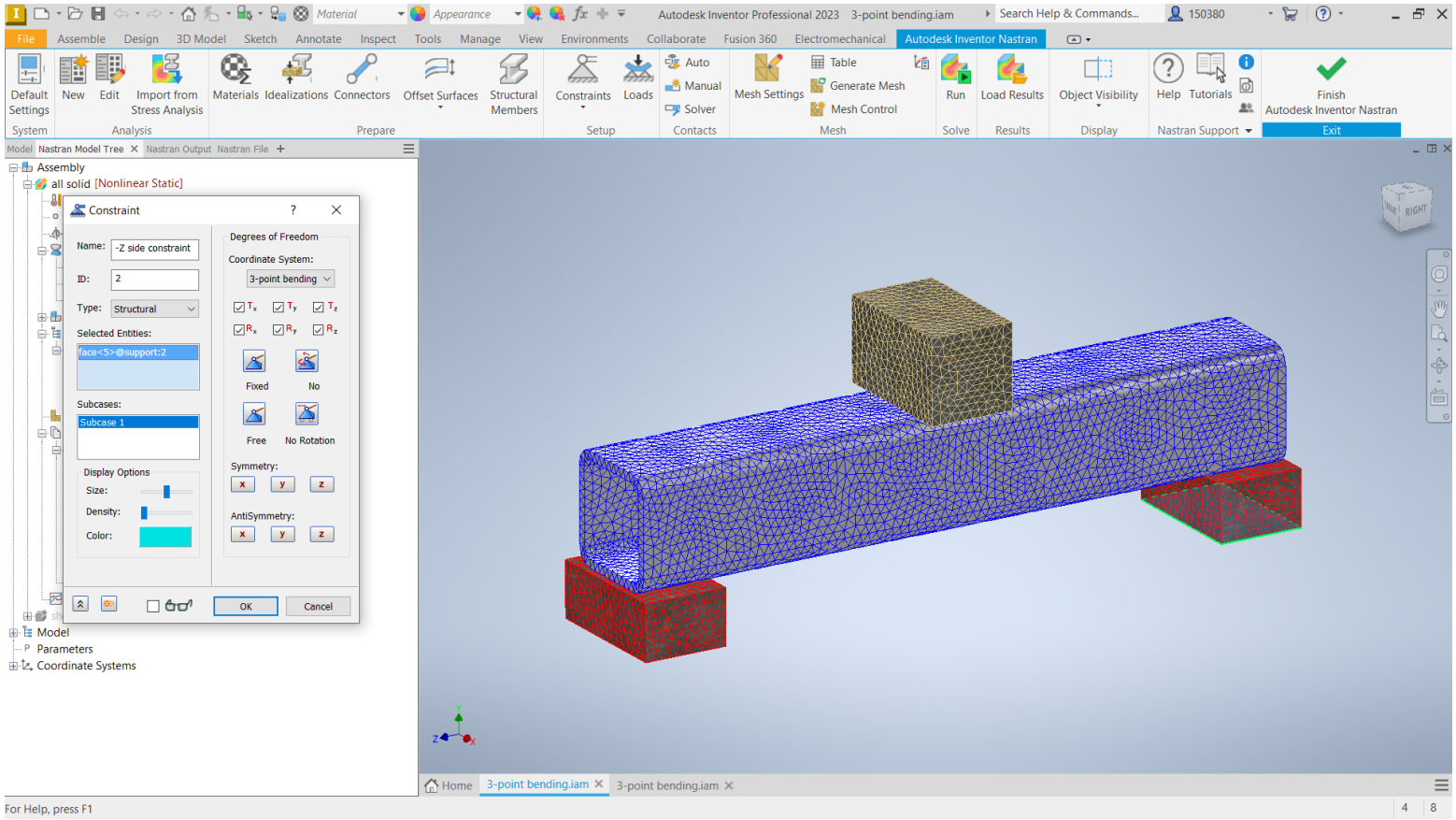
# Load Definition



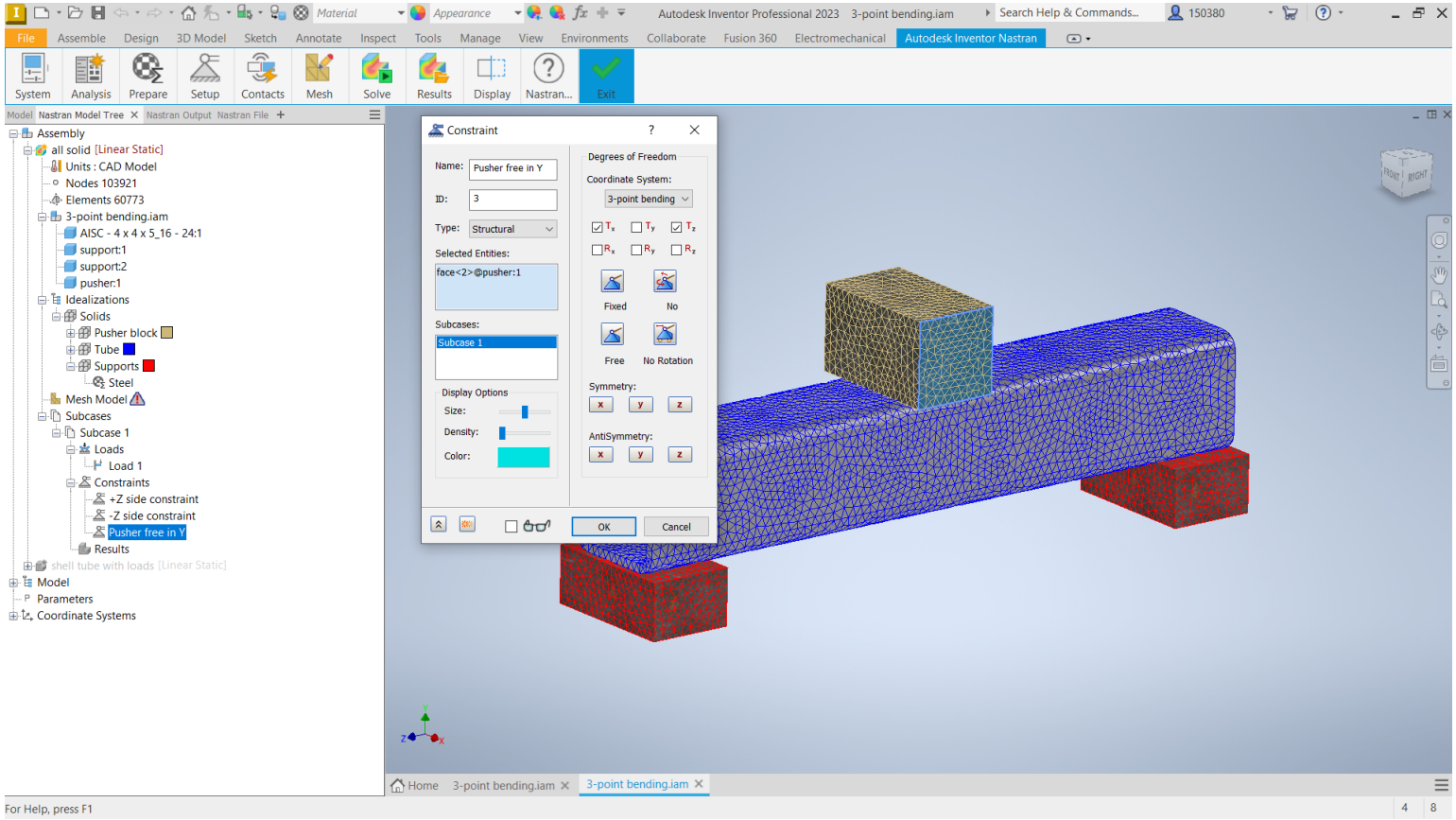
# +Z Support Constraint Definition



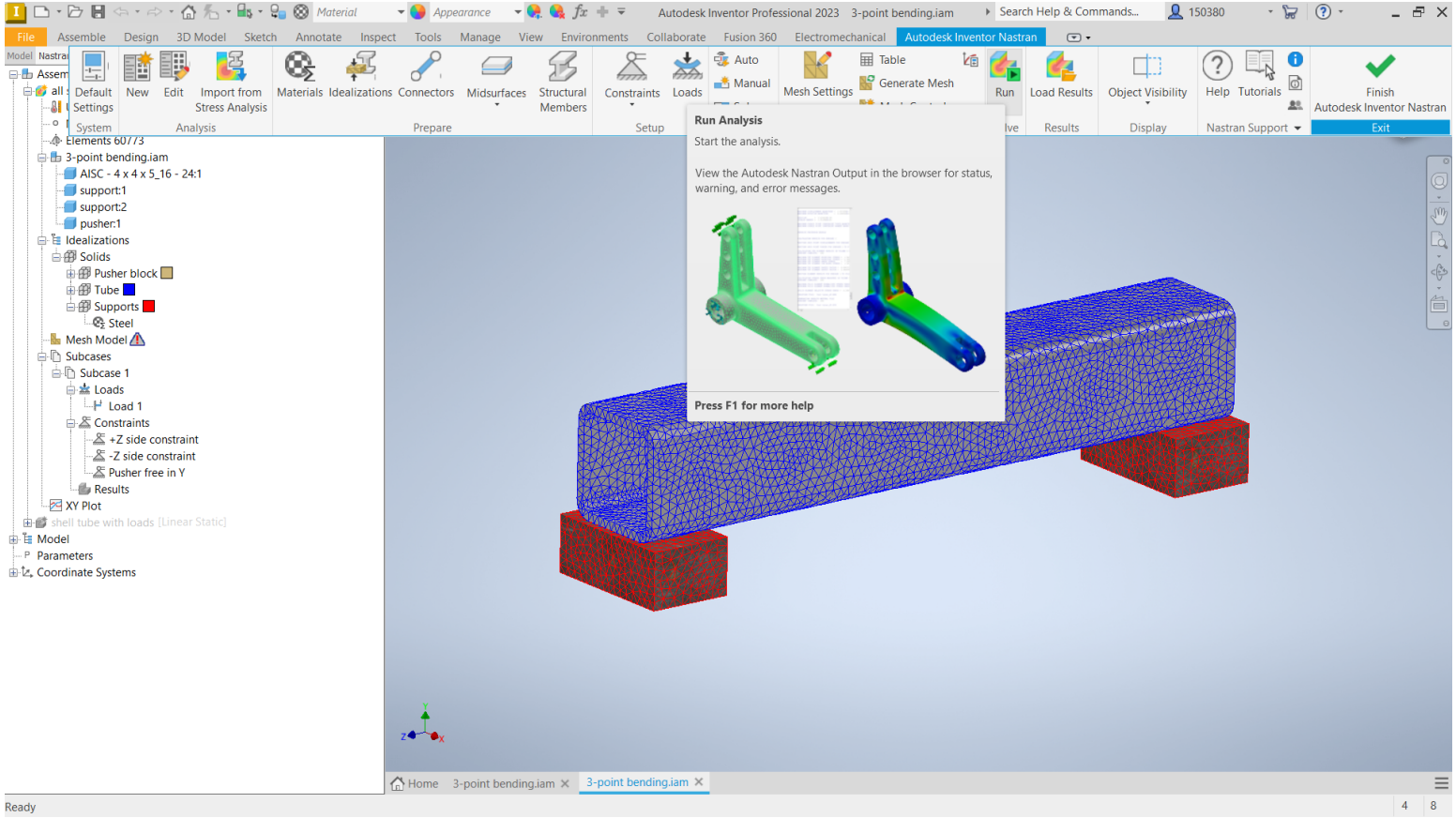
# -Z Support Constraint Definition



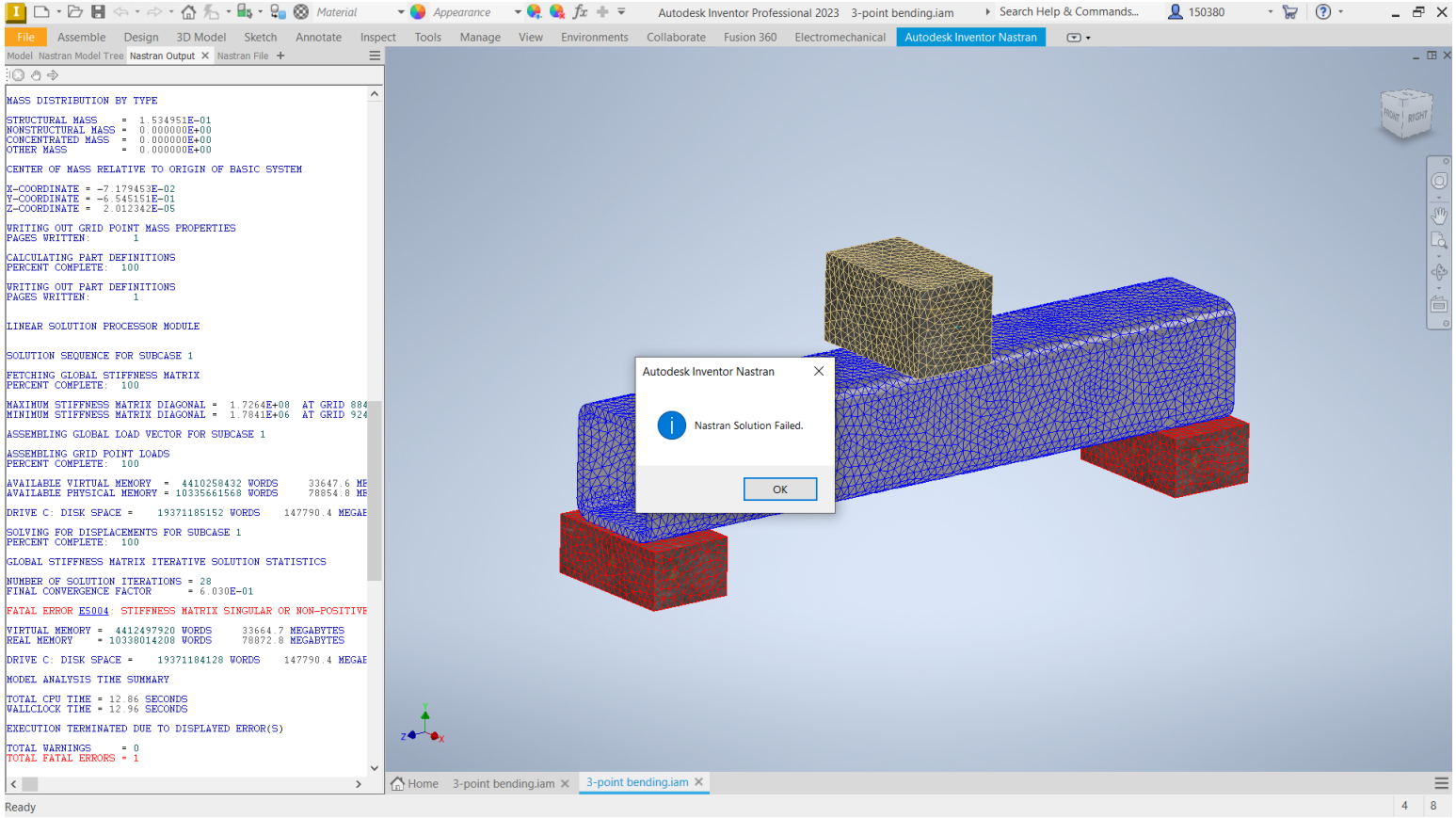
# Pusher Constraint Definition



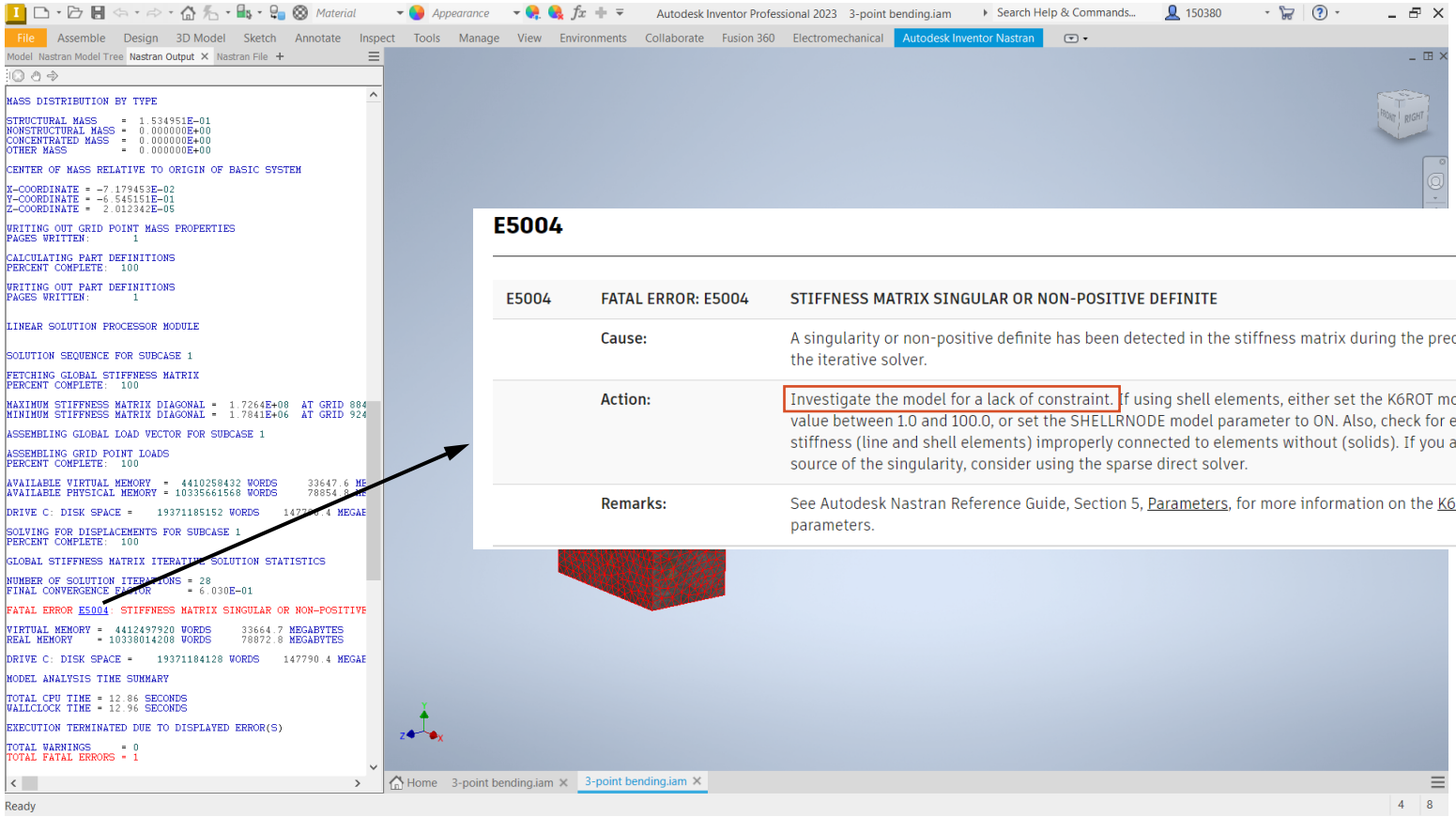
# Running Linear Statics



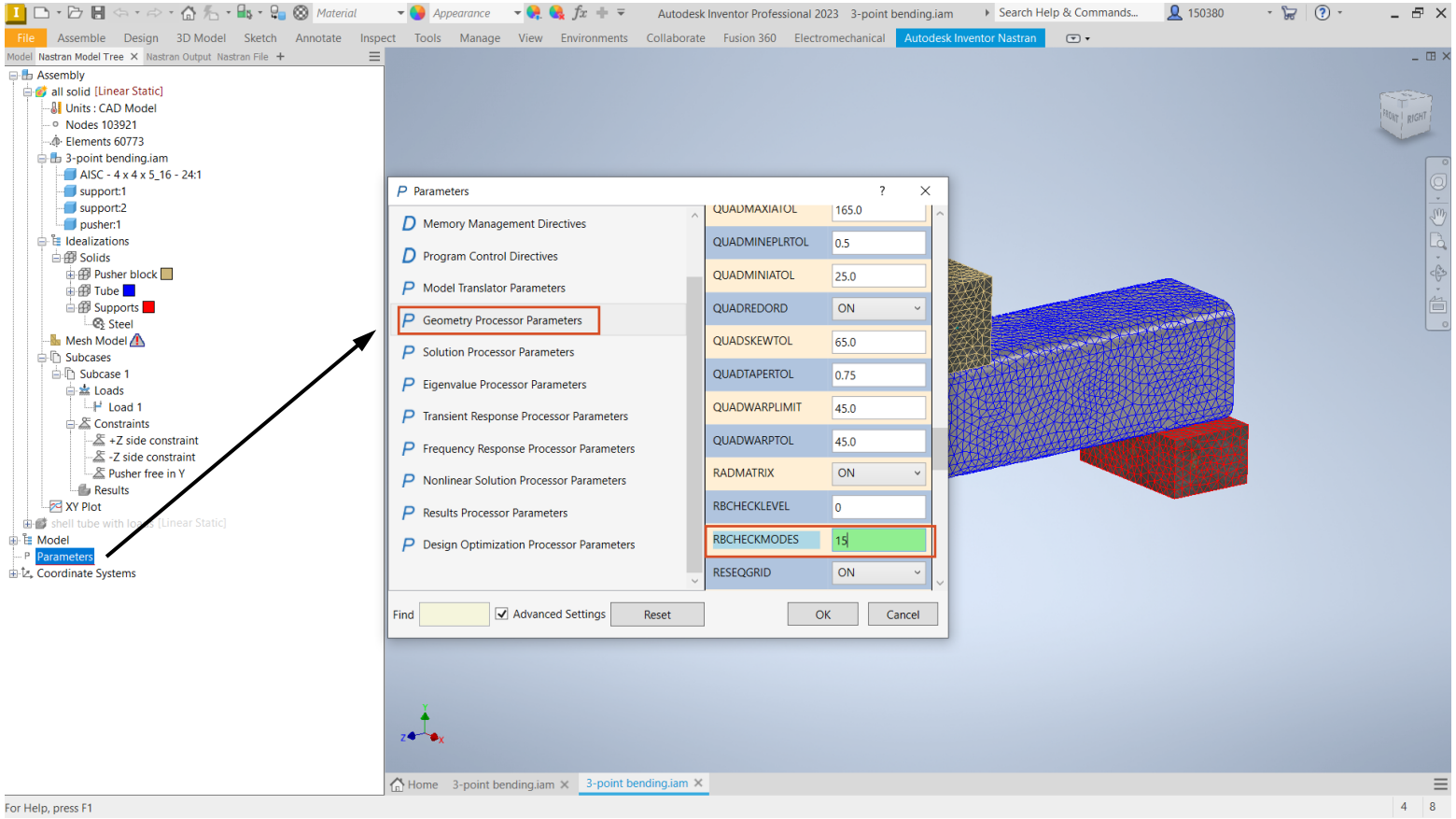
# Fatal Error!  Why?



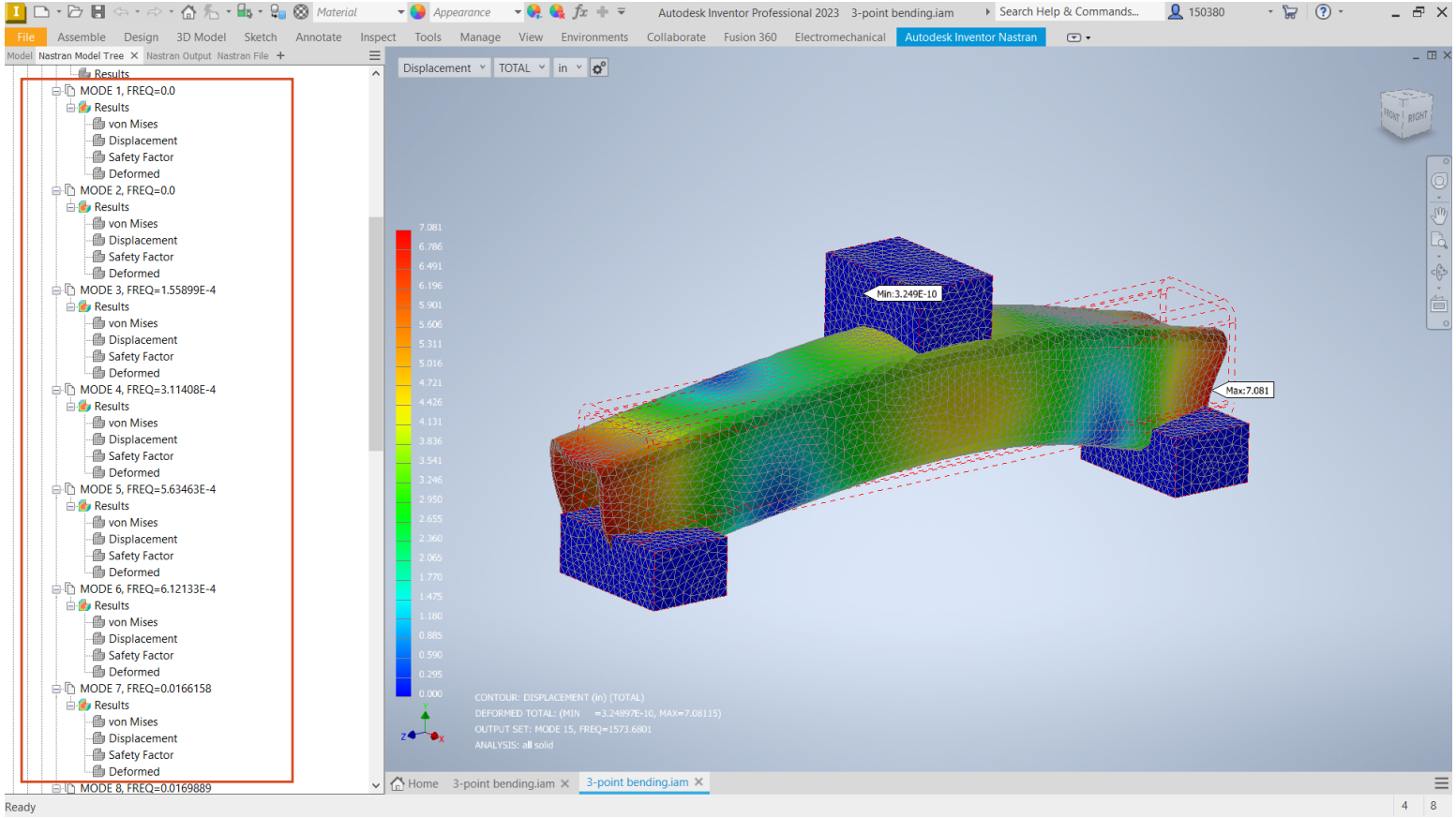
# Getting Help



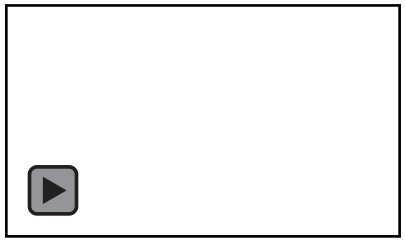
# Isolating the Issue…RBCHECKMODES Parameter



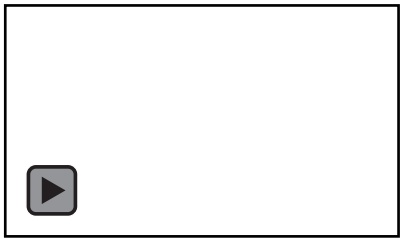
# Isolating the Issue



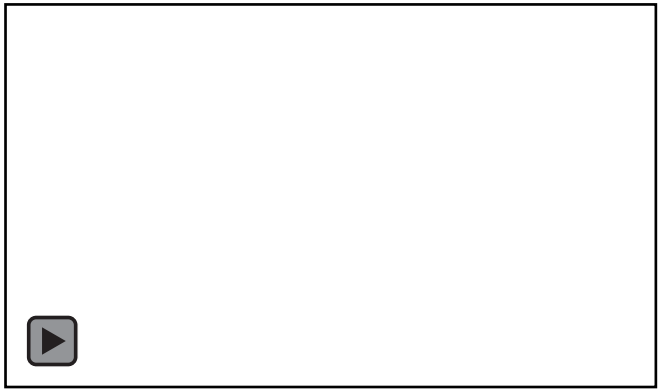
# Rigid Body Modes Results



Mode 1 – Rigid Body



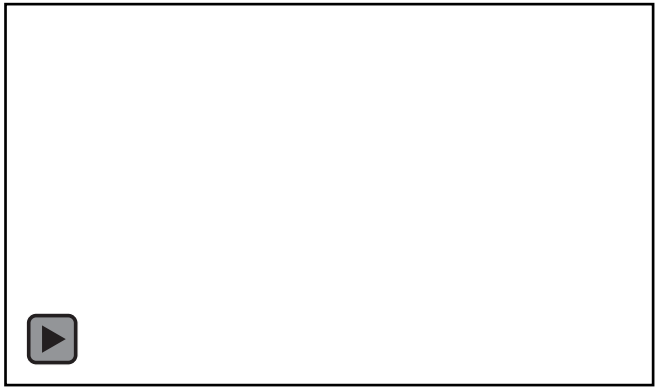
Mode 4 – Rigid Body



Mode 7 – Rigid Body – Flexible Expected

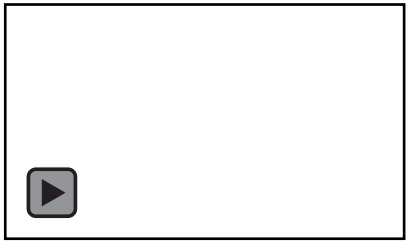
Mode 2 – Rigid Body

Mode 5 – Rigid Body



Mode 8 – Rigid Body – Flexible Expected

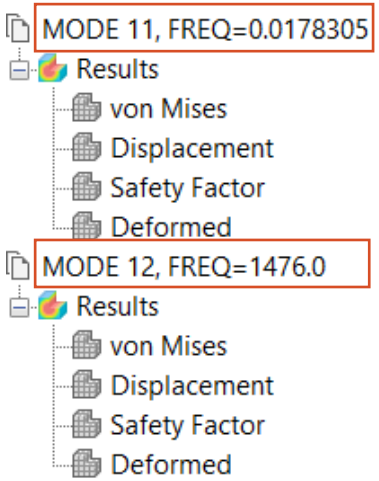
Mode 3 – Rigid Body



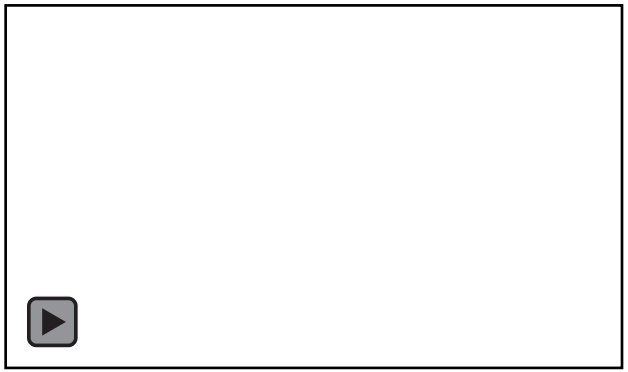
Mode 6 – Rigid Body

# 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

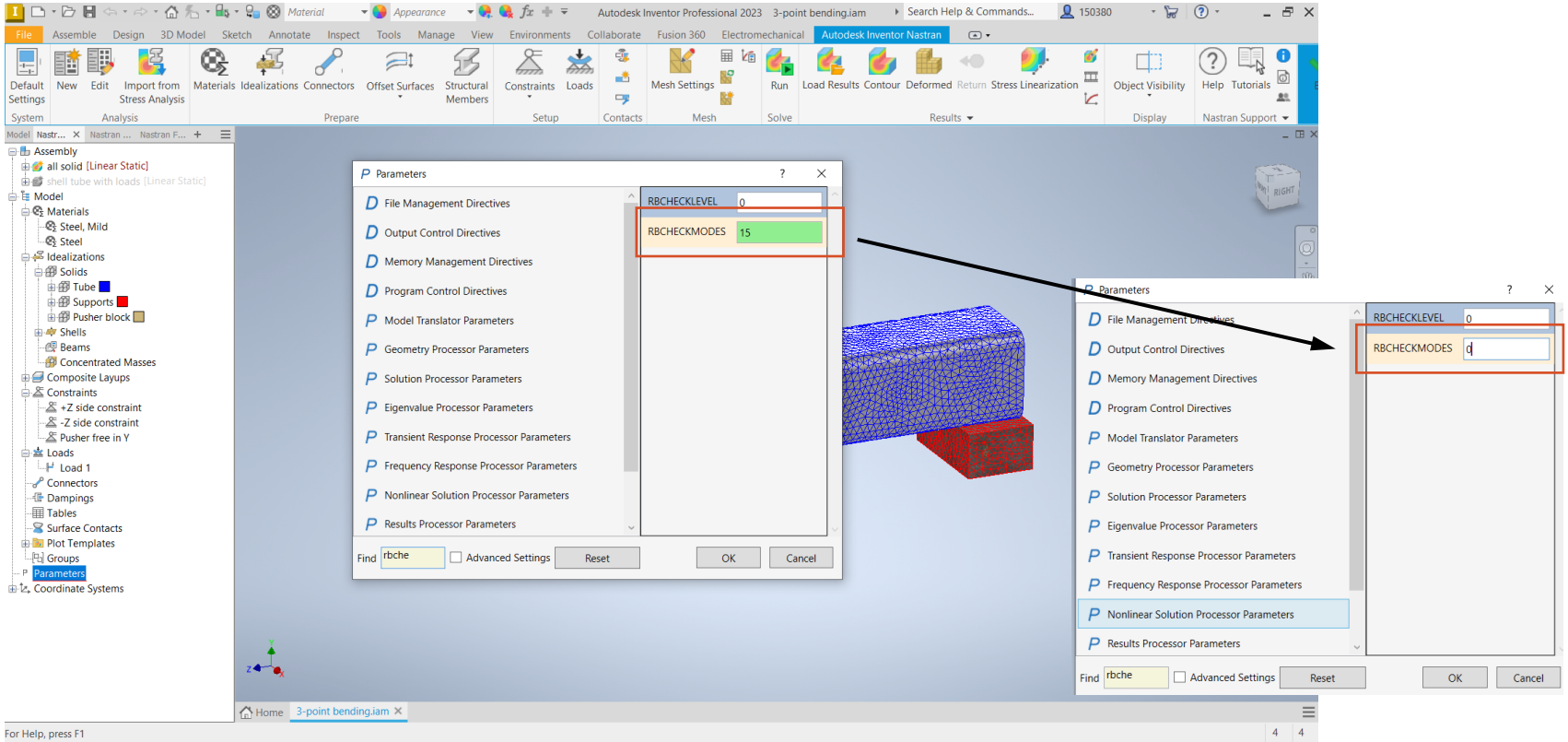



Mode 11 – Rigid Body

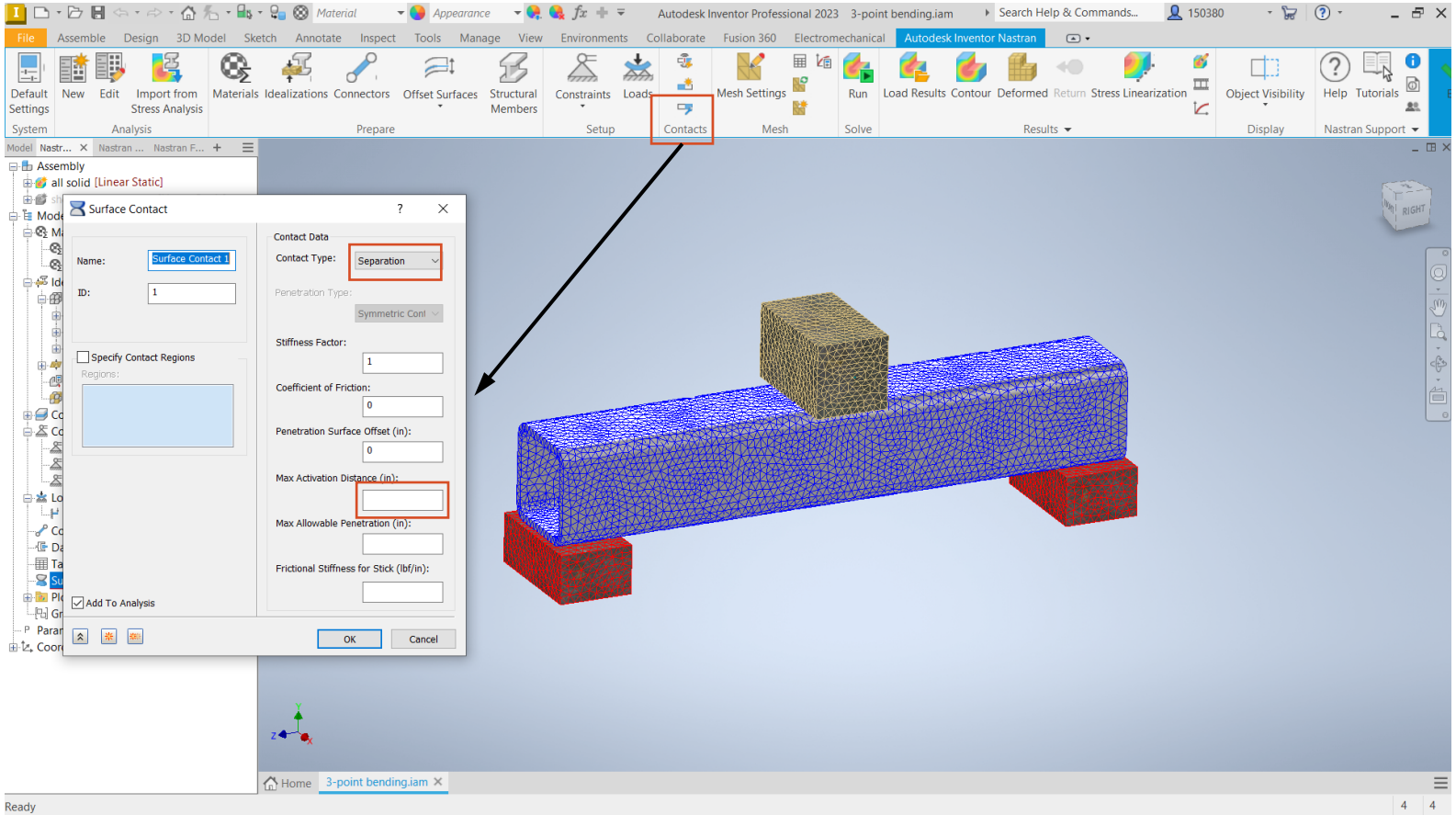


Mode 12 – Flexible

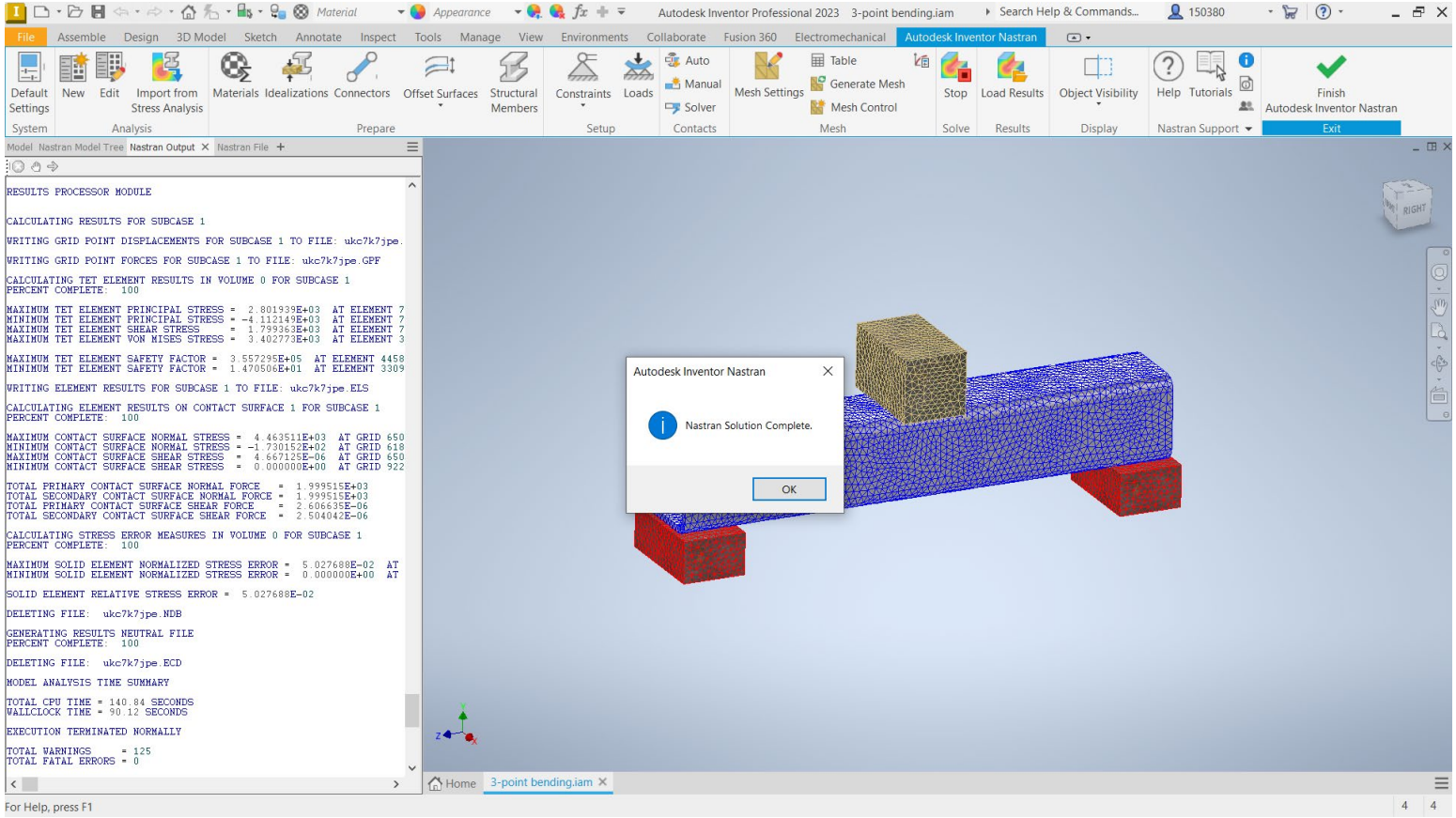
# Remember to Zero the RBCHECKMODES Parameter



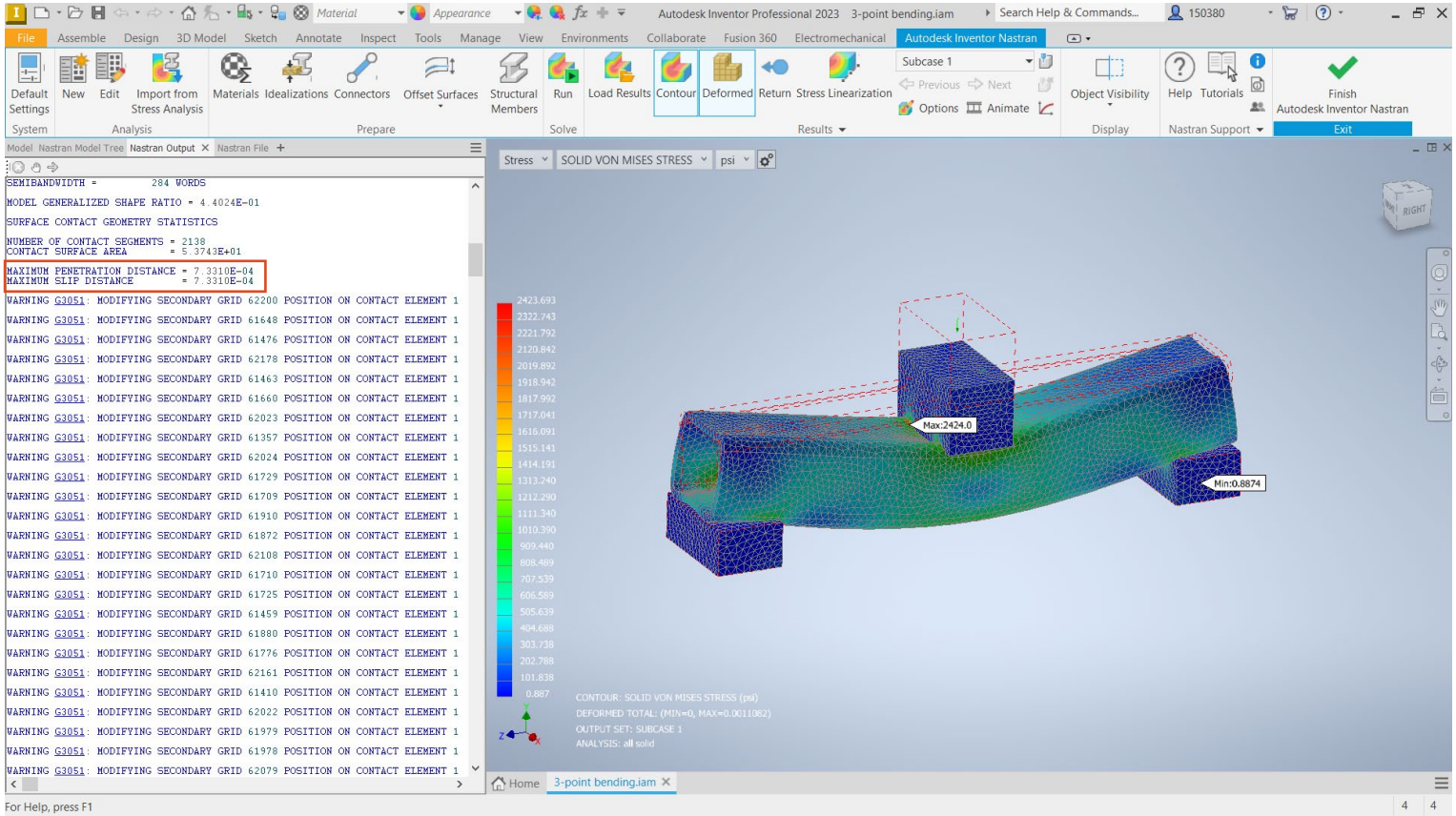
# Setting Up Solver Based Contact



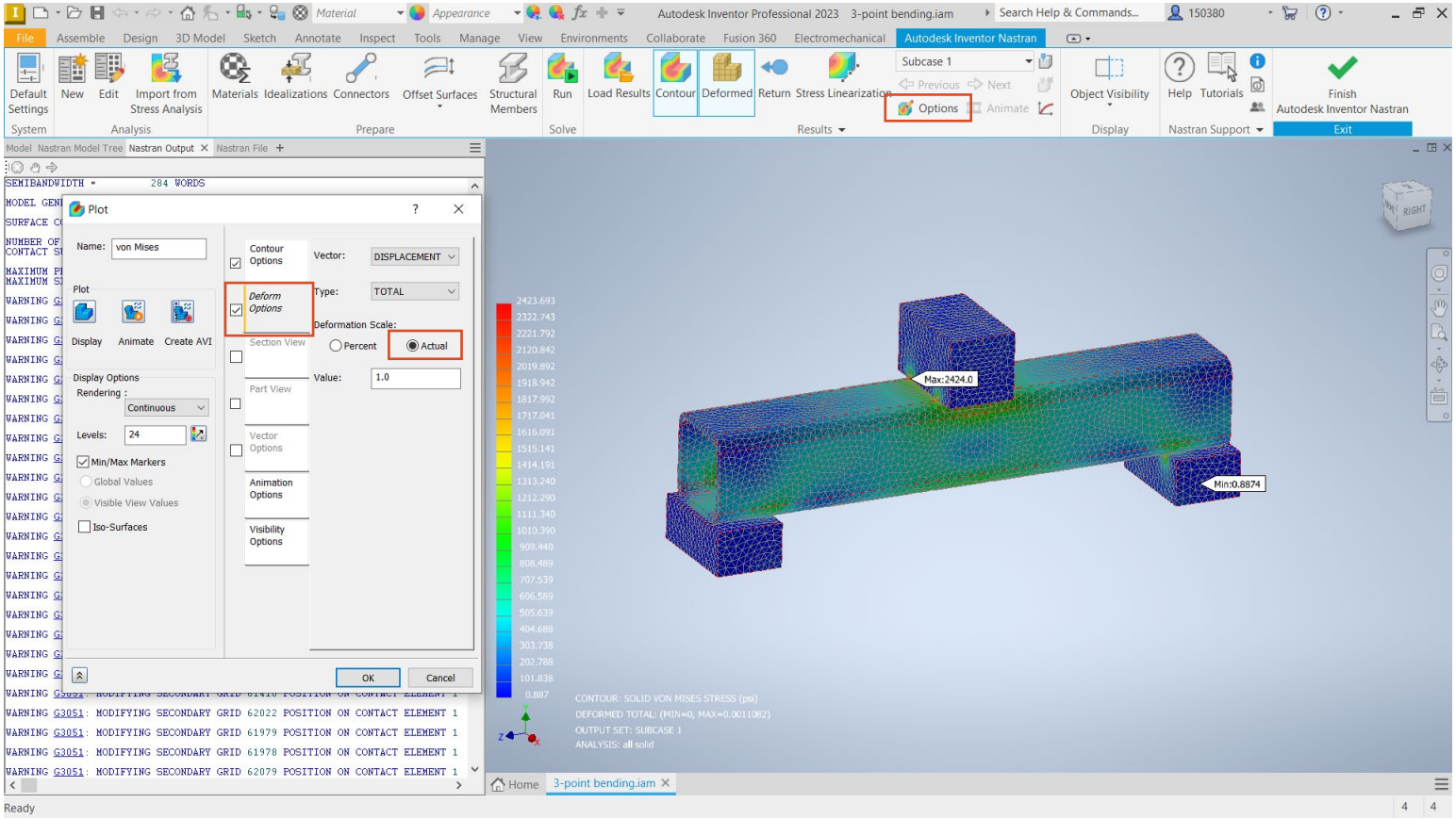
# Running with Linear Statics – Linear Contact



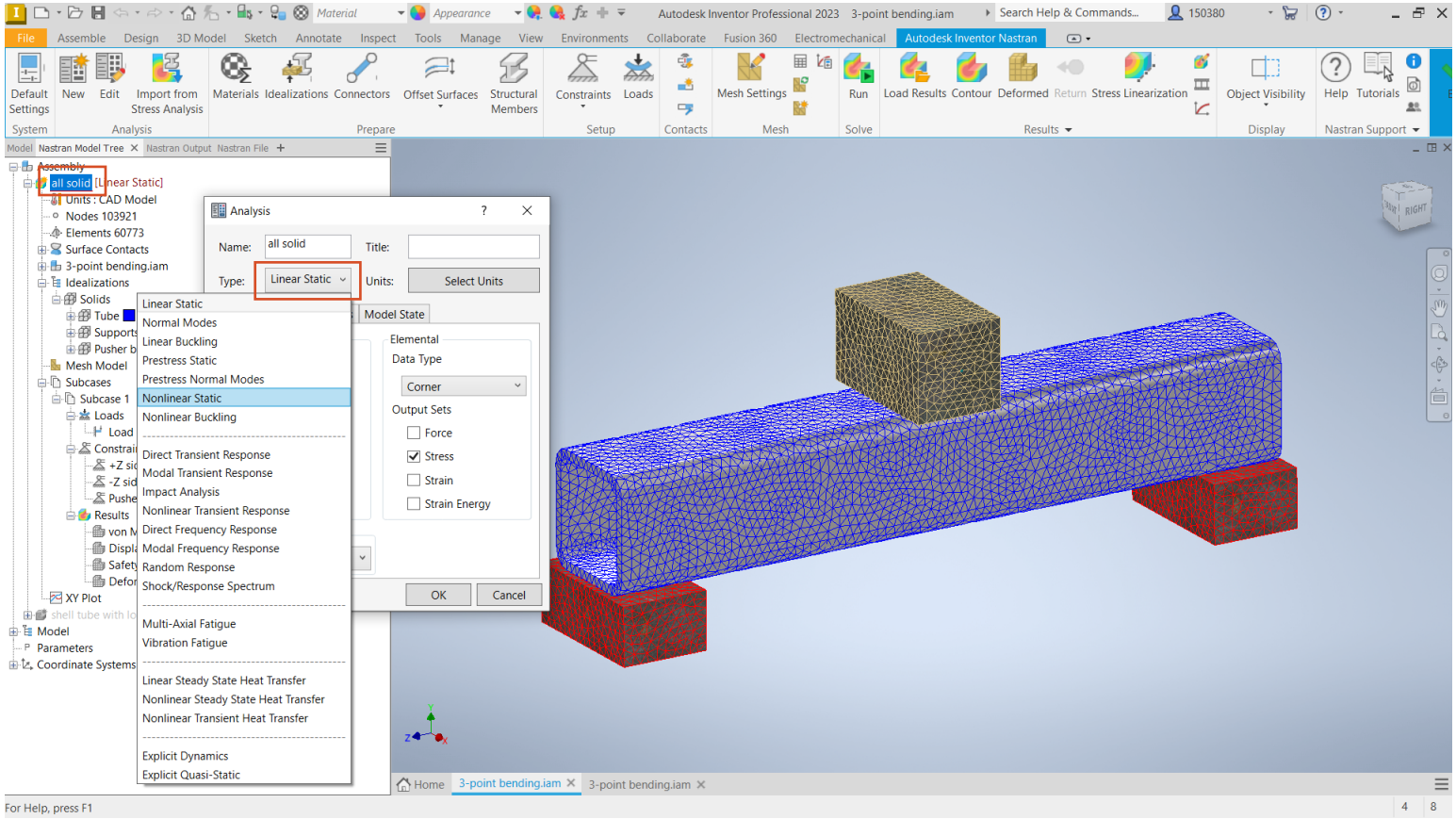
# Review Warning Messages



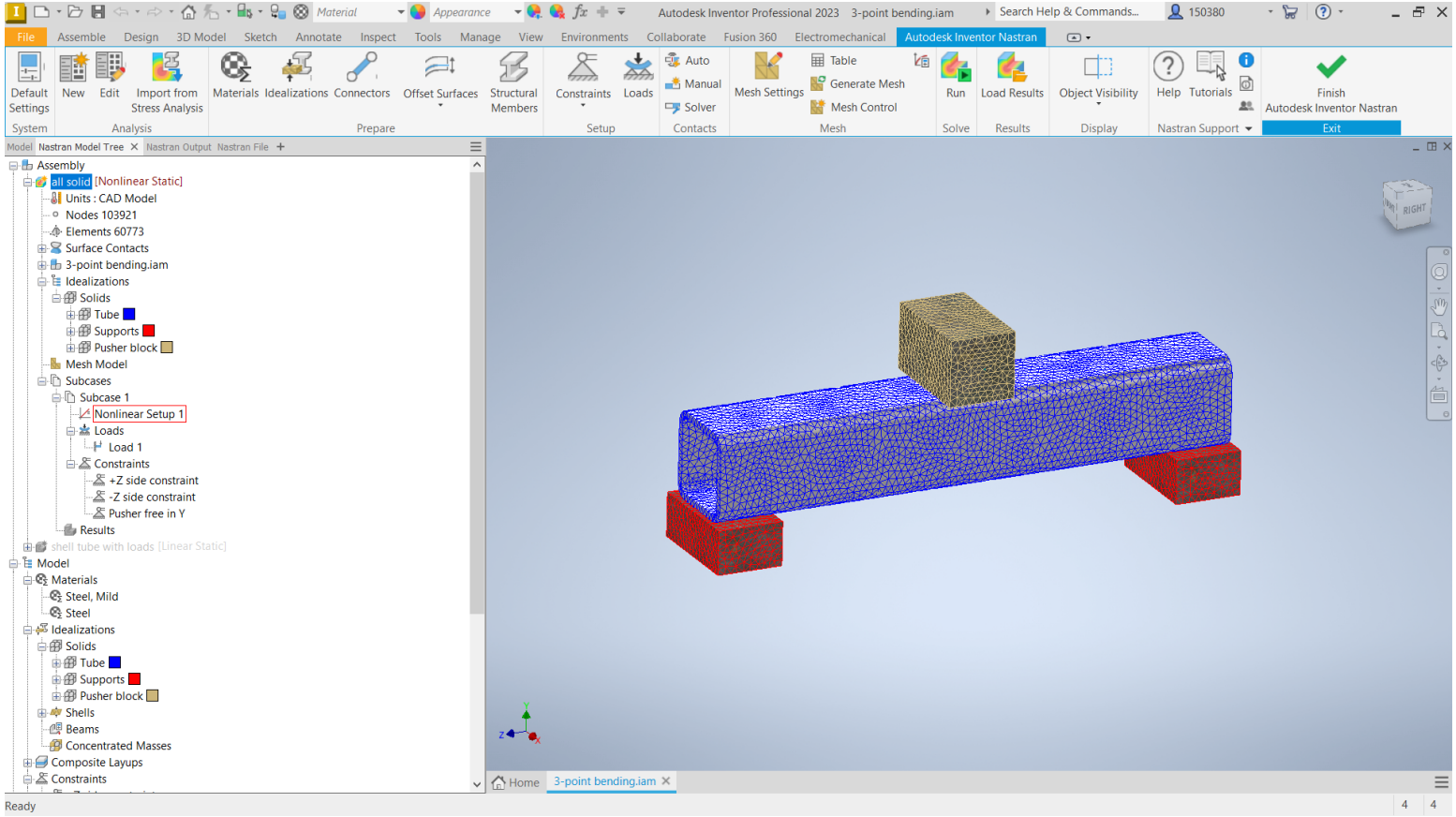
# View Deformed Shape Using Actual Deformations



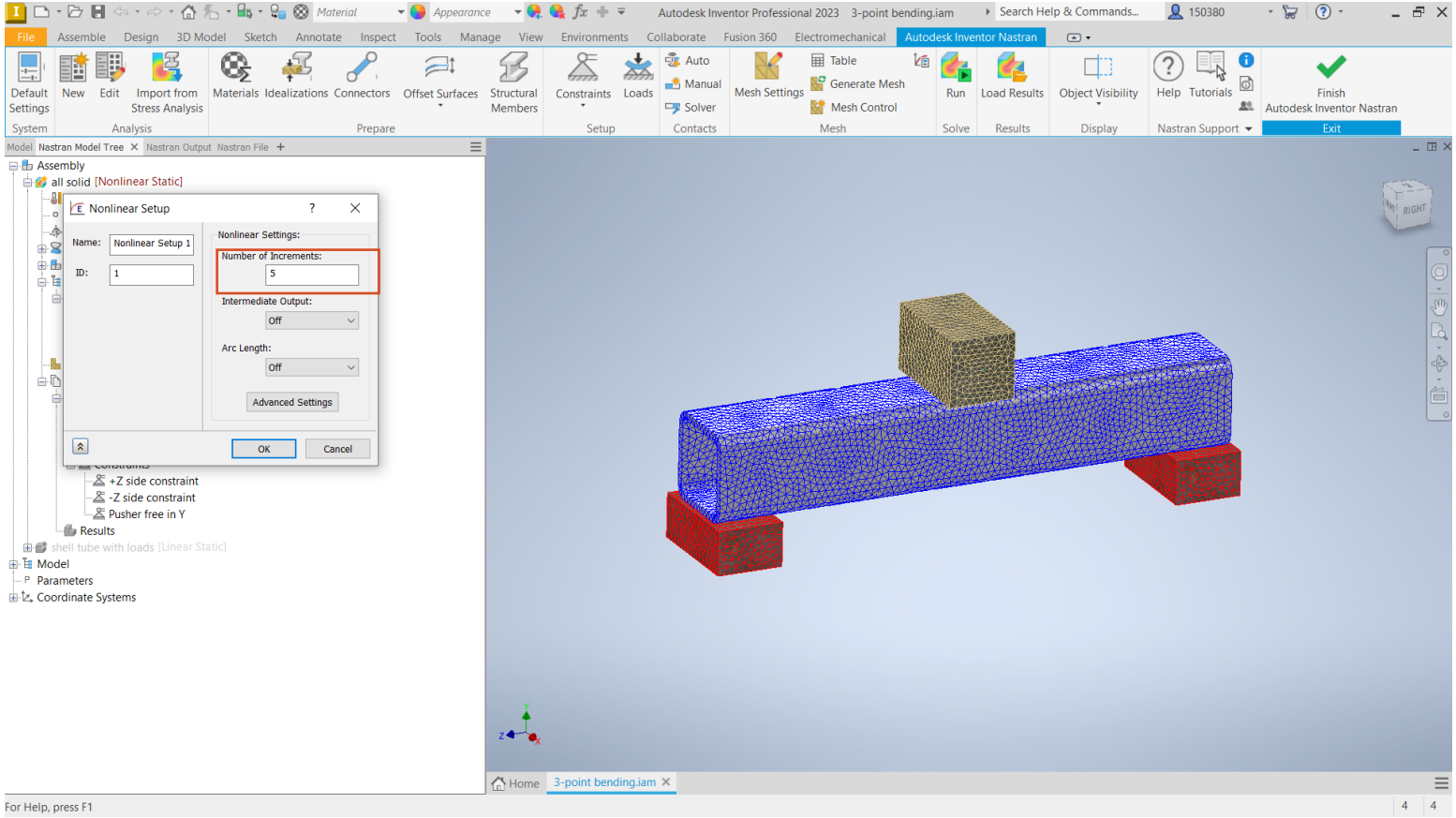
# Running Nonlinear Statics



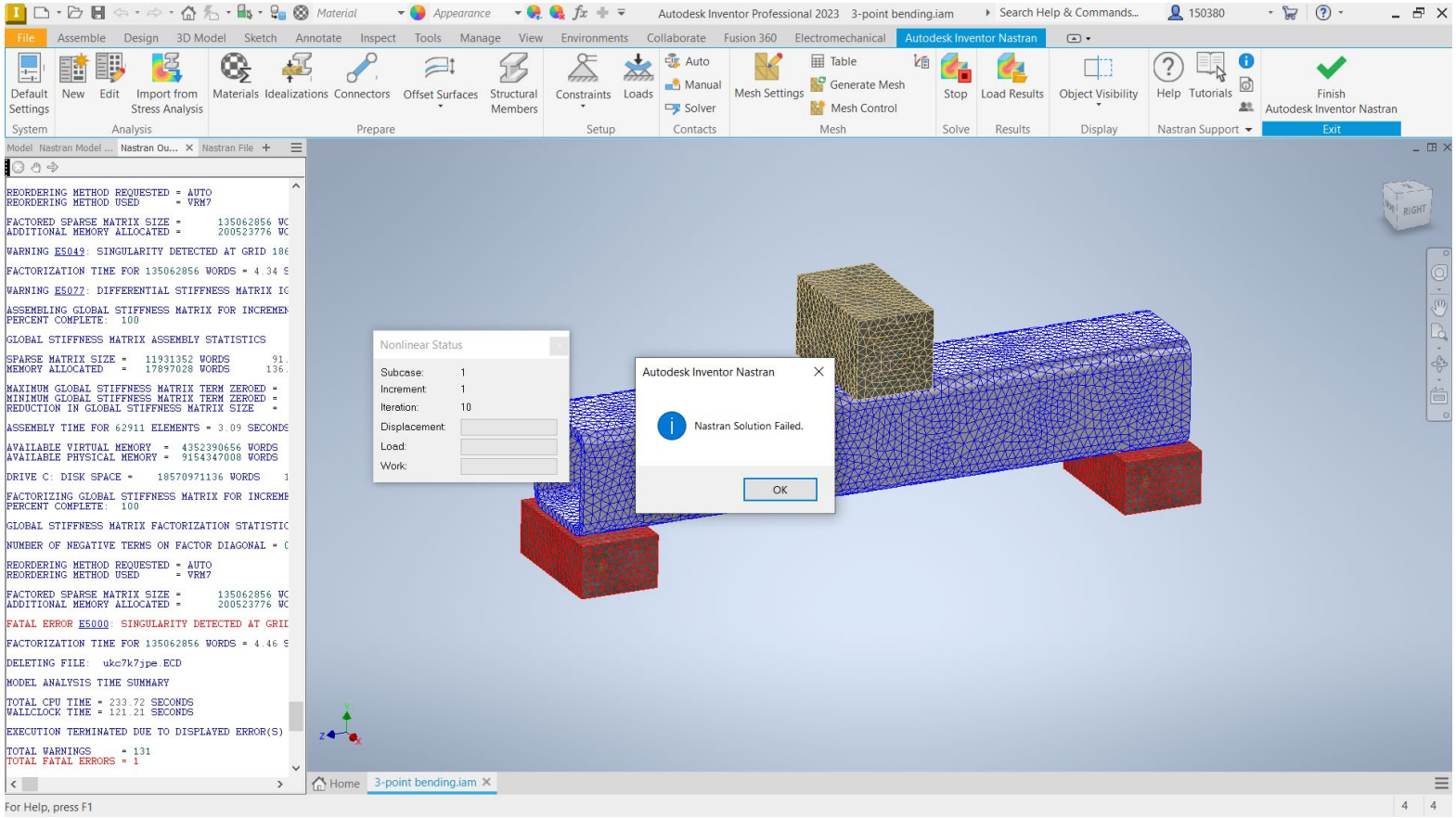
# Setting Up Nonlinear Statics



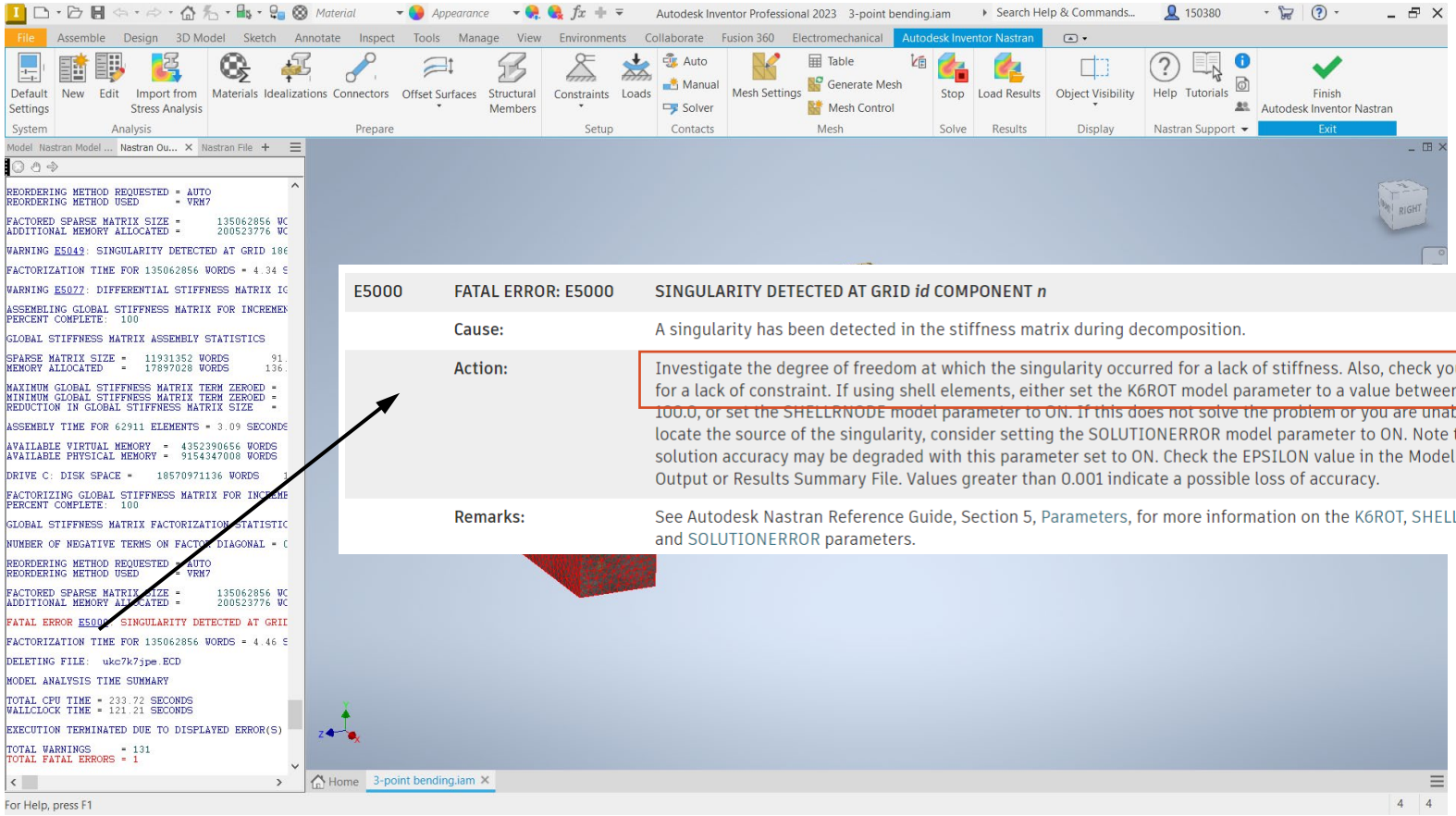
# Setting Up Nonlinear Statics



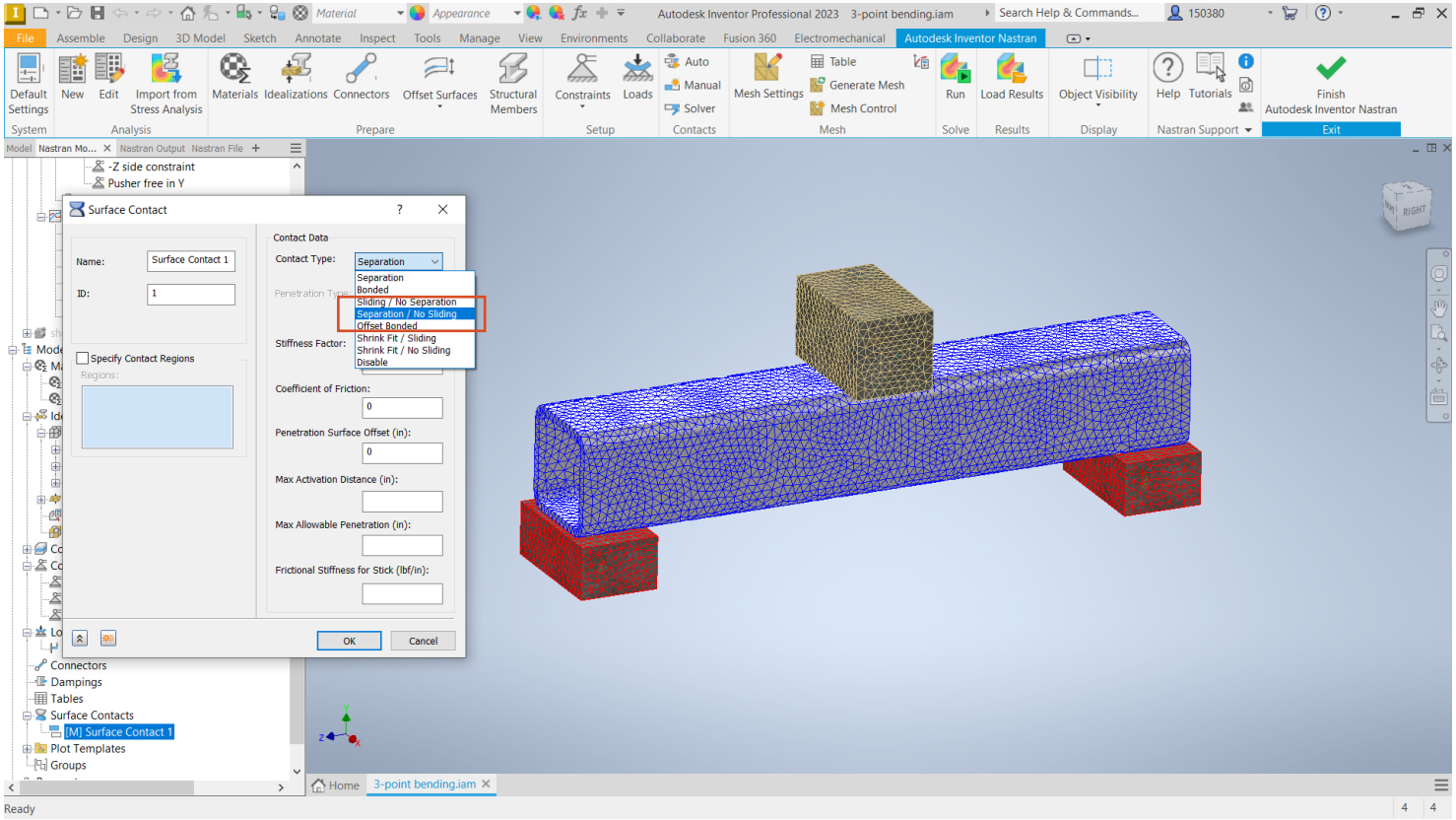
# Nonlinear Statics Fails?



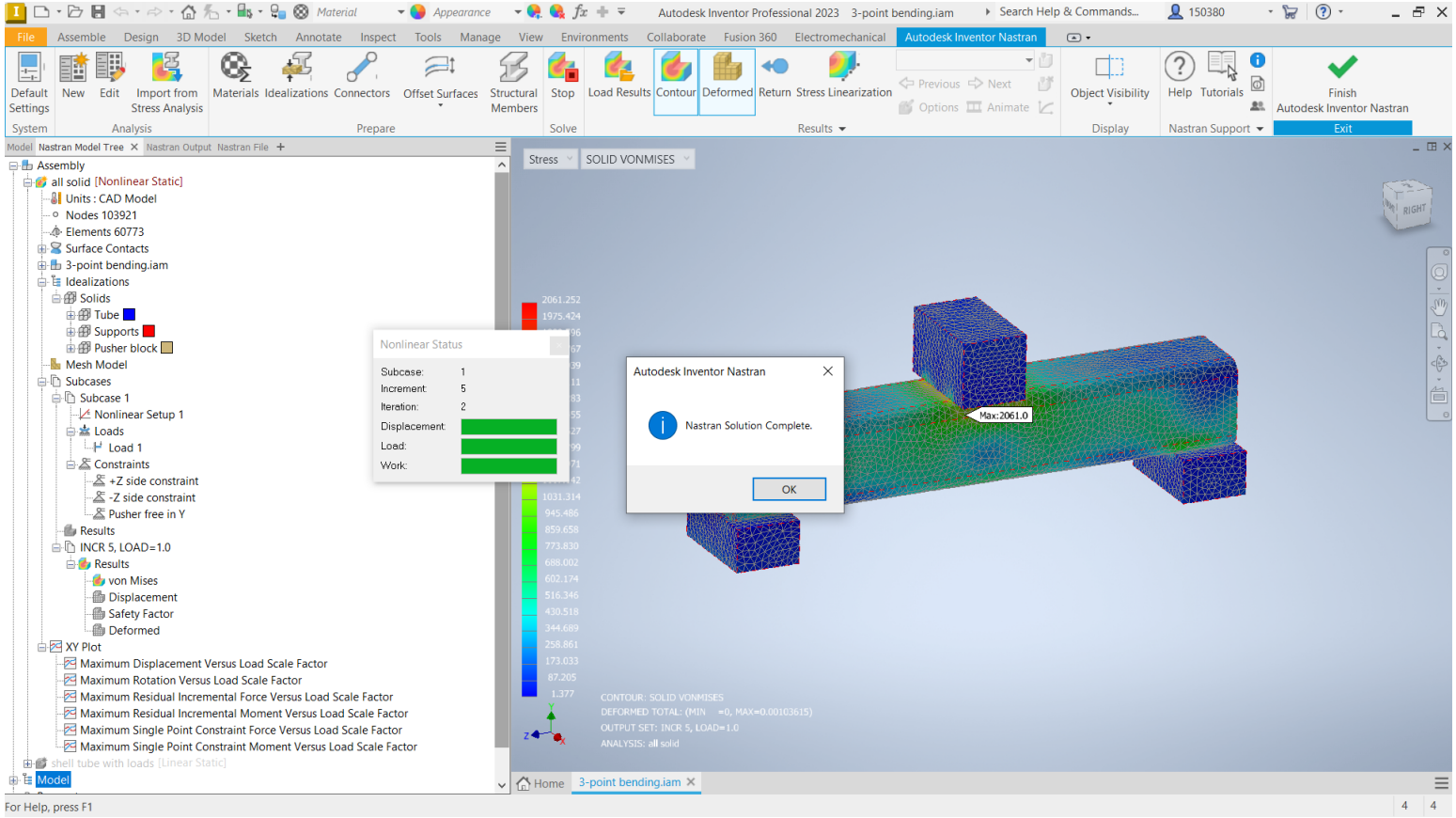
# Nonlinear Statics Fails?



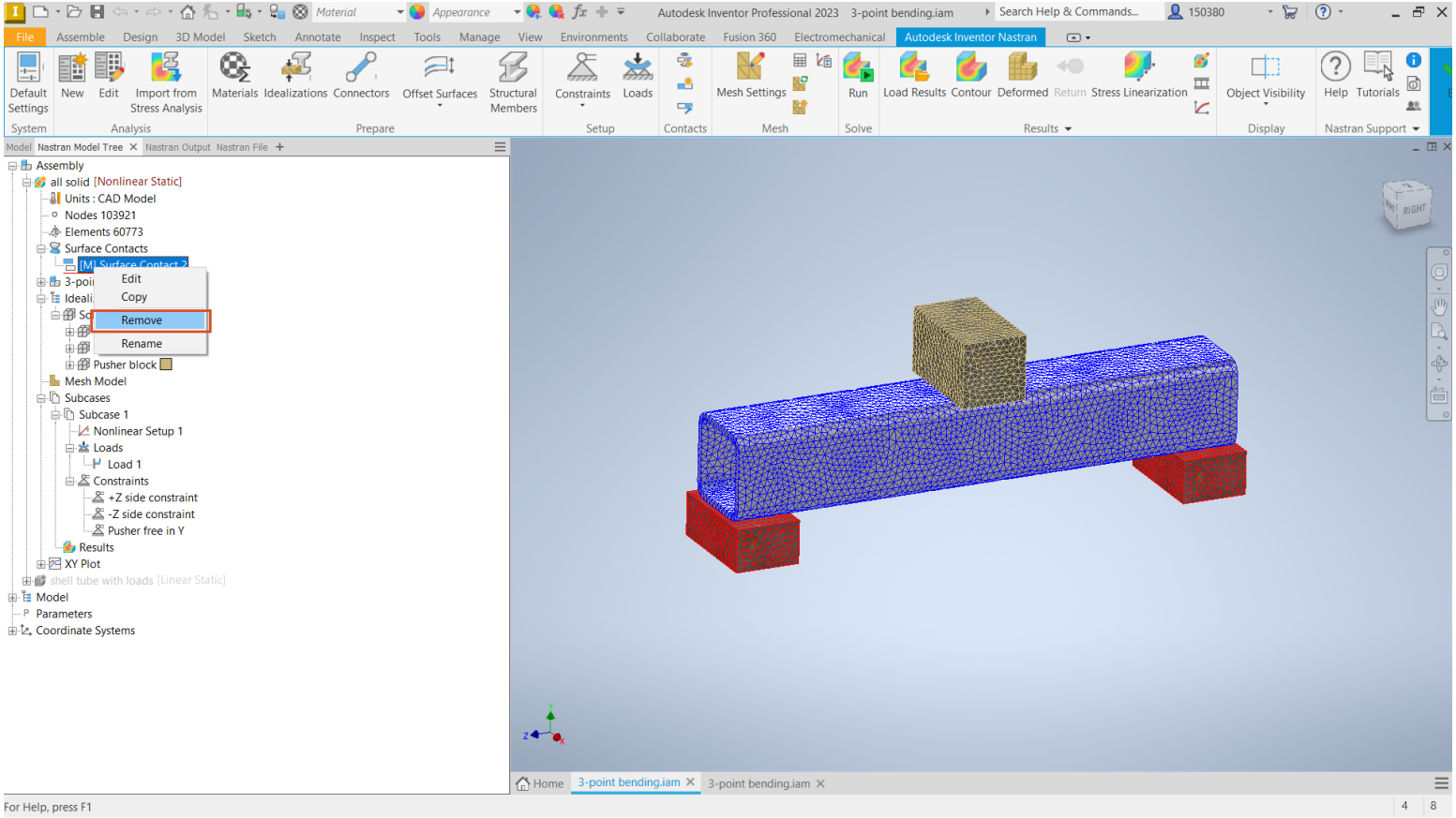
# Setting Up Rough Contact to Prevent Sliding



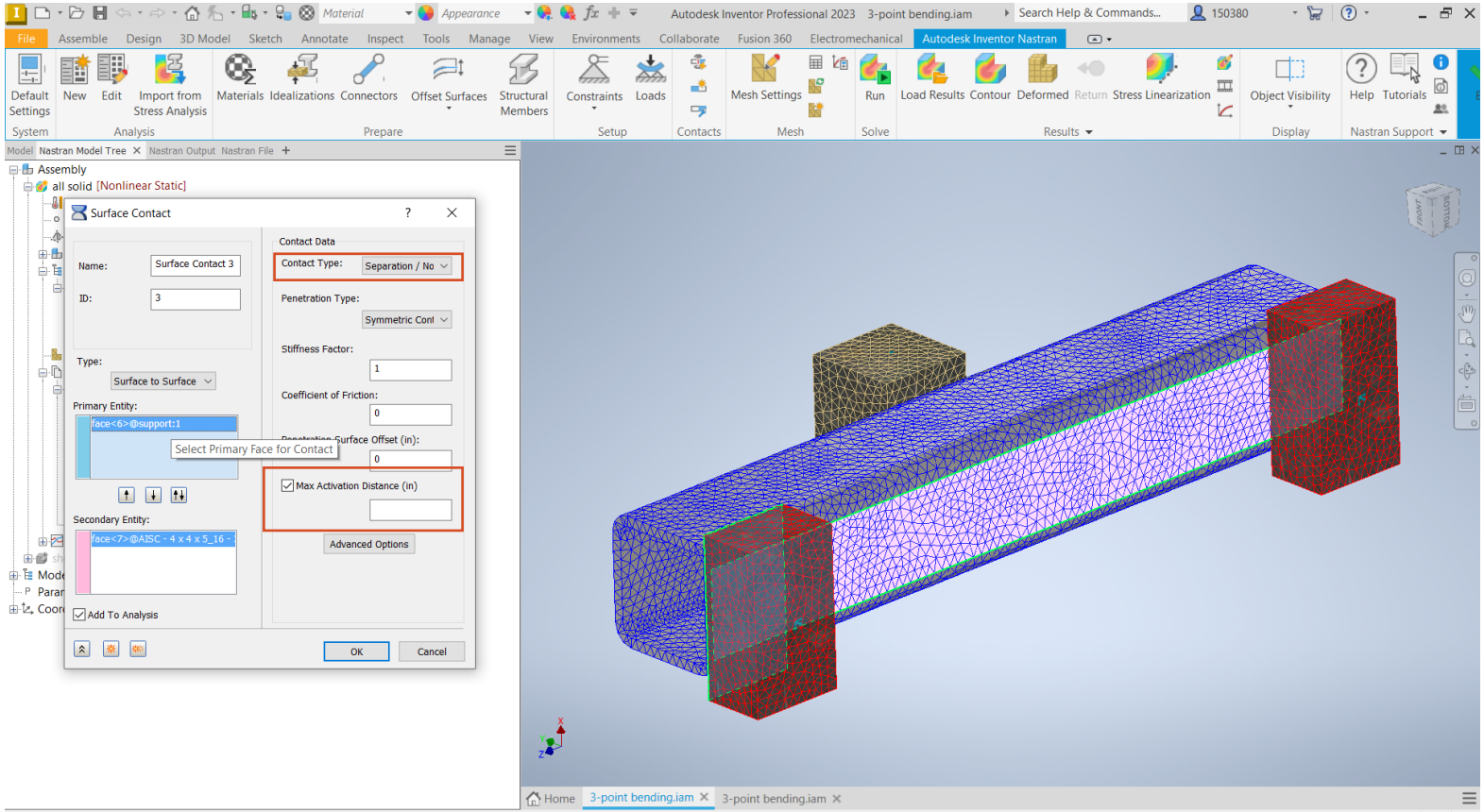
# Nonlinear Statics with Rough Contact



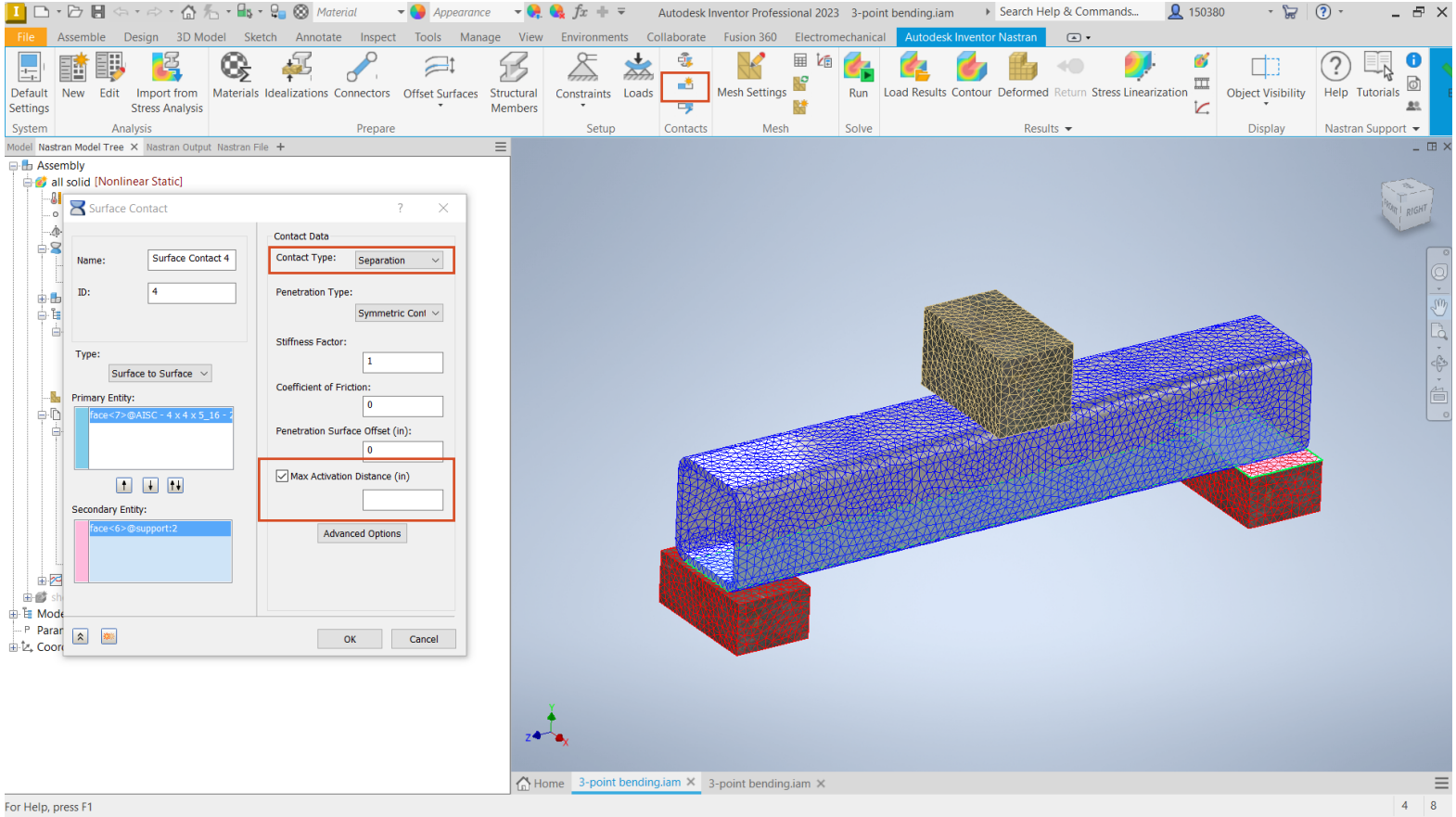
# Setting Up Manual Contact – Delete Solver Contact



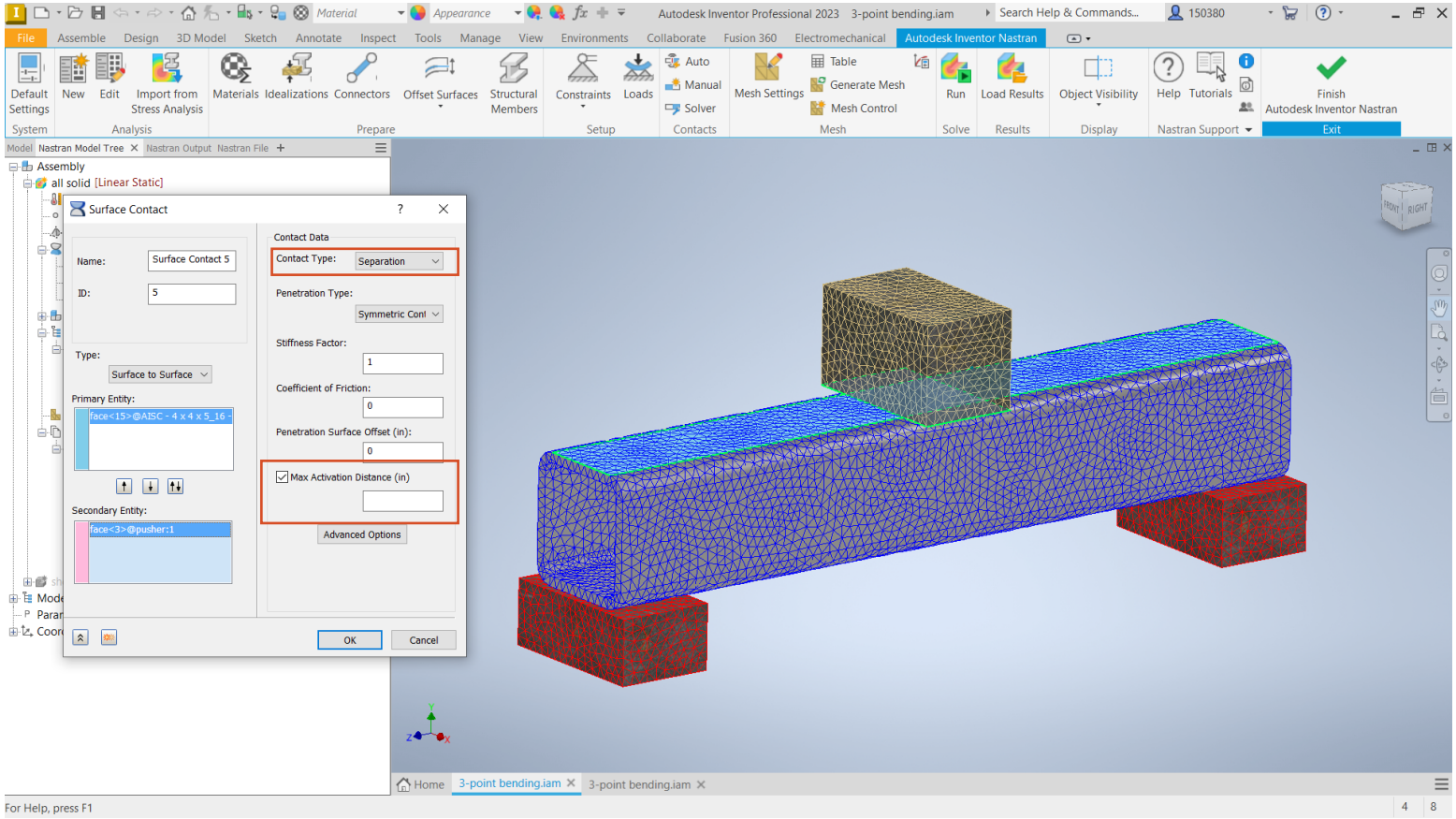
# Setting Up +Z Support Manual Contact



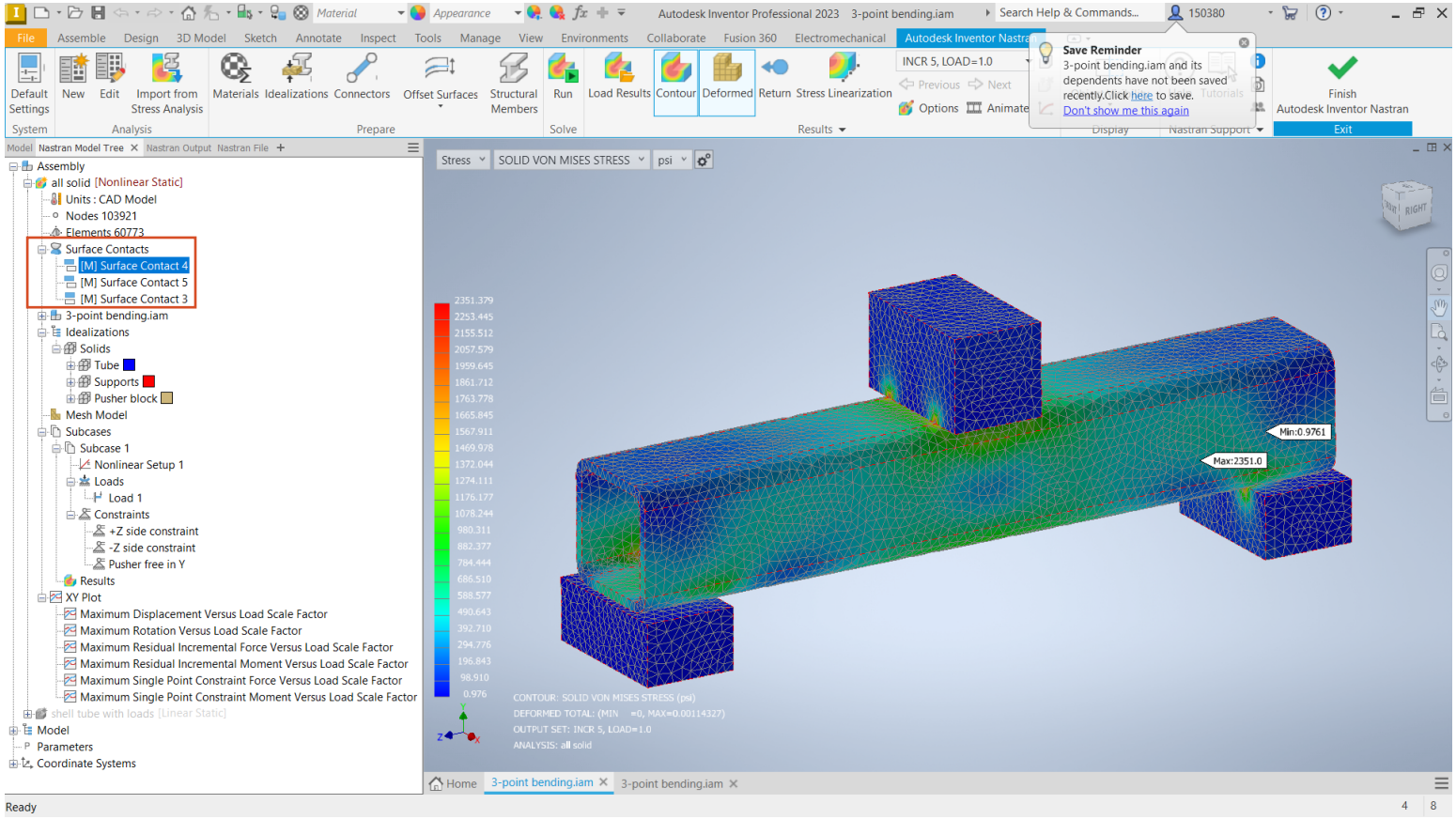
# Setting Up -Z Support Manual Contact



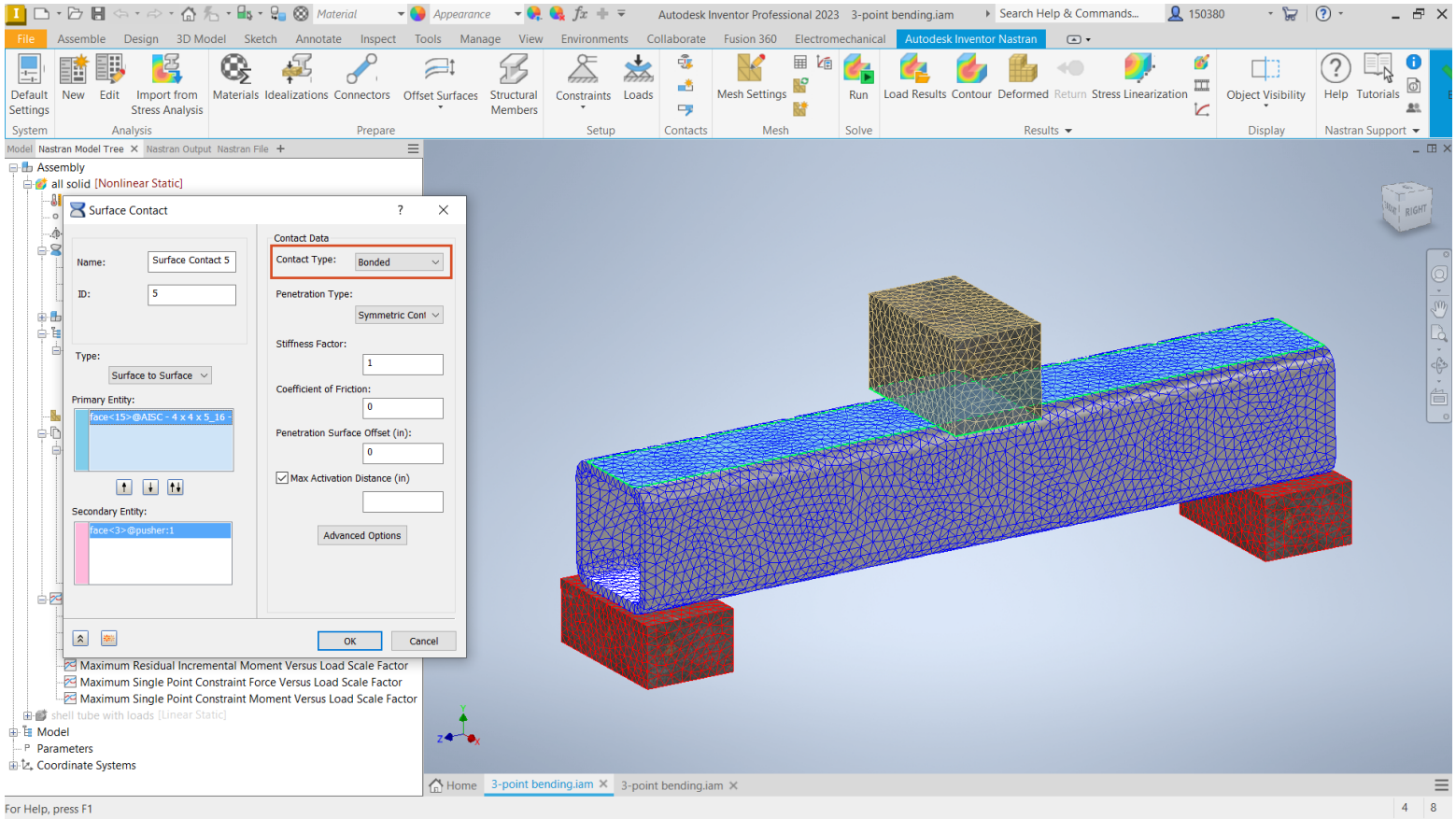
# Setting Up Pusher Manual Contact



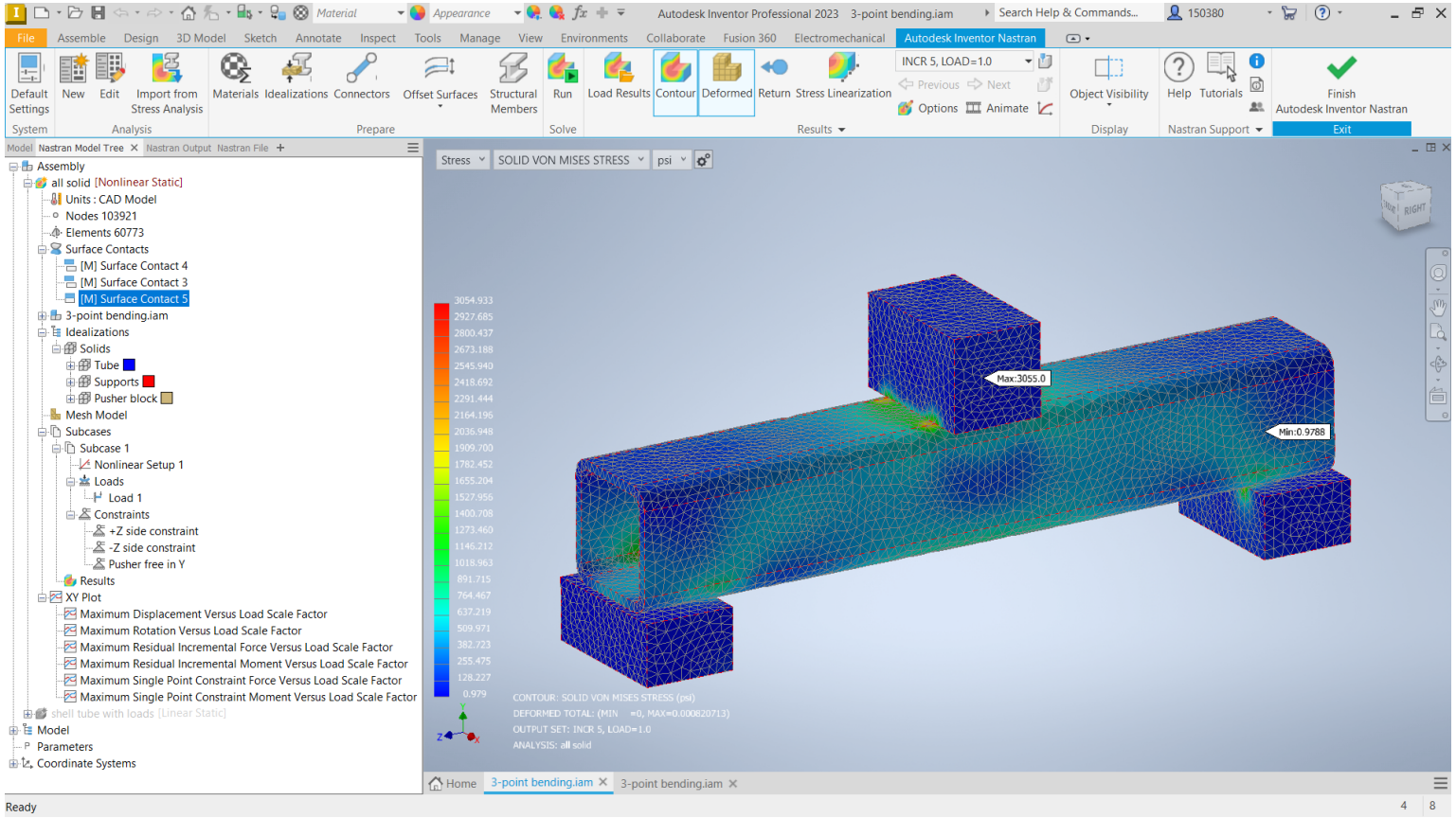
# Nonlinear Solution with No Sliding on +Z Support



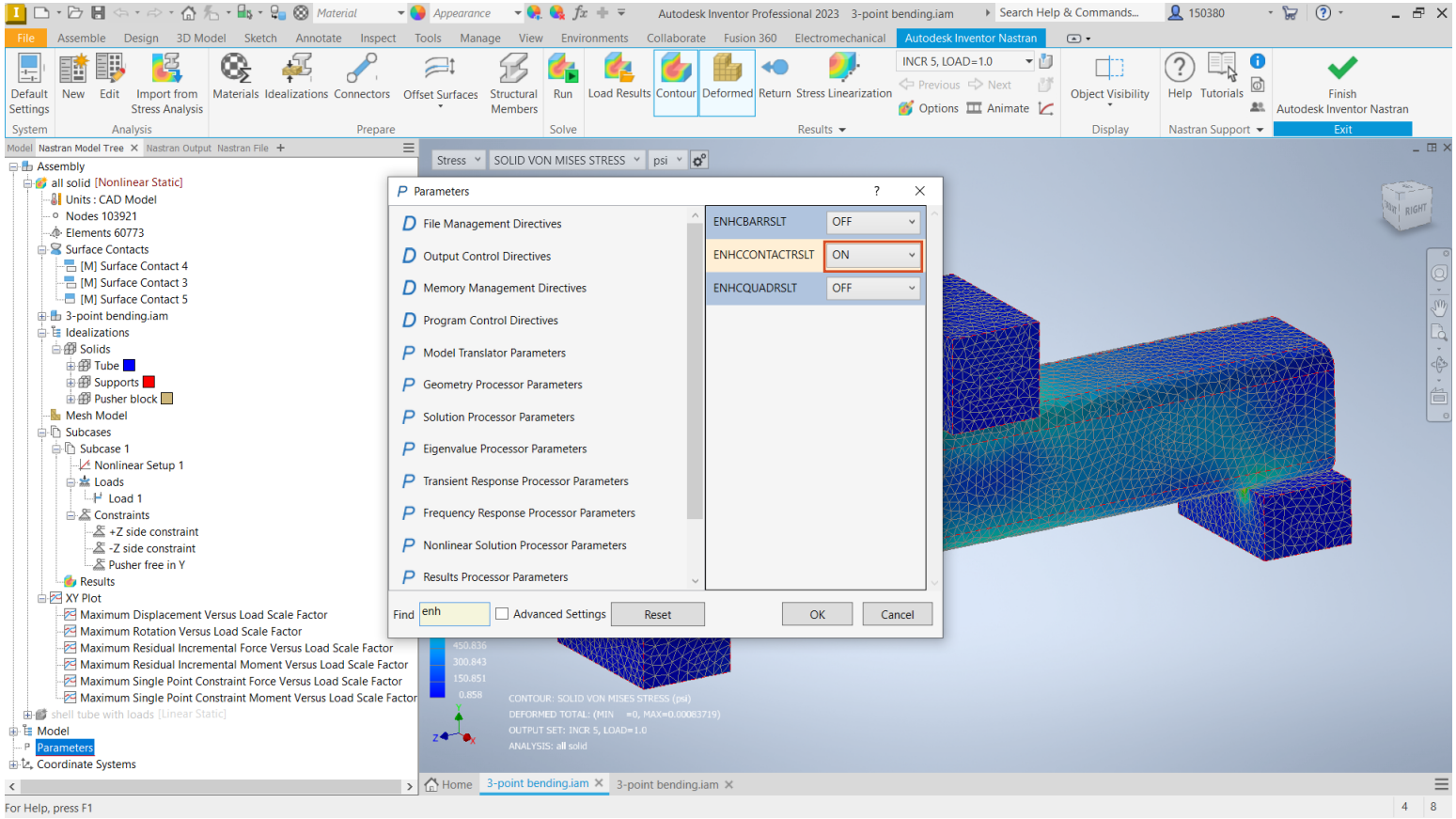
# Welding Pusher to Tube



# Nonlinear Solution with Welded Pusher

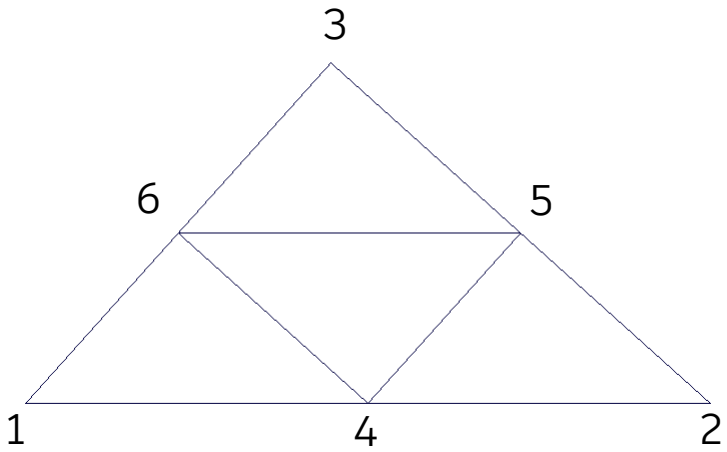


# Enhanced Contact

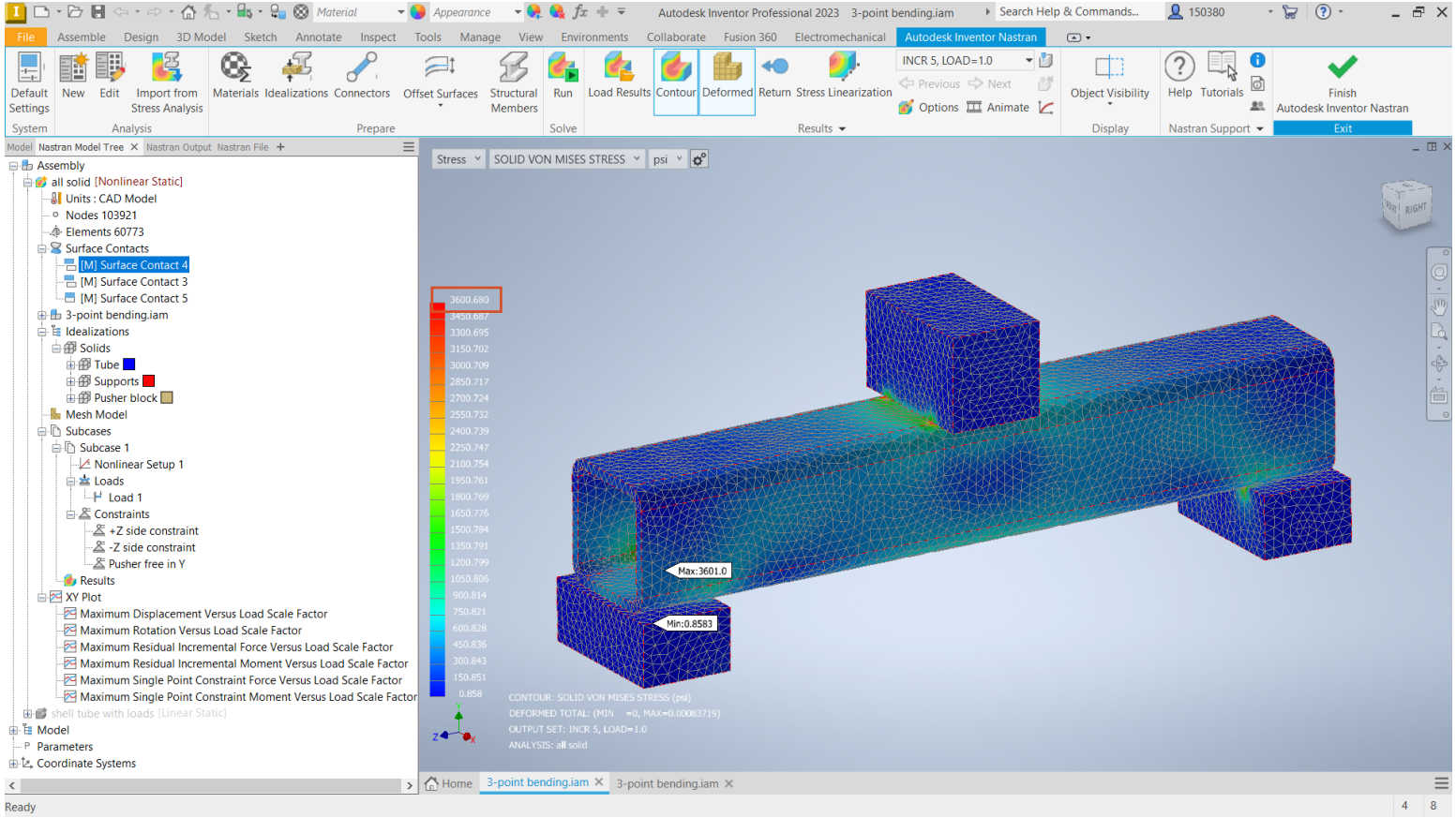


# Enhanced Contact

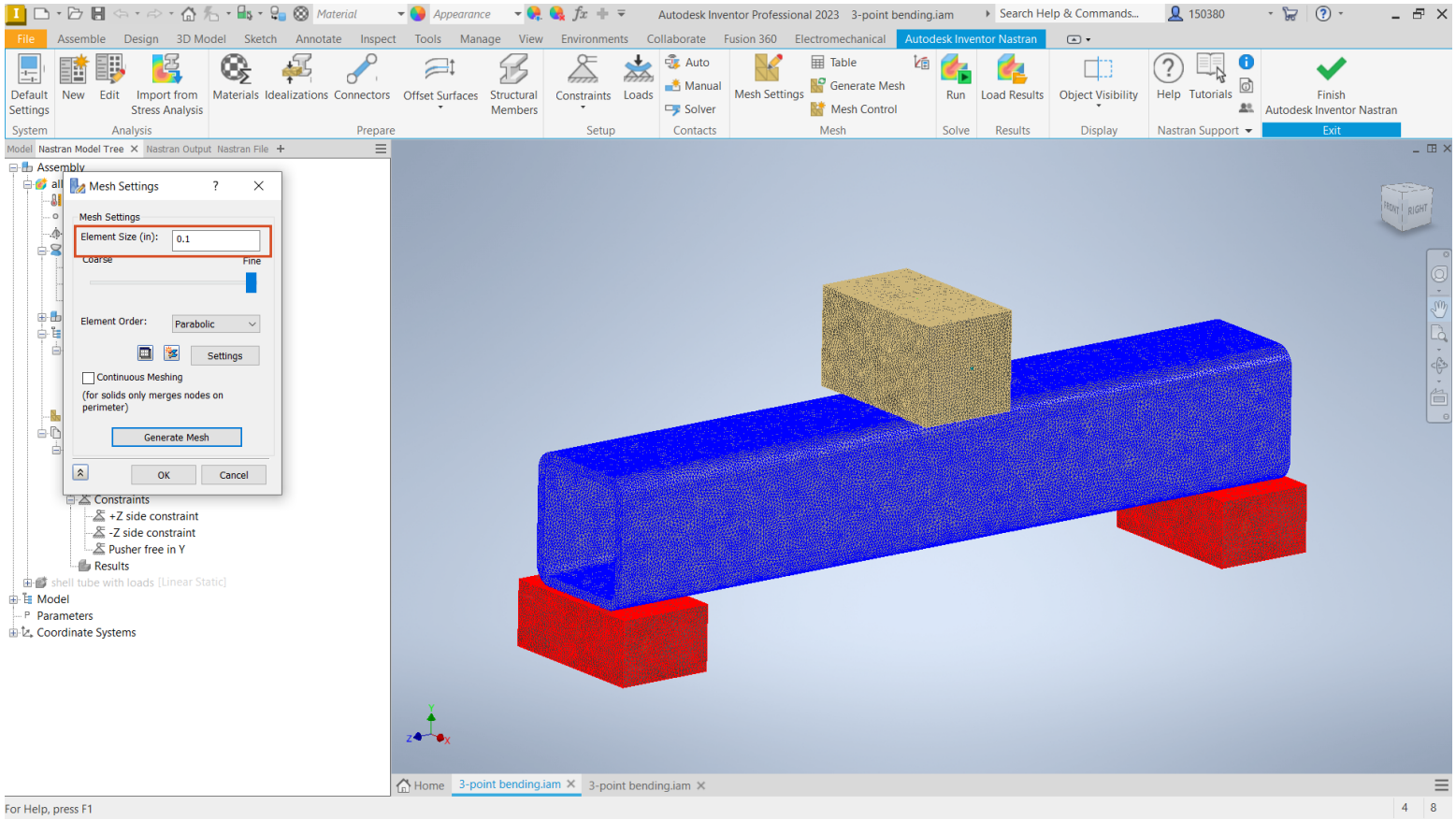
- Enhanced contact is abled with the ENHCCONTACTRSLT parameter
- Enabling this will improve accuracy for models with thin parts and only one or two TET elements through the thickness
- The big disadvantage is that performance will be much slower for larger models
- We are currently working on an improved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



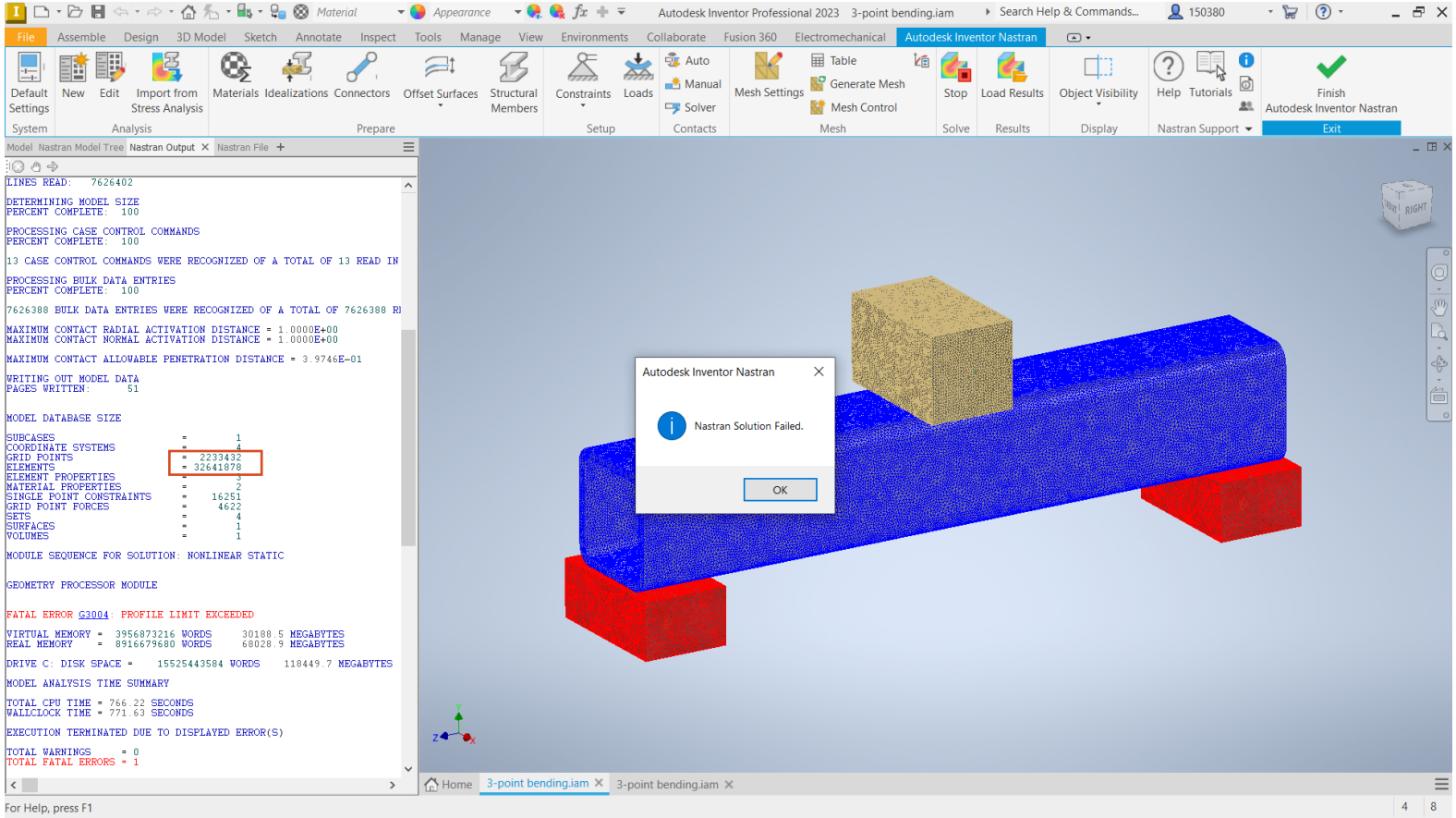
# Enhanced Contact Results in a More Accurate Stress



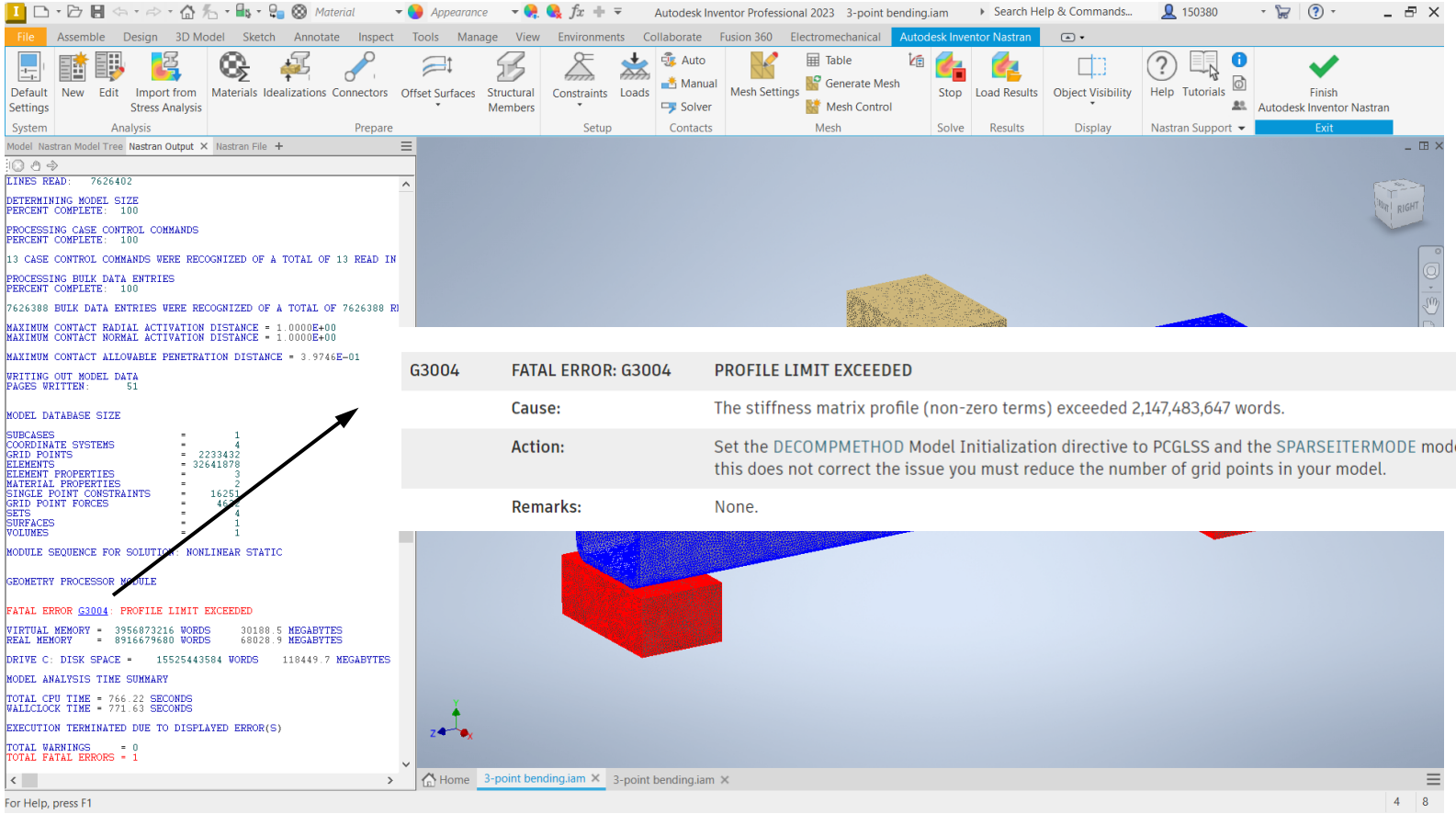
# Using a Finer Mesh for Improved Results



# Solution Fails Due to Model Size

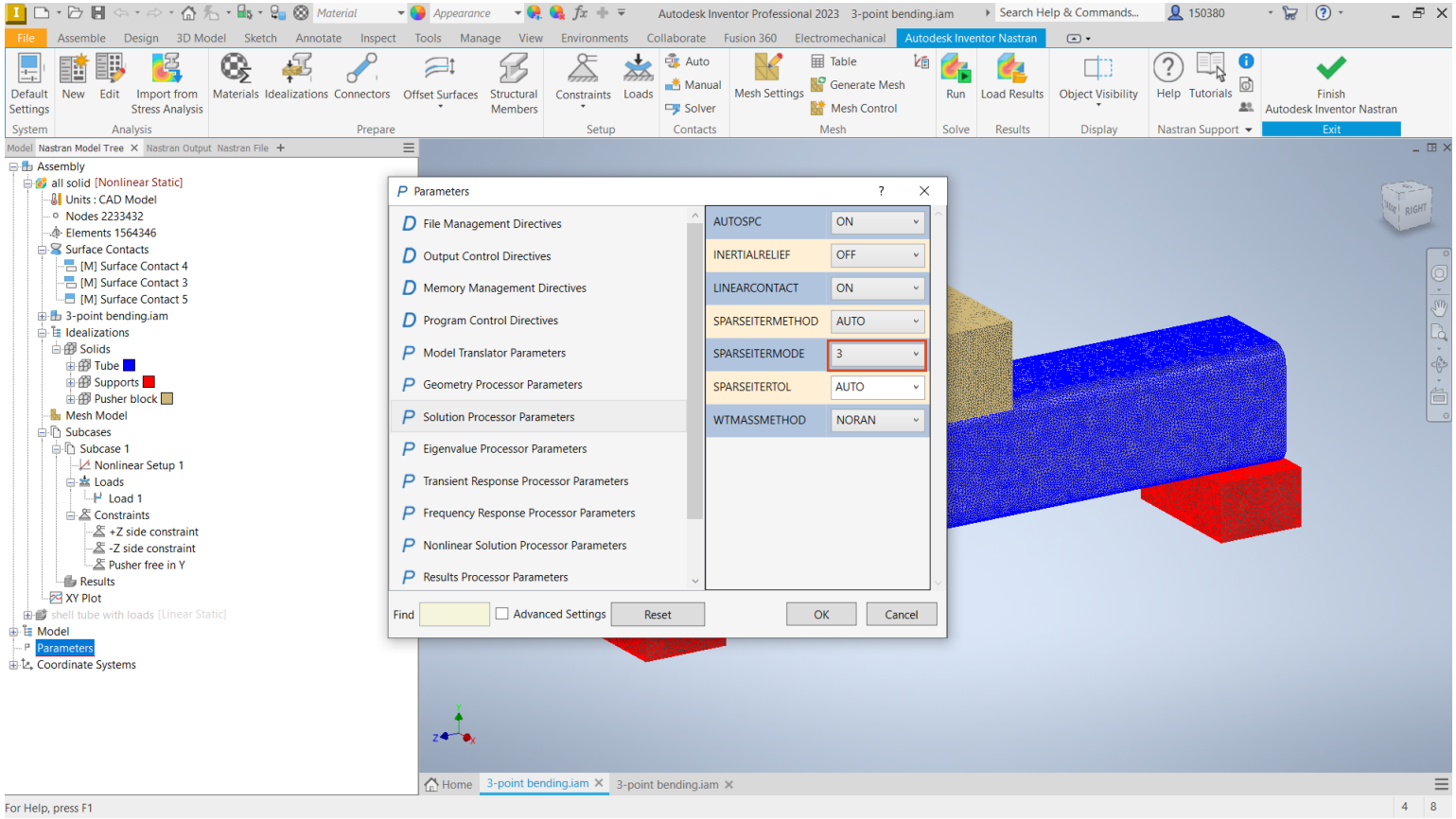


# Enhanced Contact/Max Activation Distance/Fine Mesh

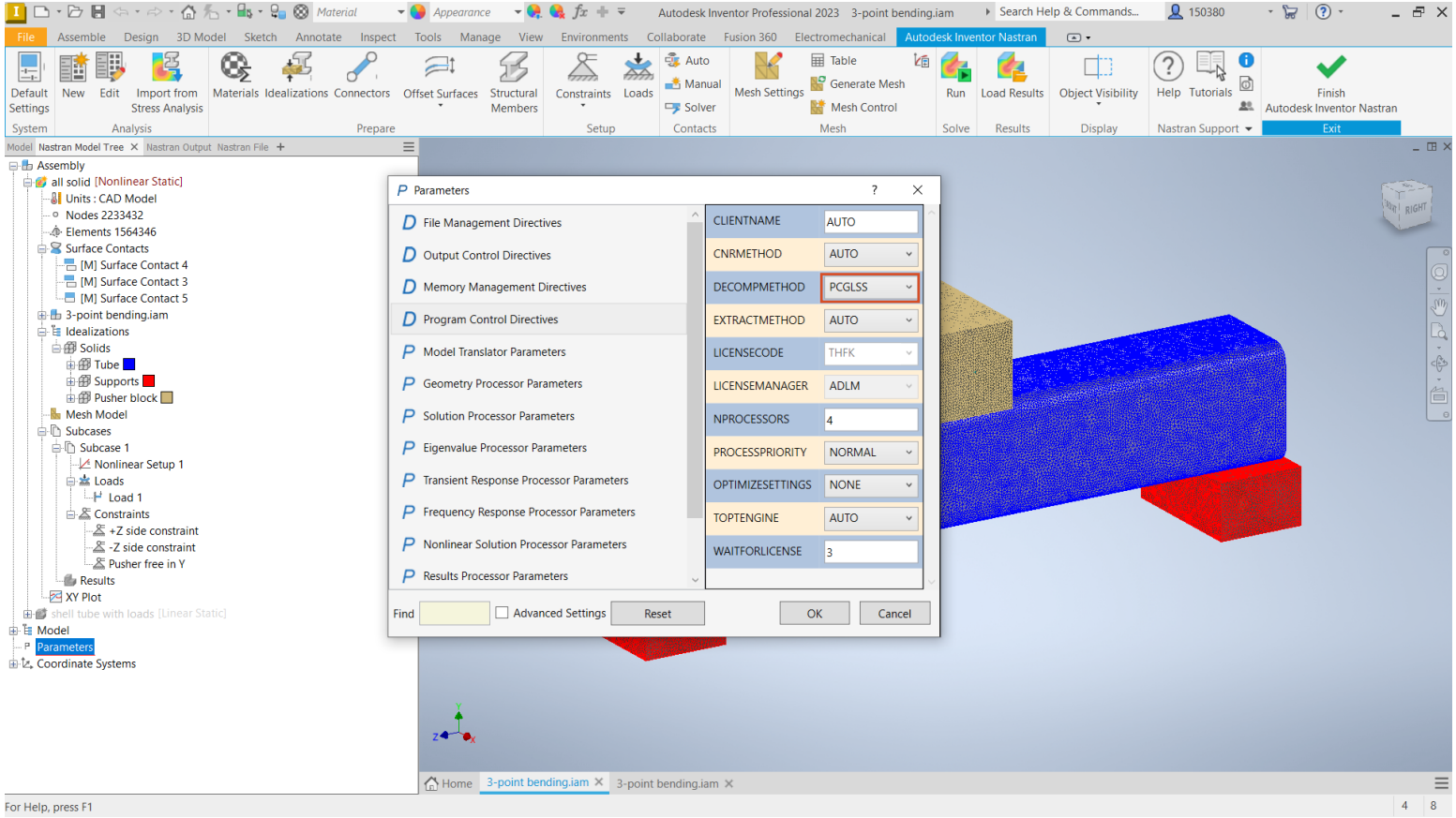



| | | |
|---|---|---|
| G3004 | FATAL ERROR: G3004 | PROFILE LIMIT EXCEEDED |
| | Cause: | The stiffness matrix profile (non-zero terms) exceeded 2,147,483,647 words. |
| | Action: | Set the DECOMPMETHOD Model Initialization directive to PCGLSS and the SPARSEITERMODE model parameter to 3. If this does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

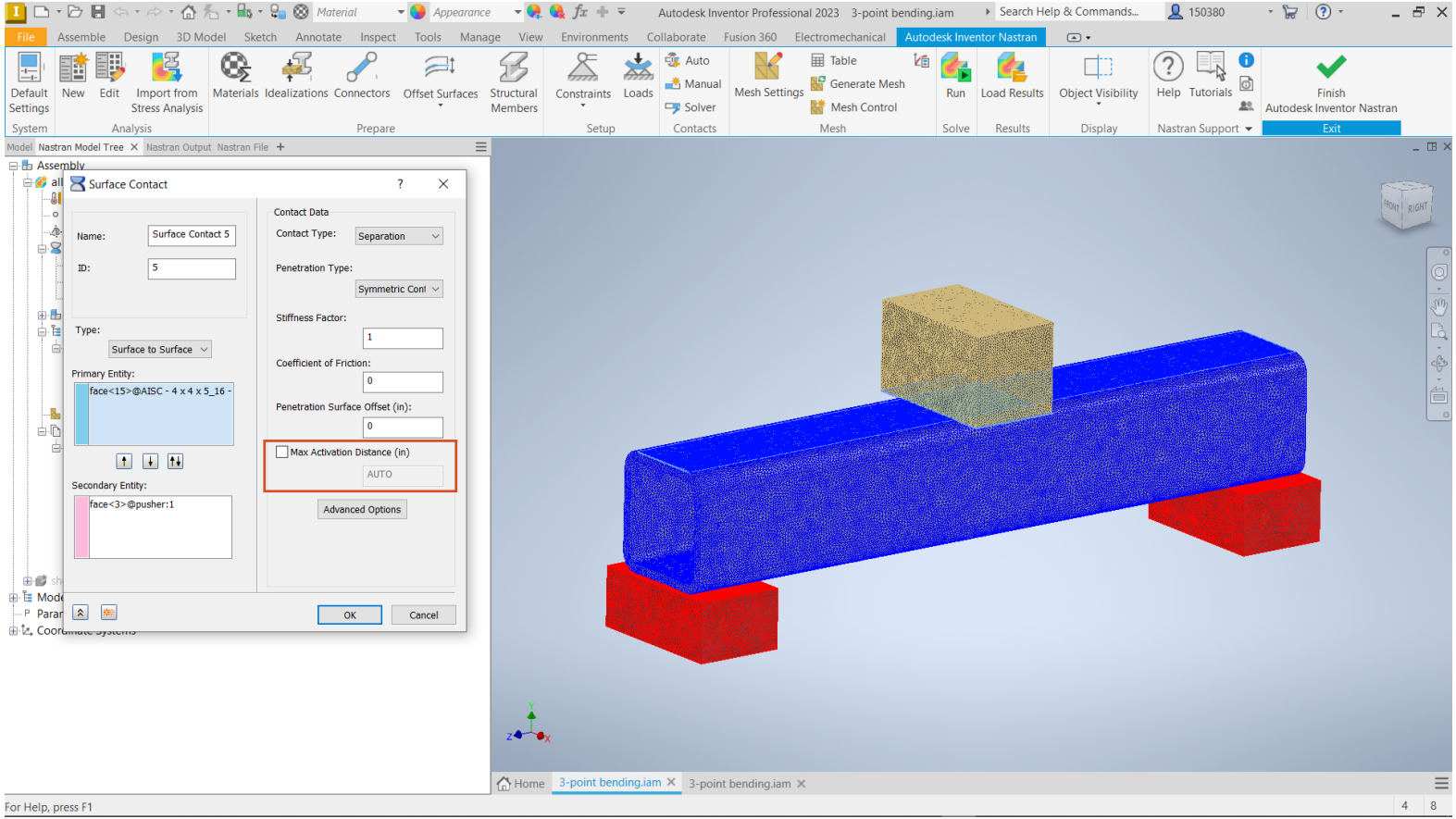
# Fixing a G3004 Error Message



# Fixing a G3004 Error Message



# Reduce the Maximum Activation Distance

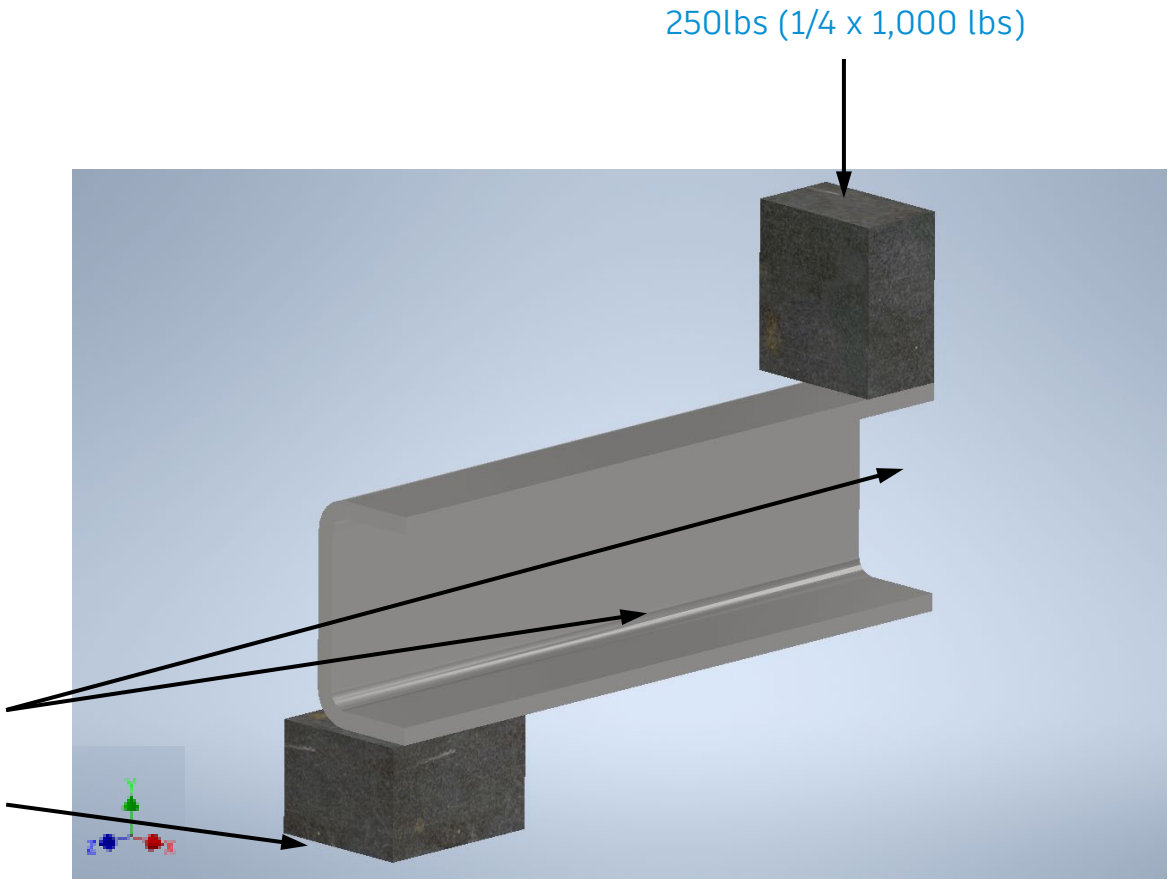


# Another Option…¼ Symmetric Model

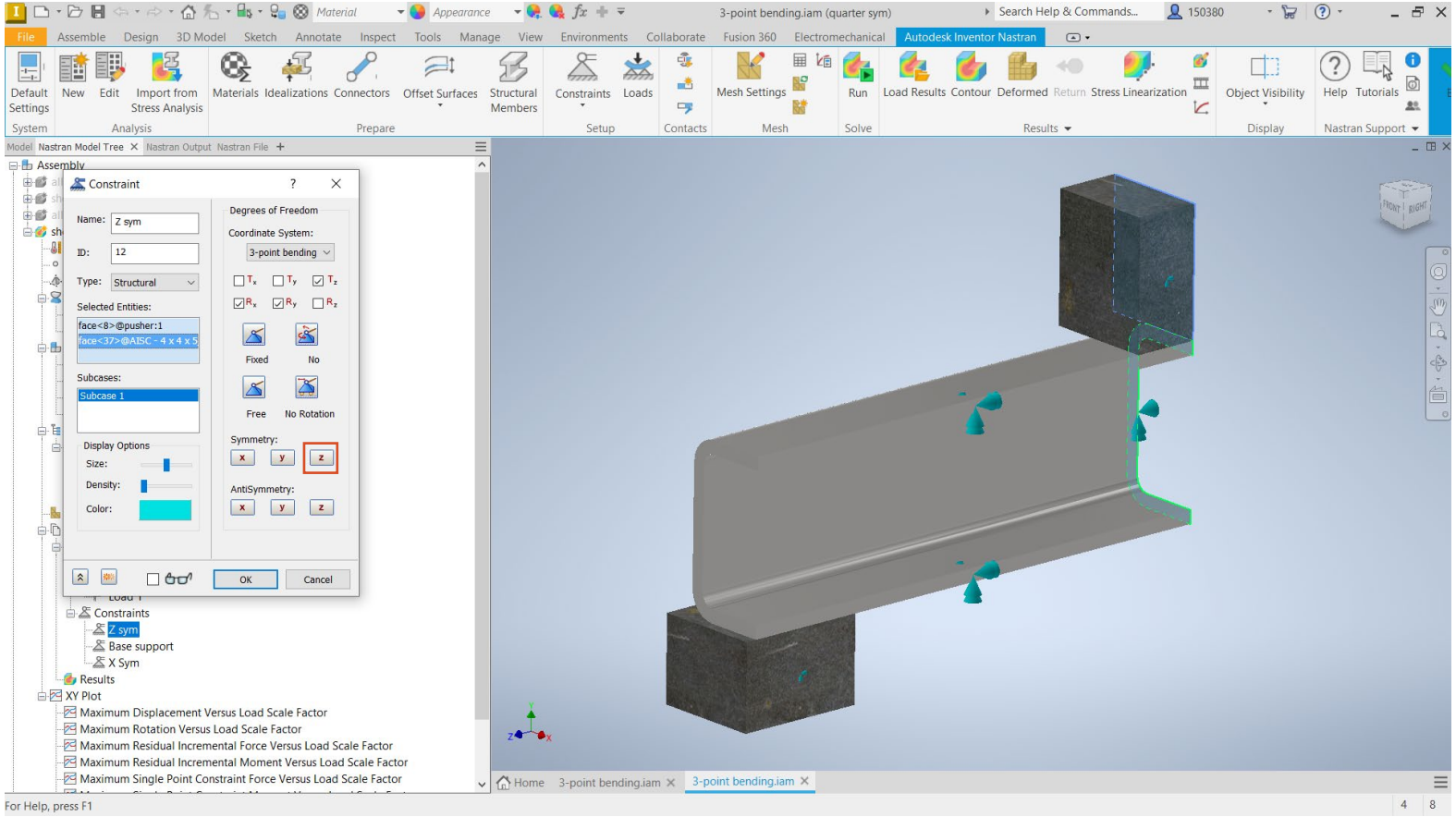
- If the loading and boundary conditions are symmetric this option makes the most sense
- The model will be cut down by ¼ improving performance and allowing a much finer mesh
- The boundary conditions will allow all surfaces to slide with no friction required
- Topology optimization can easily be performed
- Other more complicated nonlinear analyses like material nonlinear can be performed much faster

# 1/4 Symmetric Steel Tube Model

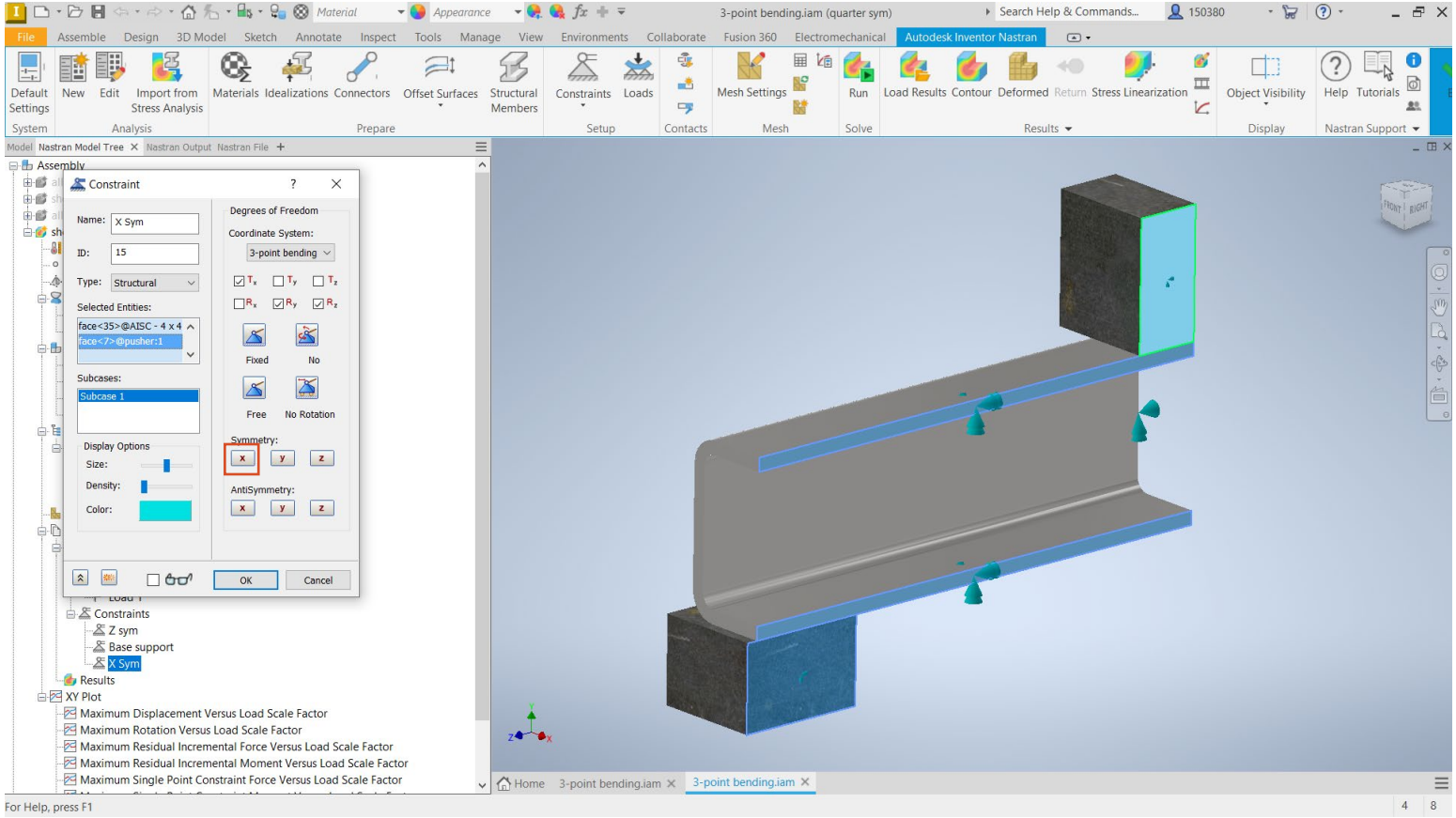
- Tube: AISC Steel:
  - 4 x 4 x 24 inches
  - 5/16 inches thick
  - Yield point 30ksi
- Blocks: Steel
- Constrained at the base of pads and side of pusher block



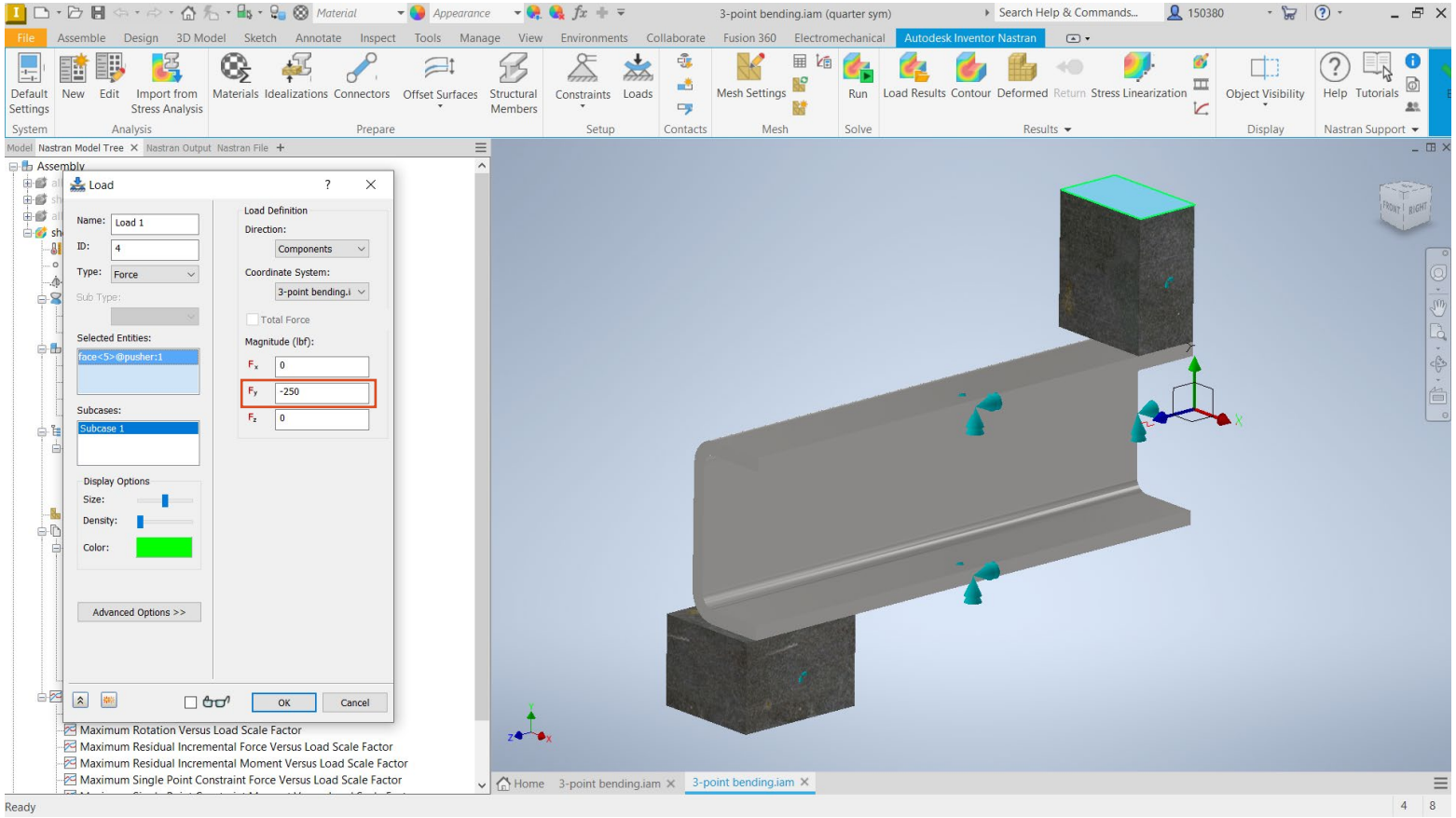
# Z-Symmetry Boundary Condition



# X-Symmetry Boundary Condition

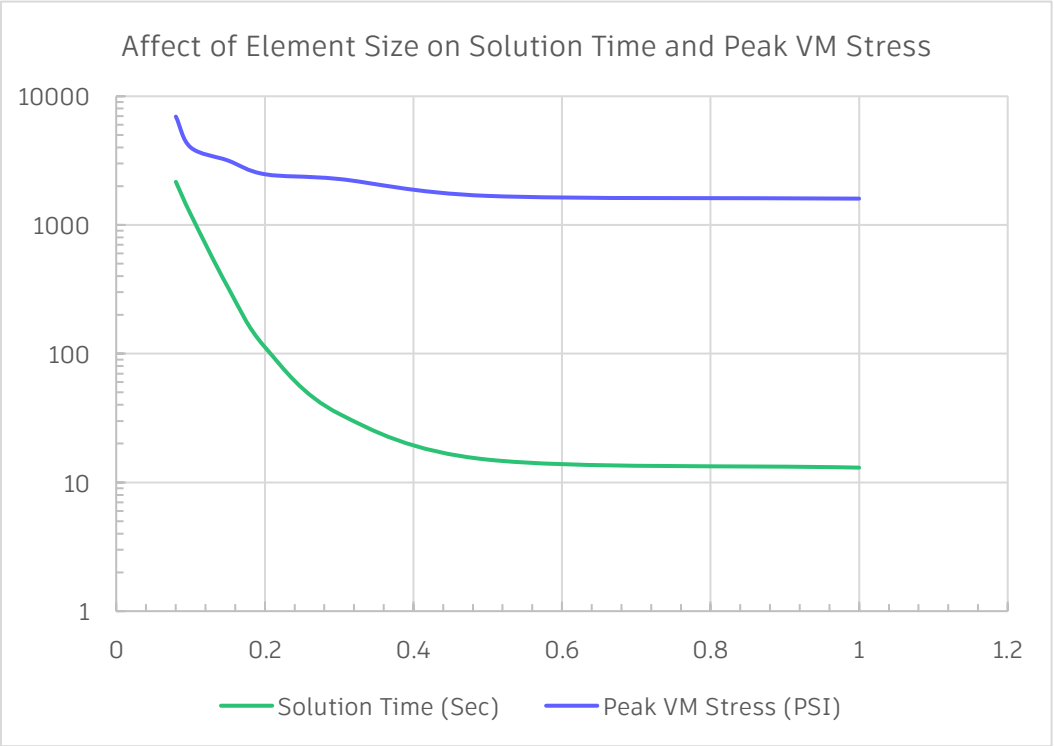


# 1/4-Symmetry Load Definition

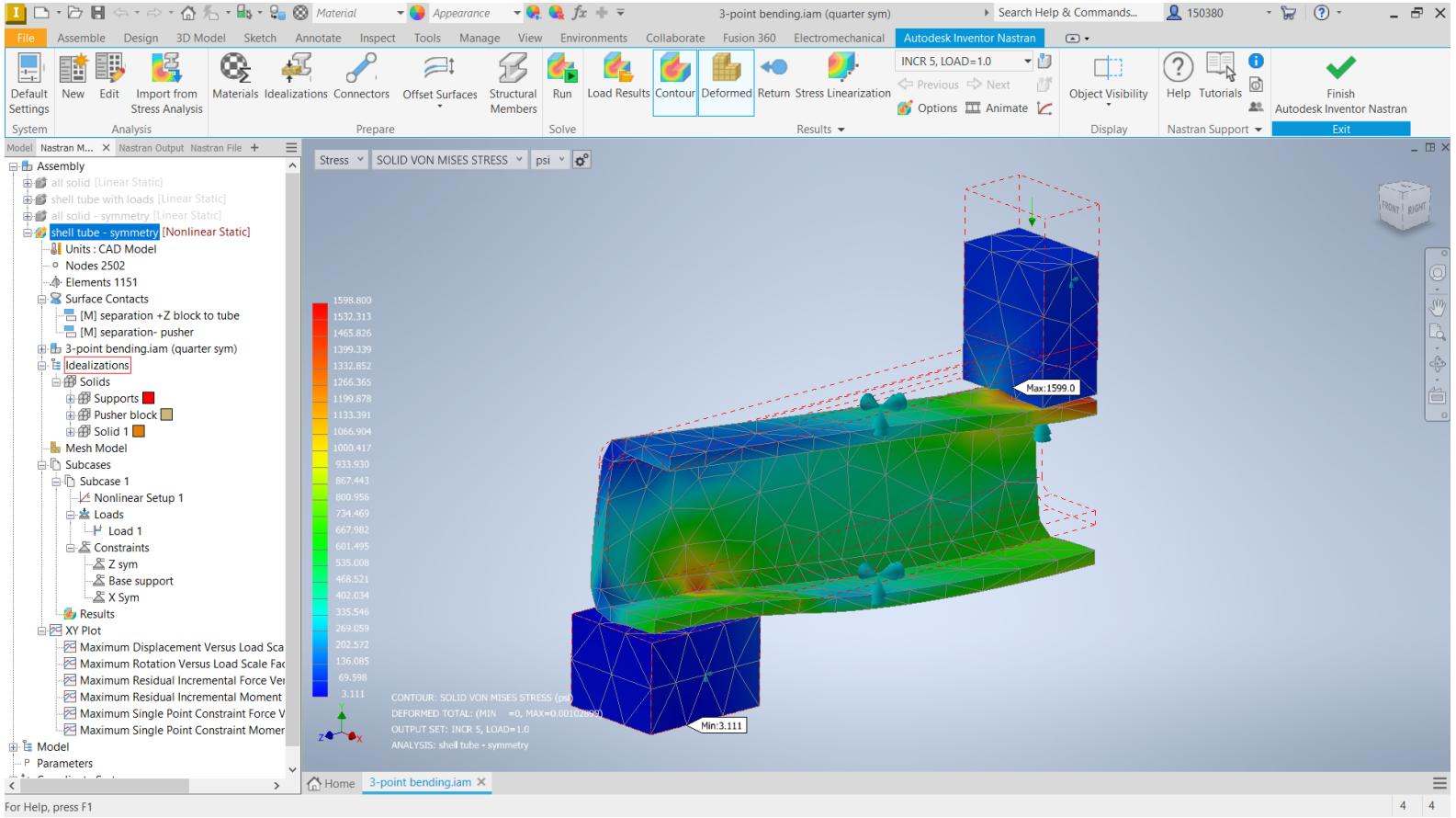


# Solid Mesh Convergence Study

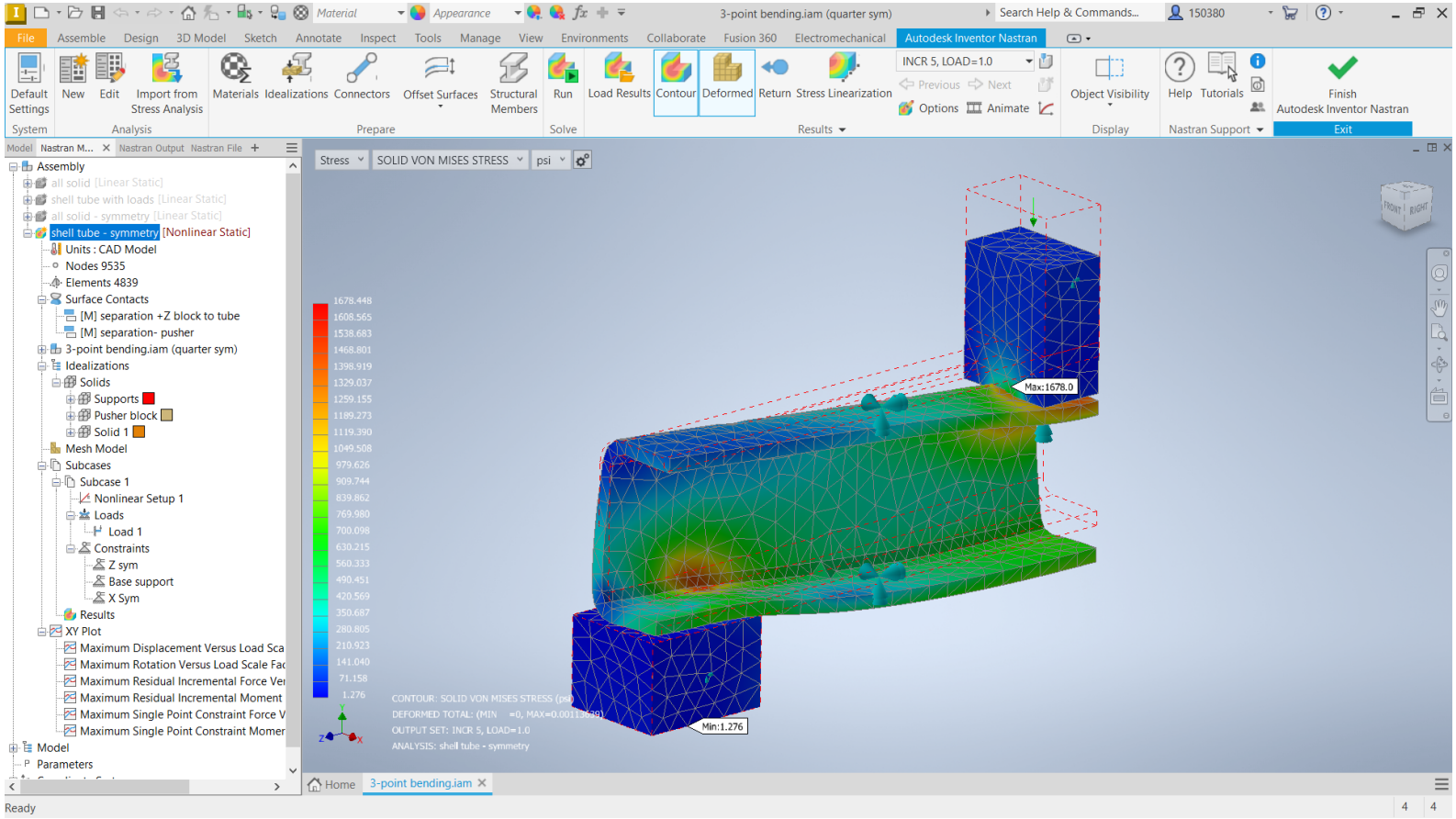
- Enhanced contact is OFF
- Mesh element size is varied from 1.0 to 0.08
- The peak stress will continue to increase due to a singularity at the edge of the support block
- Solution time goes up exponentially with model size or with the inverse of element size



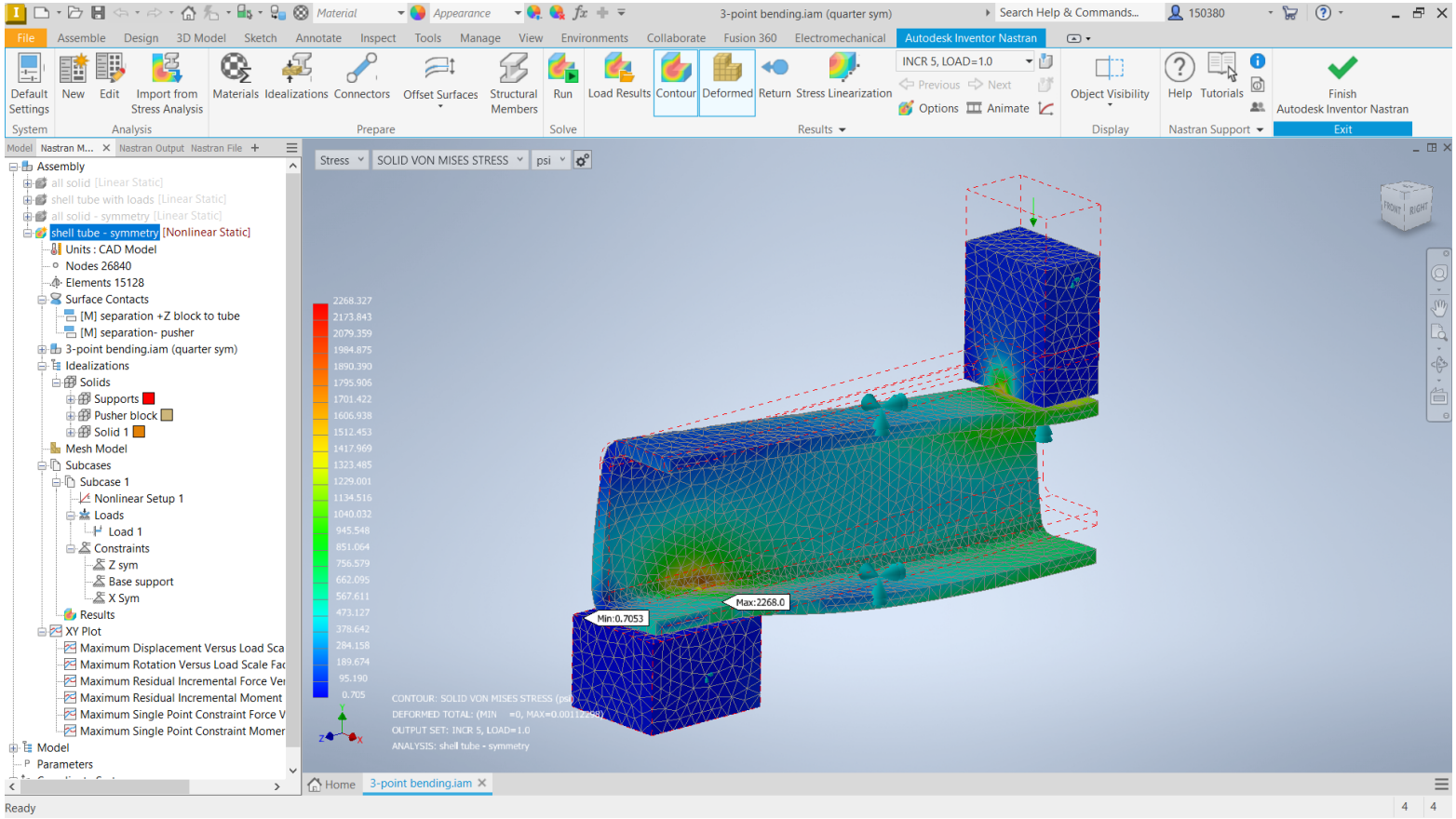
# Mesh Convergence – Element Size = 1.0in – 13Sec



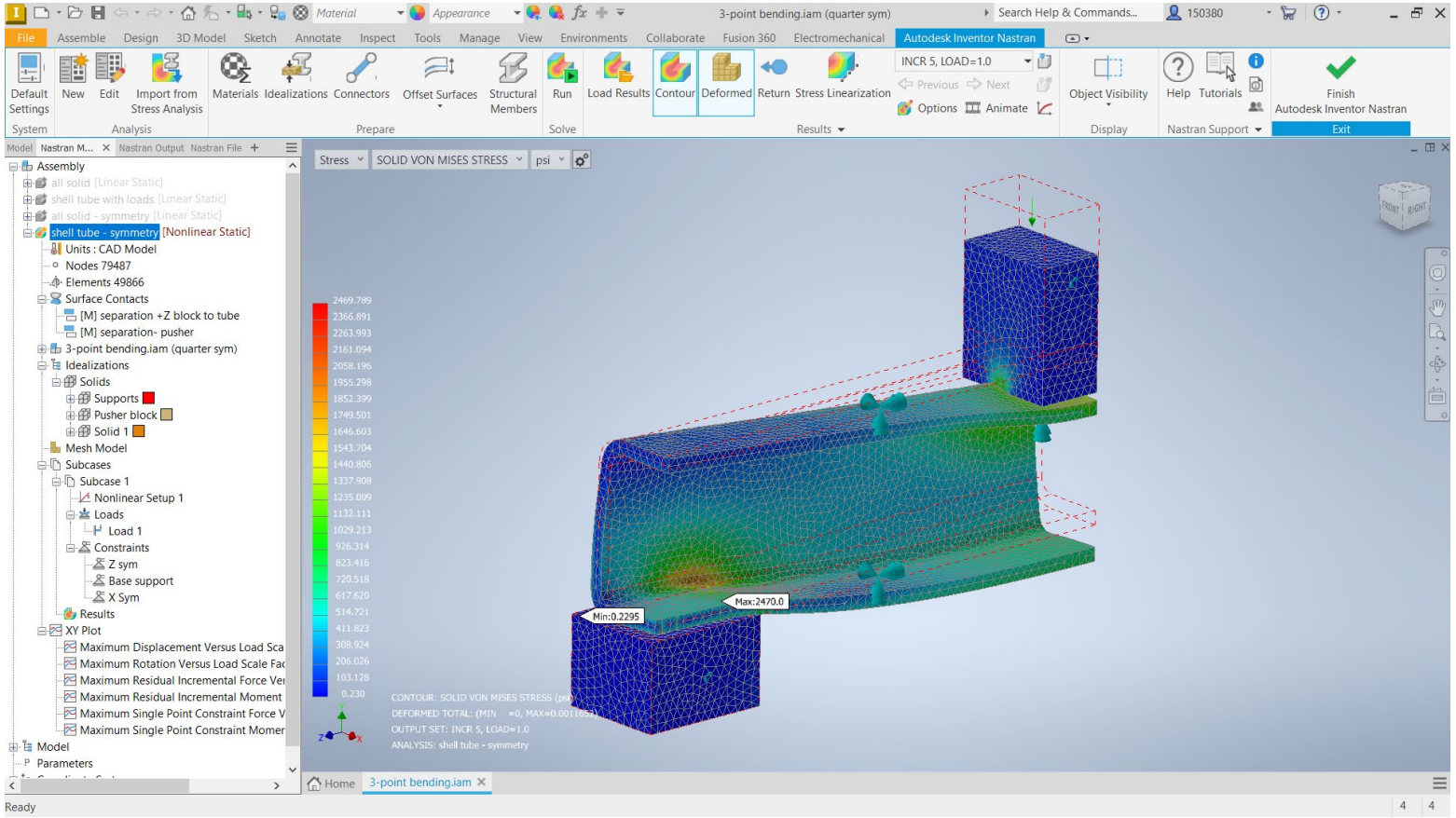
# Mesh Convergence – Element Size = 0.5in – 15Sec



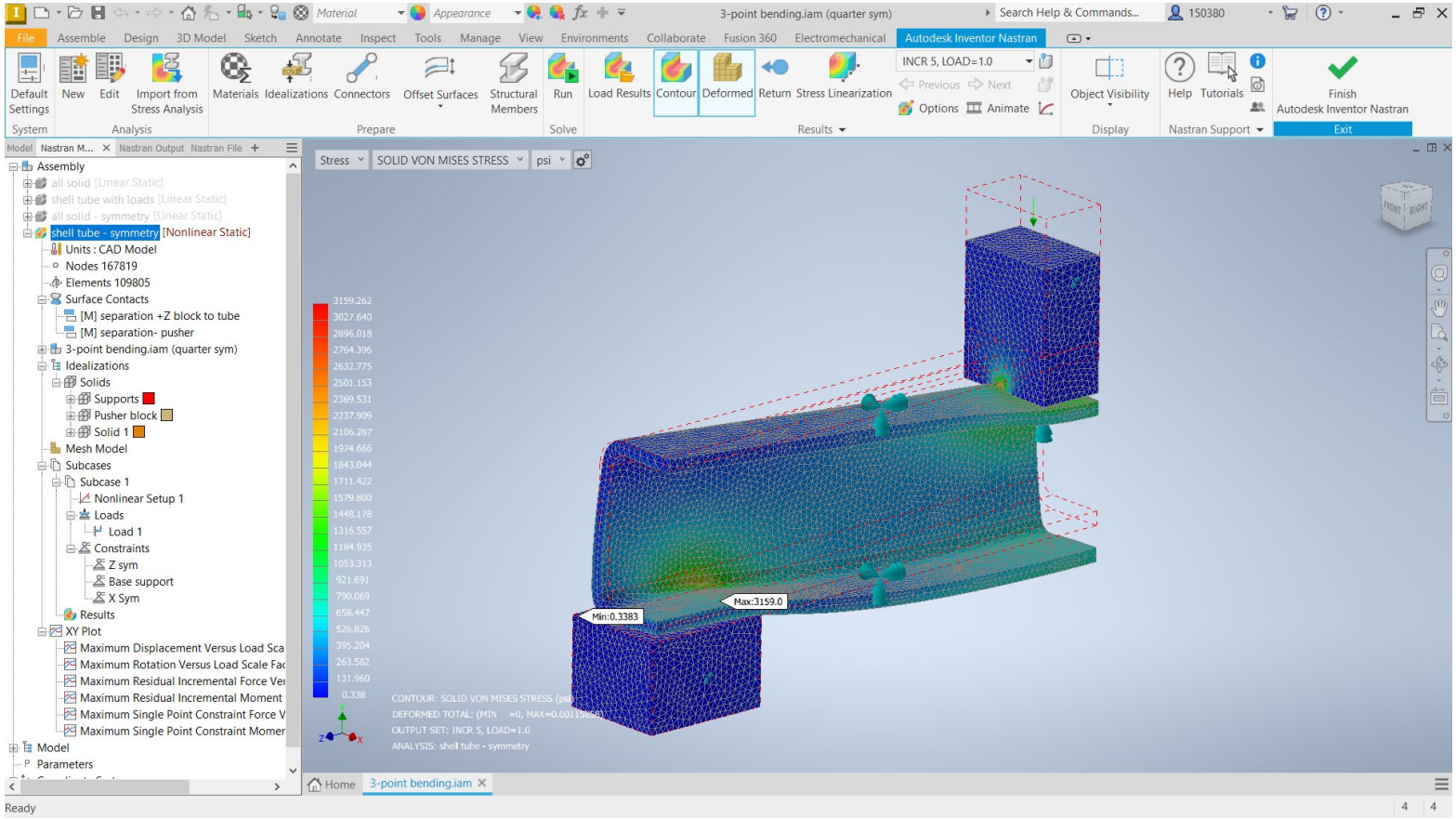
# Mesh Convergence – Element Size = 0.3in – 34Sec



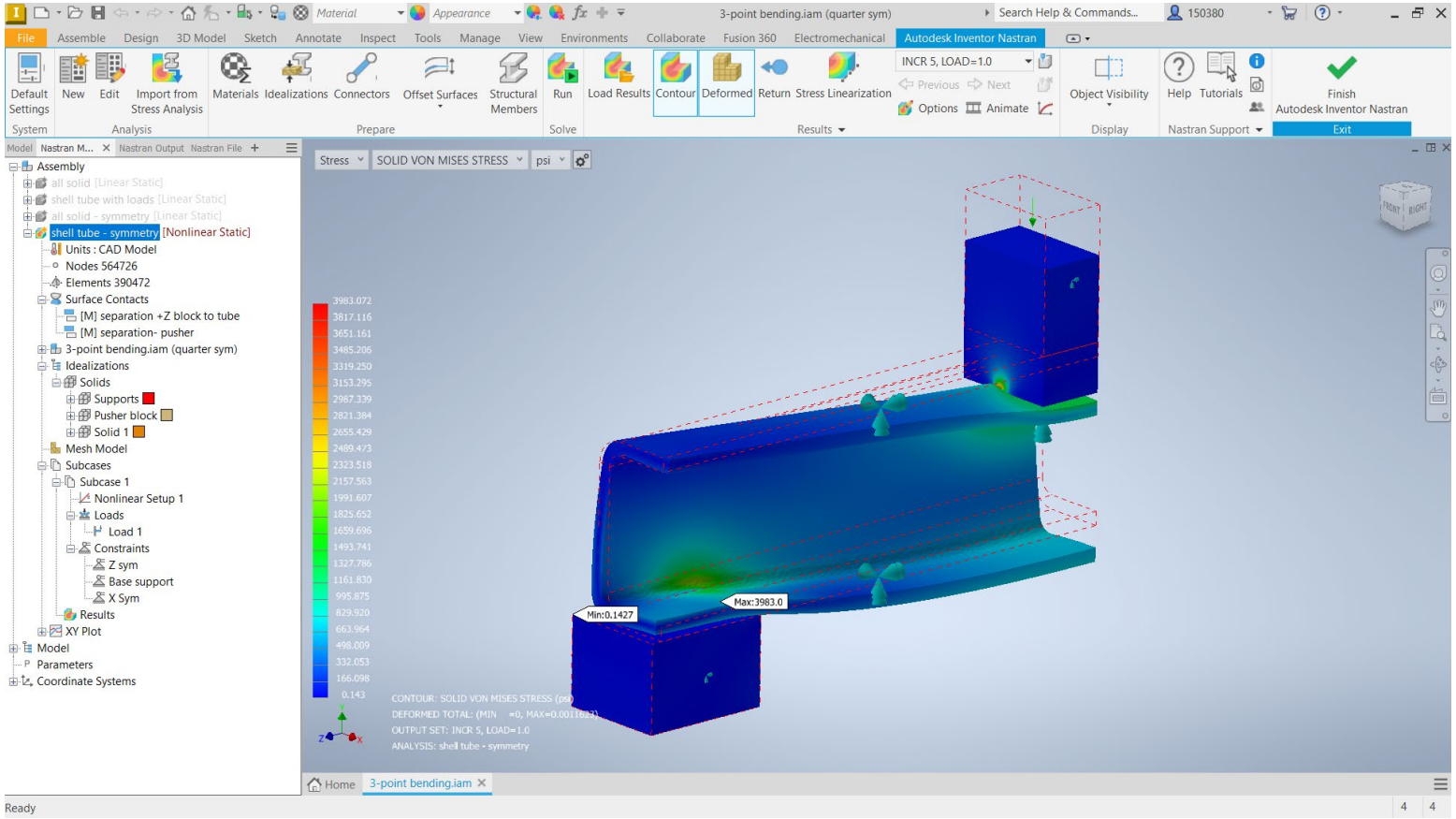
# Mesh Convergence – Element Size = 0.2in – 112Sec



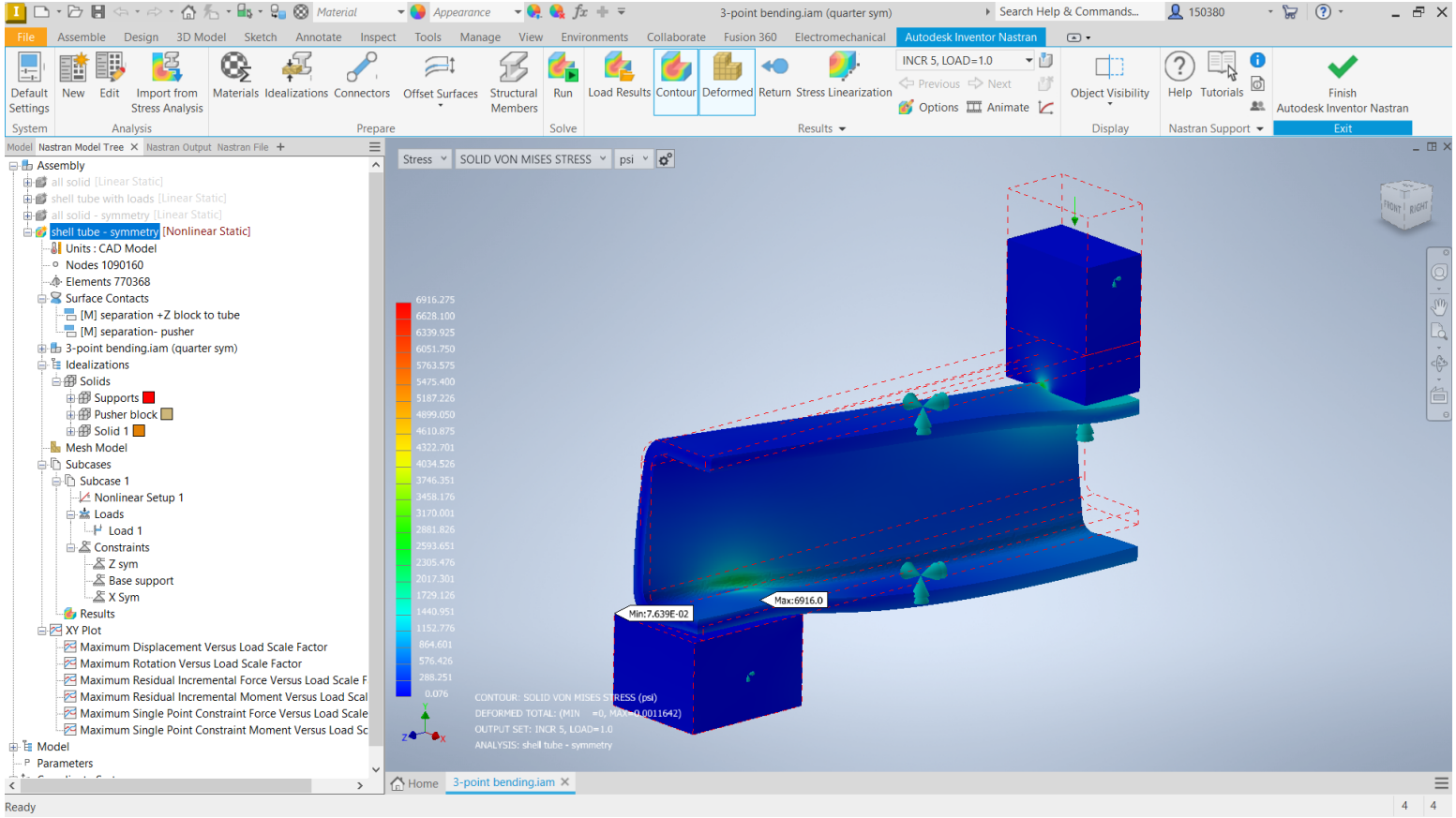
# Mesh Convergence – Element Size = 0.15in – 328Sec



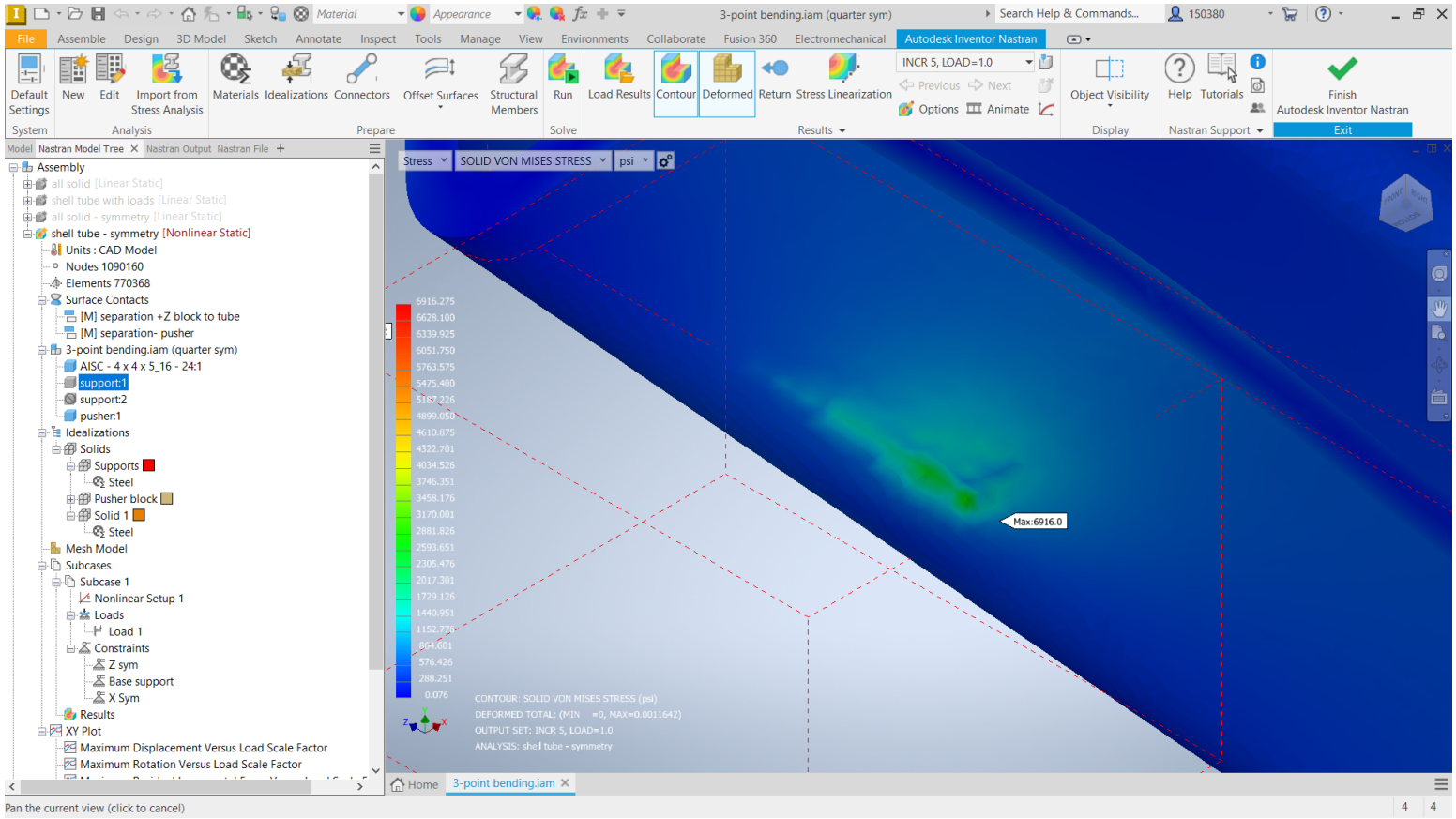
# Mesh Convergence – Element Size = 0.1in – 1205Sec

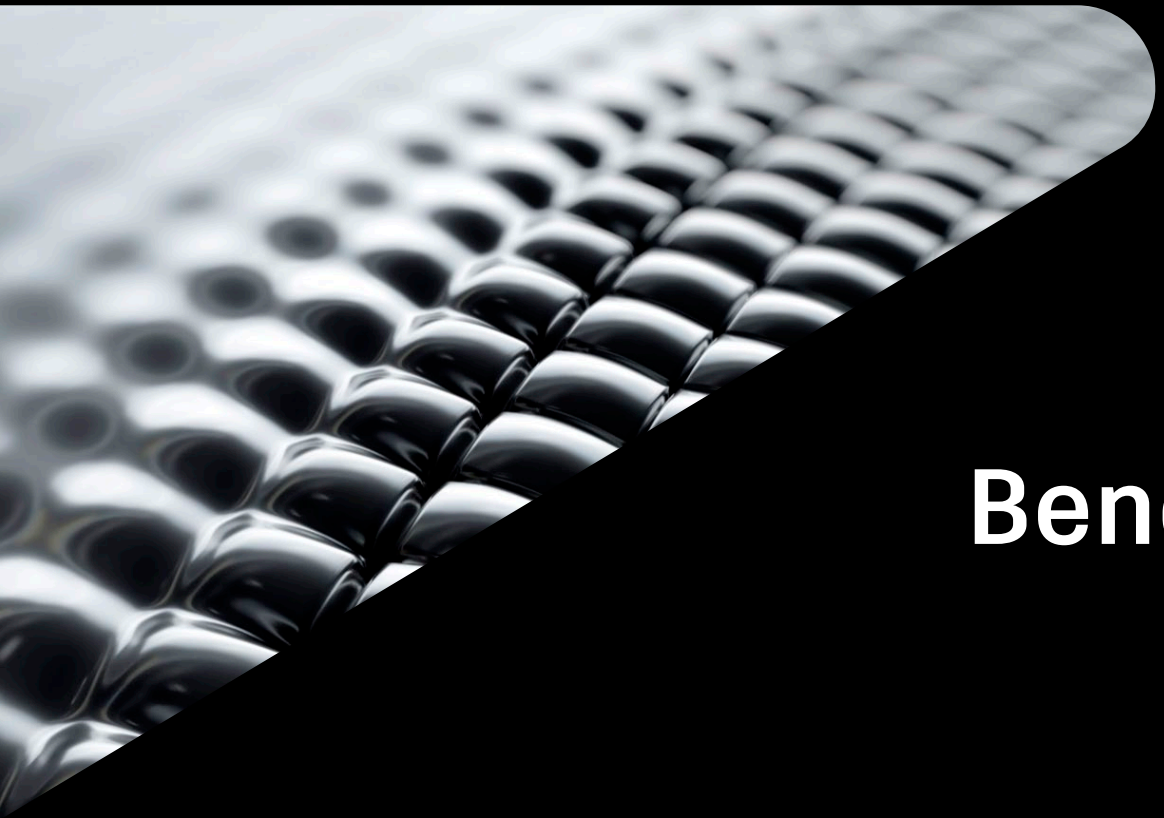


# Mesh Convergence – Element Size = 0.08in – 2154Sec



# Stress Will Continue to Increase as Mesh is Refined

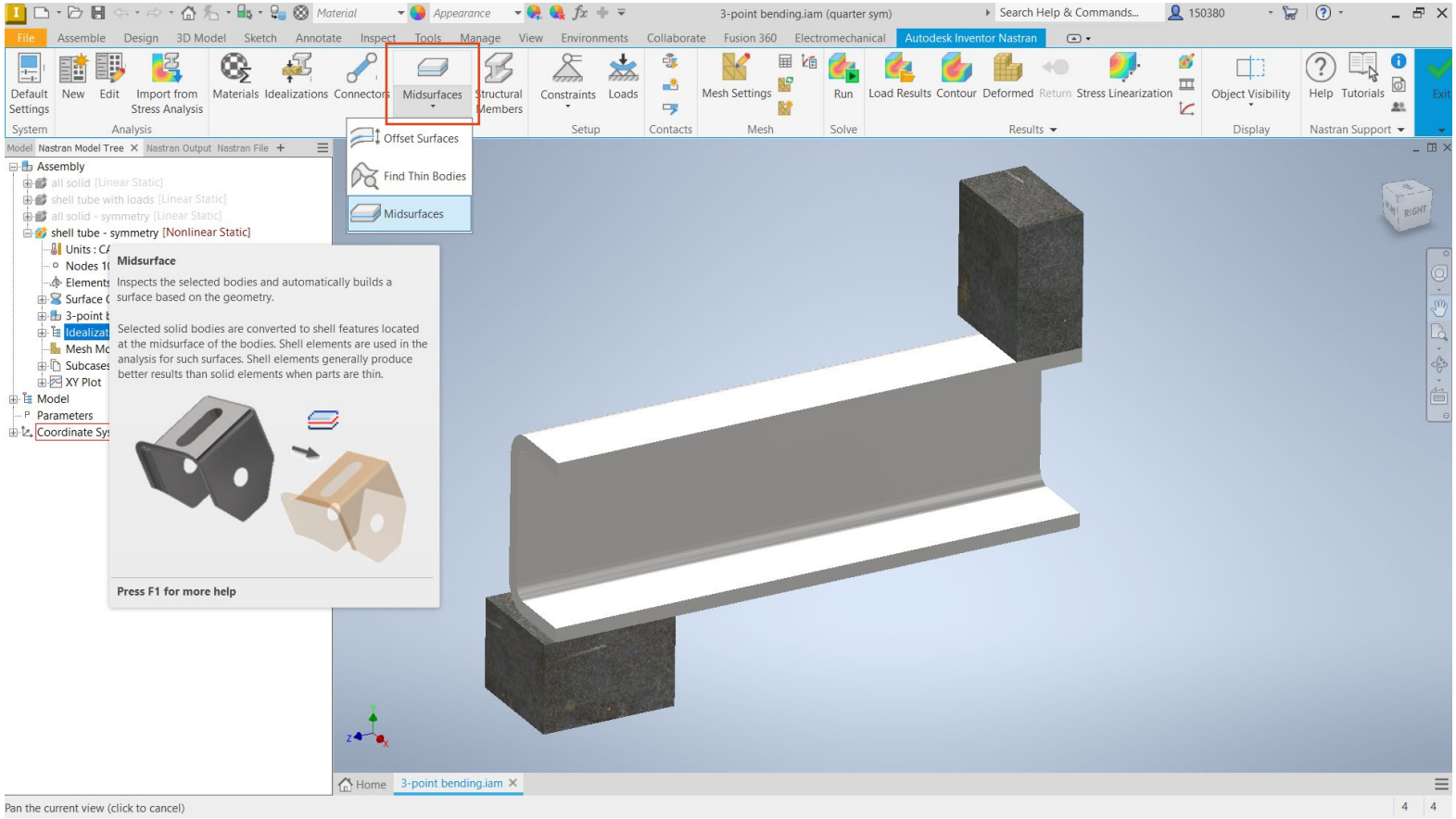




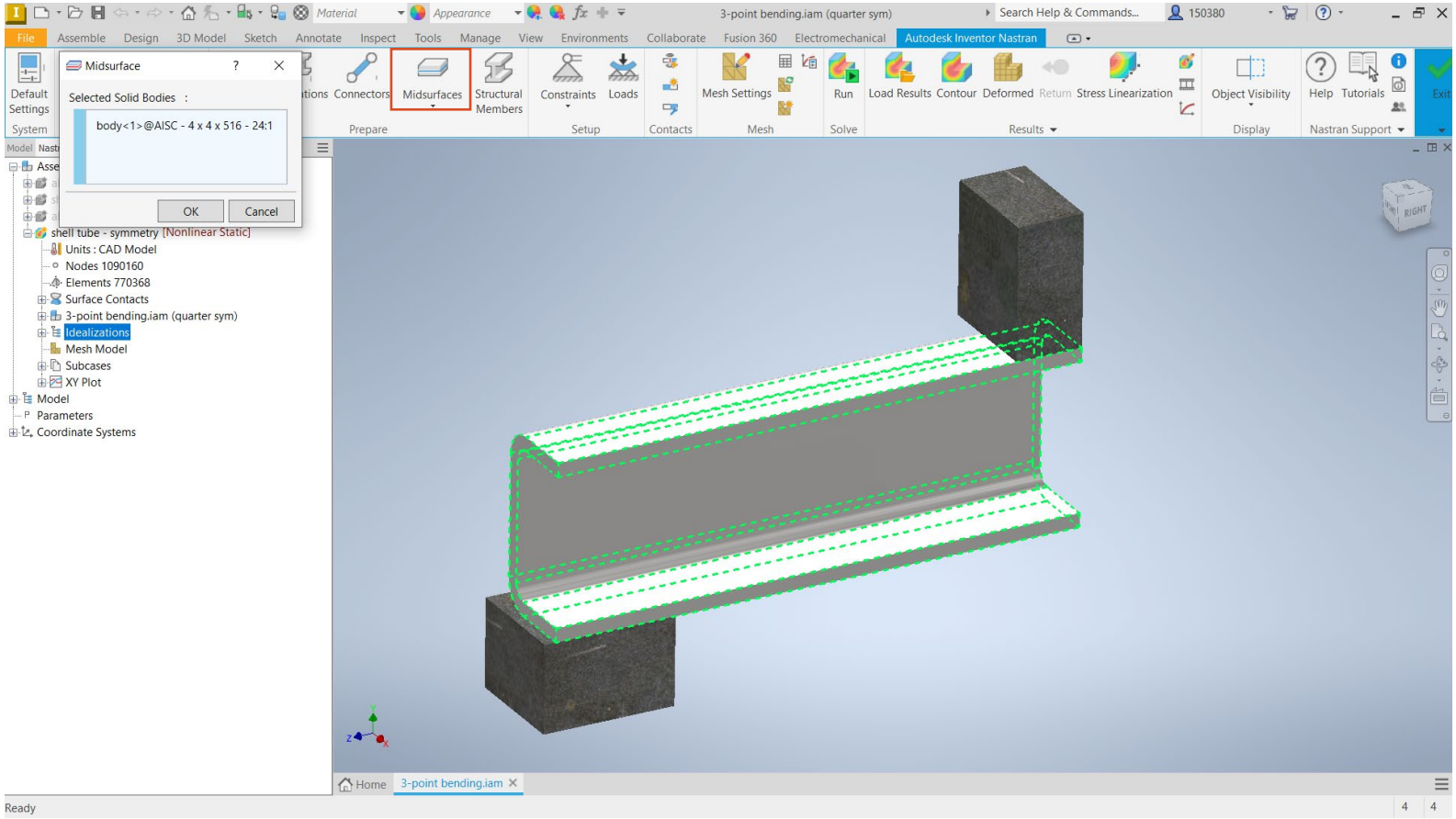
# 3-Point Tube Bending Example

Shell Elements

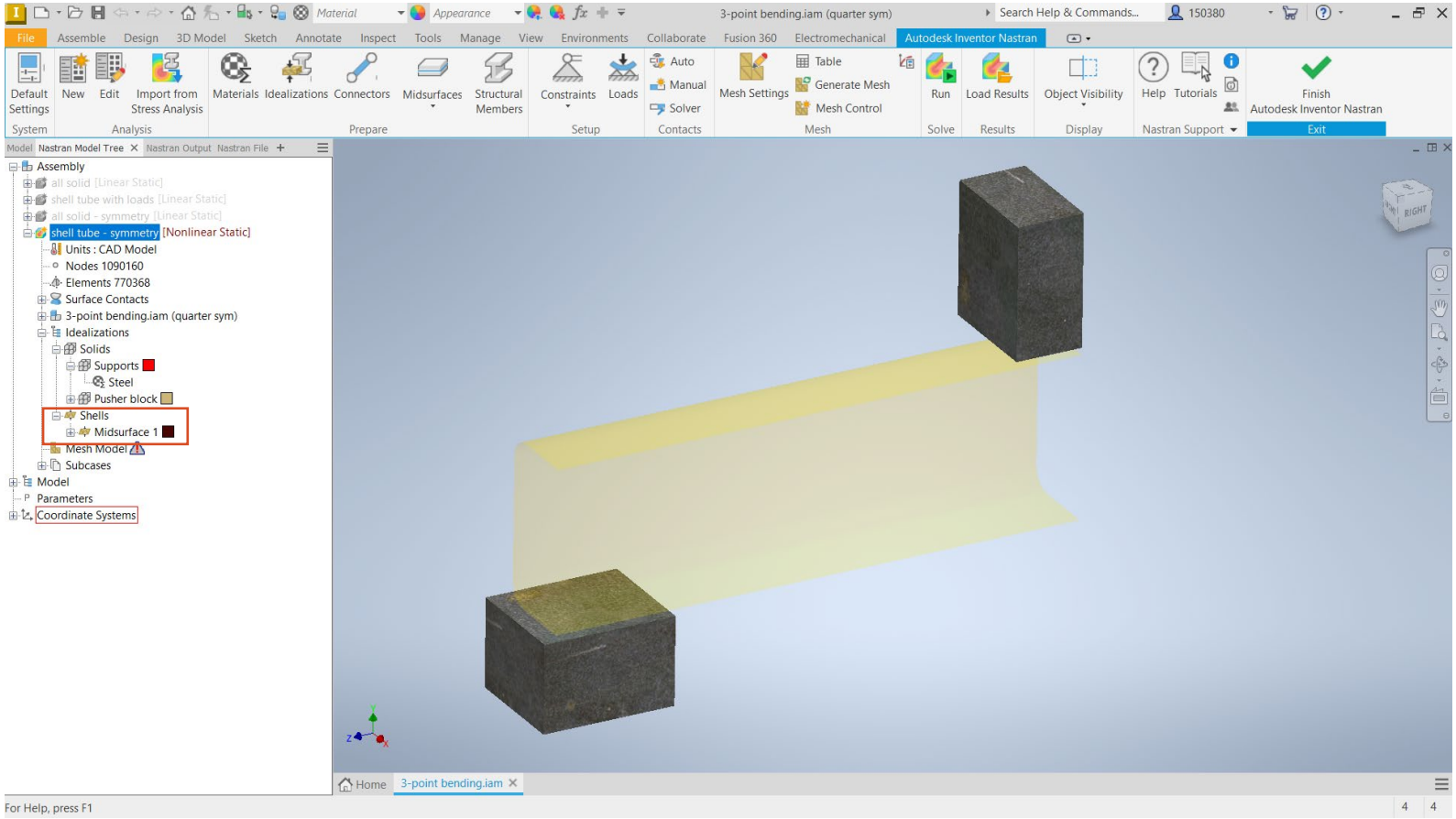
# Midsurfacing the Steel Tube



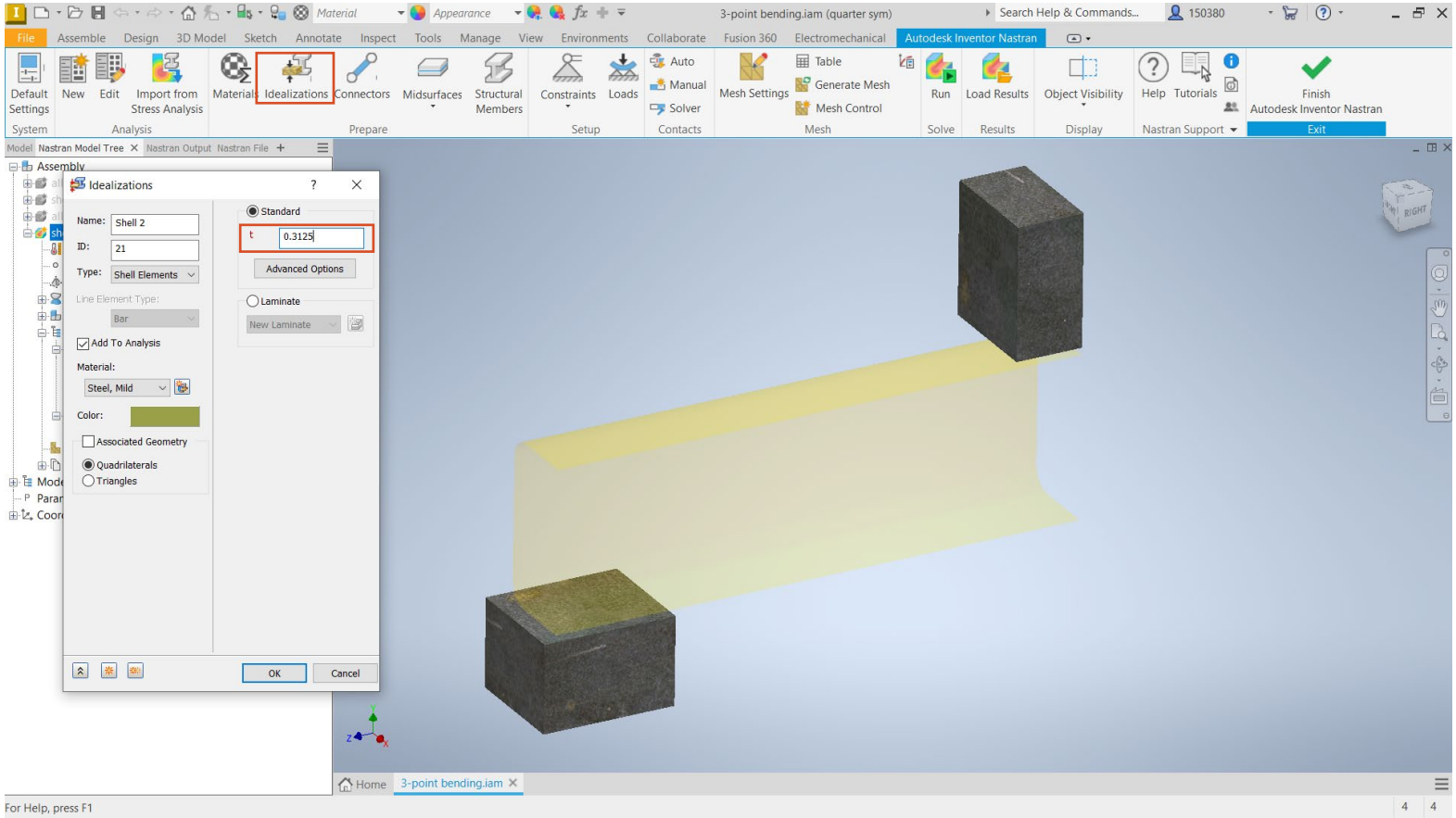
# Midsurfacing the Steel Tube



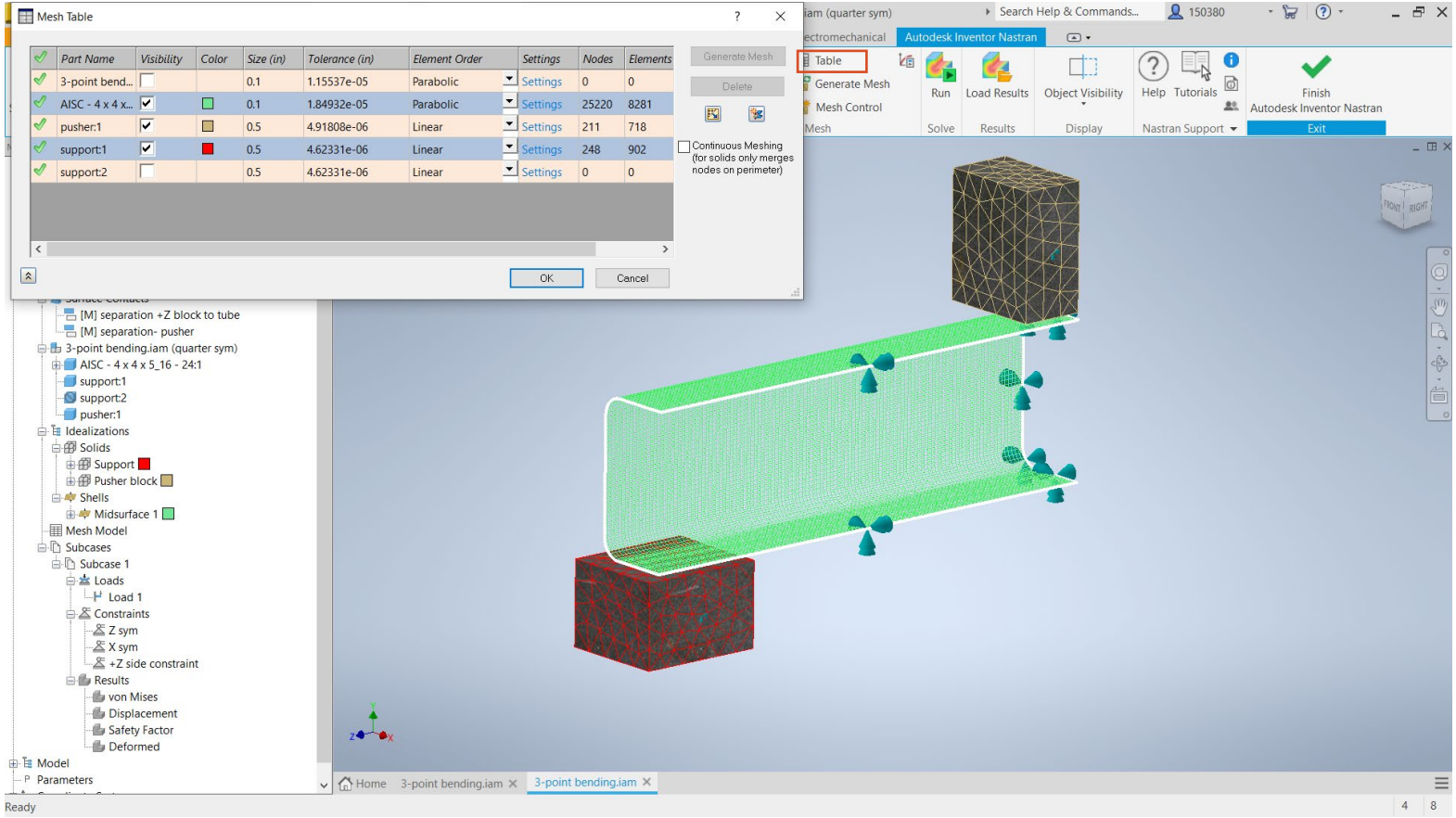
# Midsurfacing the Steel Tube – Shell Idealization



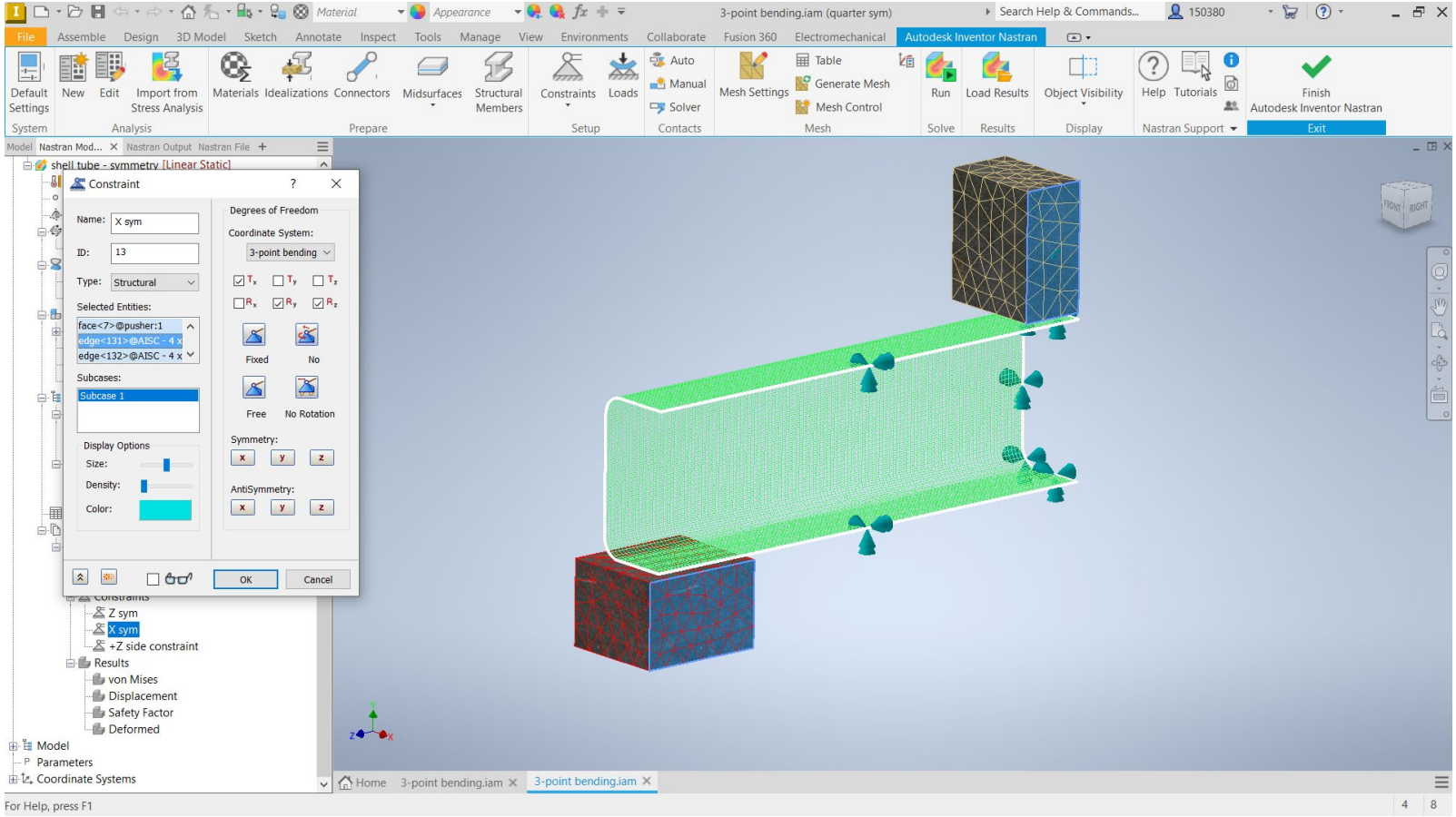
# Midsurfacing the Steel Tube – Specify Thickness



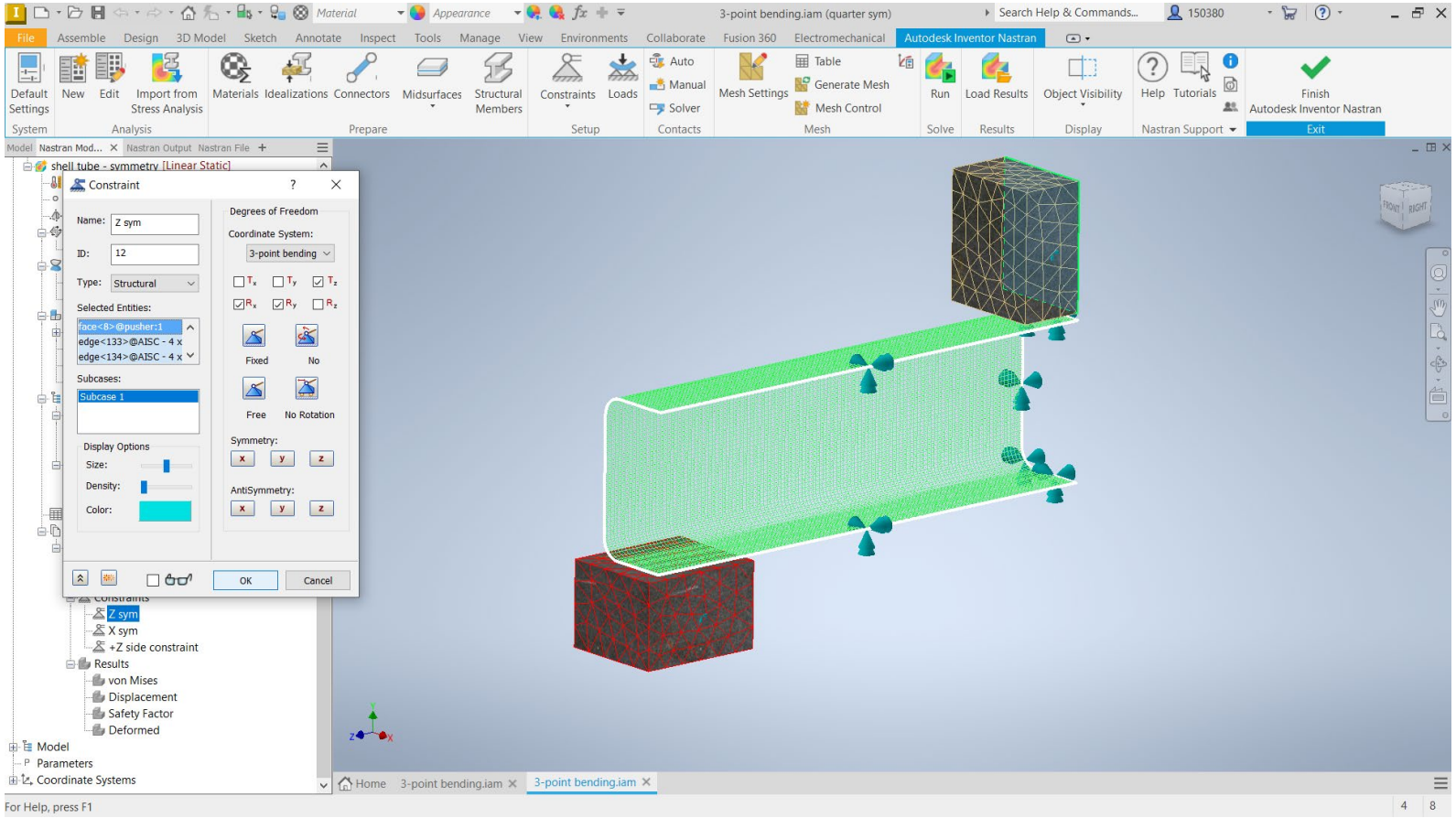
# Using the Mesh Table to Define Different Meshes



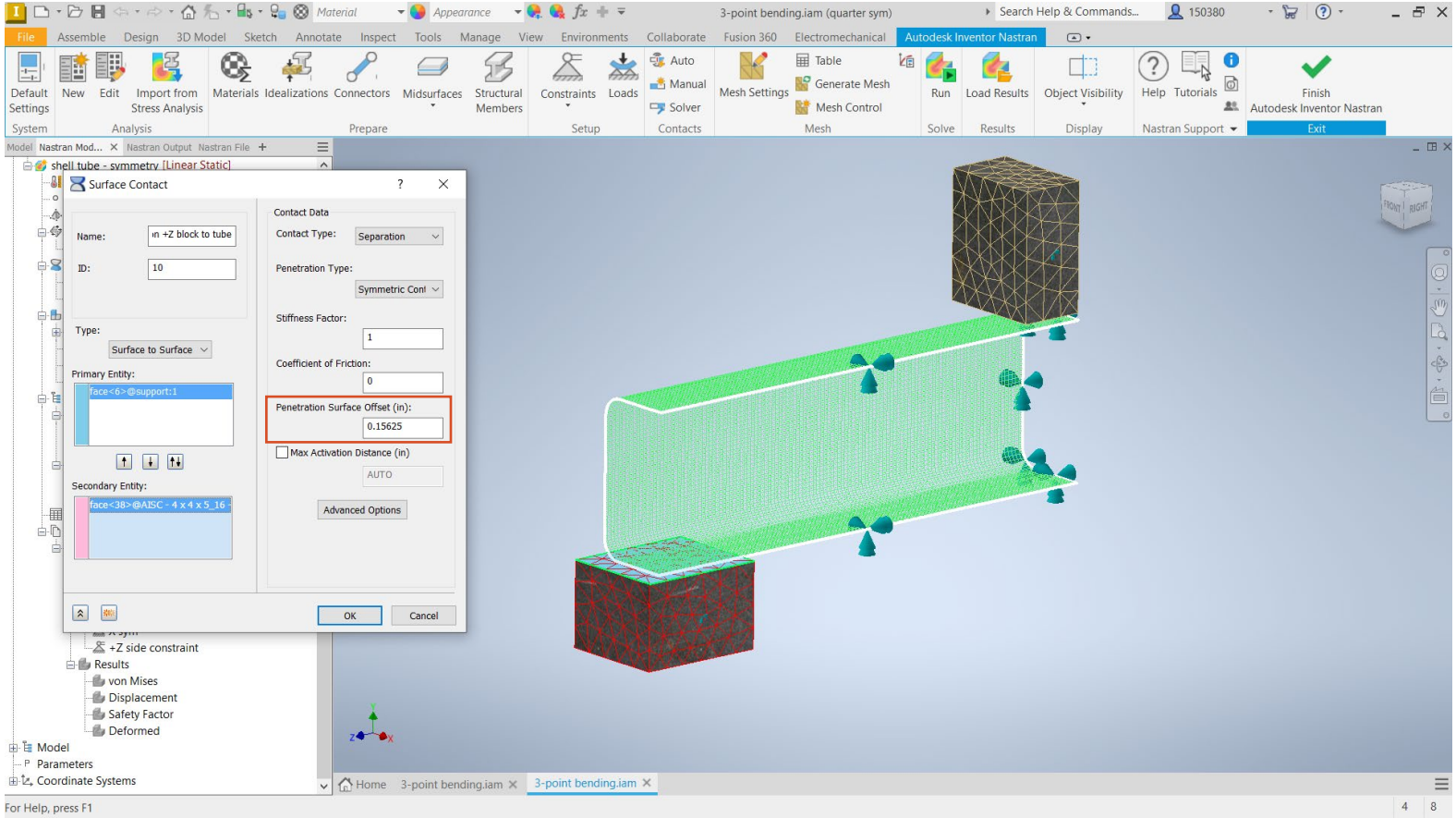
# Add Shell Edge Constraints – X-Sym



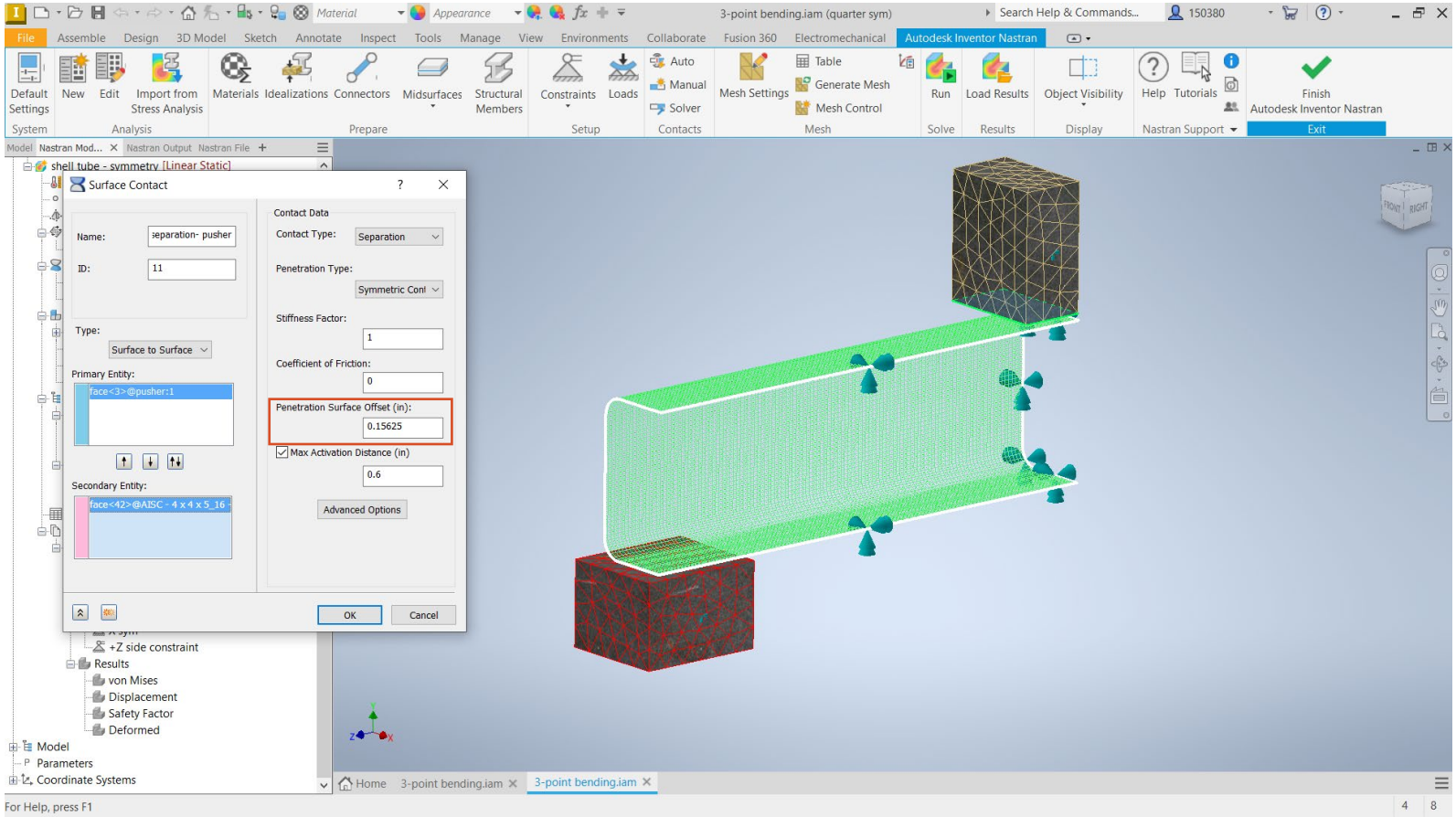
# Add Shell Edge Constraints – Z-Sym



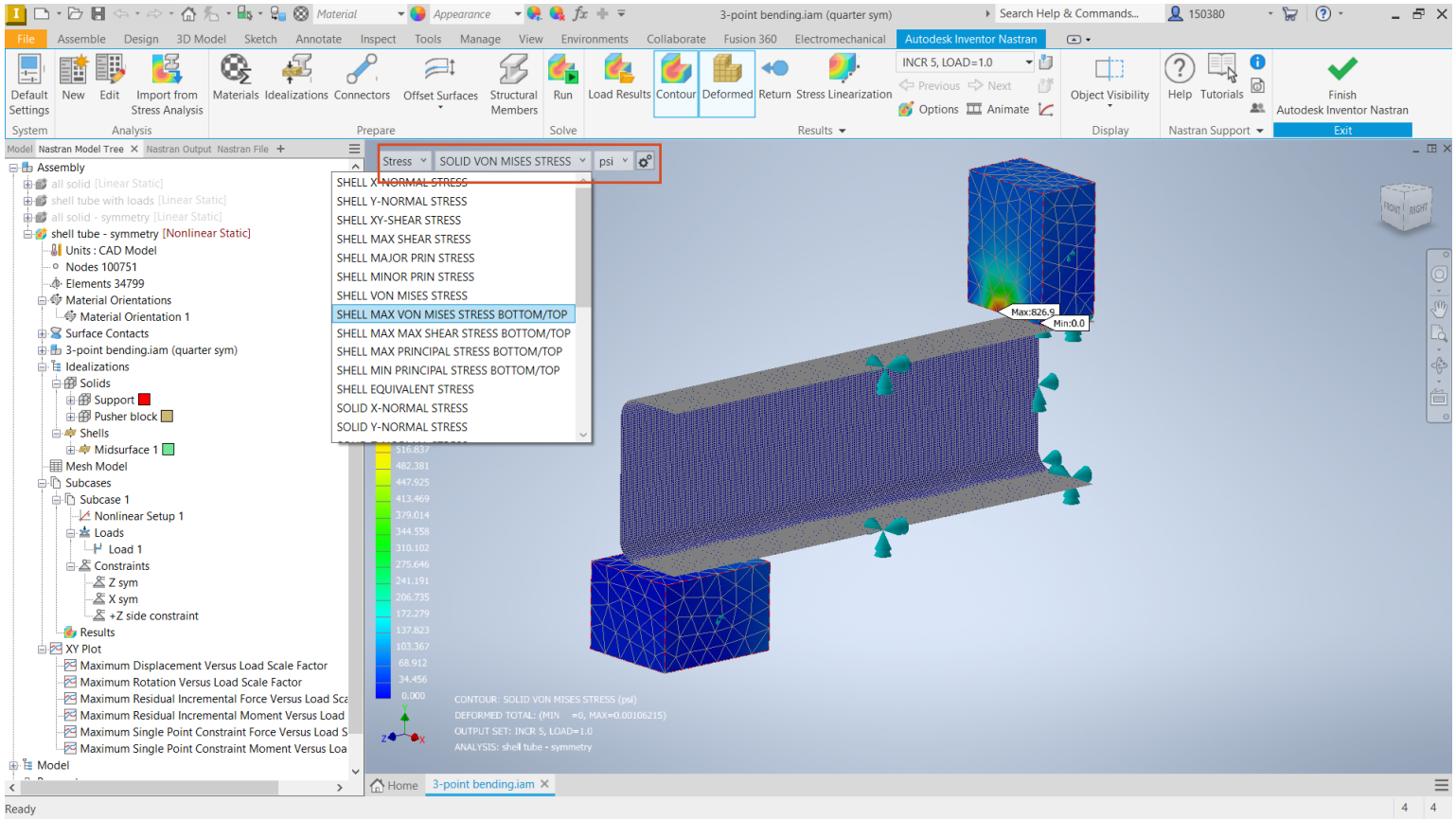
# Modify Contact Definition to Account for Shell Offset



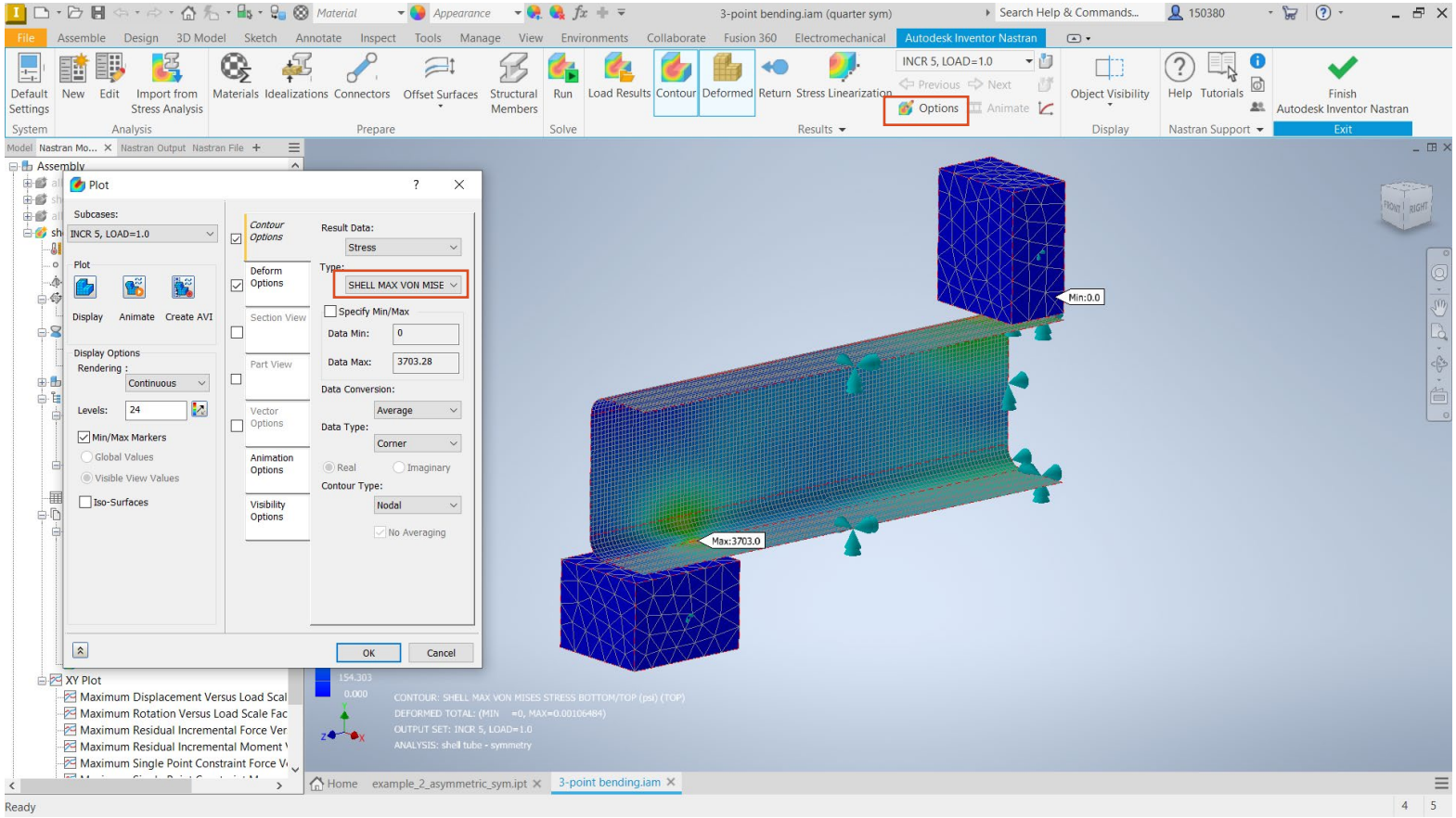
# Modify Contact Definition to Account for Shell Offset



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

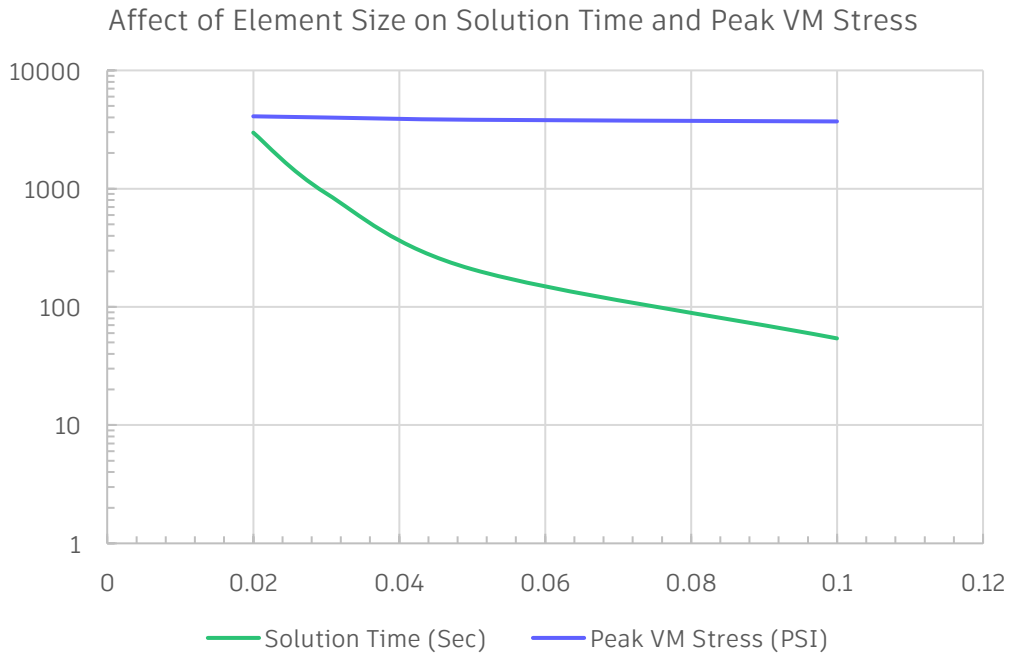


# Switch Default Solid VM Stress to Shell Max VM Top/Bottom

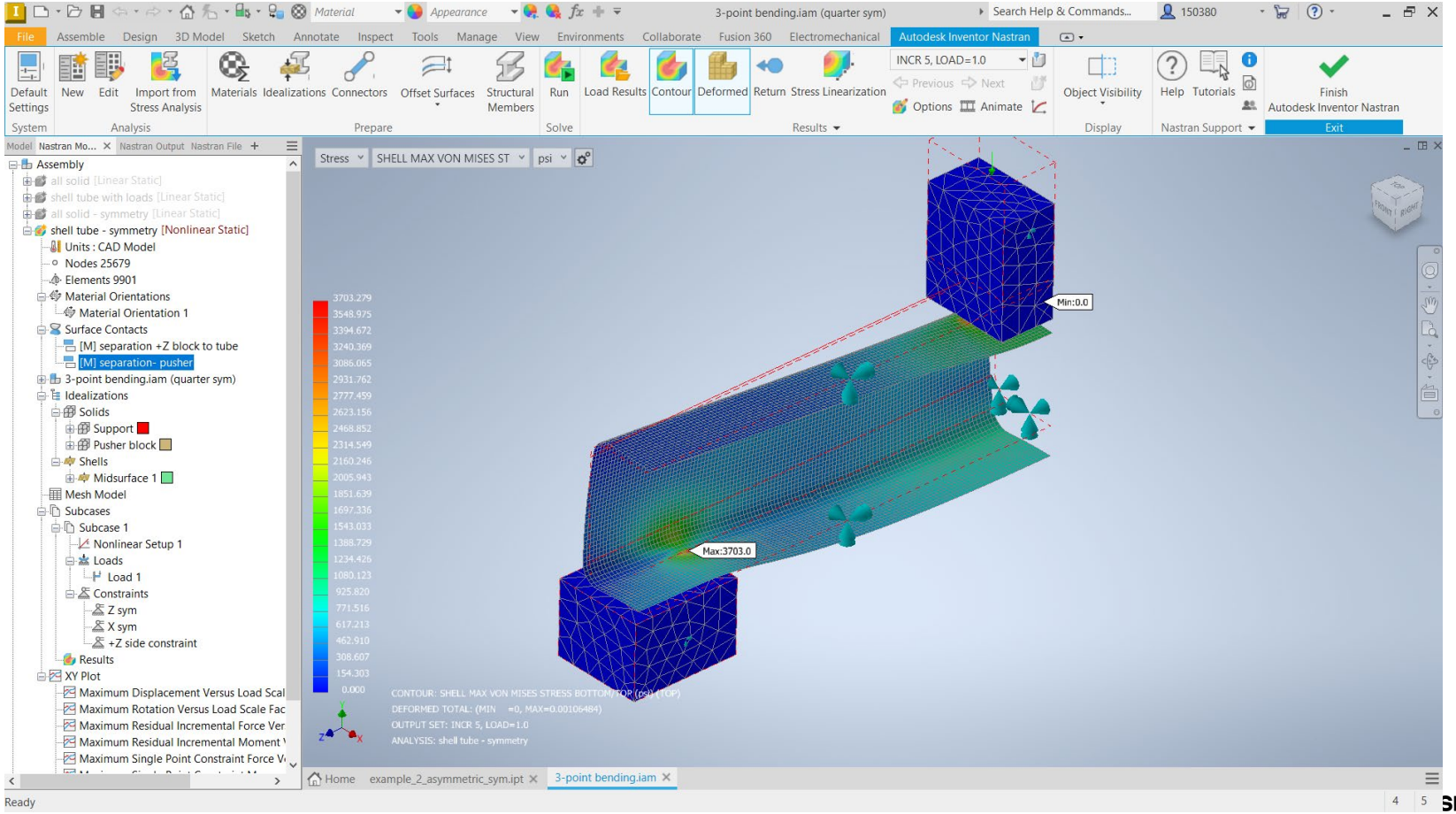


# Shell Mesh Convergence Study

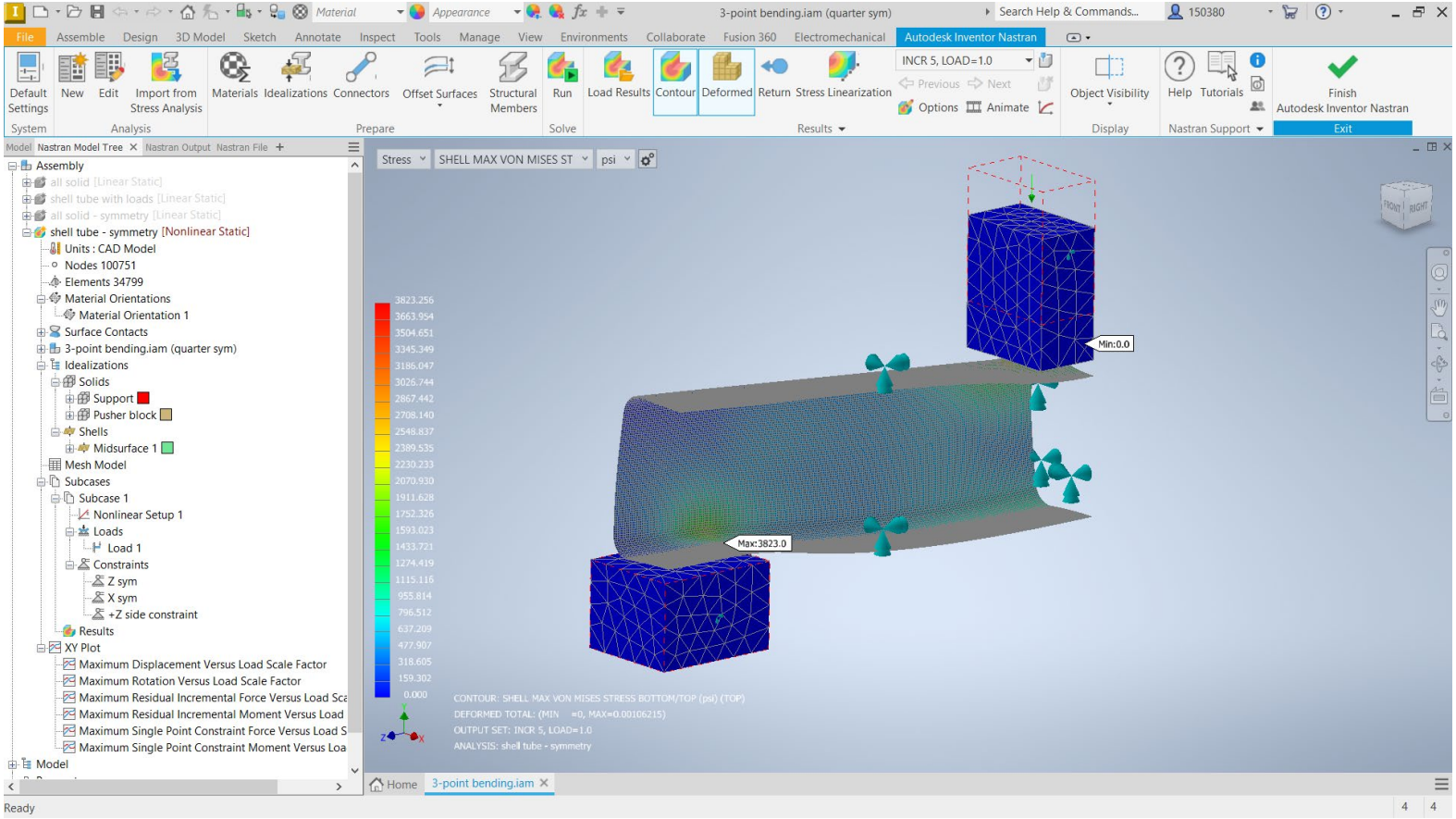
- Enhanced contact is OFF
- Mesh element size is varied from 0.1 to 0.02
- The peak stress does not increase with the same rate as it did with solids
- Solution time still goes up exponentially with model size or with the inverse of element size



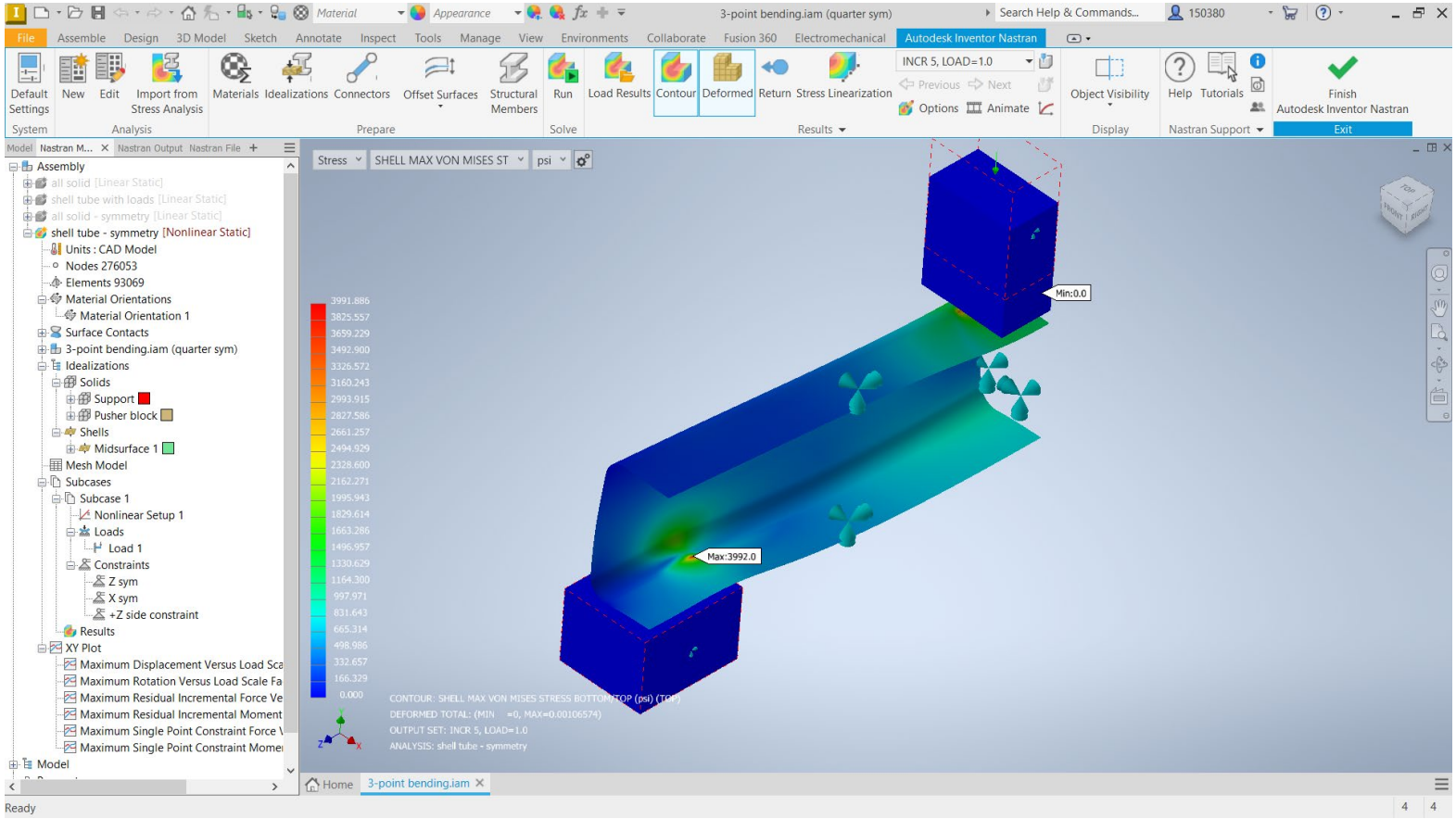
# Mesh Convergence – Element Size = 0.1in – 54Sec



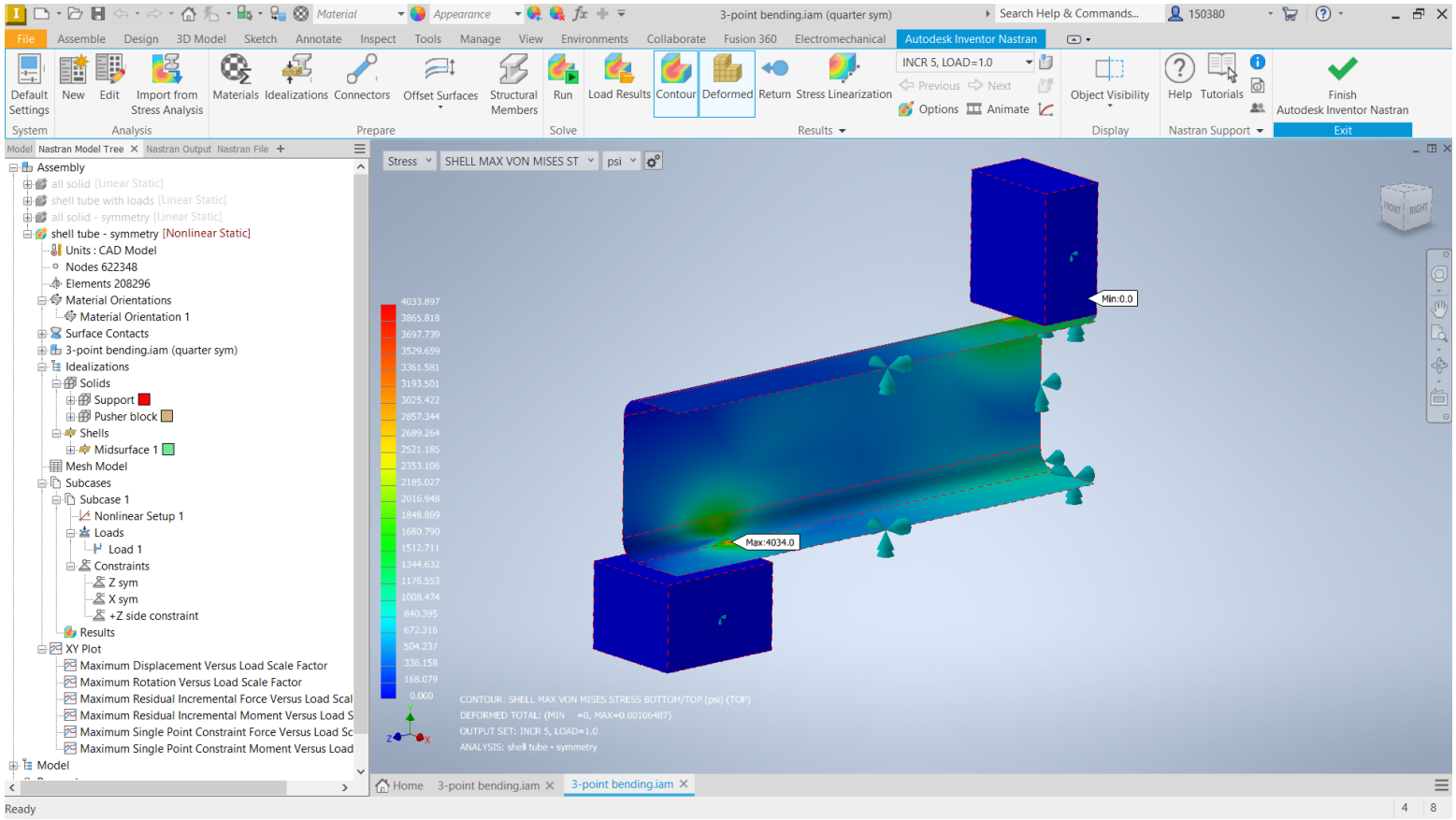
# Mesh Convergence – Element Size = 0.05in – 208Sec



# Mesh Convergence – Element Size = 0.03in – 912Sec



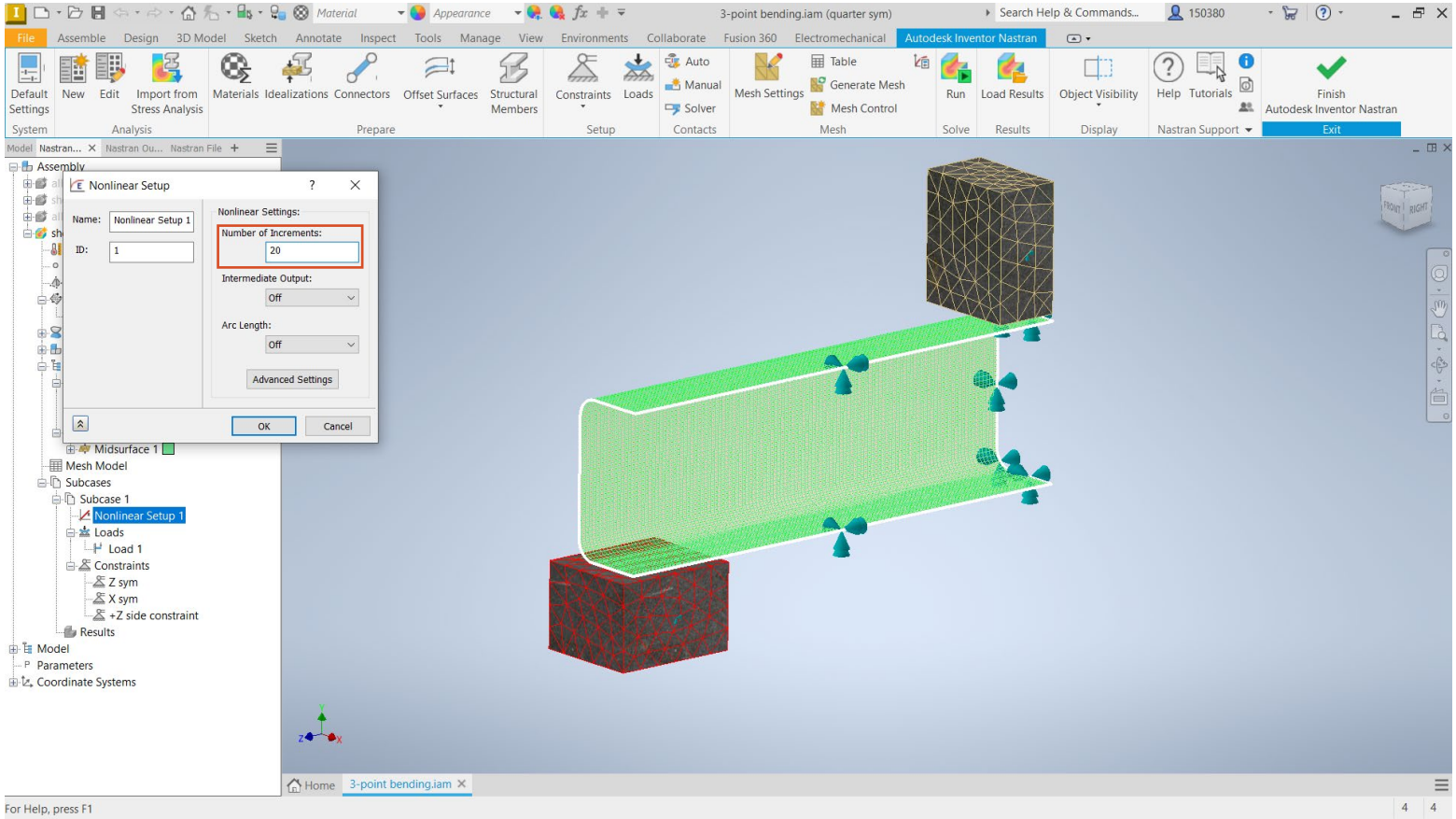
# Mesh Convergence – Element Size = 0.02in – 2983Sec



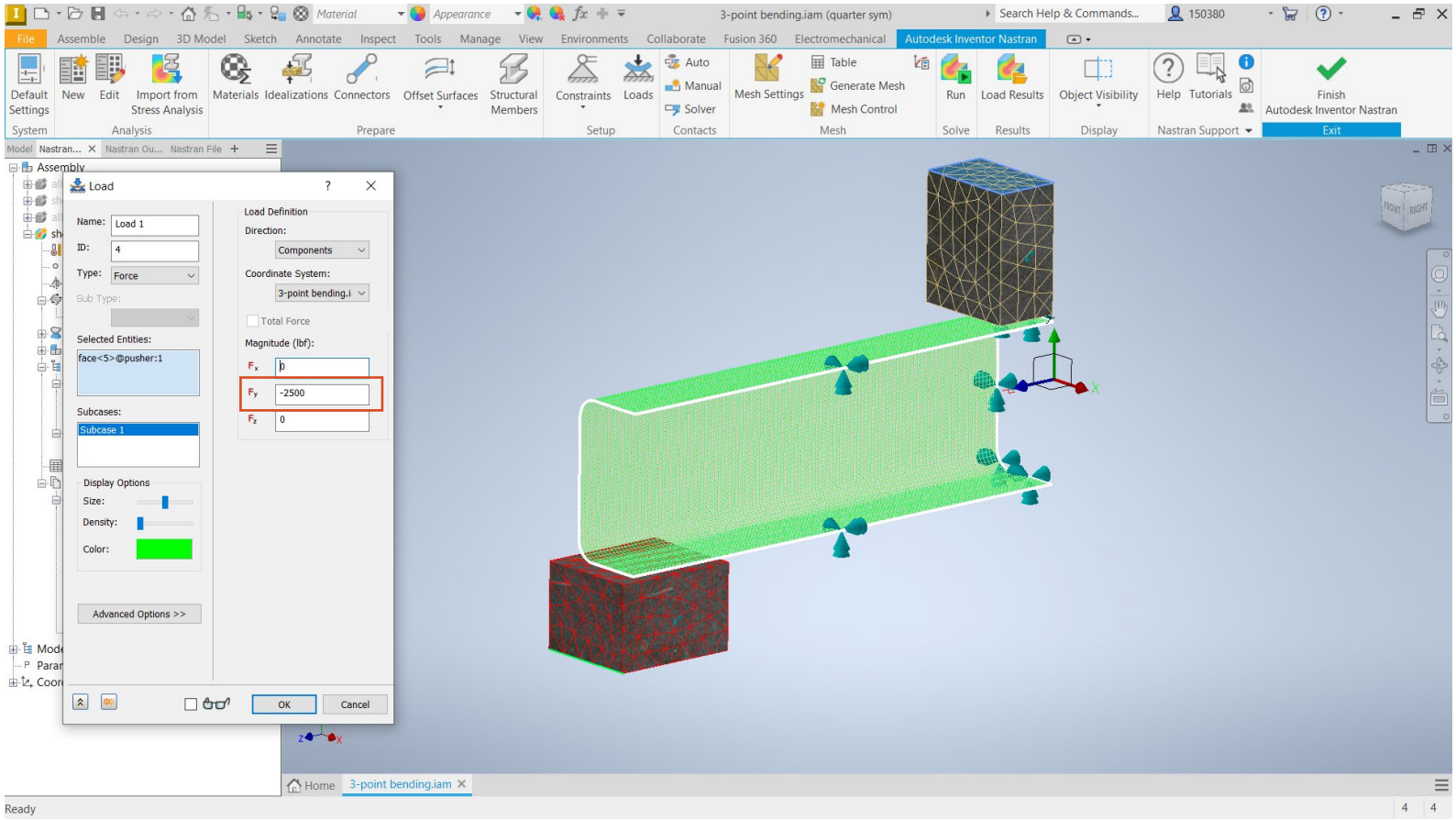
# 3-Point Tube Bending Example

Shell Element Material Nonlinear Analysis

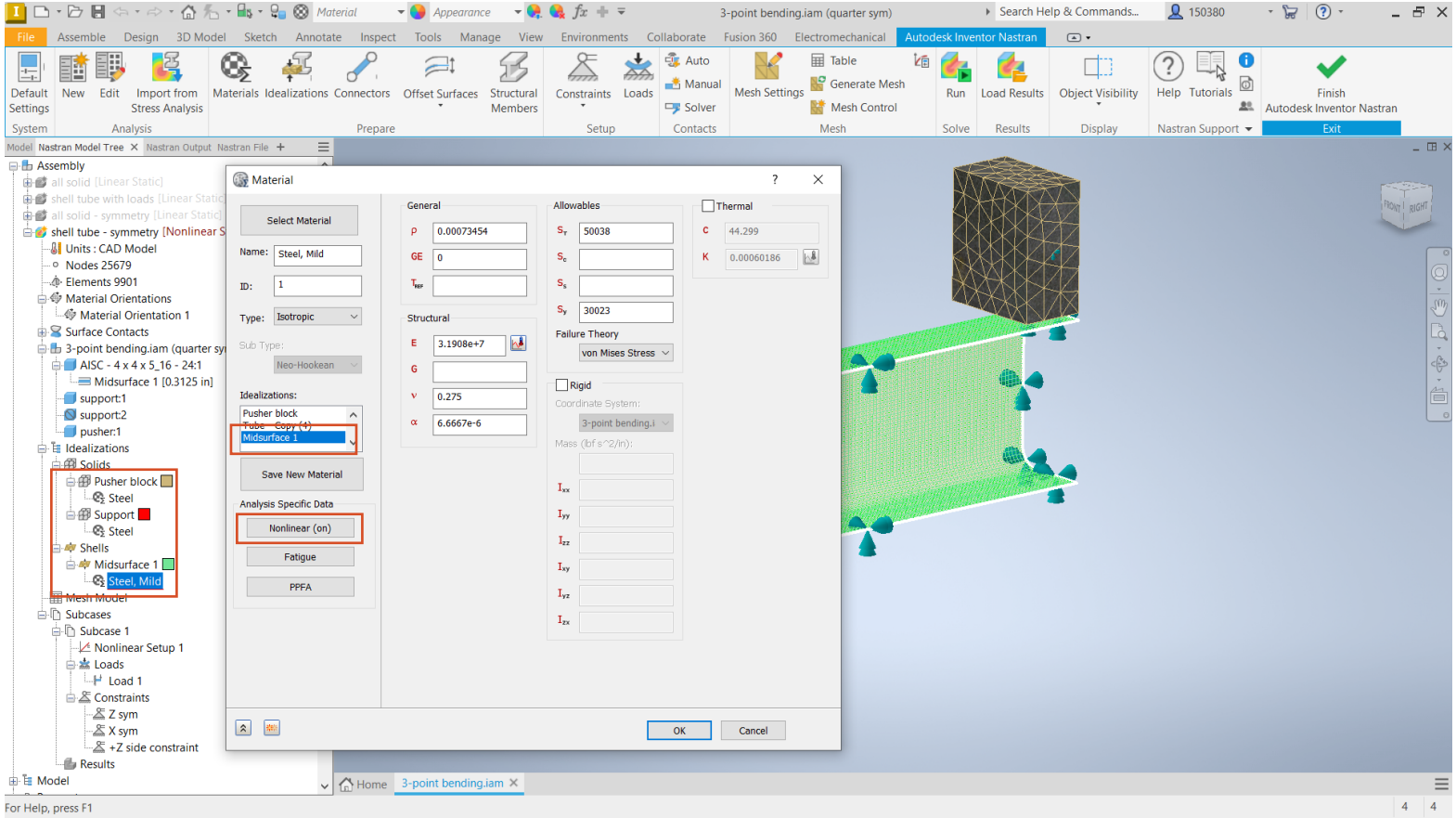
# For Material Nonlinear Use 20-40 Increments



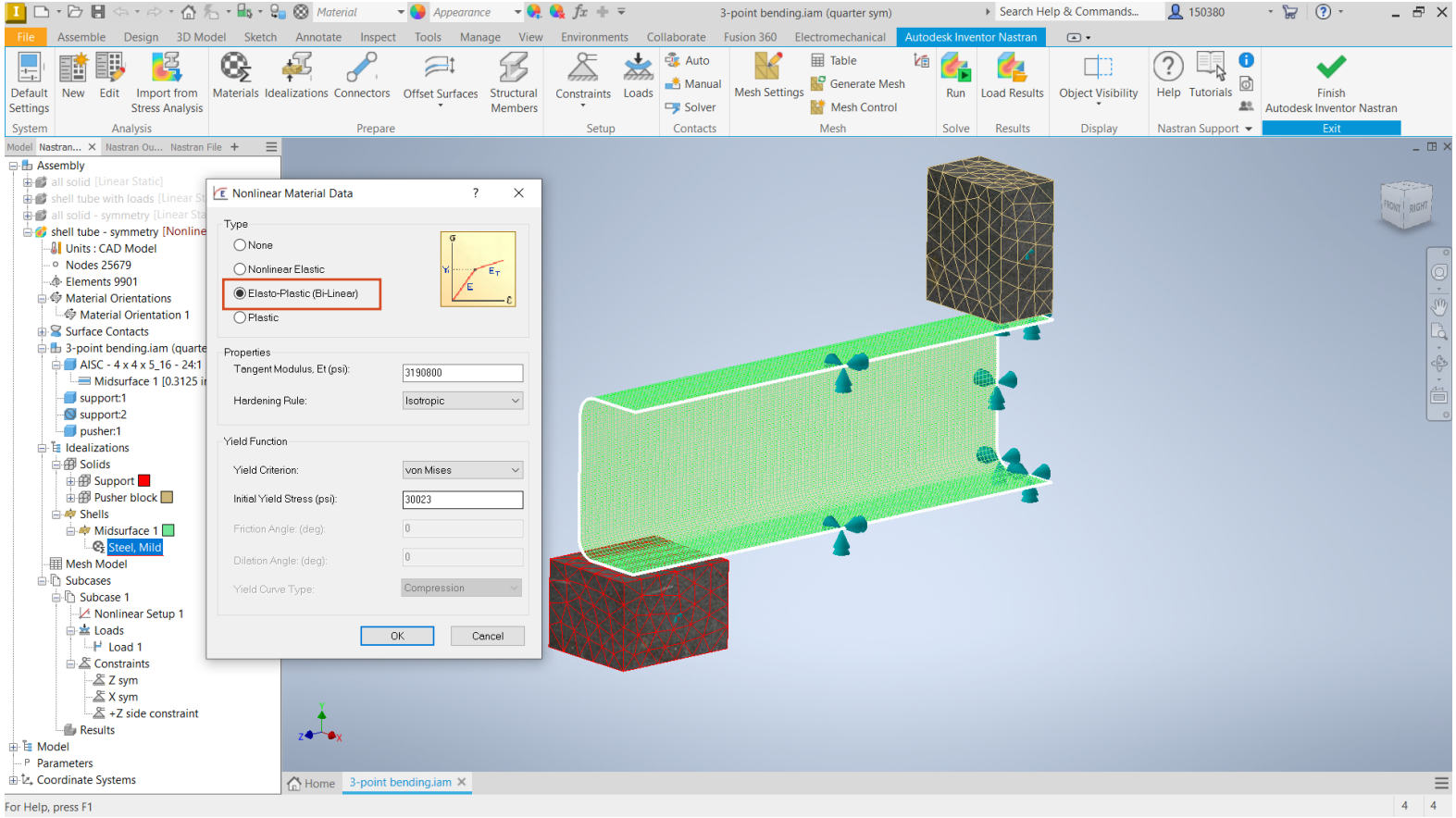
# Increase the Load 10x So the Tube Will Yield



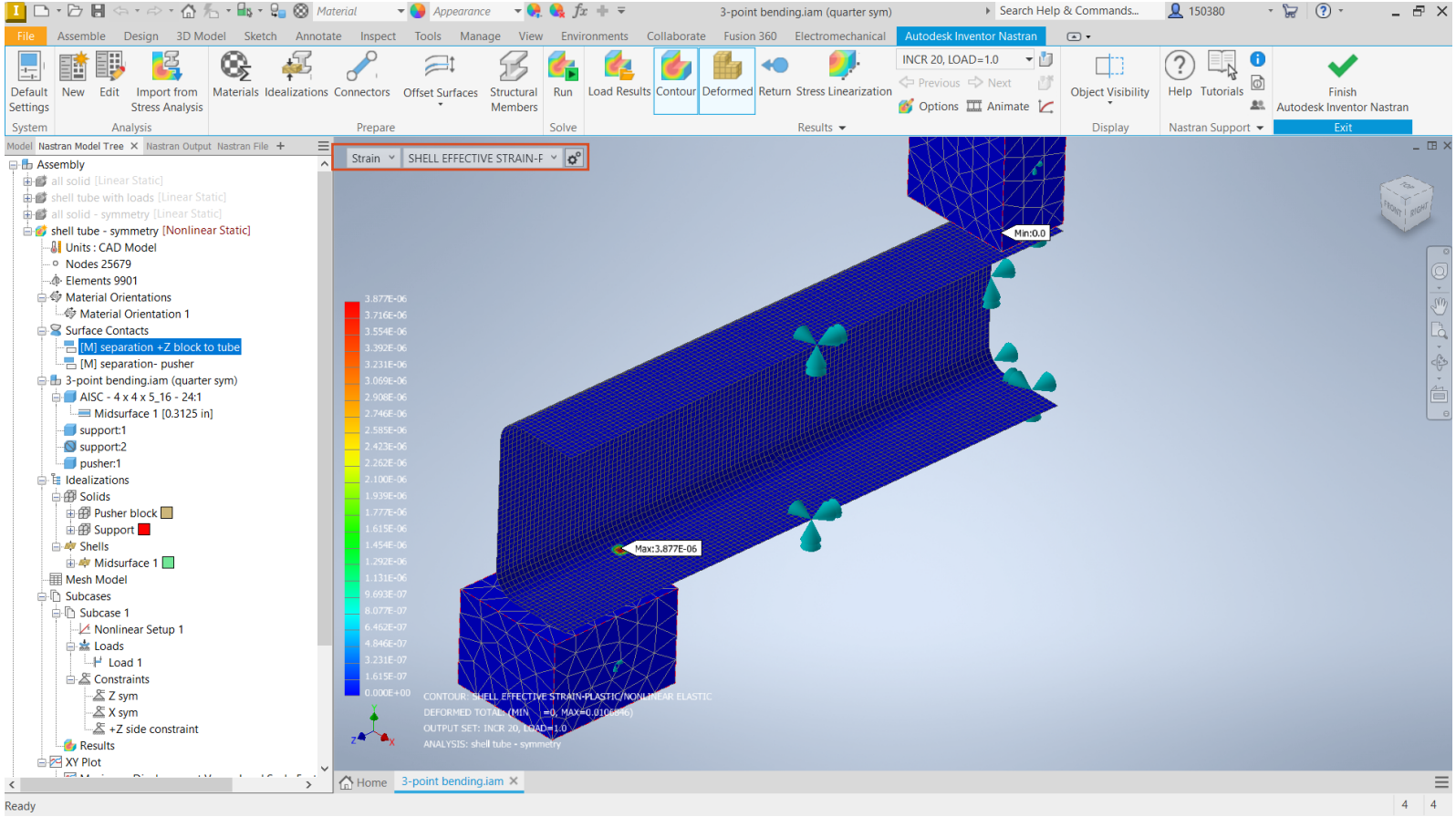
# Enable a Nonlinear Material for Tube Only



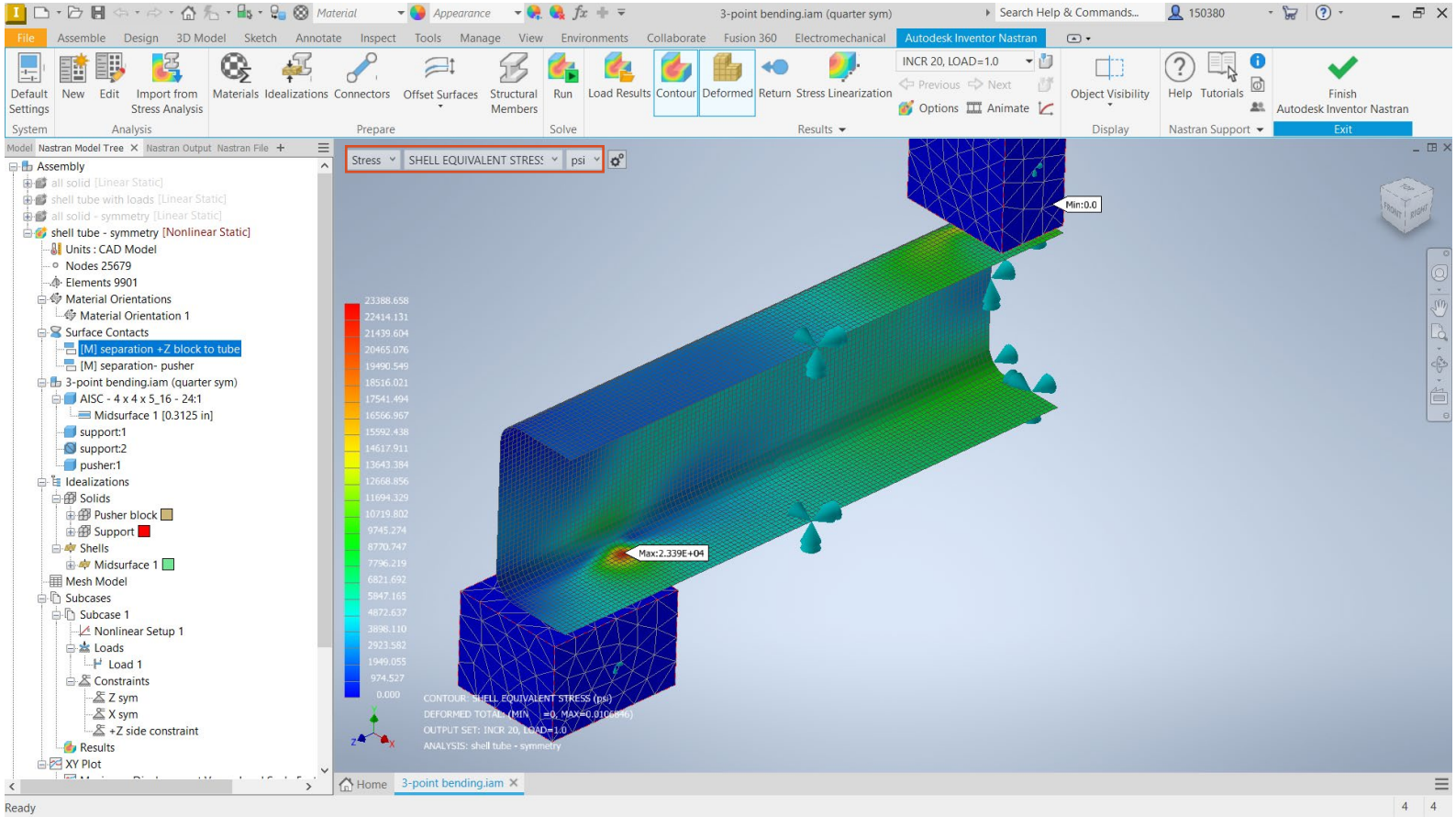
# Enable a Nonlinear Material



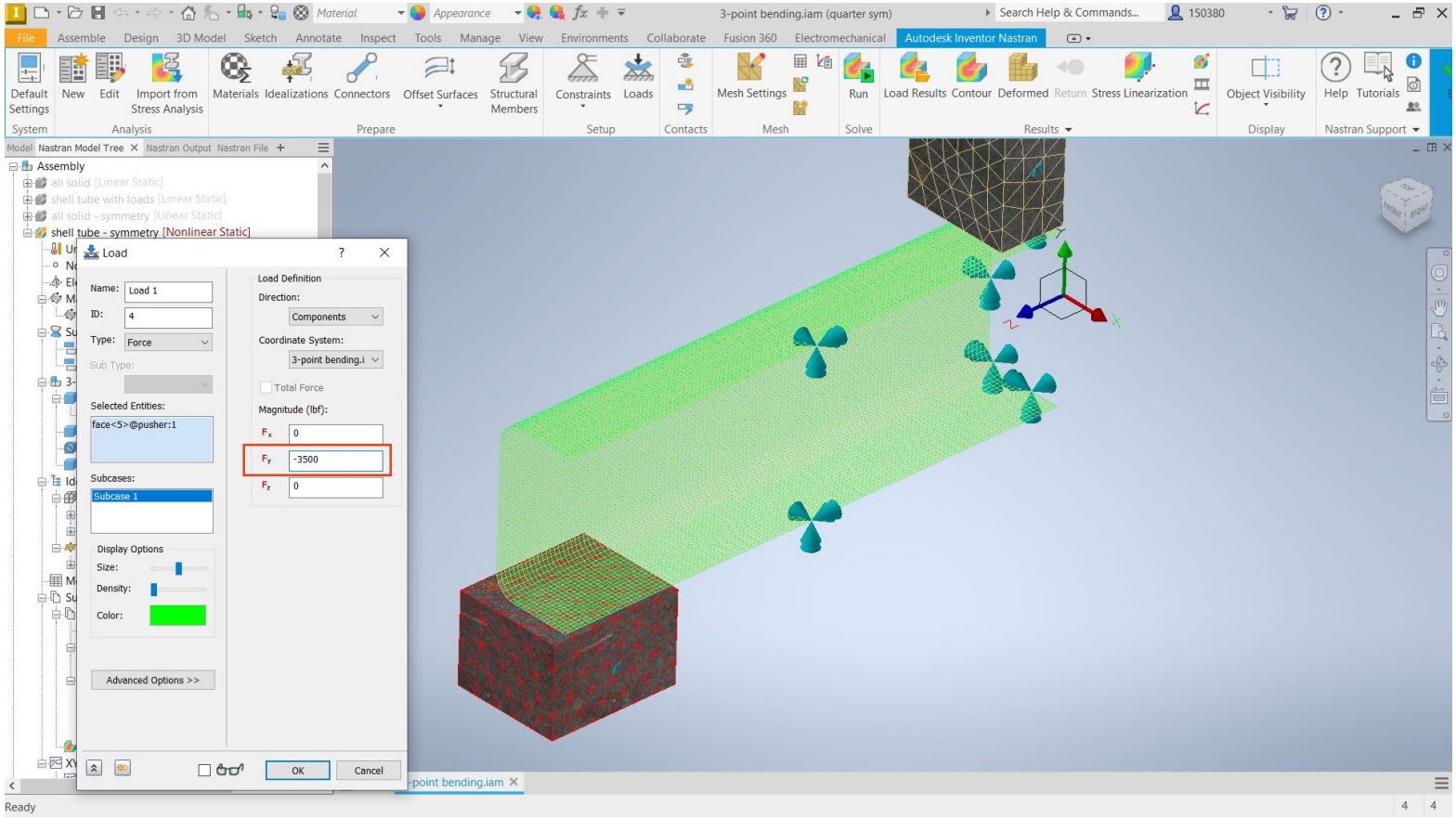
# Plastic Strain Due to Yielding



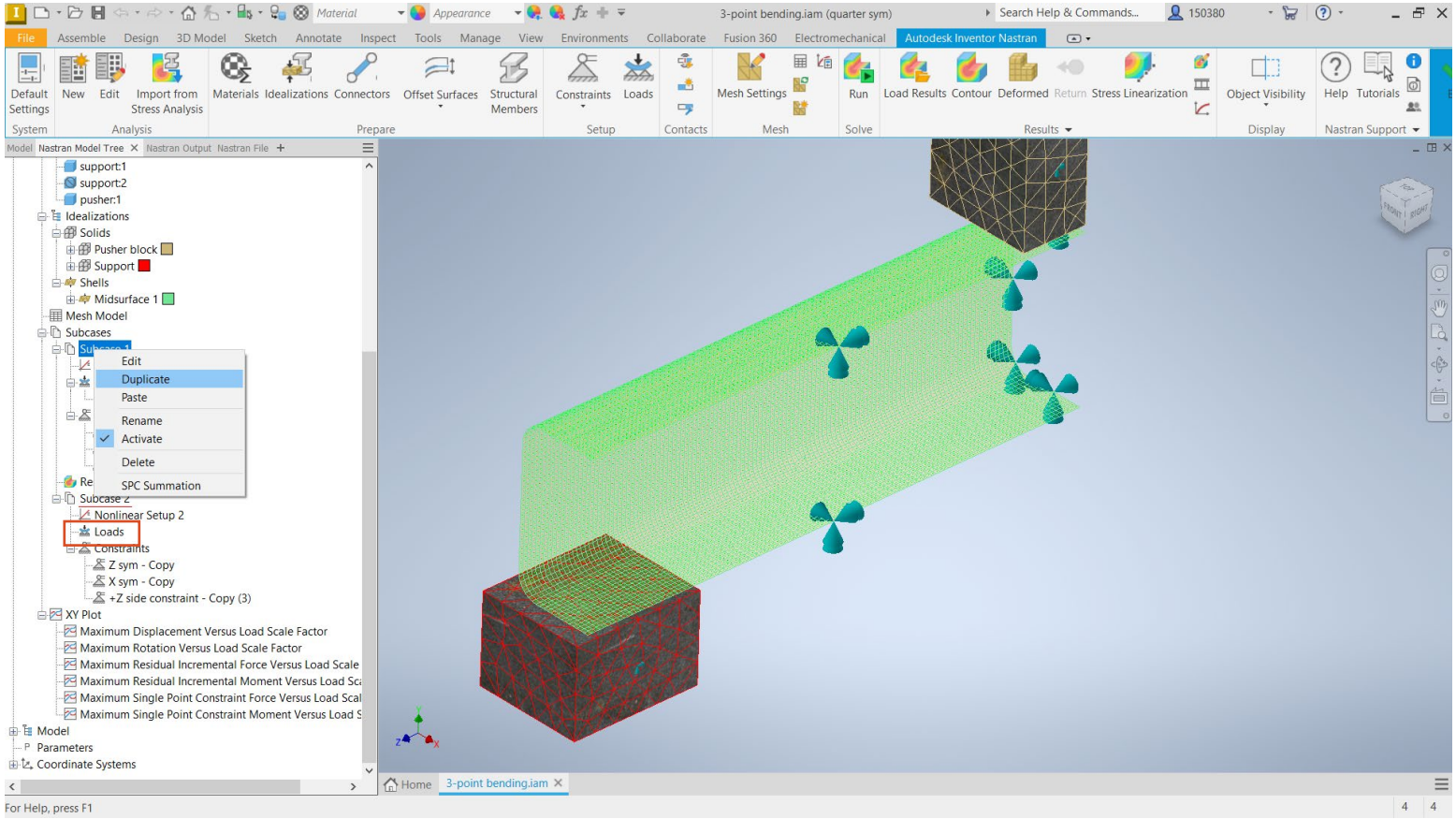
# 10x Stress is Effectively Less Due to Plastic Strain



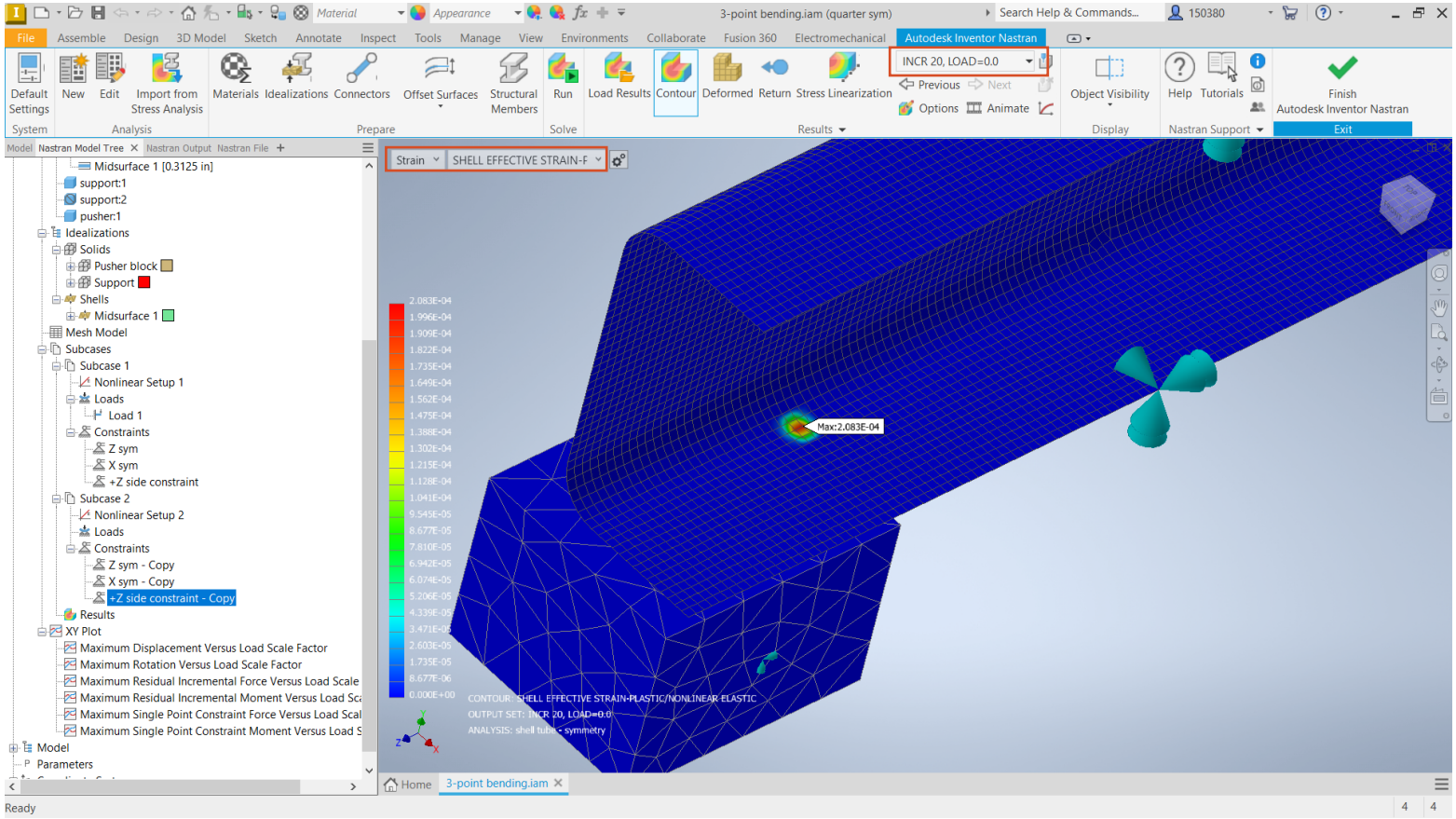
# Increase Load Further to See More Plastic Strain



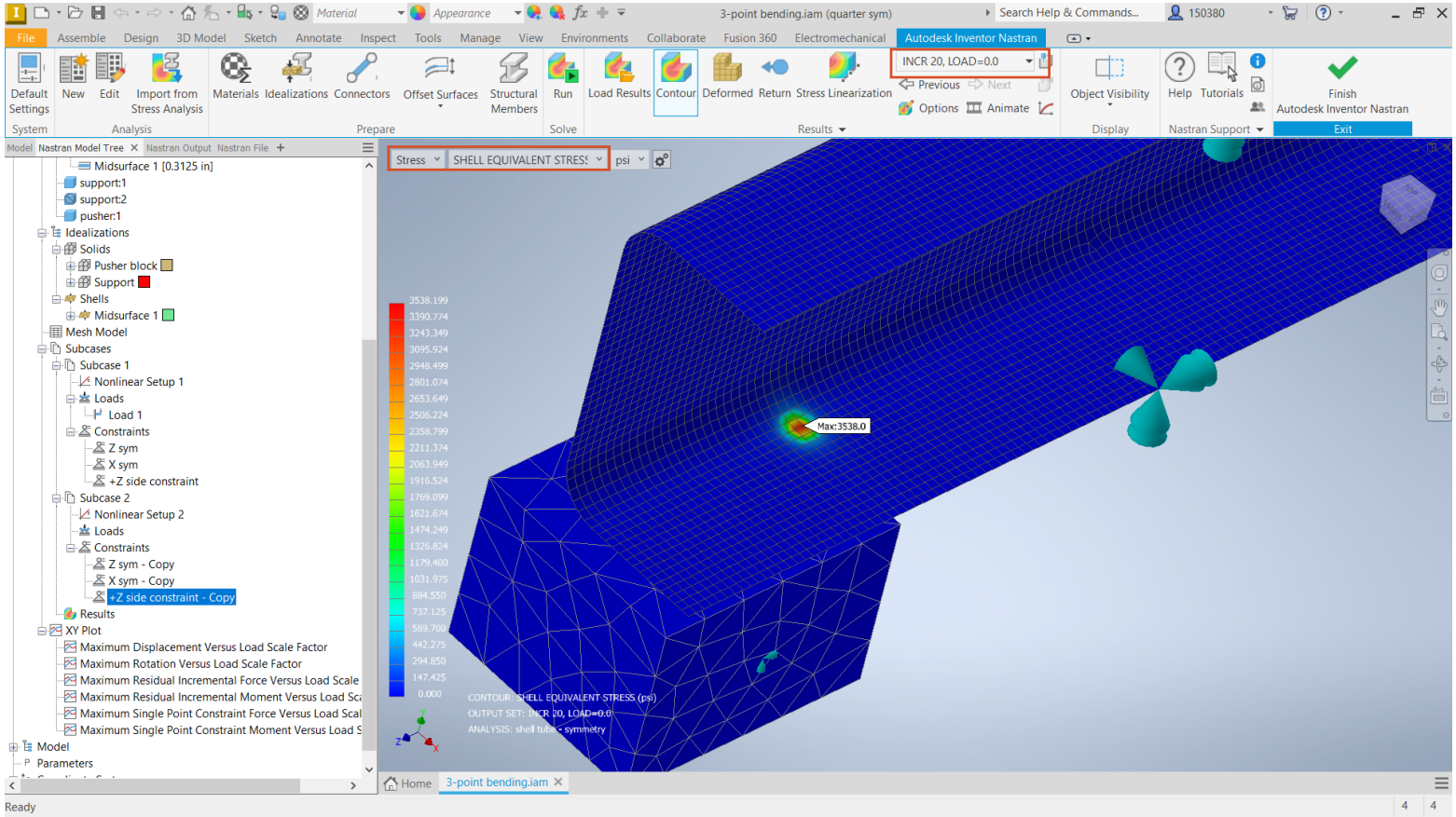
# Duplicate Subcase 1 and Delete Load to See Unload



# Select the Last Load Case to See Residual Effective Strain



# Select the Last Load Case to See Residual Effective Stress



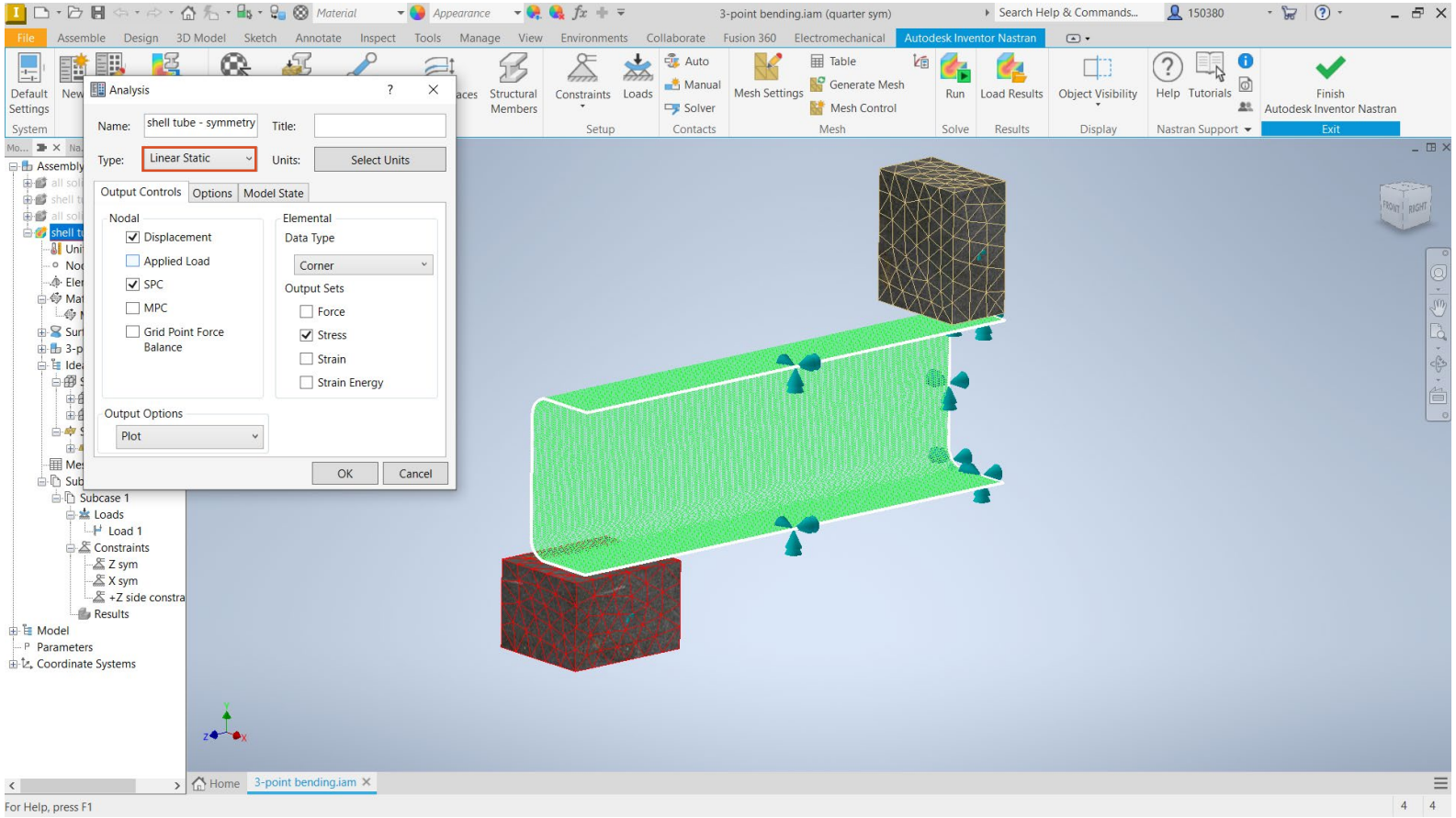
# 3-Point Tube Bending Example

## Shell Element and Solid Element Topology Optimization and Light-Weighting

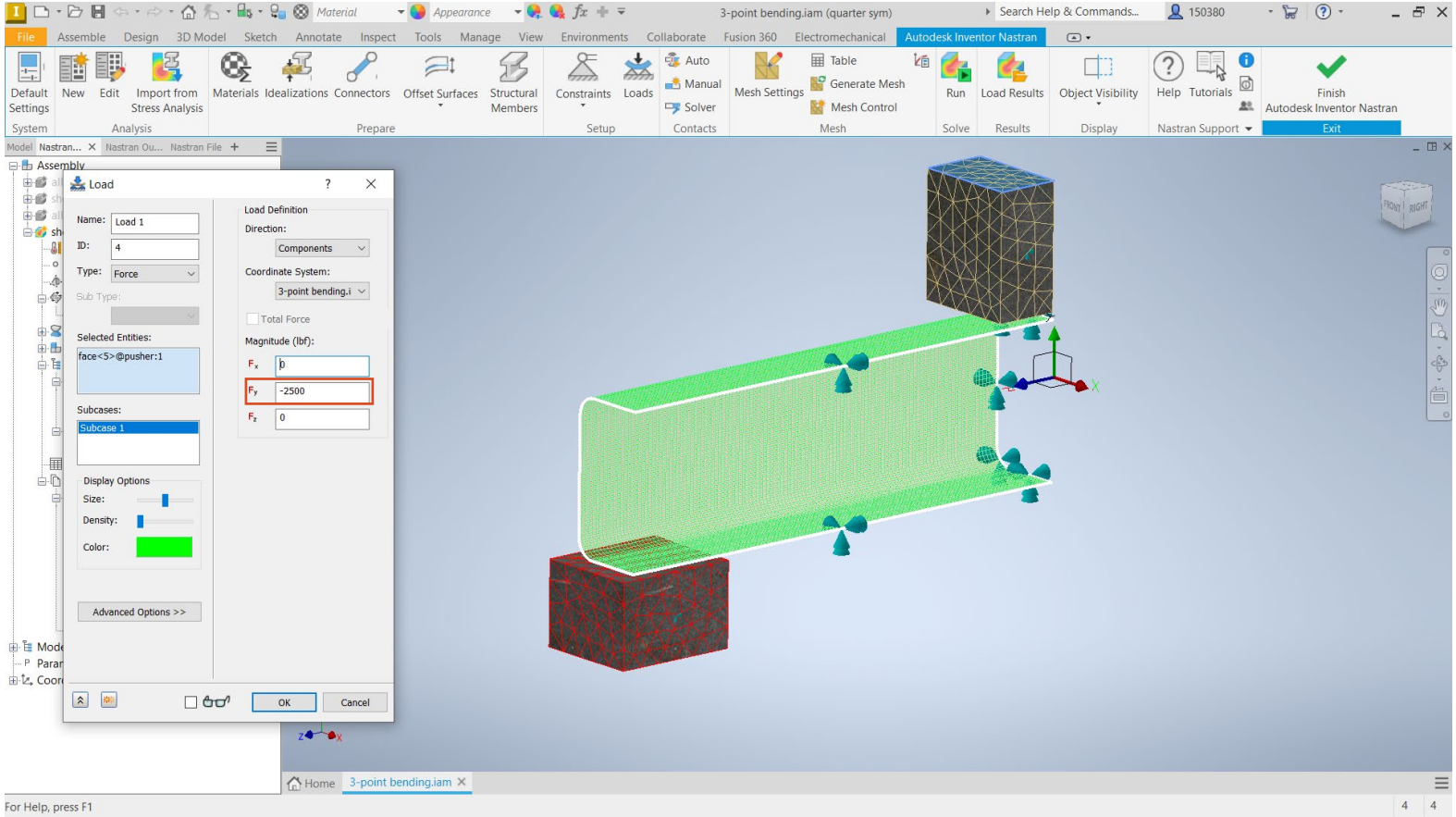
# Definitions

- **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 and 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)

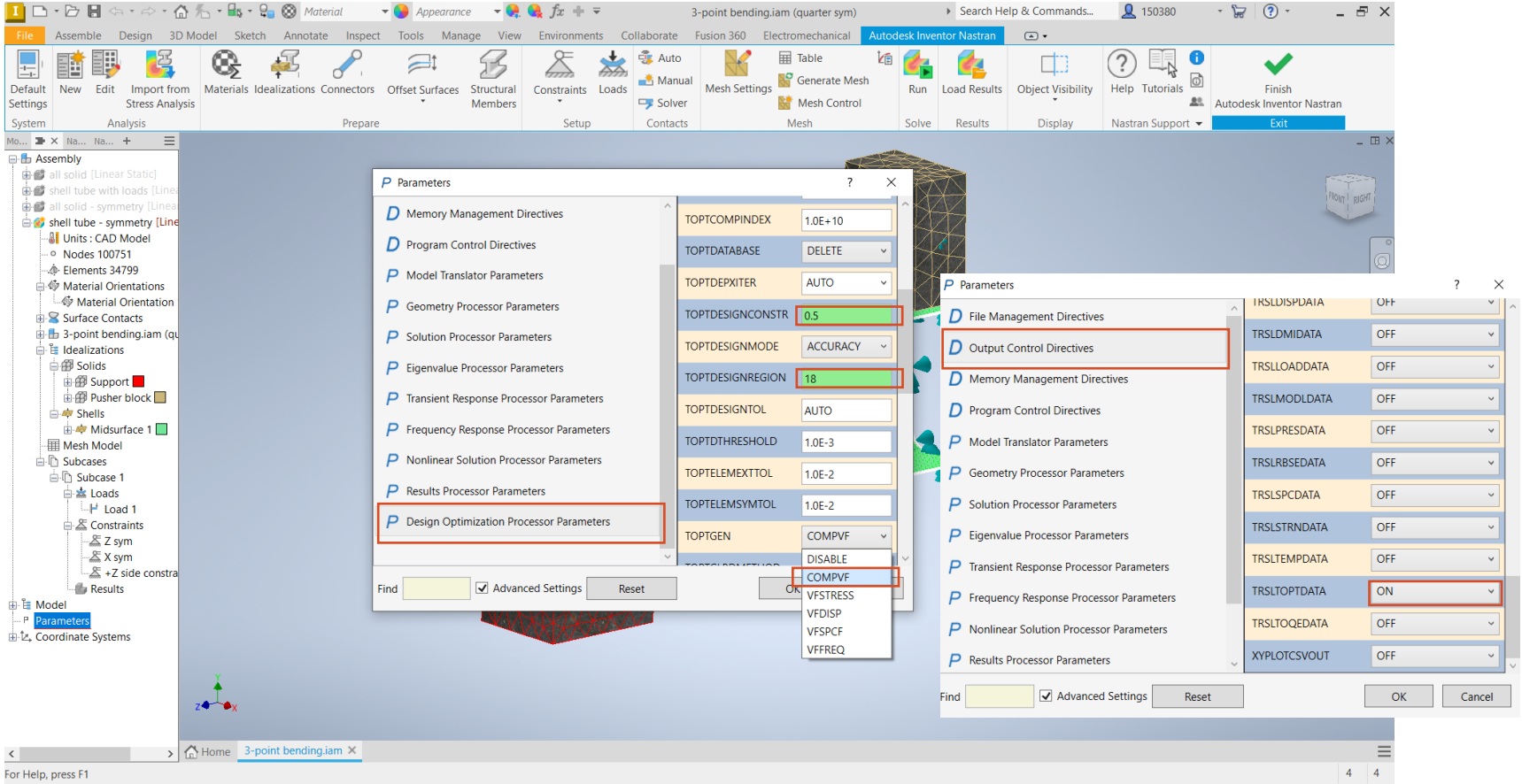
# Change Solution Type Back to Linear Statics



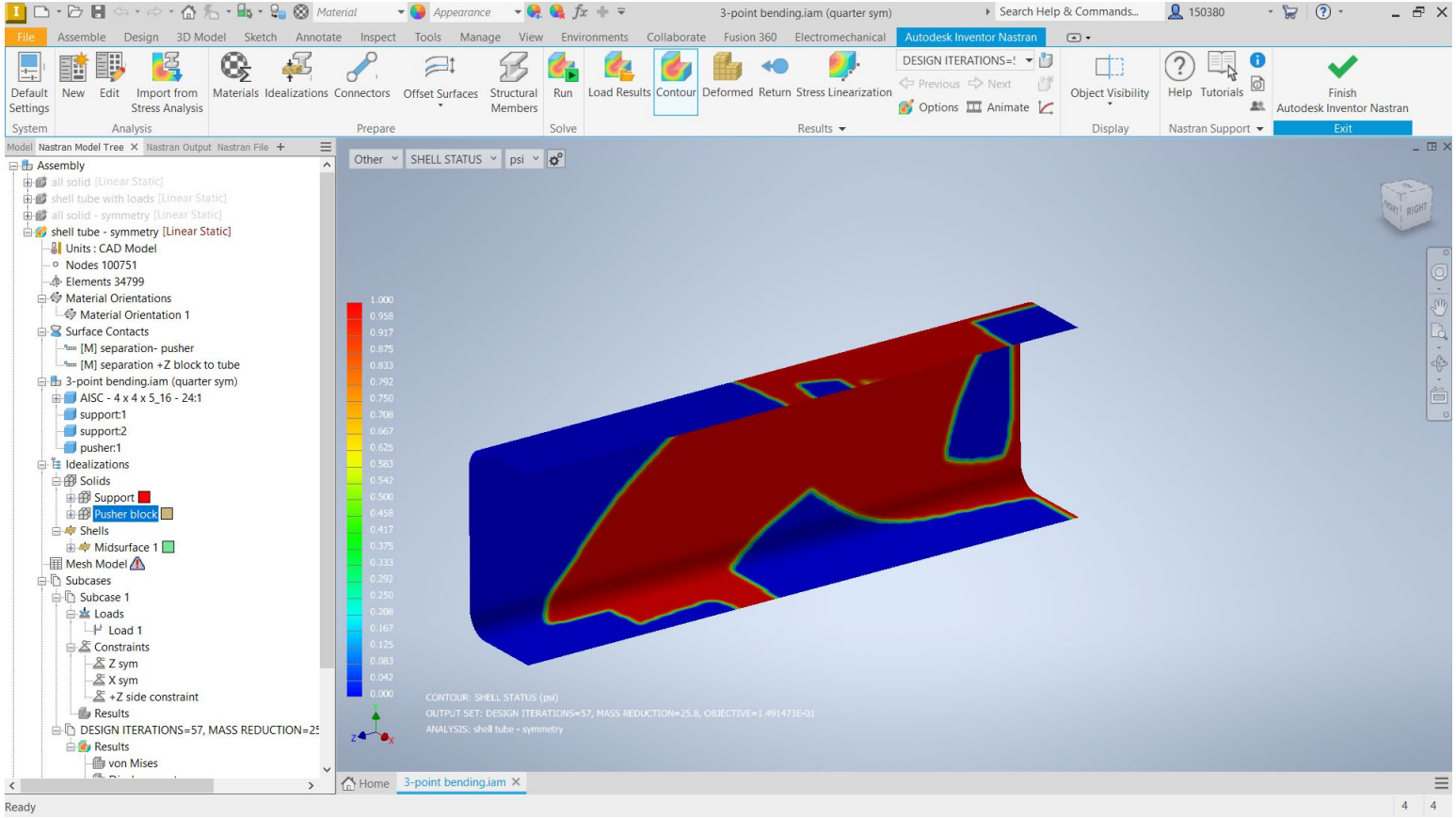
# Change Solution Type Back to Linear Statics



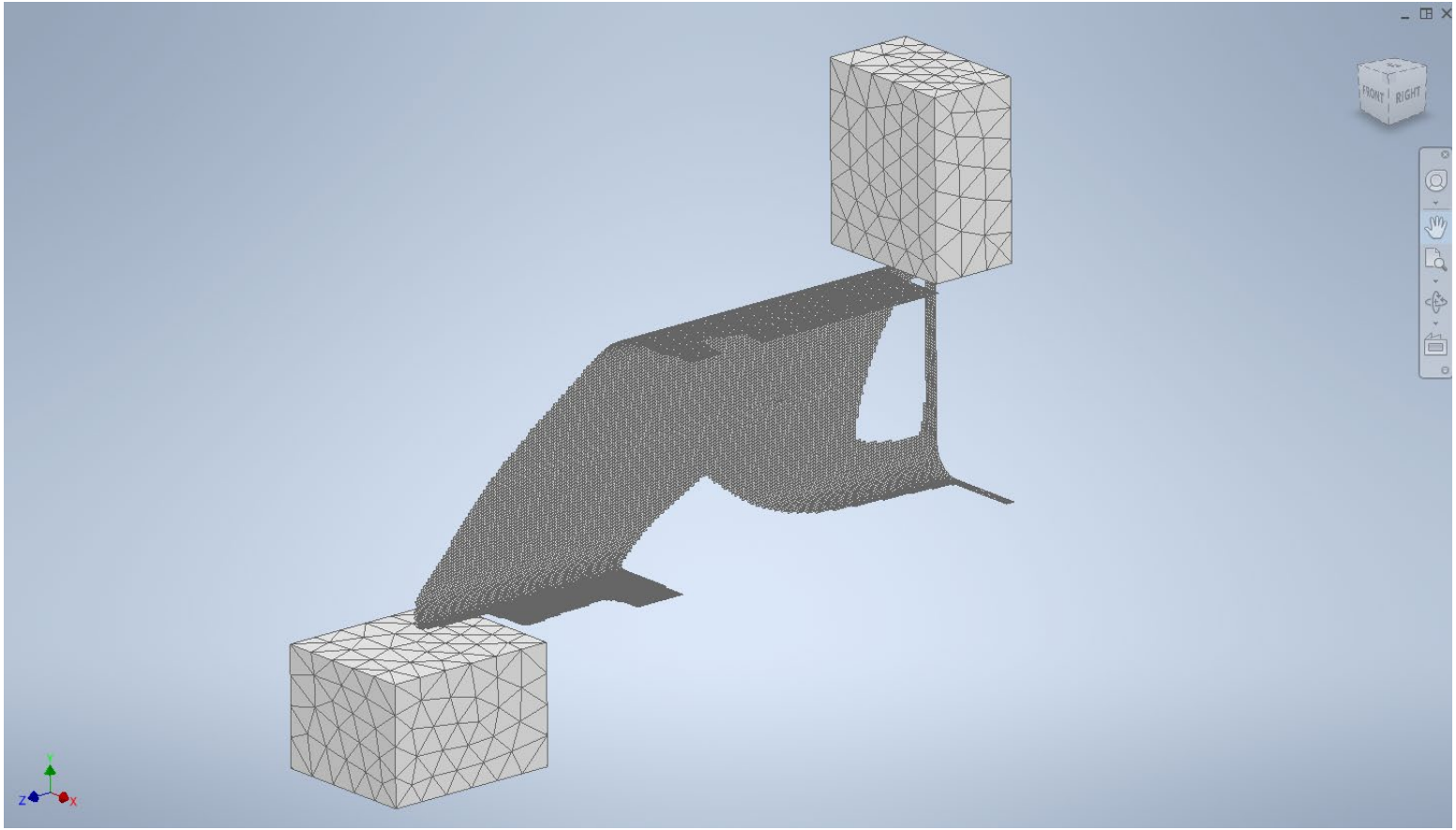
# Enable Topology Optimization Using the TOPGEN Parameter



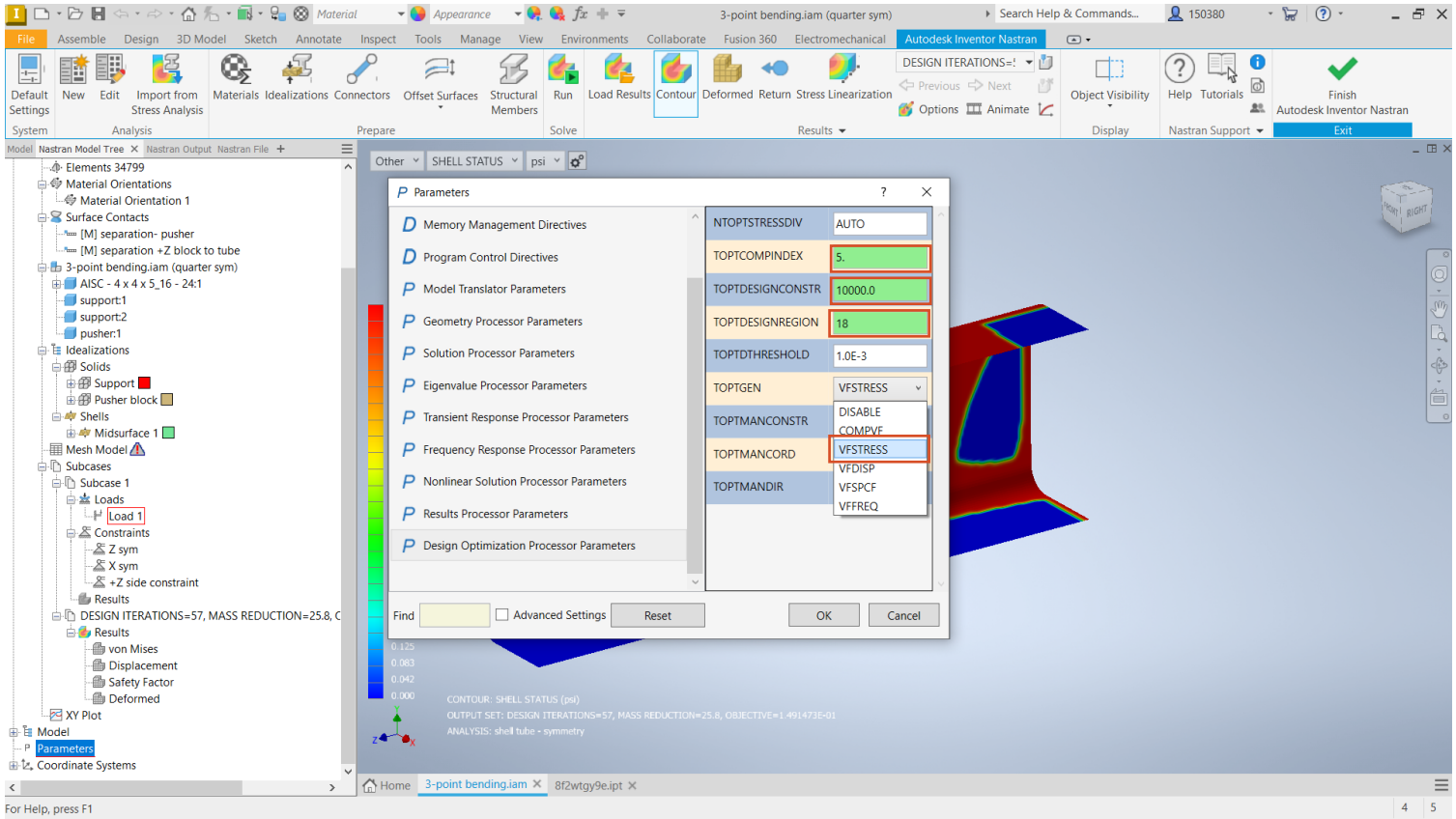
# Min Compliance with a Fixed VF Constraint

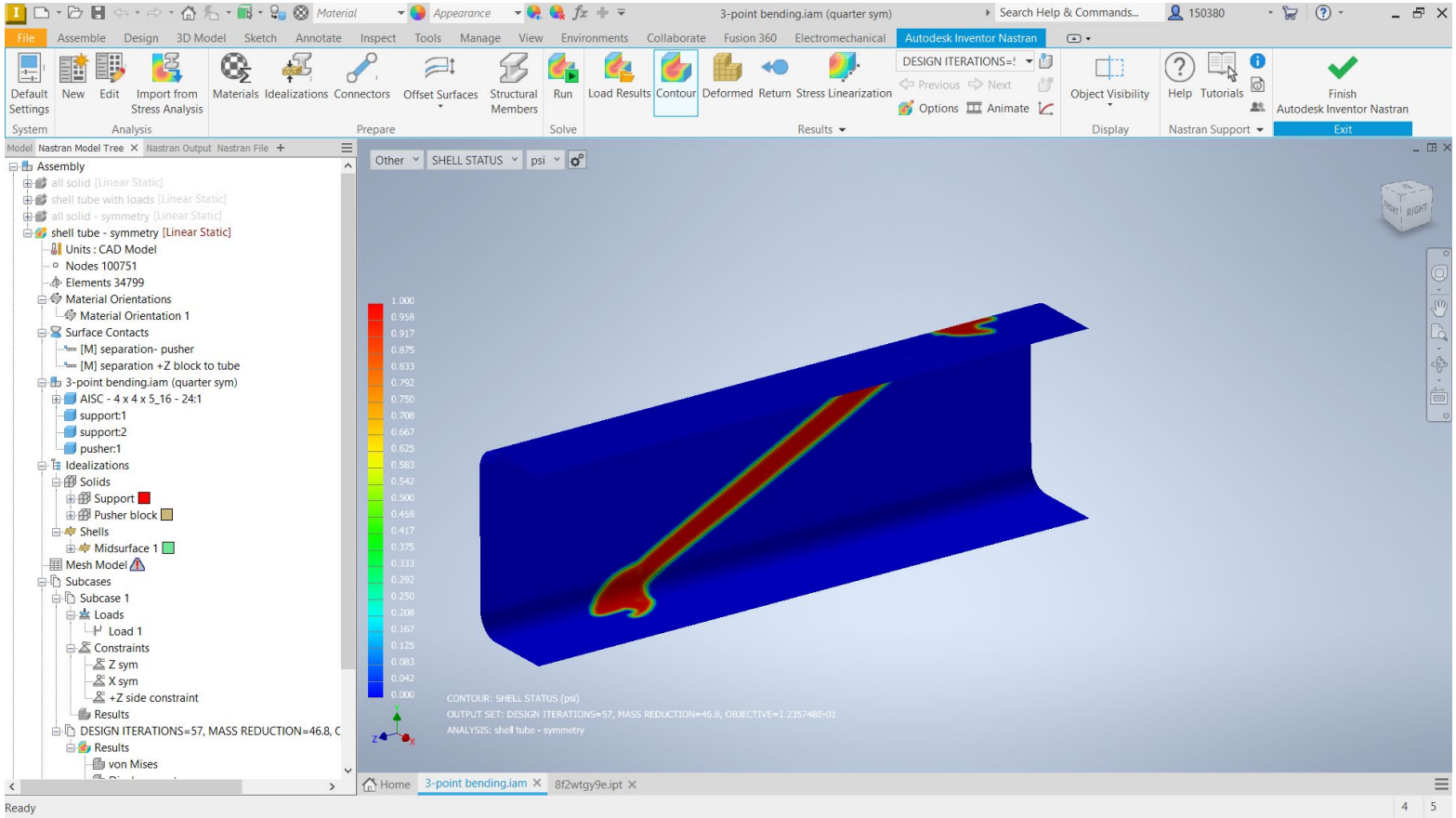


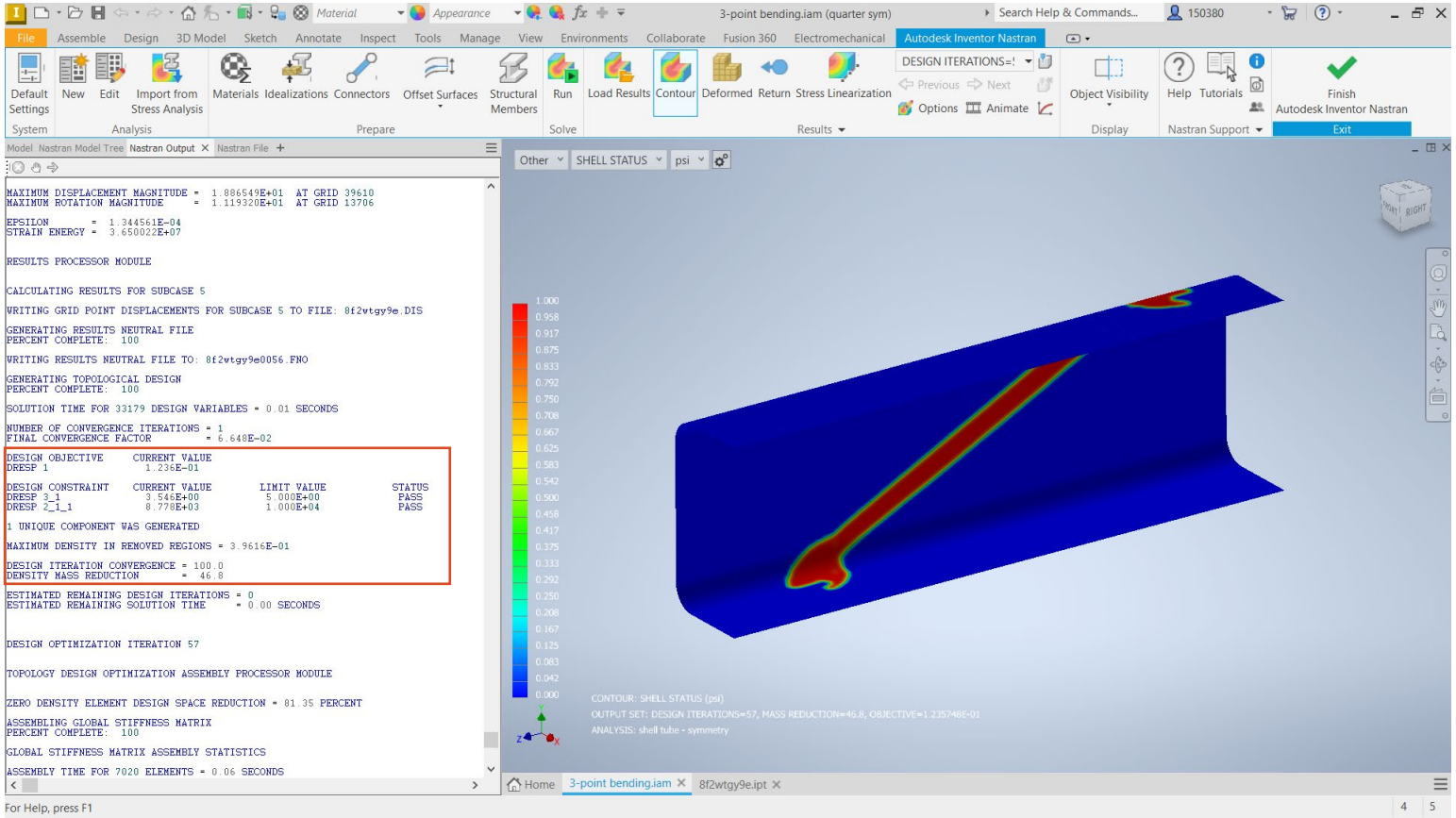
# Generated .STL File



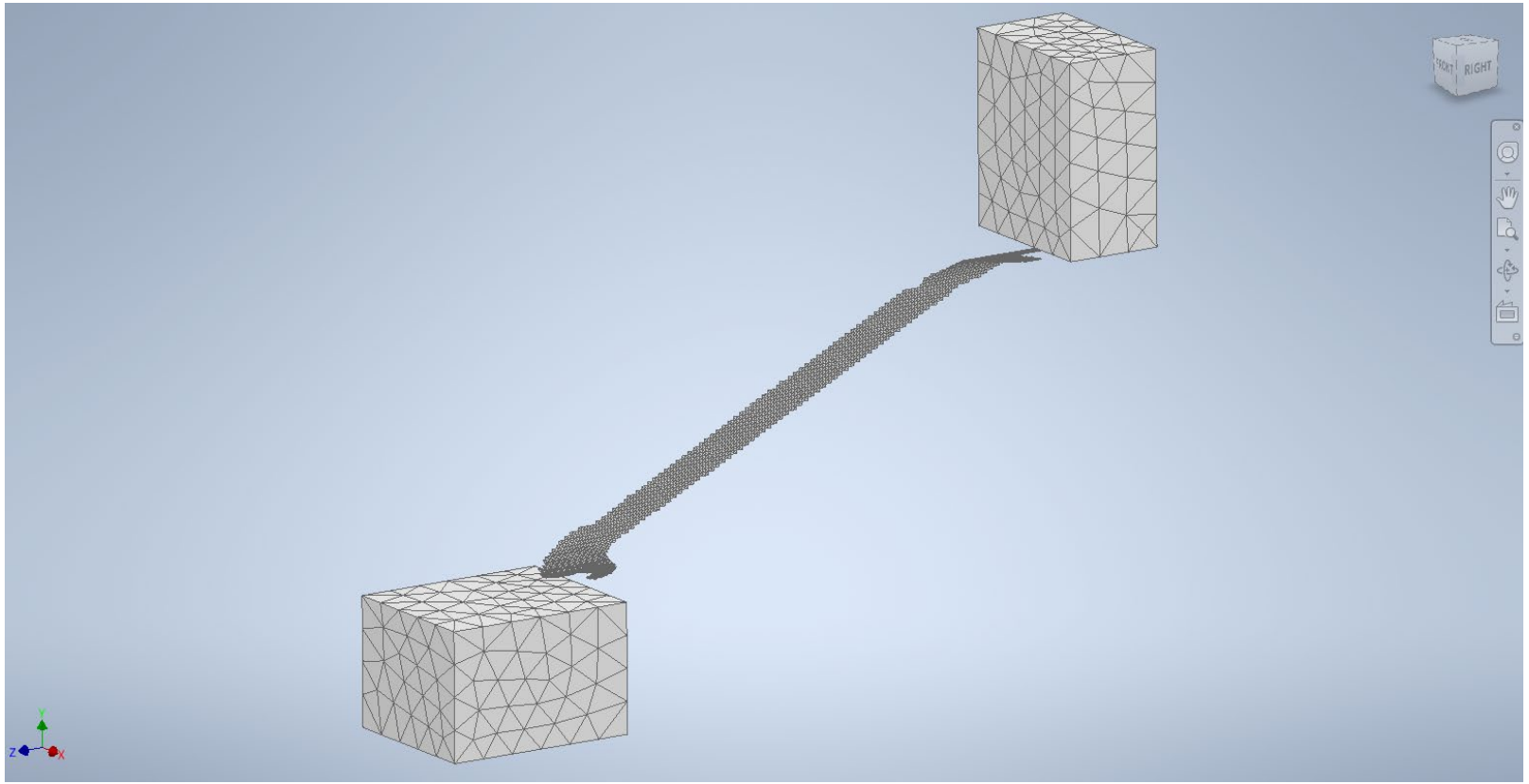
# Min Mass with a Stress and Compliance Index Constraint

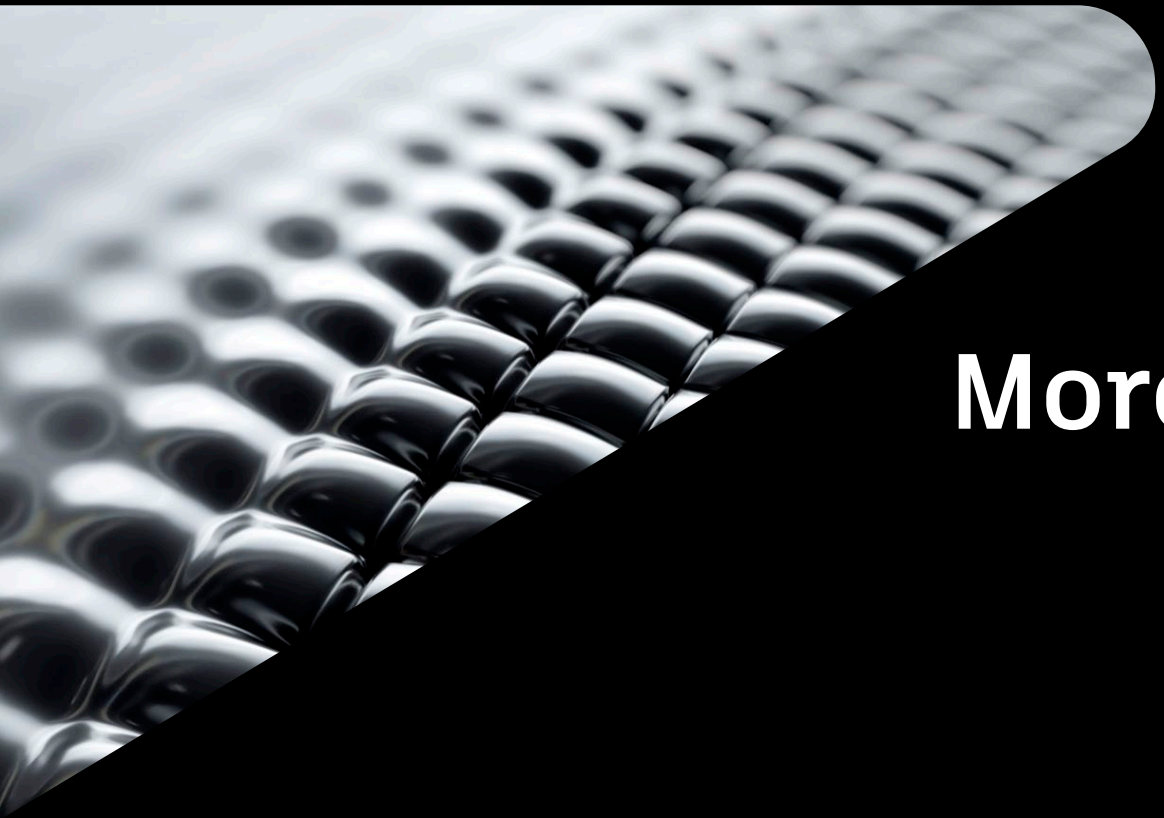



3-point bending.iam (quarter sym)
Search Help & Commands...
File
Assemble
Design
3D Model
Sketch
Annotate
Inspect
Tools
Manage
View
Environments
Collaborate
Fusion 360
Electromechanical
Autodesk Inventor Nastran
Default Settings
New
Edit
Import from Stress Analysis
Materials
Idealizations
Connectors
Offset Surfaces
Structural Members
Run
Load Results
Contour
Deformed
Return
Stress Linearization
DESIGN ITERATIONS=!
Previous
Next
Options
Animate
Object Visibility
Help
Tutorials
Finish Autodesk Inventor Nastran
System
Analysis
Prepare
Solve
Results
Display
Nastran Support
Exit
Model
Nastran Model Tree
Nastran Output
Nastran File
Assembly
all solid [Linear Static]
shell tube with loads [Linear Static]
all solid - symmetry [Linear Static]
shell tube - symmetry [Linear Static]
Units : CAD Model
Nodes 100751
Elements 34799
Material Orientations
Material Orientation 1
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=46.8, C
Results
von Mises
Other
SHELL STATUS
psi
1.000
0.958
0.917
0.875
0.833
0.792
0.750
0.708
0.667
0.625
0.583
0.542
0.500
0.458
0.417
0.375
0.333
0.292
0.250
0.208
0.167
0.125
0.083
0.042
0.000
CONTOUR: SHELL STATUS (psi)
OUTPUT SET: DESIGN ITERATIONS=57, MASS REDUCTION=46.8, OBJECTIVE=1.235748E-01
ANALYSIS: shell tube - symmetry
Home
3-point bending.iam
8f2wtgy9e.ipt
Ready


3-point bending.iam (quarter sym)
Search Help & Commands...
Autodesk Inventor Nastran
Default Settings
New
Edit
Import from Stress Analysis
Materials
Idealizations
Connectors
Offset Surfaces
Structural Members
Run
Load Results
Contour
Deformed
Return
Stress Linearization
DESIGN ITERATIONS=!
Previous
Next
Options
Animate
Object Visibility
Help
Tutorials
Finish Autodesk Inventor Nastran
System
Analysis
Prepare
Solve
Results
Display
Nastran Support
Exit
Model
Nastran Model Tree
Nastran Output
Nastran File
MAXIMUM DISPLACEMENT MAGNITUDE = 1.886549E+01 AT GRID 39610
MAXIMUM ROTATION MAGNITUDE = 1.119320E+01 AT GRID 13706
EPSILON = 1.344561E-04
STRAIN ENERGY = 3.650022E+07
RESULTS PROCESSOR MODULE
CALCULATING RESULTS FOR SUBCASE 5
WRITING GRID POINT DISPLACEMENTS FOR SUBCASE 5 TO FILE: 8f2wtgy9e.DIS
GENERATING RESULTS NEUTRAL FILE
PERCENT COMPLETE: 100
WRITING RESULTS NEUTRAL FILE TO: 8f2wtgy9e0056.FNO
GENERATING TOPOLOGICAL DESIGN
PERCENT COMPLETE: 100
SOLUTION TIME FOR 33179 DESIGN VARIABLES = 0.01 SECONDS
NUMBER OF CONVERGENCE ITERATIONS = 1
FINAL CONVERGENCE FACTOR = 6.648E-02
DESIGN OBJECTIVE CURRENT VALUE
DRESP 1 1.236E-01
DESIGN CONSTRAINT CURRENT VALUE LIMIT VALUE STATUS
DRESP 3_1 3.546E+00 5.000E+00 PASS
DRESP 2_1_1 8.770E+03 1.000E+04 PASS
1 UNIQUE COMPONENT WAS GENERATED
MAXIMUM DENSITY IN REMOVED REGIONS = 3.9616E-01
DESIGN ITERATION CONVERGENCE = 100.0
DENSITY MASS REDUCTION = 46.8
ESTIMATED REMAINING DESIGN ITERATIONS = 0
ESTIMATED REMAINING SOLUTION TIME = 0.00 SECONDS
DESIGN OPTIMIZATION ITERATION 57
TOPOLOGY DESIGN OPTIMIZATION ASSEMBLY PROCESSOR MODULE
ZERO DENSITY ELEMENT DESIGN SPACE REDUCTION = 81.35 PERCENT
ASSEMBLING GLOBAL STIFFNESS MATRIX
PERCENT COMPLETE: 100
GLOBAL STIFFNESS MATRIX ASSEMBLY STATISTICS
ASSEMBLY TIME FOR 7020 ELEMENTS = 0.06 SECONDS
Other
SHELL STATUS
psi
1.000
0.958
0.917
0.875
0.833
0.792
0.750
0.708
0.667
0.625
0.583
0.542
0.500
0.458
0.417
0.375
0.333
0.292
0.250
0.208
0.167
0.125
0.083
0.042
0.000
CONTOUR: SHELL STATUS (psi)
OUTPUT SET: DESIGN ITERATIONS=57, MASS REDUCTION=46.8, OBJECTIVE=1.235748E-01
ANALYSIS: shell tube - symmetry
Home
3-point bending.iam
8f2wtgy9e.ipt
For Help, press F1

# Generated .STL File

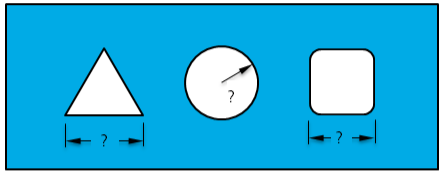




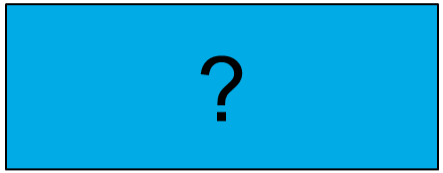
# More On Inventor Optimization

# Topology Optimization

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

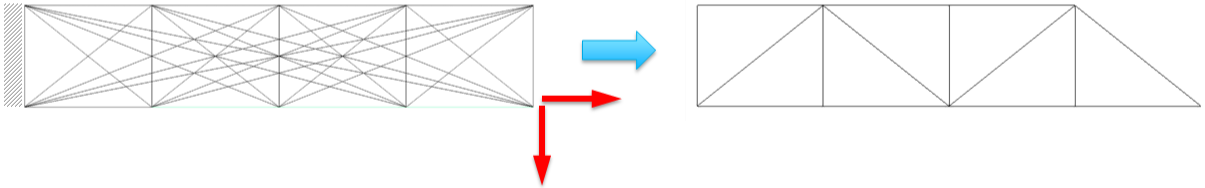



Shape Optimization




Topology Optimization

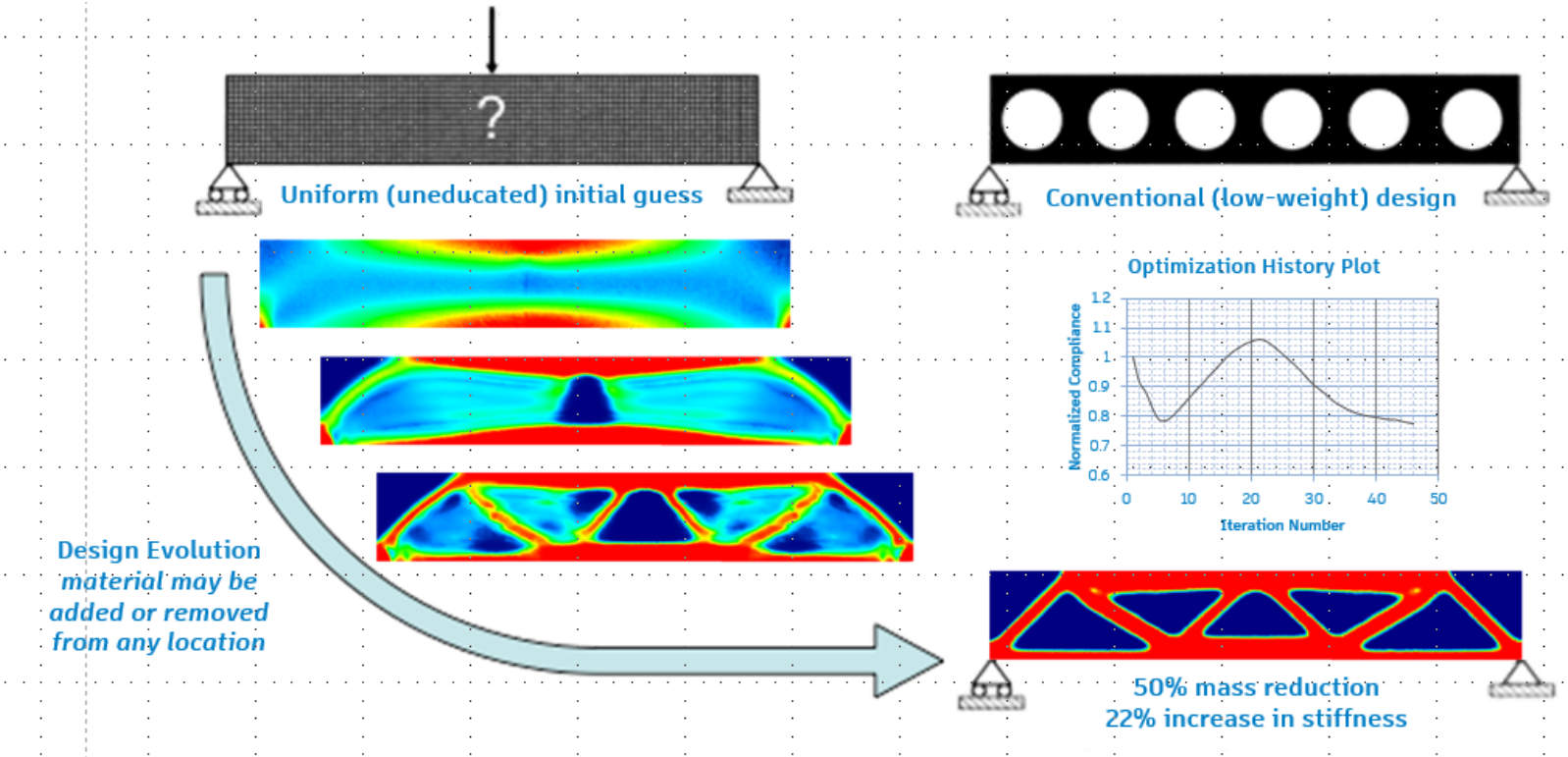
- Method with Finite Element Analysis
  - Initial method…truss



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

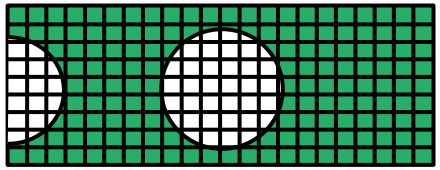
# Topology Optimization with FEA

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



# Topology Optimization Using SIMP - Nastran

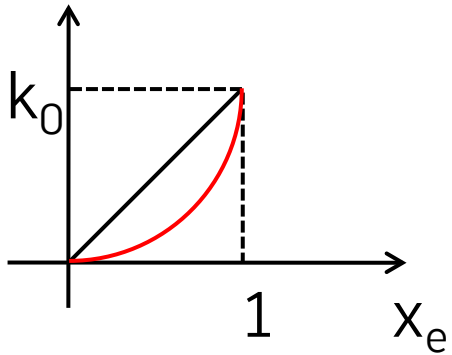
- For fixed mesh, determine density ($x_e$) of each element



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



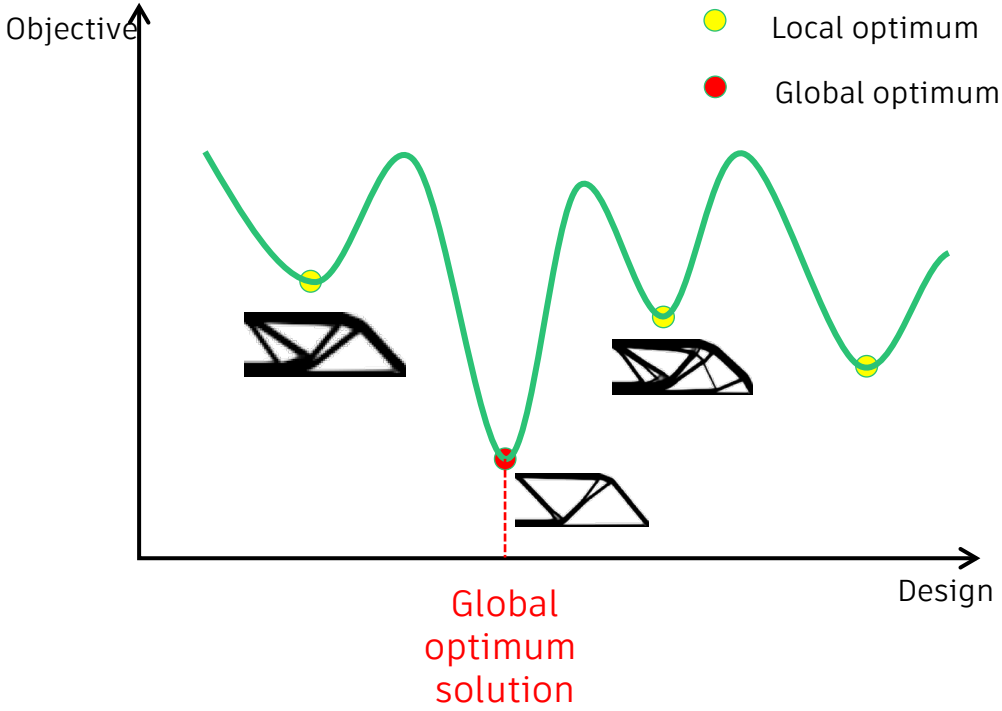

- SIMP = Solid Isotropic Material Penalization

(Not limited to isotropic materials)

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

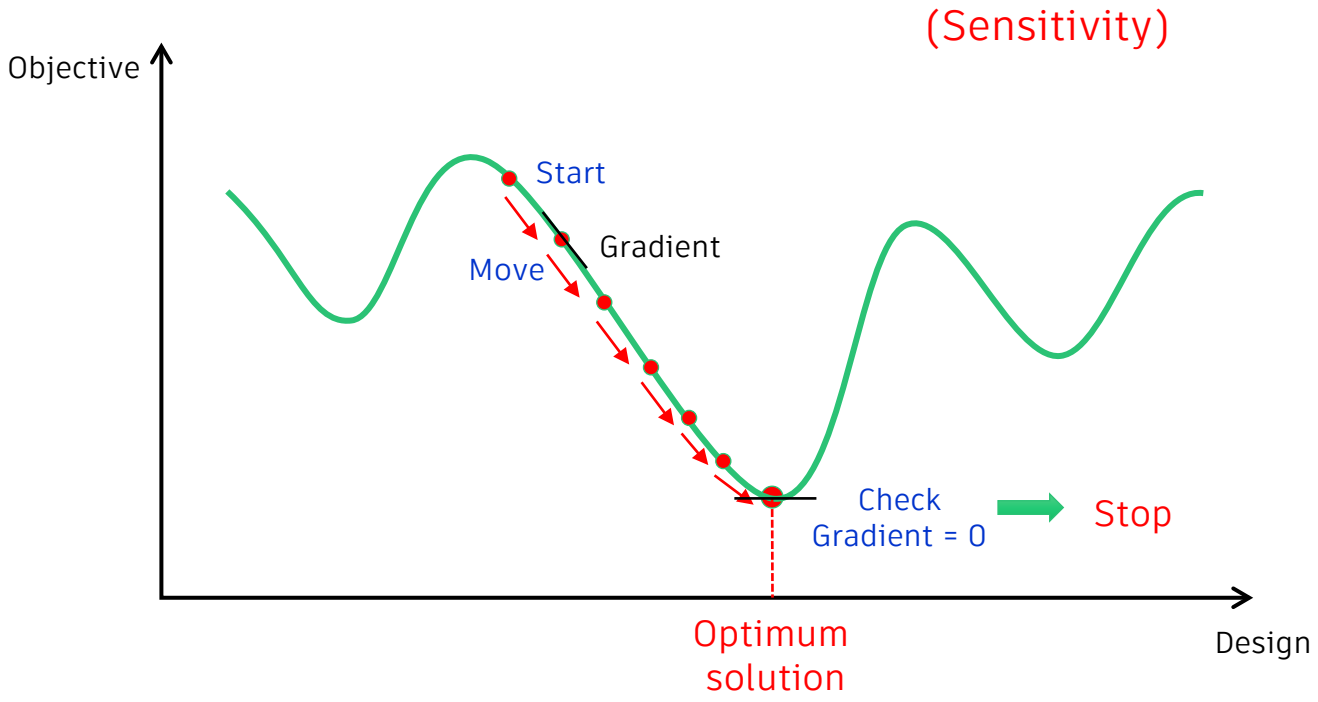
# Global Versus Local Minimum

- 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



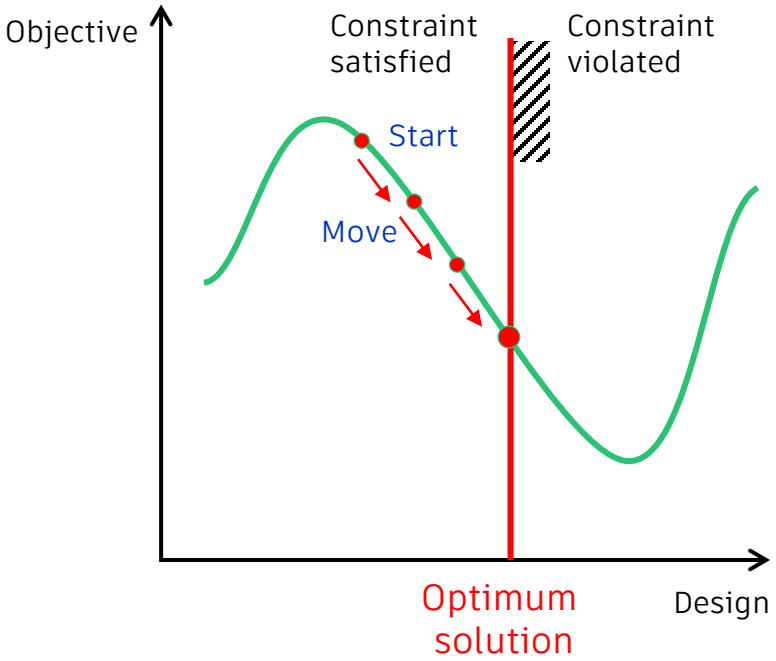
# Gradient-based Methods

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

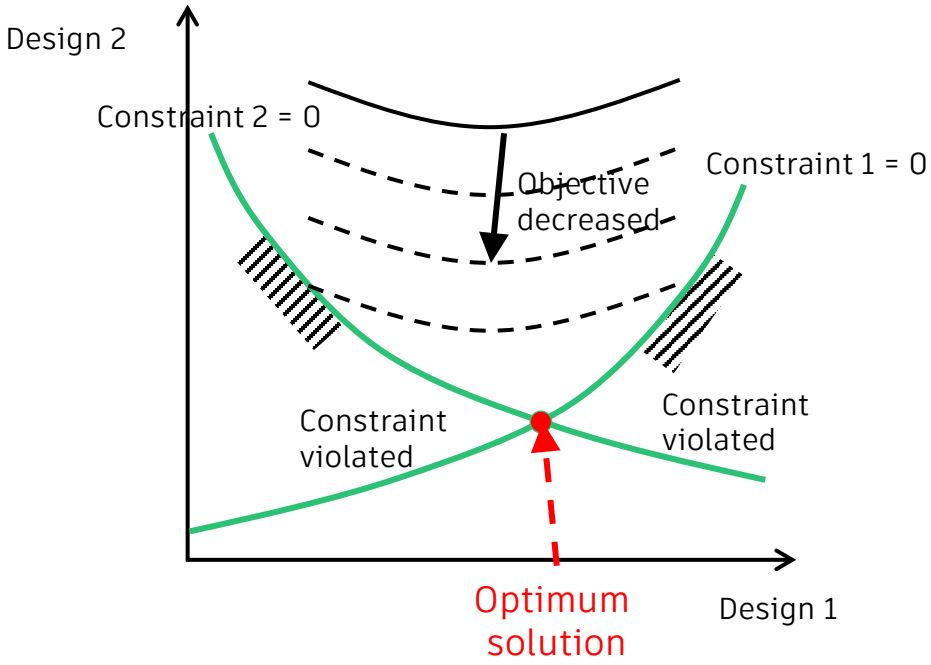


# How Constraints Play in Optimization?

Most cases, constraints determine optimal design




Single constraint example




Two constraints example

# Nastran Topology Optimization Objectives

| Objective | Min/Max/Either | 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

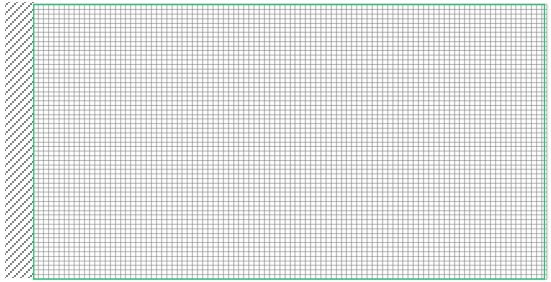
| Design Constraints | Range | Multiple Load Cases | Individual Load Cases | Solution Sequence |
|---|---|---|---|---|
| Compliance | Range | Yes | Yes | LS |
| Compliance Index | < Upper | Yes | Yes | LS |
| Max Displacement Component in Model | < Upper | Yes | Yes | LS |
| Specific Grid Point Displacement 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

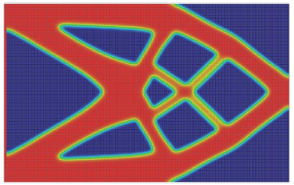
# Nastran Topology Optimization Manufacturing Constraints

| 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 |

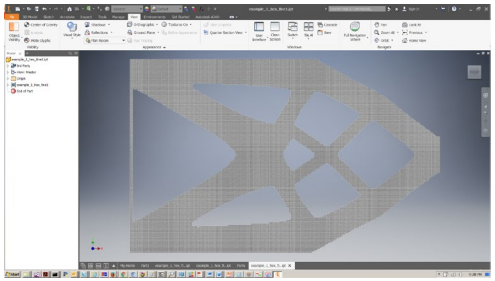
# Minimum Member Size Manufacturing Constraint

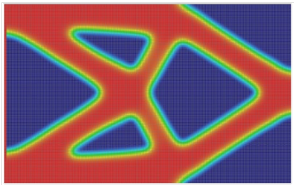


- Fixed at one end and edge loaded at the other end
- Objective is minimize mass
- Constraint is maximum vertical displacement at loaded edge
- Manufacturing constraint: minimum member size (prevents non-designable feature generation)

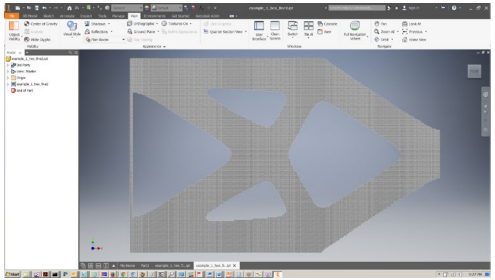


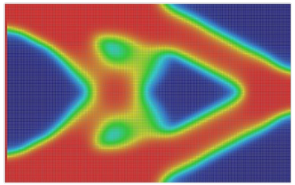
Min member size = 2.0
50.6% mass reduction



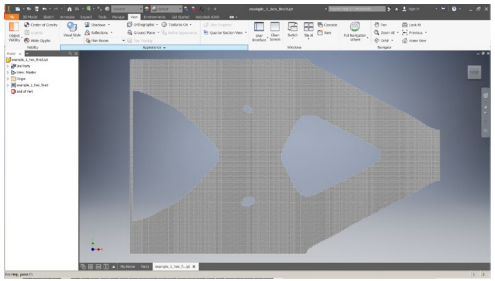


Min member size = 4.0
47.2% mass reduction

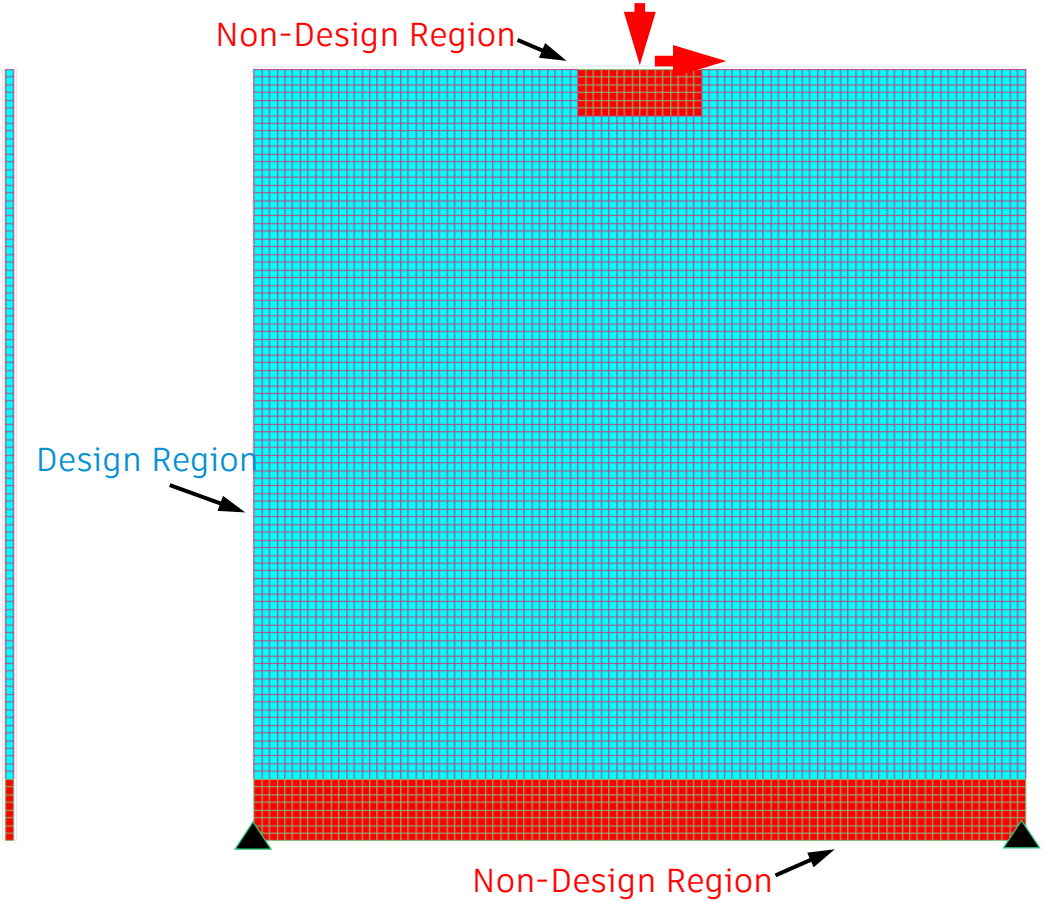




Min member size = 6.0
40.4% mass reduction

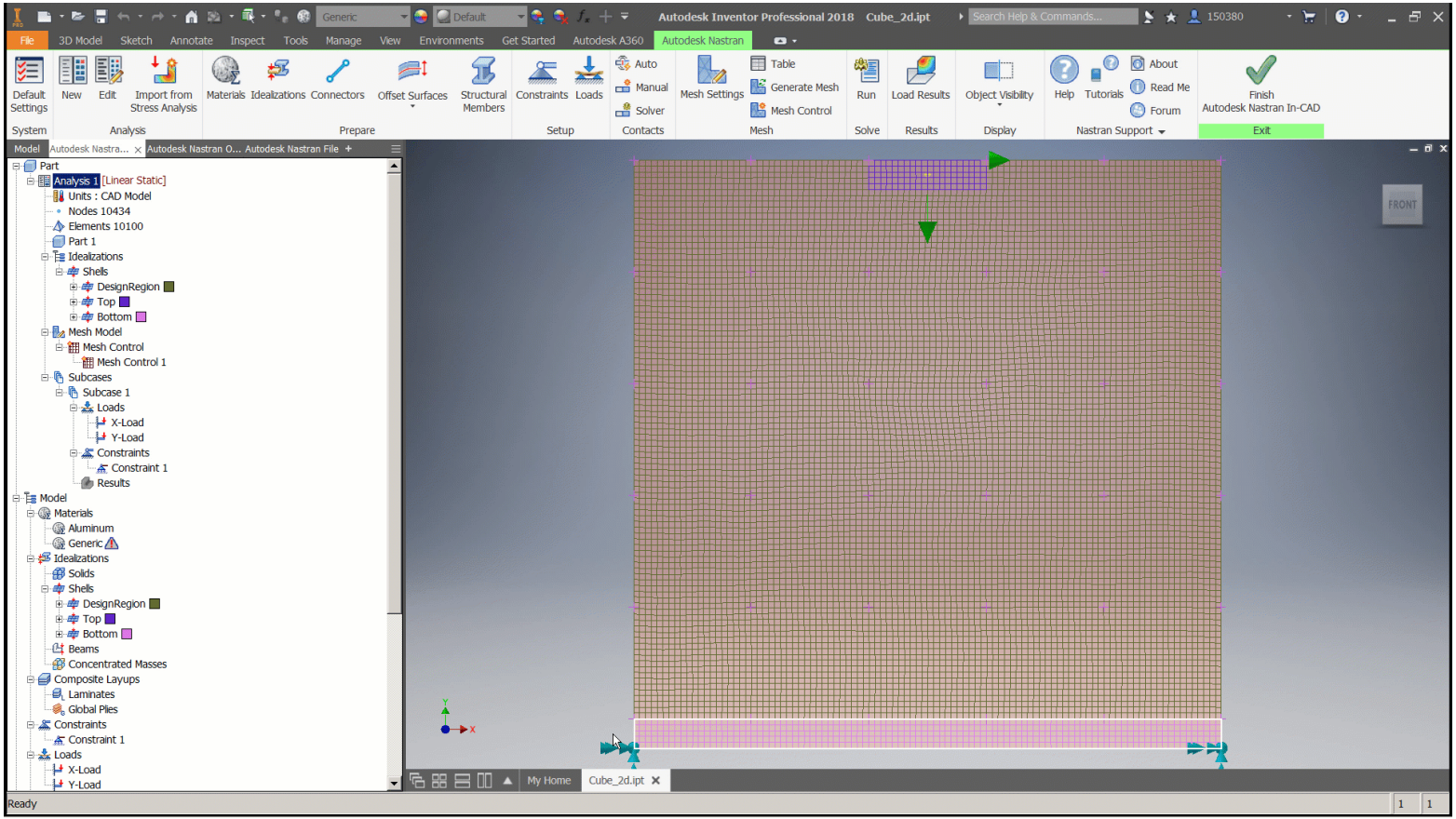


# Example #1 Model Definition

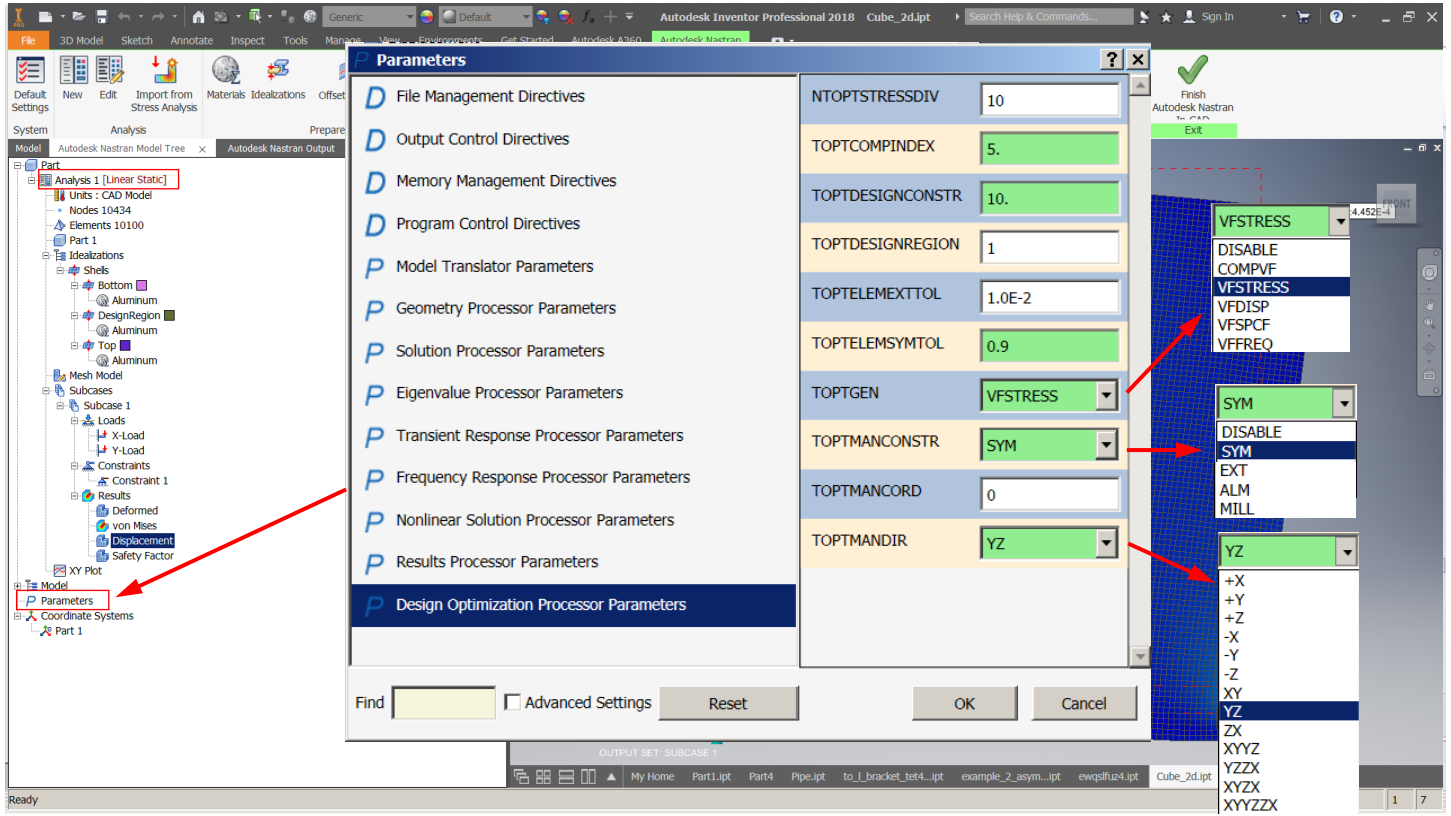



- **Boundary Condition:** Fixed at bottom corners
- **Loading:**
  - Point load in vertical and shear directions
- **Design constraints:**
  - Desired volume fraction
  - Stress limit
  - Displacement limit
  - Lowest frequency
- **Objectives:**
  - Minimize compliance
  - Minimize mass/volume
- **Manufacturing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

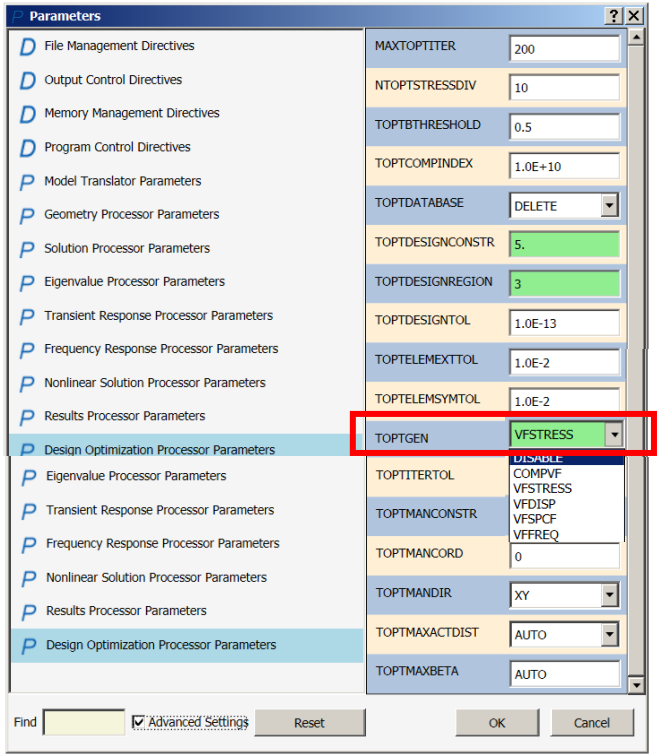
# Topology Optimization Example #1



# Topology Optimization Example #1

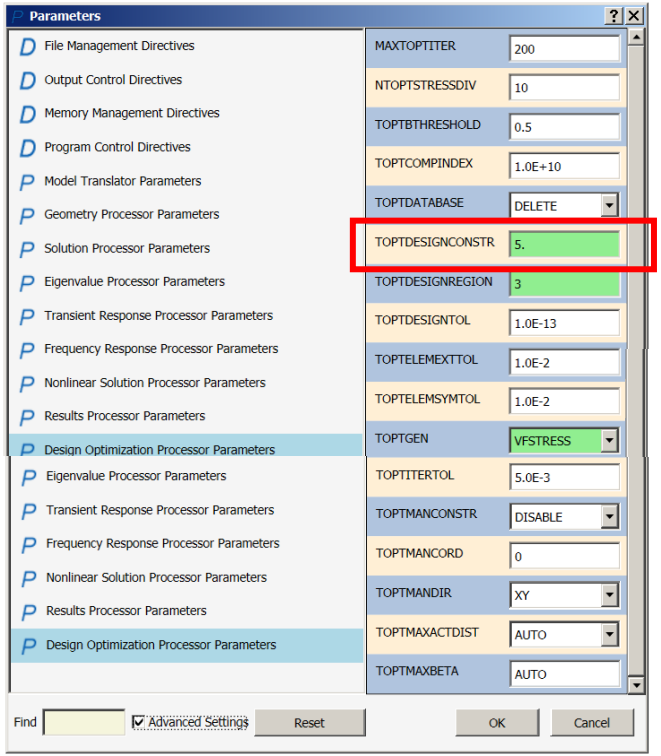


# Inventor Nastran Optimization Parameters - TOPTGEN




| 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 |

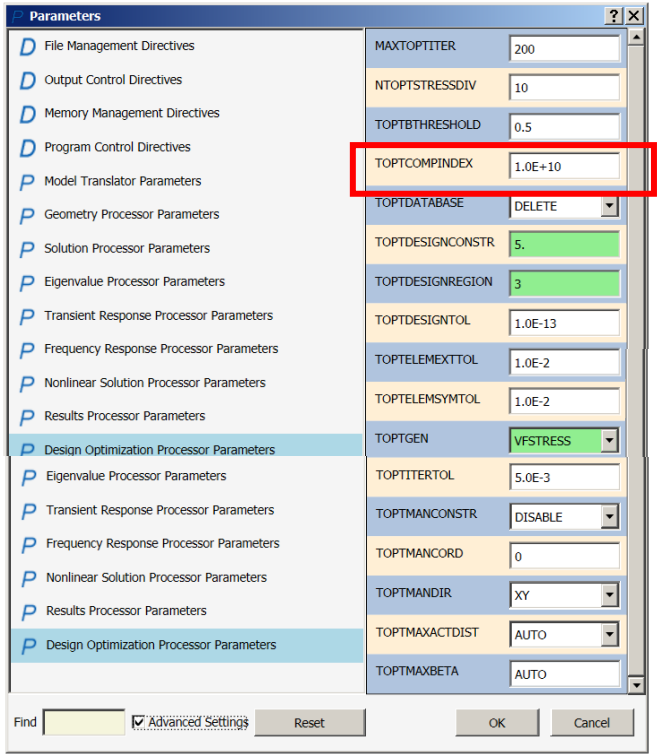
# Inventor Nastran Optimization Parameters - TOPTDESIGNCONSTR



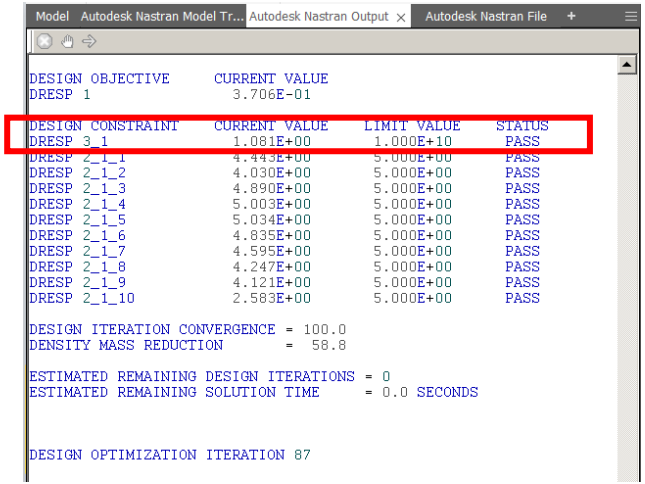

| Description | Type | Default |
|---|---|---|
| Topology design optimization design constraint value based on the TOPTGEN setting. See TOPTGEN. | Real > 0.0 | 1.0E+10 |

| 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

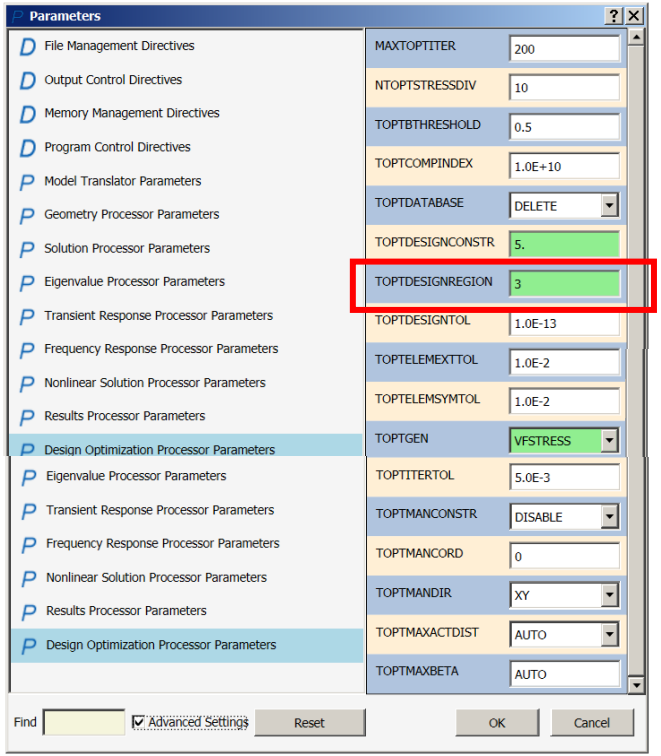


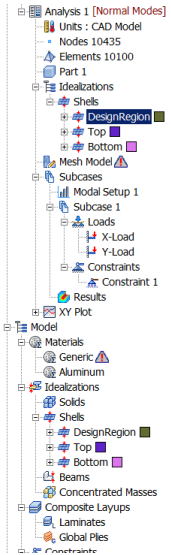

| Description | Type | Default |
|---|---|---|
| Topology design optimization compliance index design constraint value. Applicable only when TOPTGEN is set to 2, 3, or 4. | Real > 1.0 | 1.0E+10 |

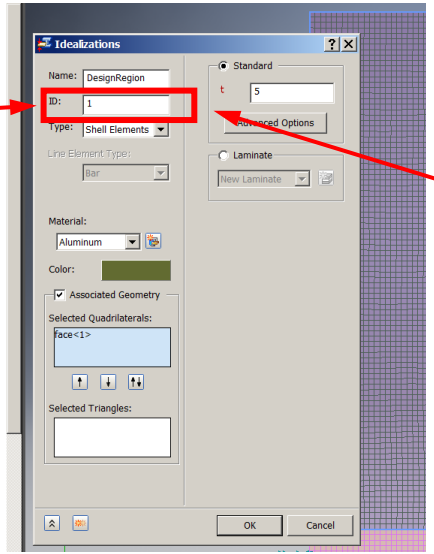


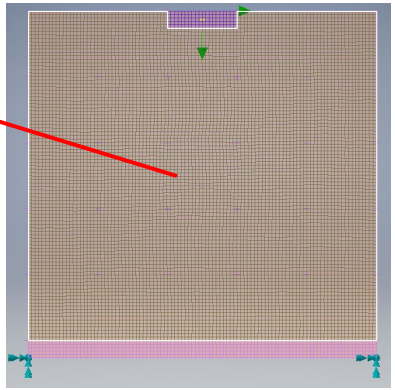
# Inventor Nastran Optimization Parameters - TOPTDESIGNREGION

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



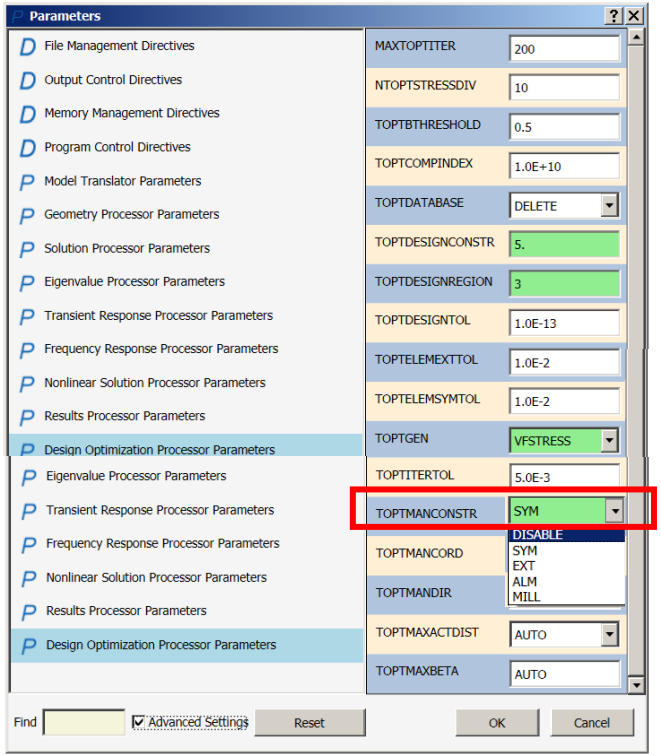



- Note: Specifying the wrong ID may result in an 2299 or 5125 fatal error

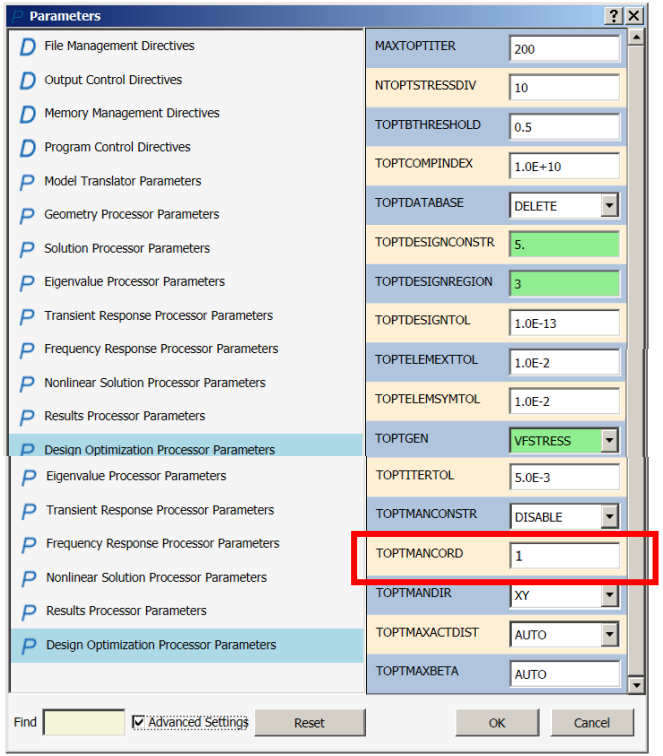
# Inventor Nastran Optimization Parameters - TOPTMANCONSTR

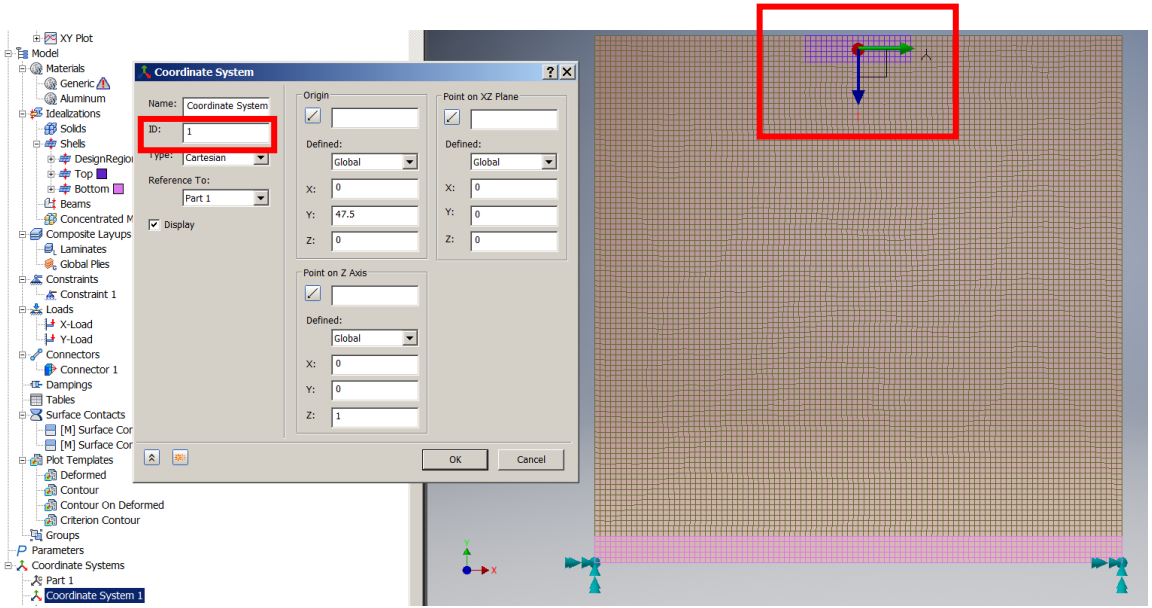



| Description | Type | Default |
|---|---|---|
| Defines the type of manufacturing constraint to be used in Automated Topology Optimization Generation (ATOG). There are three options:<br><br>0 - No manufacturing constraint is specified.<br><br>1 - Symmetry using either 1, 2, or 3 planes of symmetry is specified.<br><br>2 - Extrude design constraint is specified.<br><br>3 - Additive layer manufacturing design constraint is specified.<br><br>4 - 3-axis milling manufacturing design constraint is specified.<br><br>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. | 0 ≤ Integer ≤ 3<br><br>DISABLE/<br><br>SYM/<br><br>EXT/<br><br>ALM/<br><br>MILL | 0 |

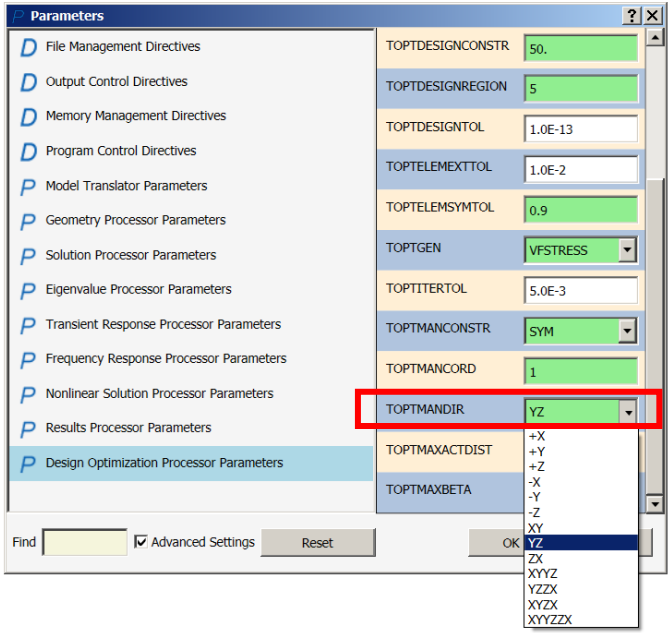
# Inventor Nastran Optimization Parameters - TOPTMANCORD

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



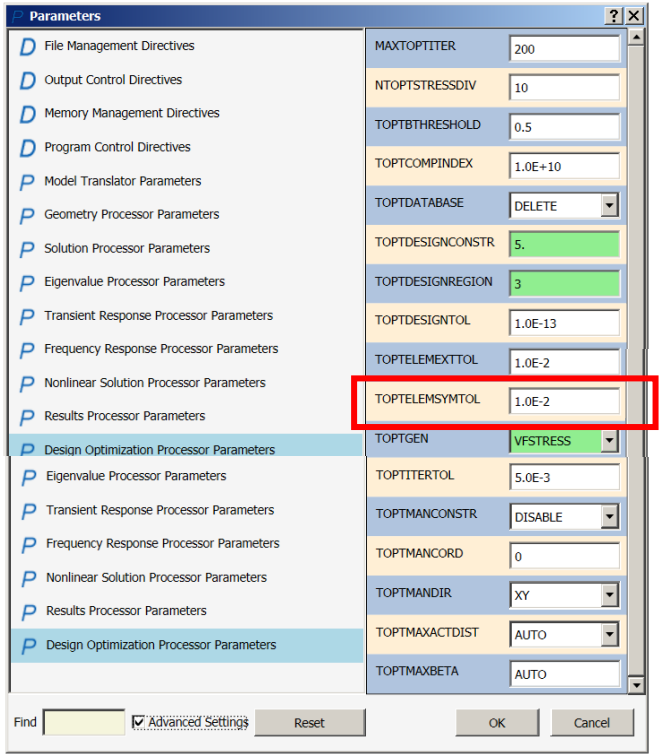
# Inventor Nastran Optimization Parameters - TOPTMANDIR




| 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/<br>-X/-Y/-Z/<br>XY/YZ/ZX/<br>XYYZ/YZZX/<br>XYZX/<br>XYYZZX | XY |

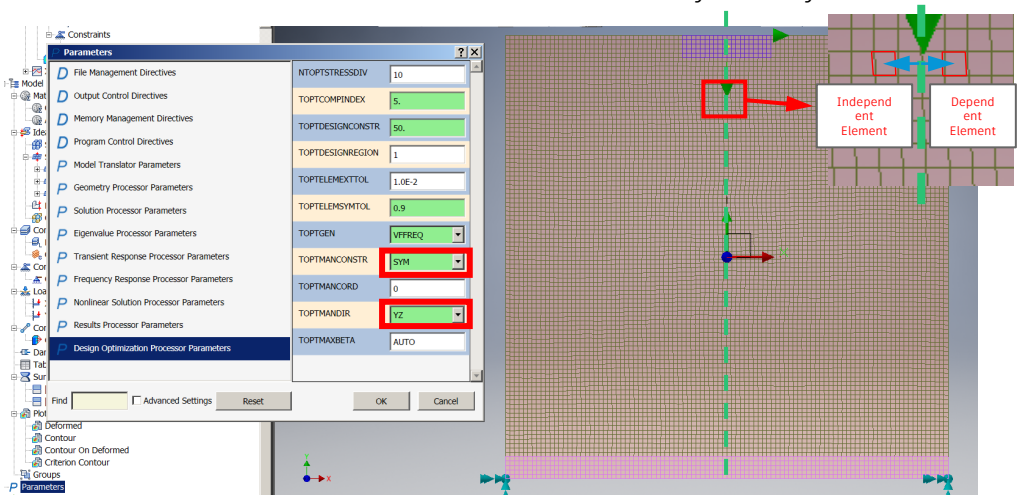
| Keyword | TOPTMANDIR Definition |
|---|---|
| DISABLE | No manufacturing constraints specified |
| SYM | Symmetry plane or planes specified in the TOPTMANCORD system |
| EXT | Extrude direction axis specified in the TOPTMANCORD system |
| ALM | Print direction axis specified in the TOPTMANCORD system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

# Inventor Nastran Optimization Parameters - TOPTELEMSYMTOL



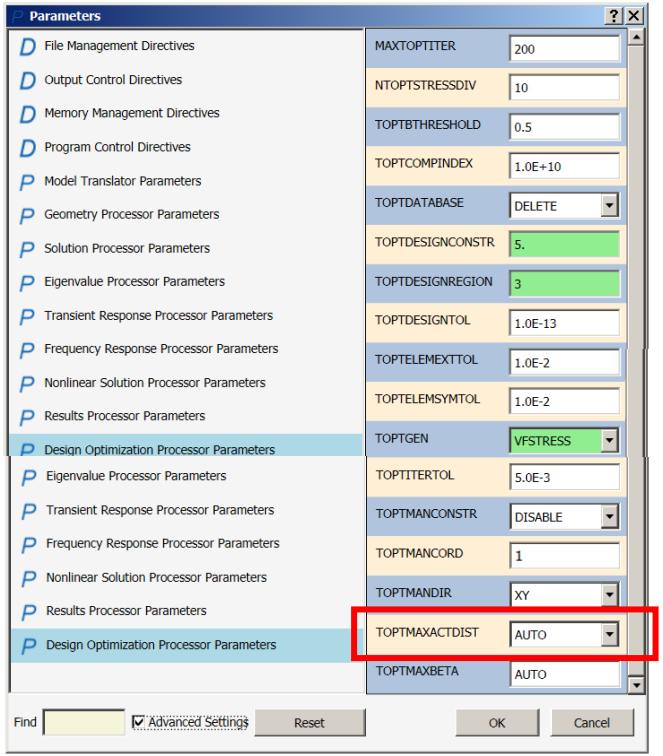

| Description | Type | Default |
|---|---|---|
| Near tolerance used to identify elements which are symmetric with respect to the specified TOPVAR Bulk Data entry mirror symmetry plane. The actual tolerance is derived using TOPTELEMSYMTOL and an element reference dimension. | Real | 1.0E-2 |

YZ Symmetry



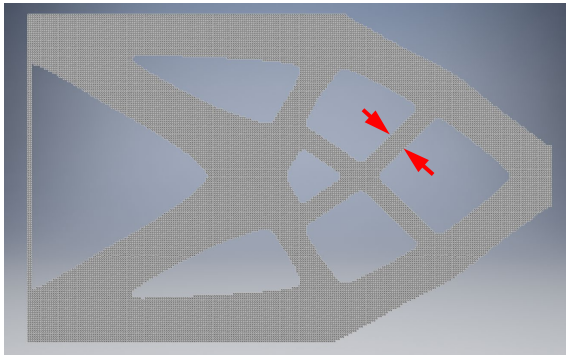
# Inventor Nastran Optimization Parameters - TOPTMAXACTDIST

Use this to specify a minimum member size

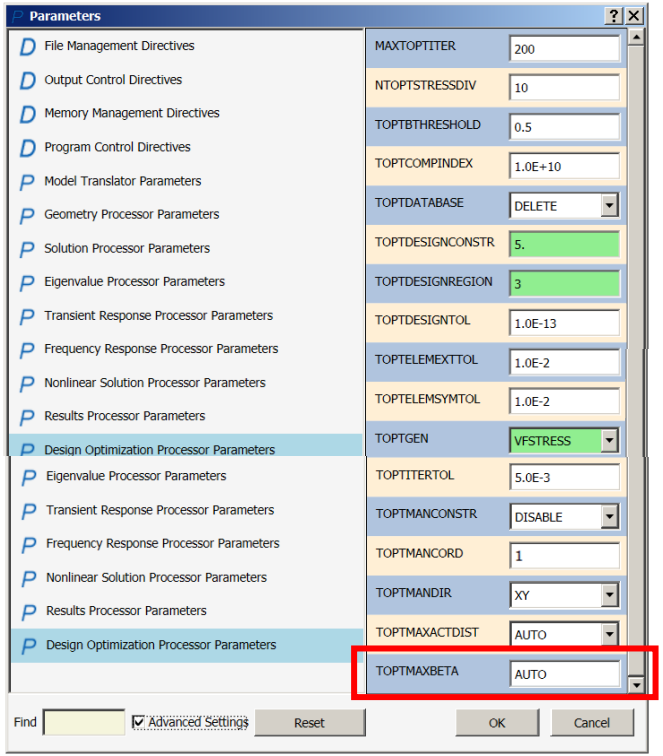



| Description | Type | Default |
|---|---|---|
| Topology design optimization maximum distance for identifying adjacent elements. Elements within distance TOPTMAXACTDIST are used for sensitivity filtering. The default AUTO setting is recommended since large values may result in slower performance and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

- TOPTMAXACTDIST is ½ minimum member size
- To specify a minimum member size of 1 use 0.5.

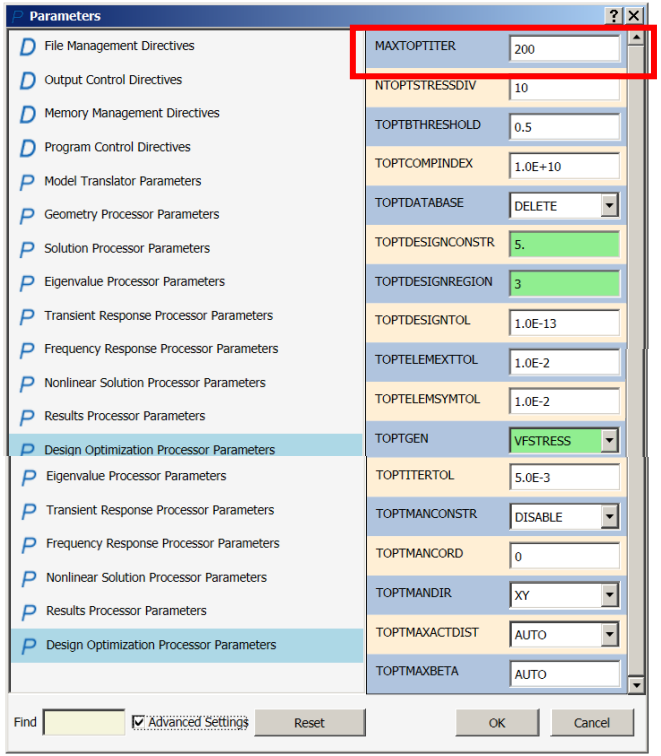


# Inventor Nastran Optimization Parameters - TOPTMAXBETA

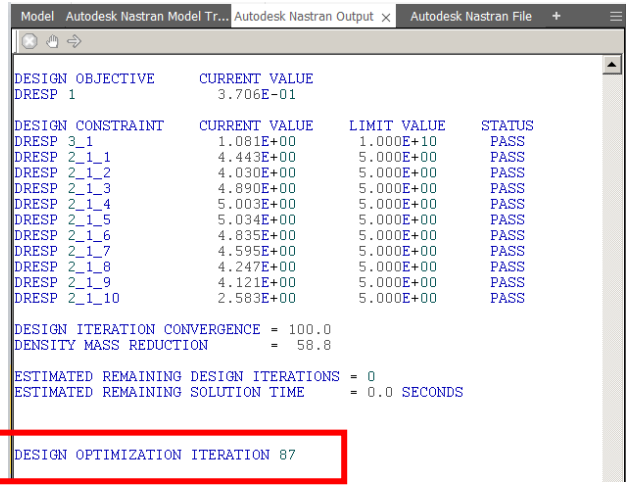



| Description | Type | Default |
|---|---|---|
| Specifies the penalty value for enforcing minimum member size manufacturing constraints. A value between 1.0 and 16.0 is recommended. The default AUTO value selects the best value depending on what other constraints are specified. | Real<br><br>AUTO |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

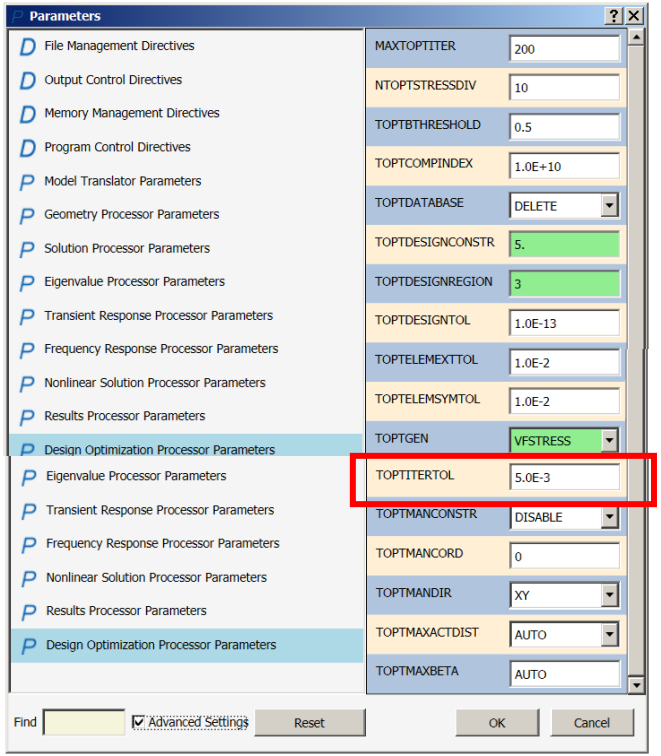
# Inventor Nastran Optimization Parameters - MAXTOPTITER



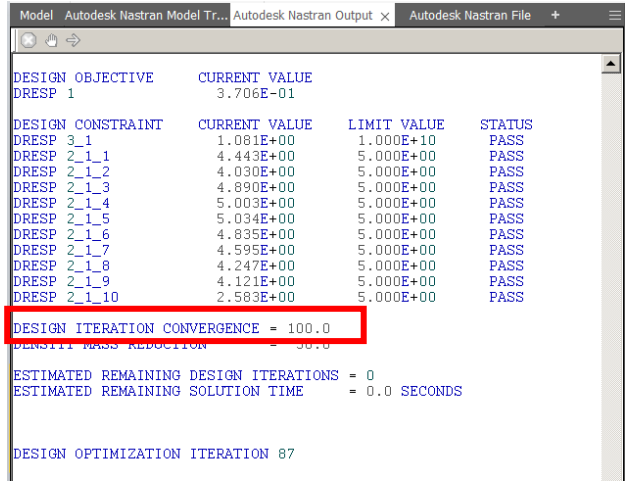

| Description | Type | Default |
|---|---|---|
| Topology design optimization maximum number of convergence iterations permitted. The solver will iterate until the convergence factor set by TOPTITERTOL is reached or MAXOPTITER iterations have been performed. A zero setting will result in iteration until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



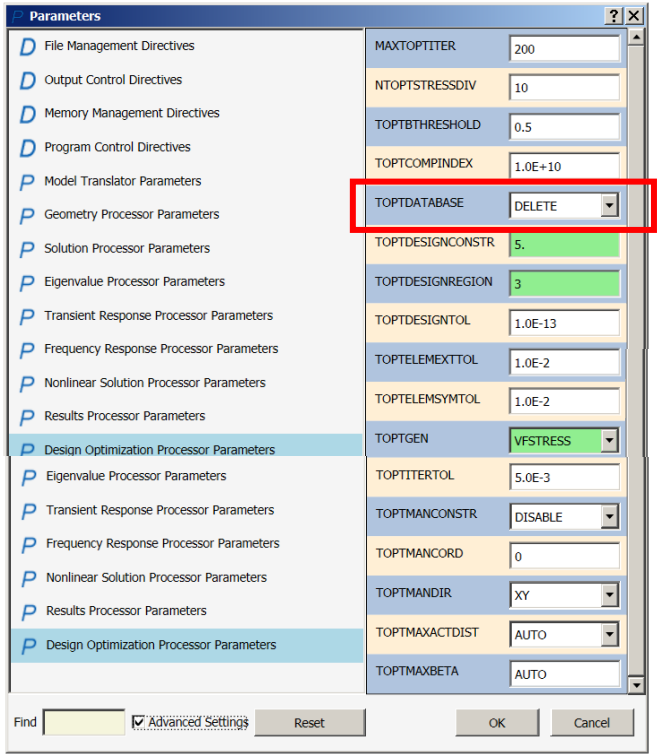
# Inventor Nastran Optimization Parameters - TOPTITERTOL




| 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 |



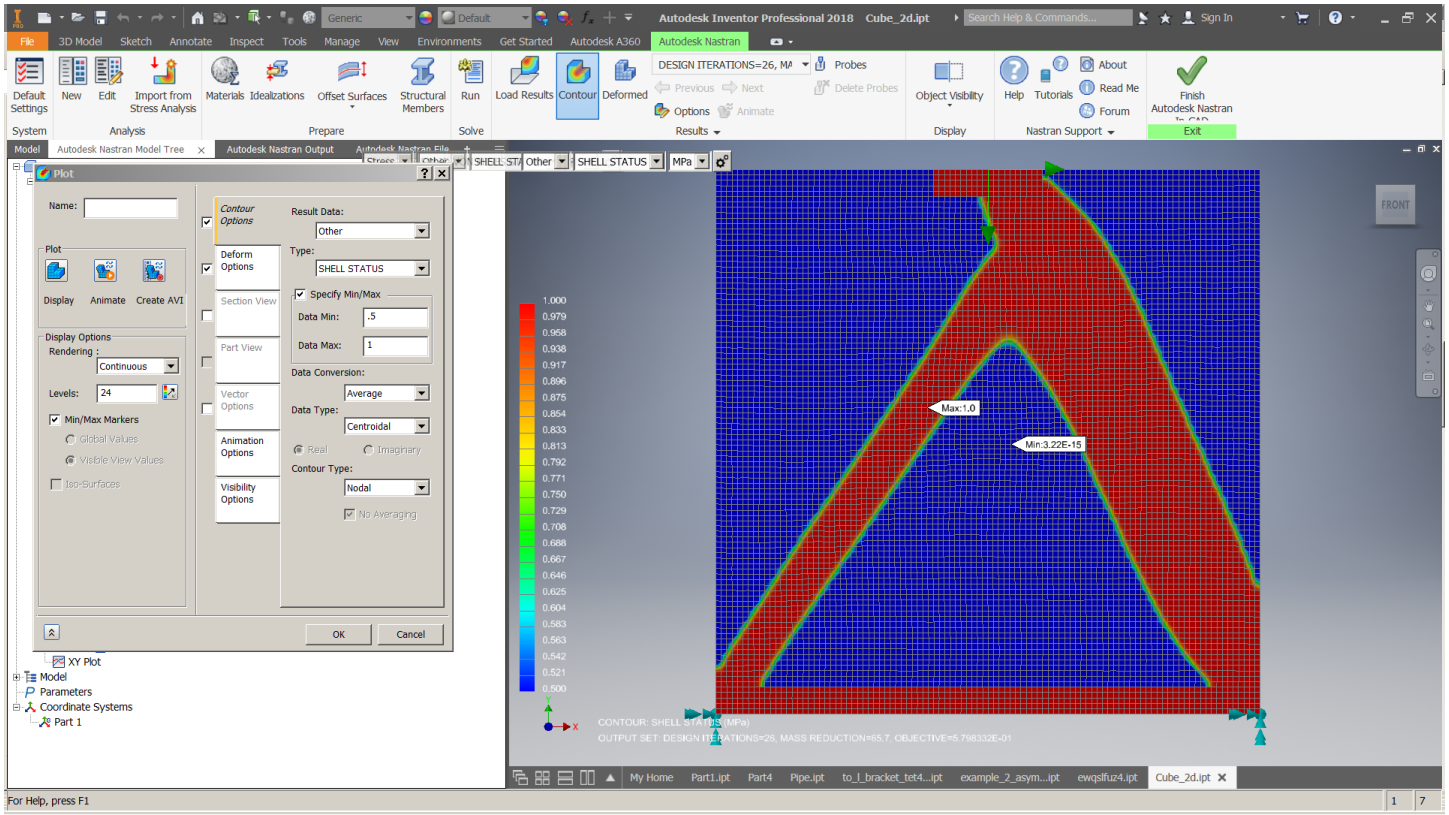
# Inventor Nastran Optimization Parameters - TOPTDATABASE



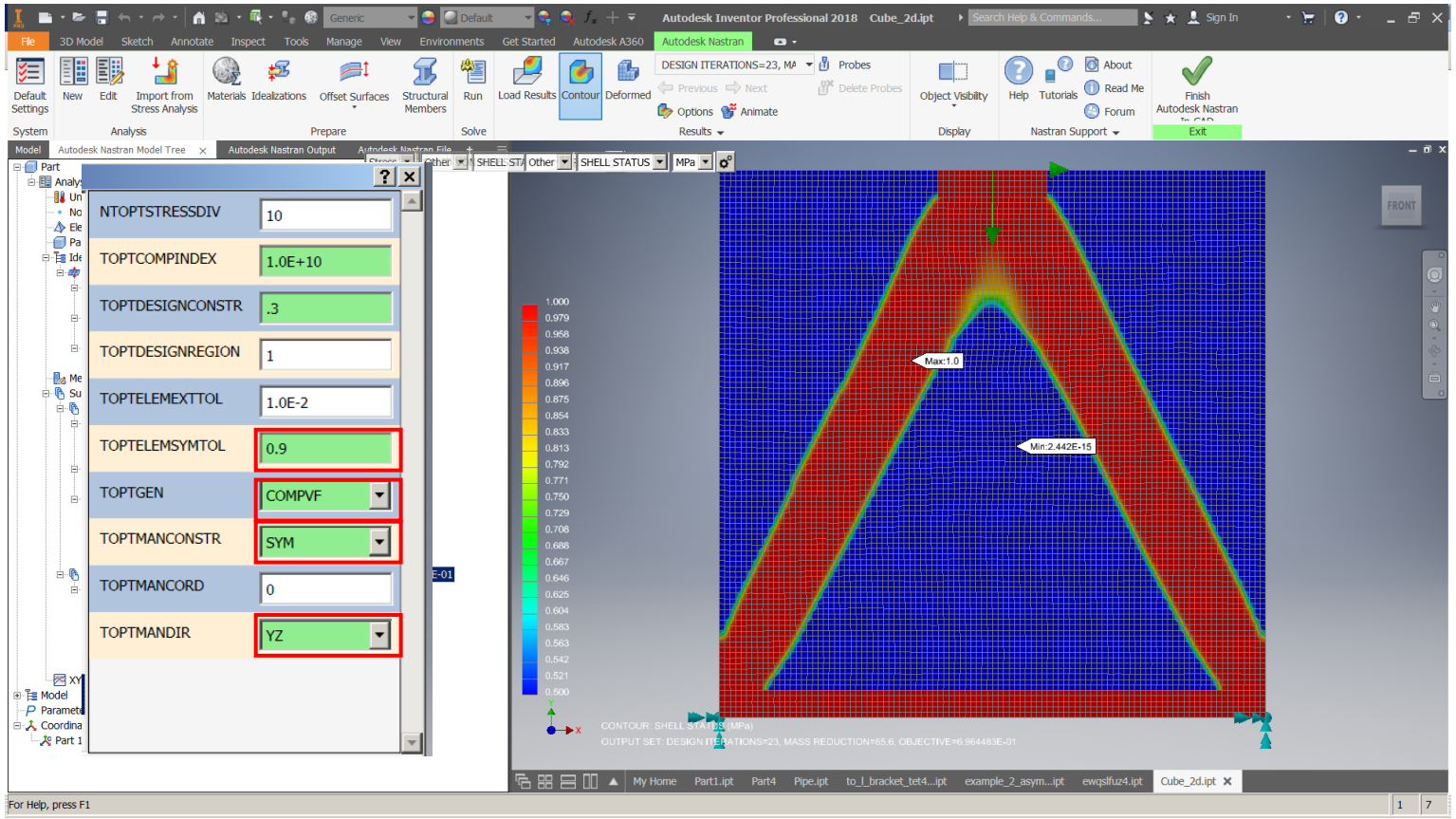

| Description | Type | Default |
|---|---|---|
| Controls the storage and retrieval of topology design optimization density data. The default value DELETE purges all element density data when the program terminates normally. When set to STORE, the converged optimized design is stored in a single file 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<br>FETCH<br>STORE<br>UPDATE | DELETE |

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

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

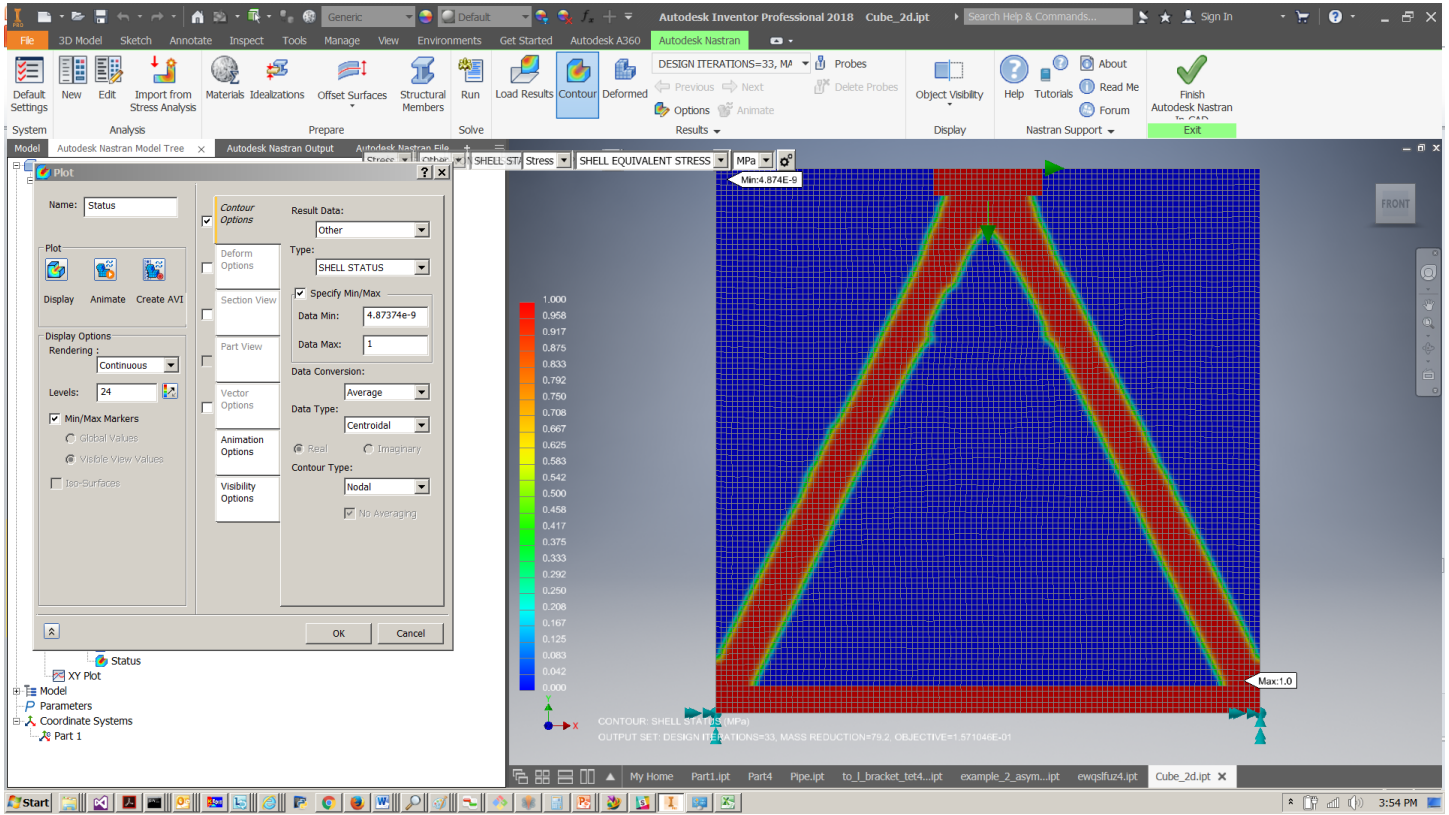


# Obj: Min. Compliance,  Constraint: Desired VF, with Sym



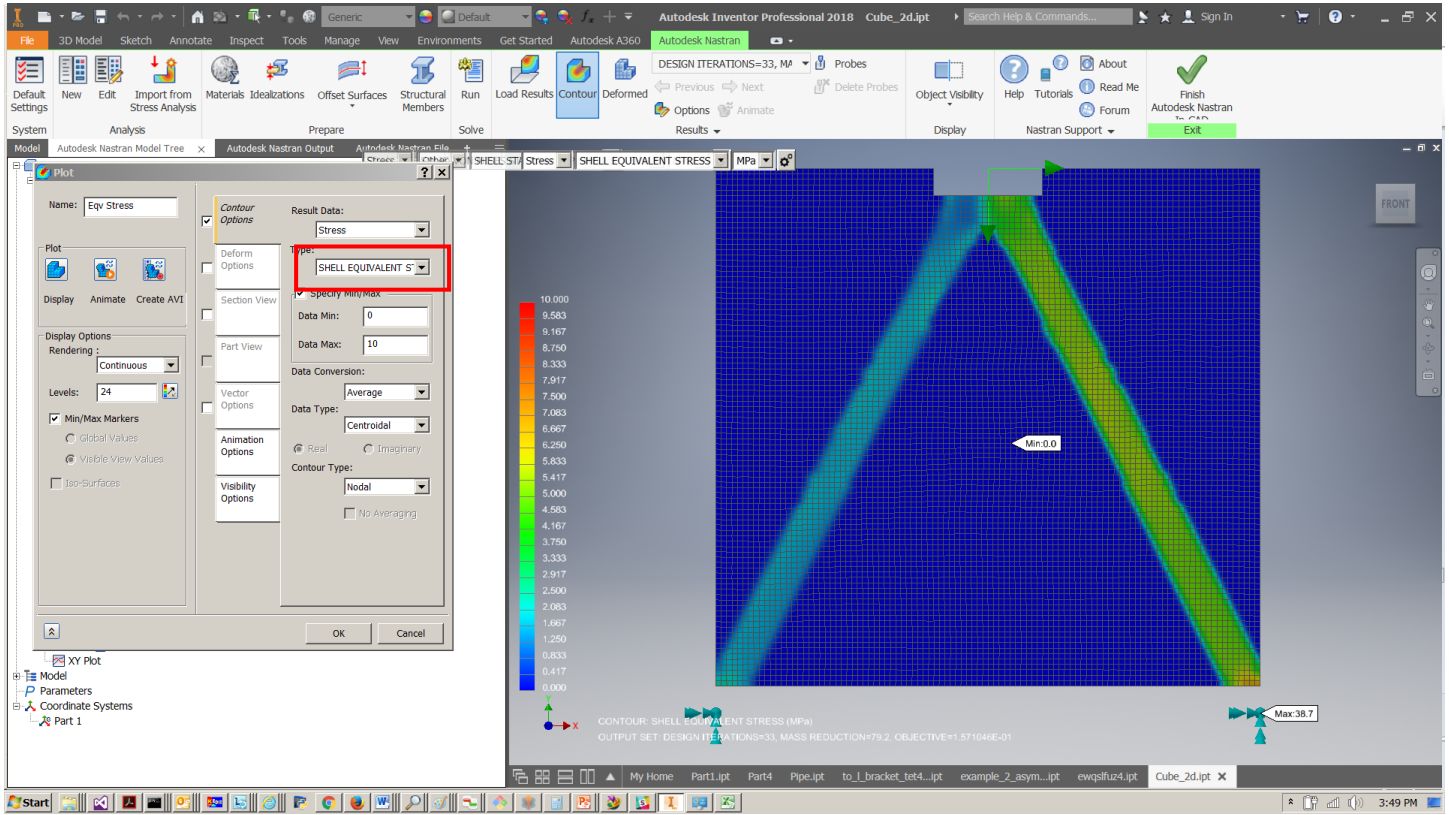
# Obj: Min. VF (mass), Constraint: Stress & Comp. Index

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

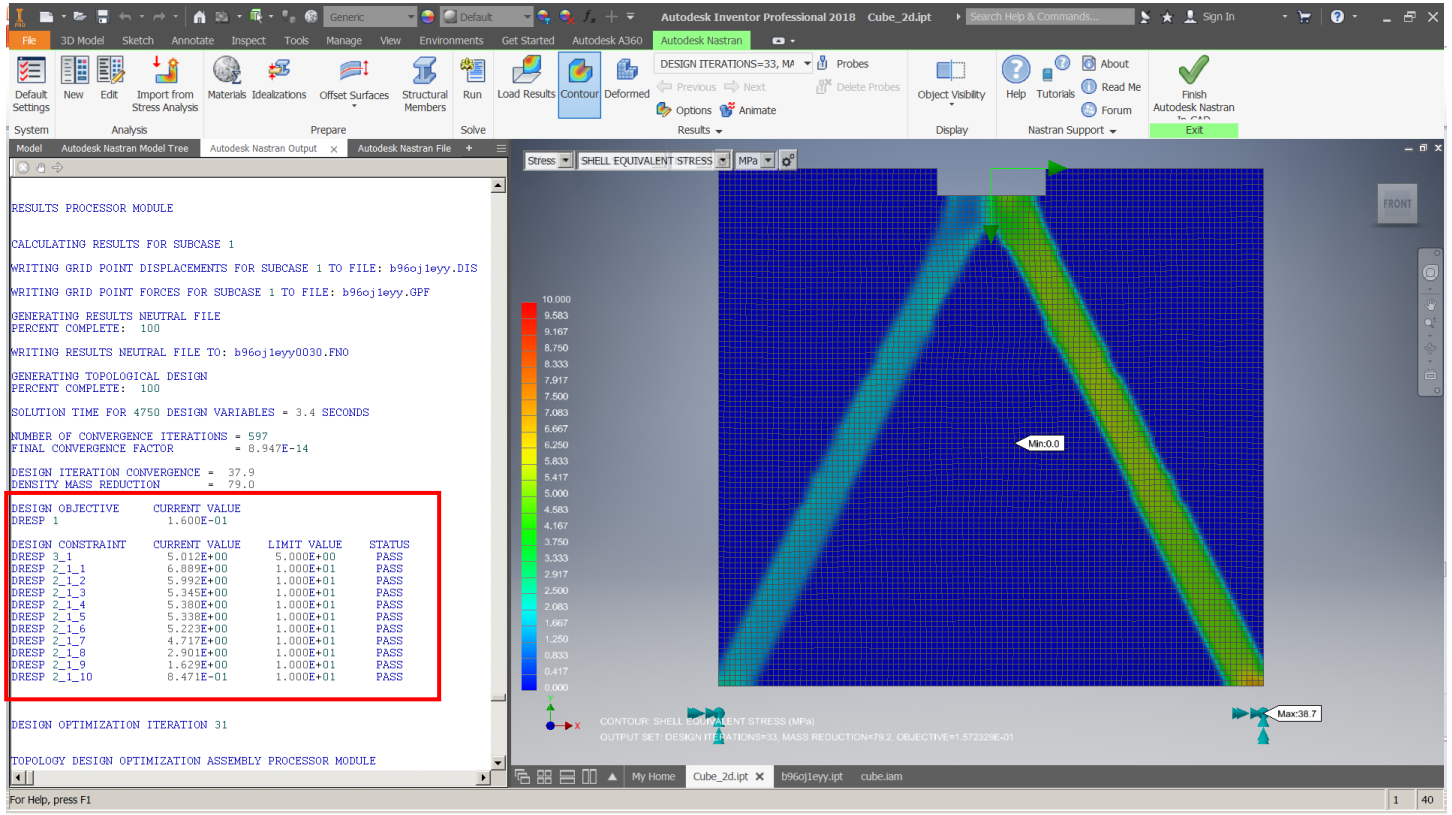


# Verify Stress Constraint Using Equivalent Stress

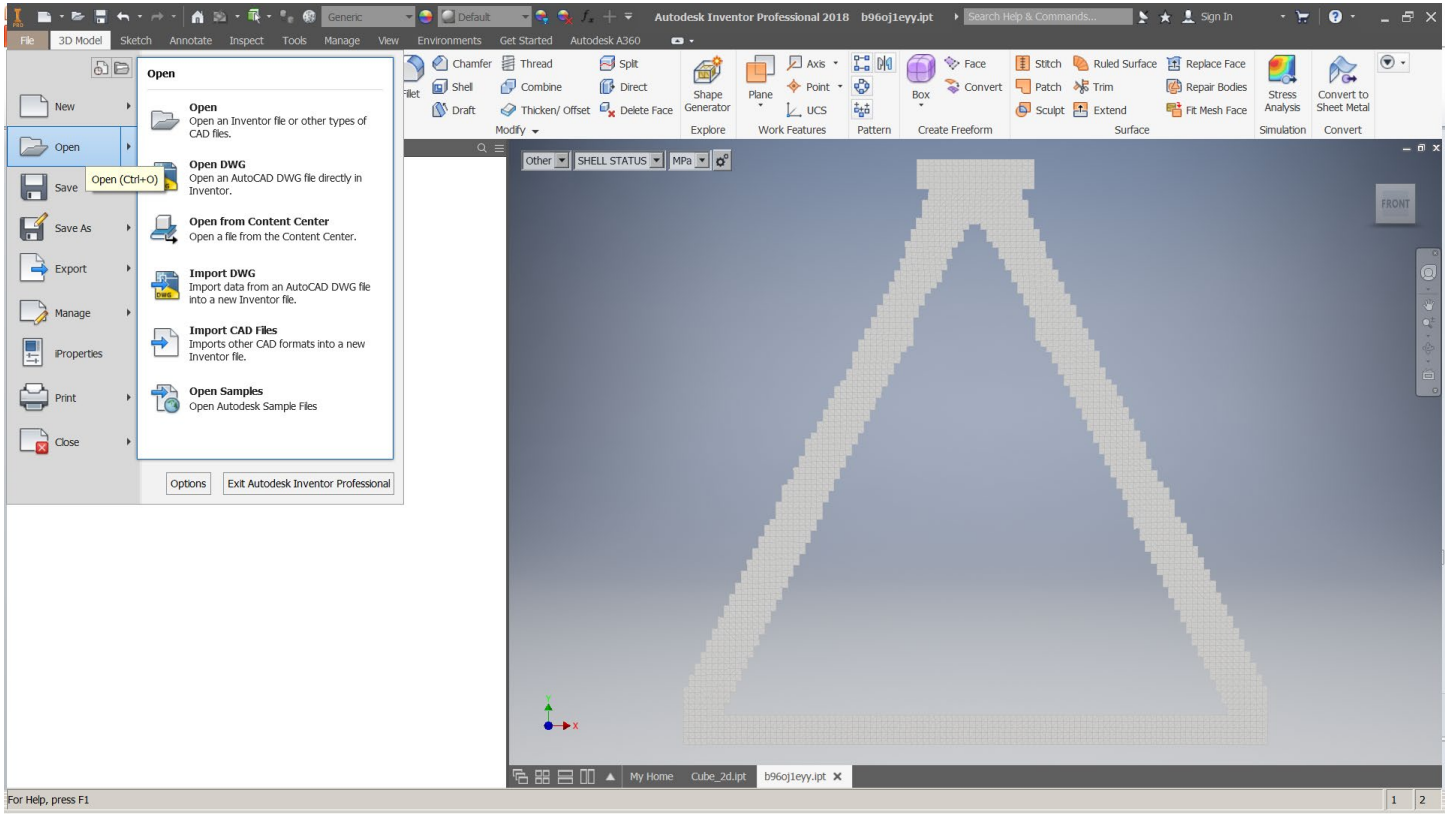
| | |
|---|---|
| NTOPTSTRESSDIV | 10 |
| TOPTCOMPINDEX | 5. |
| TOPTDESIGNCONSTR | 10. |
| TOPTDESIGNREGION | 1 |
| TOPTELEMEXTTOL | 1.0E-2 |
| TOPTELEMSYMTOL | 0.9 |
| TOPTGEN | VFSTRESS |
| TOPTMANCONSTR | SYM |
| TOPTMANCORD | 0 |
| TOPTMANDIR | YZ |



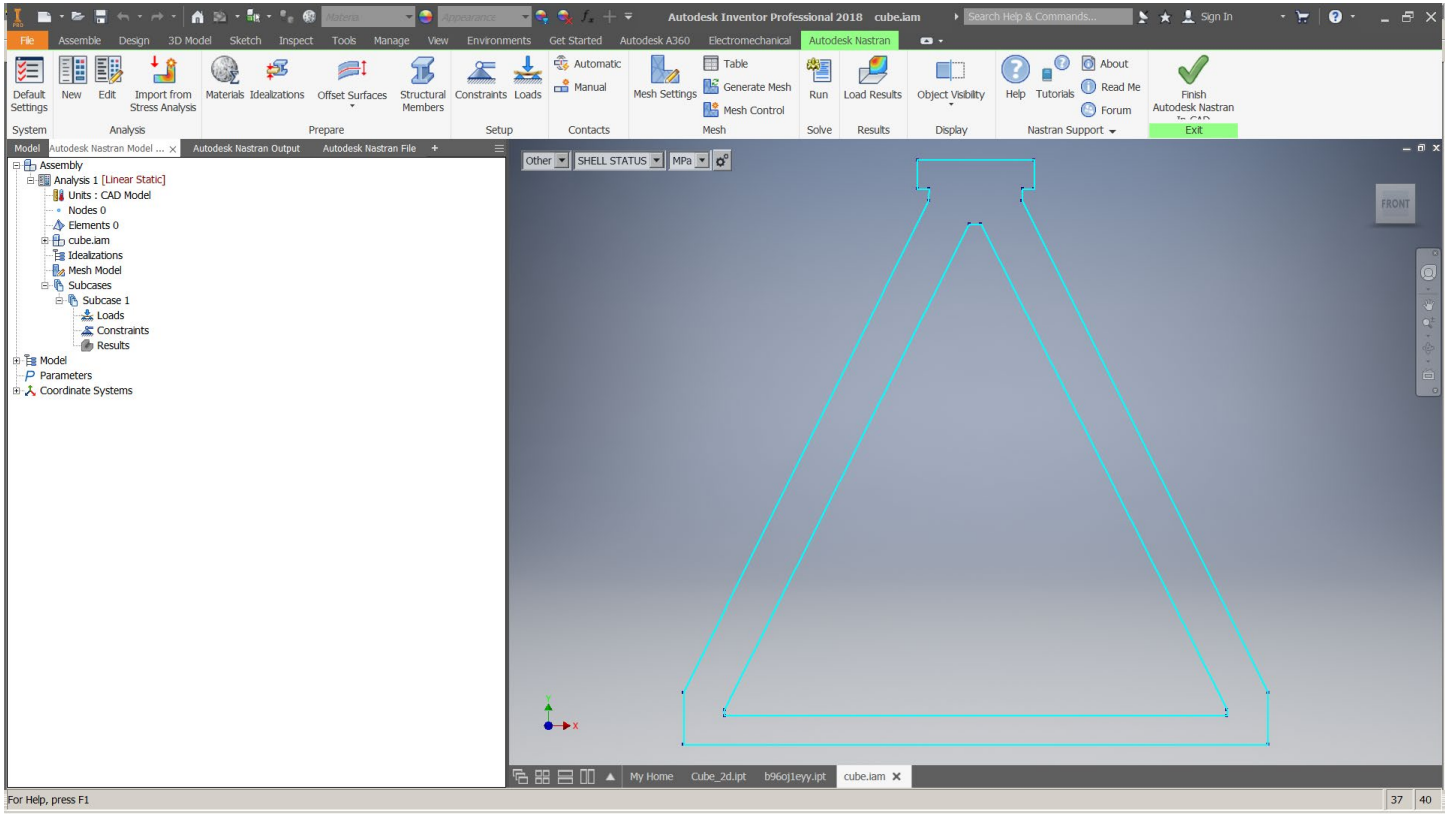
# Verify Stress Constraint Using Equivalent Stress



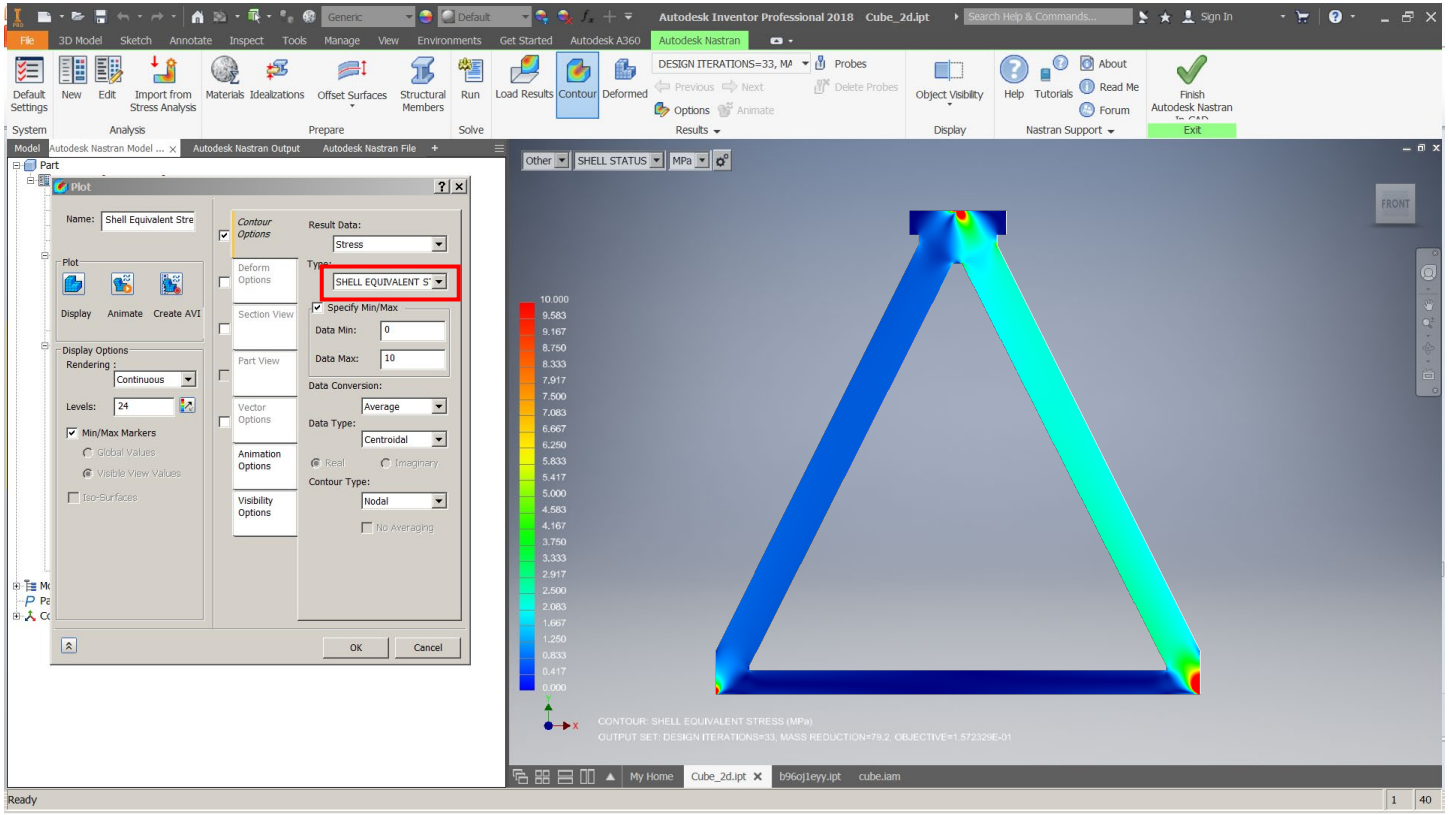
# Import Generated Geometry to Build New Design



# Import Generated Geometry to Build New Design

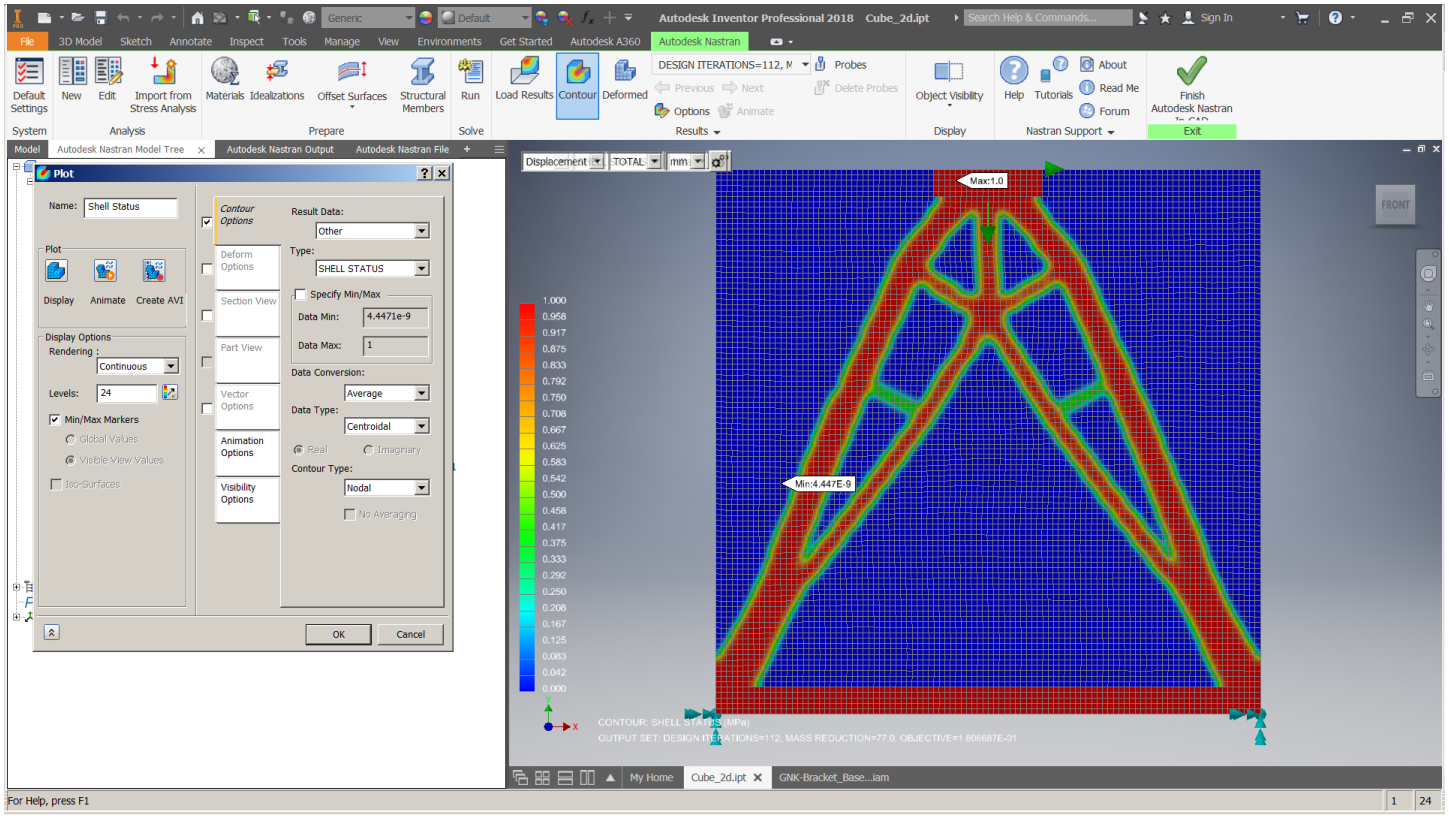


# Import Generated Geometry to Build New Design



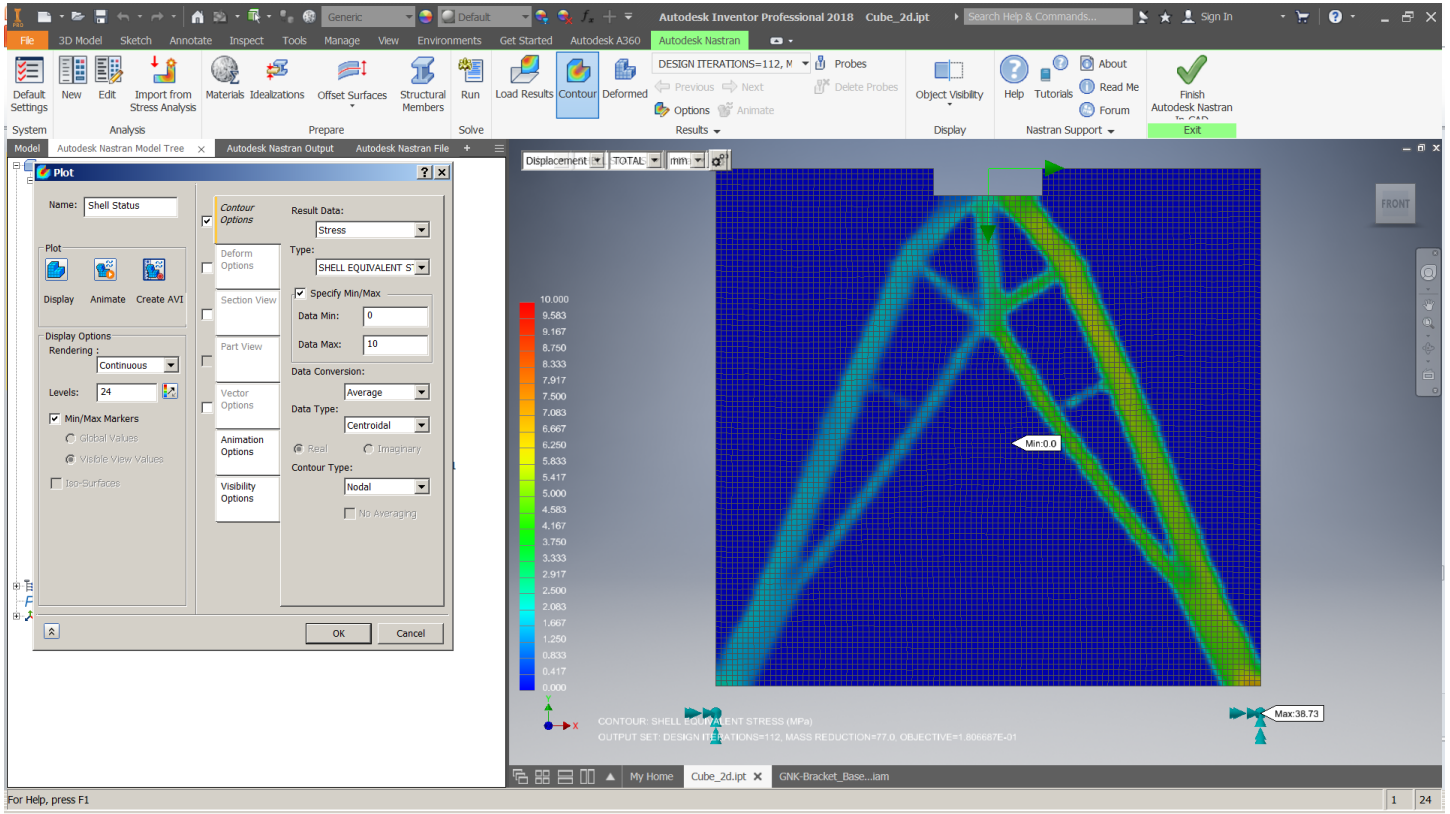
# Obj: Min. VF (mass), Constraint: Stress & Comp. Index

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



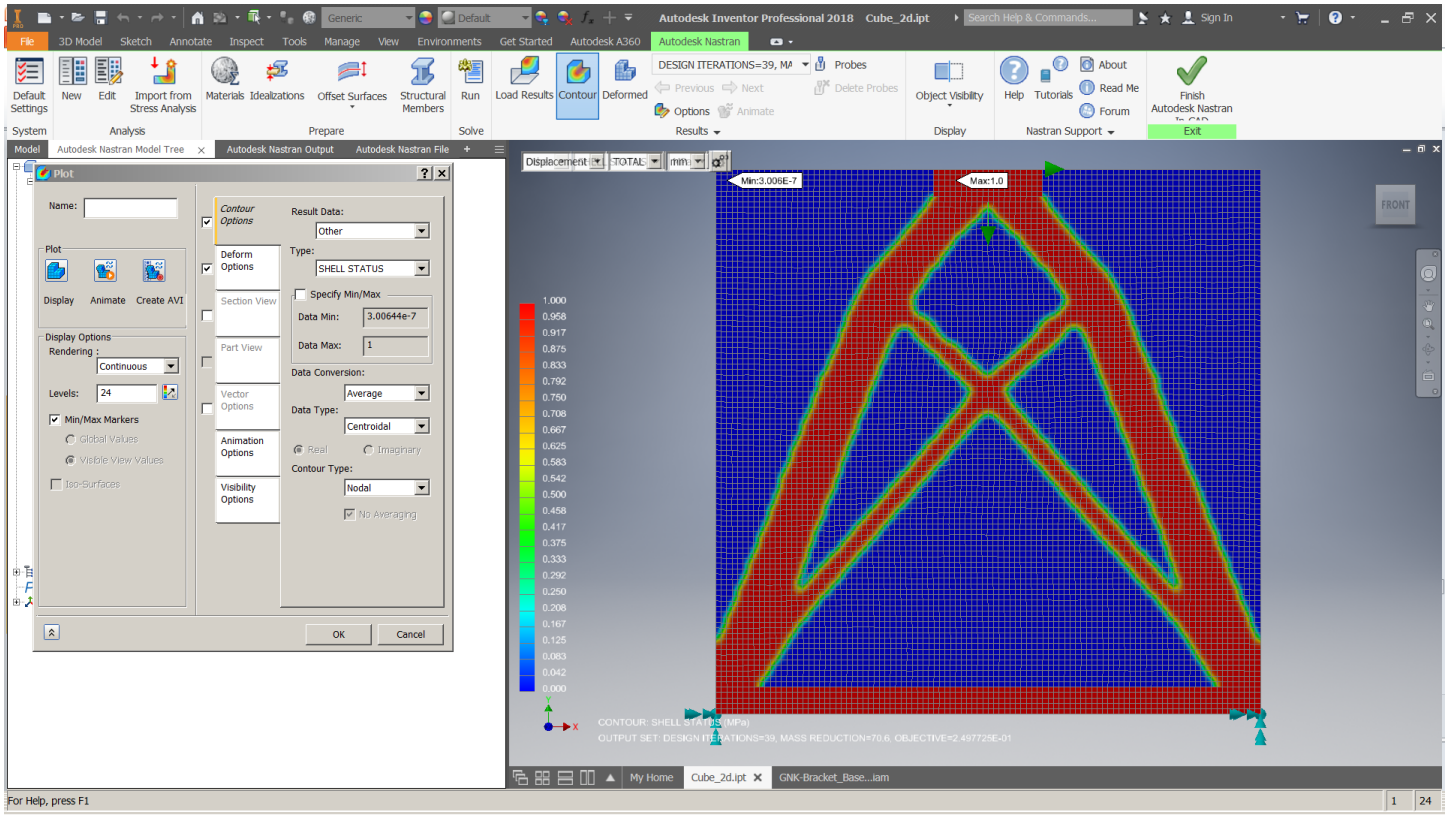
# Obj: Min. VF (mass), Constraint: Stress & Comp. Index

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



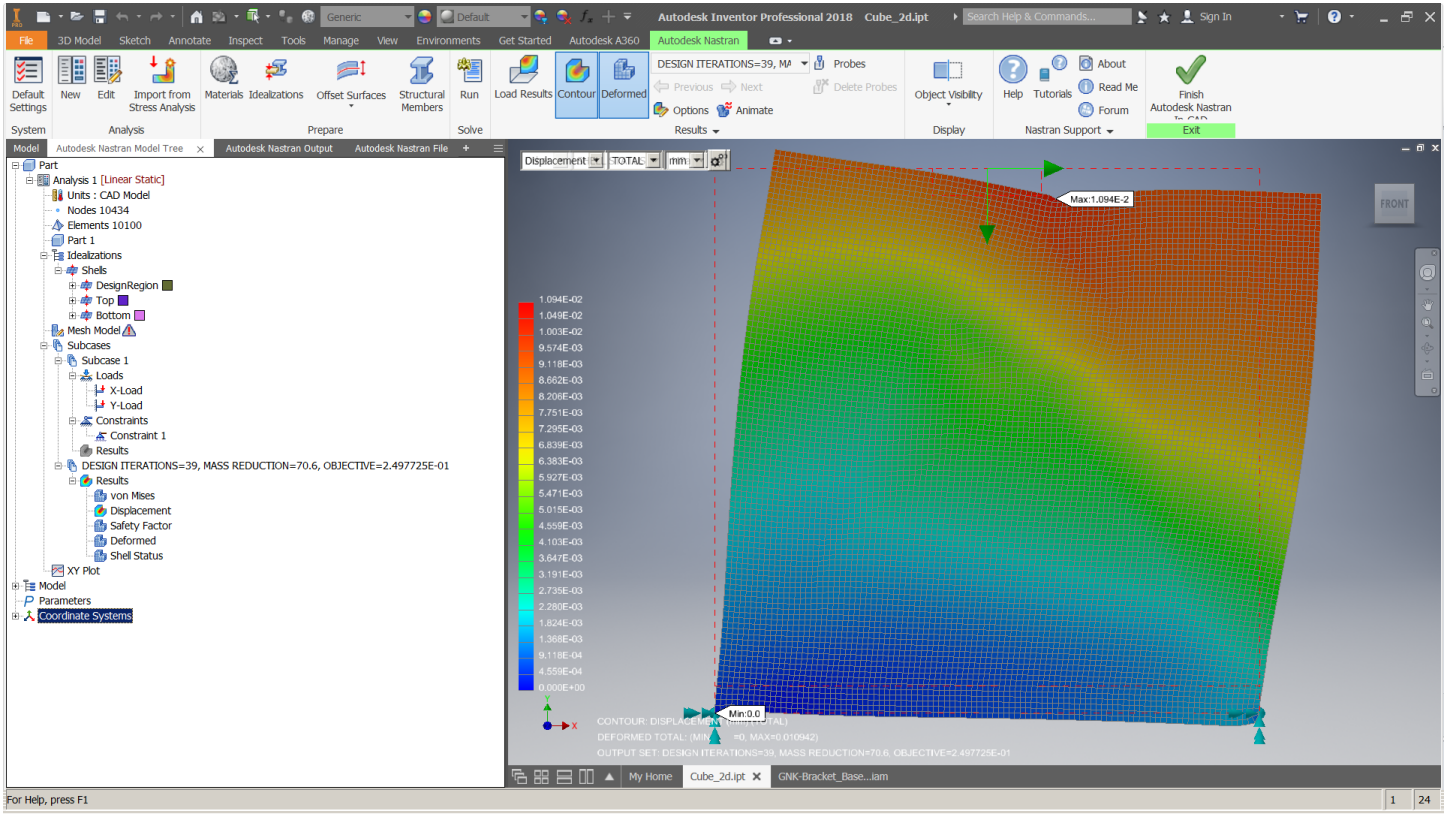
# Obj: Min. VF (mass), Constraint: Disp. & Comp. Index

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

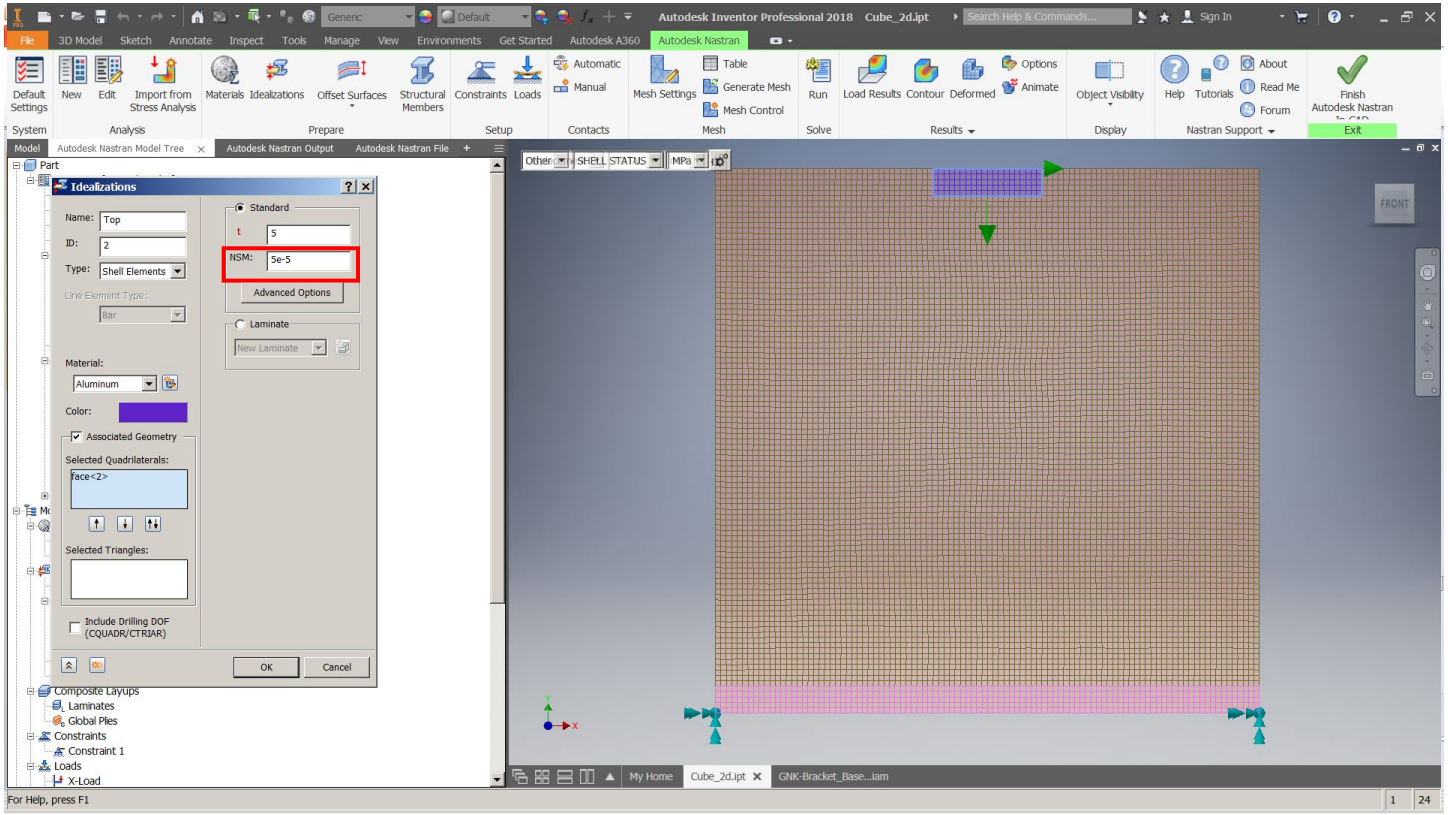


# Obj: Min. VF (mass), Constraint: Disp. & Comp. Index

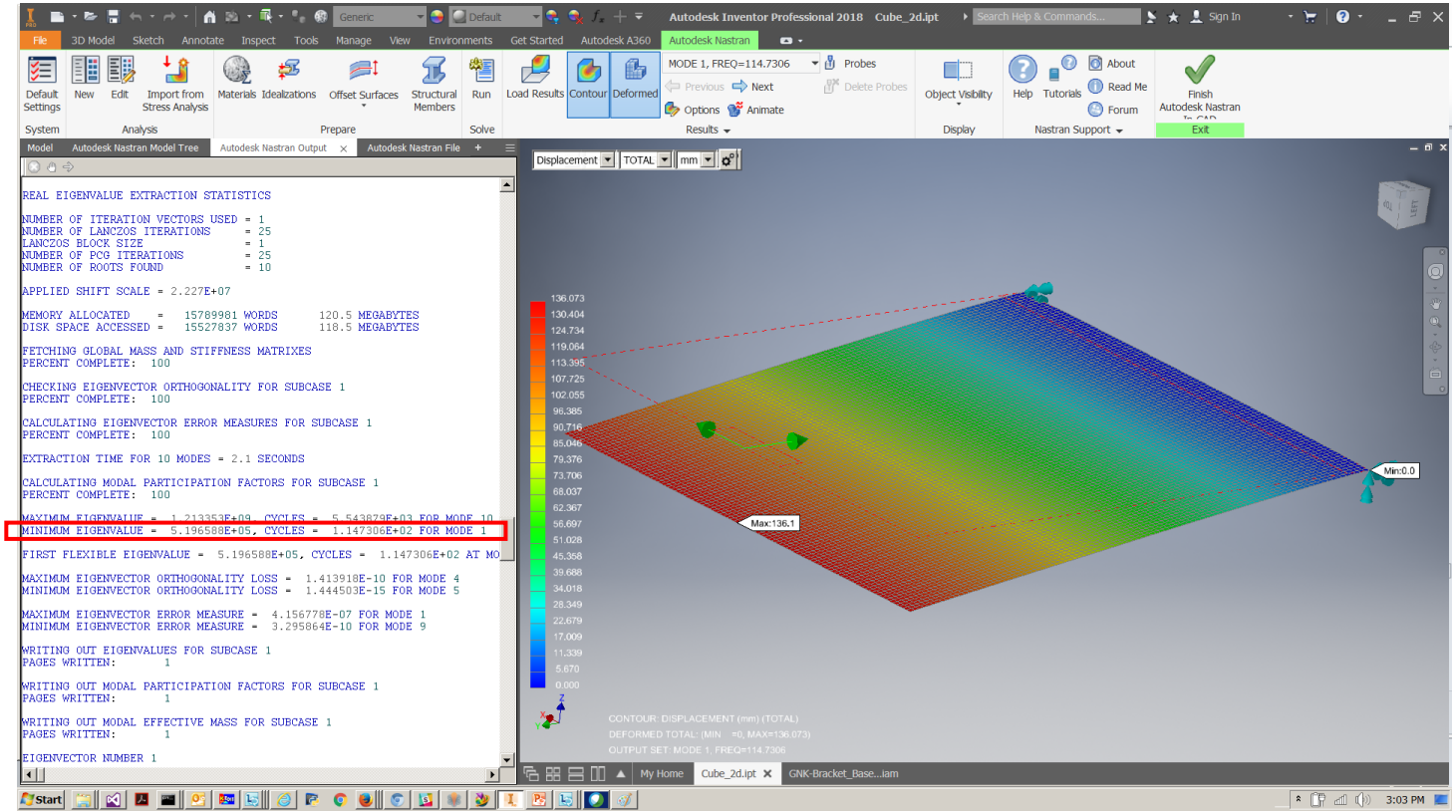
| | |
|---|---|
| NTOPTSTRESSDIV | 10 |
| TOPTCOMPINDEX | 5. |
| TOPTDESIGNCONSTR | 0.01 |
| TOPTDESIGNREGION | 1 |
| TOPTELEMEXTTOL | 1.0E-2 |
| TOPTELEMSYMTOL | 0.9 |
| TOPTGEN | VFDISP |
| TOPTMANCONSTR | SYM |
| TOPTMANCORD | 0 |
| TOPTMANDIR | YZ |



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

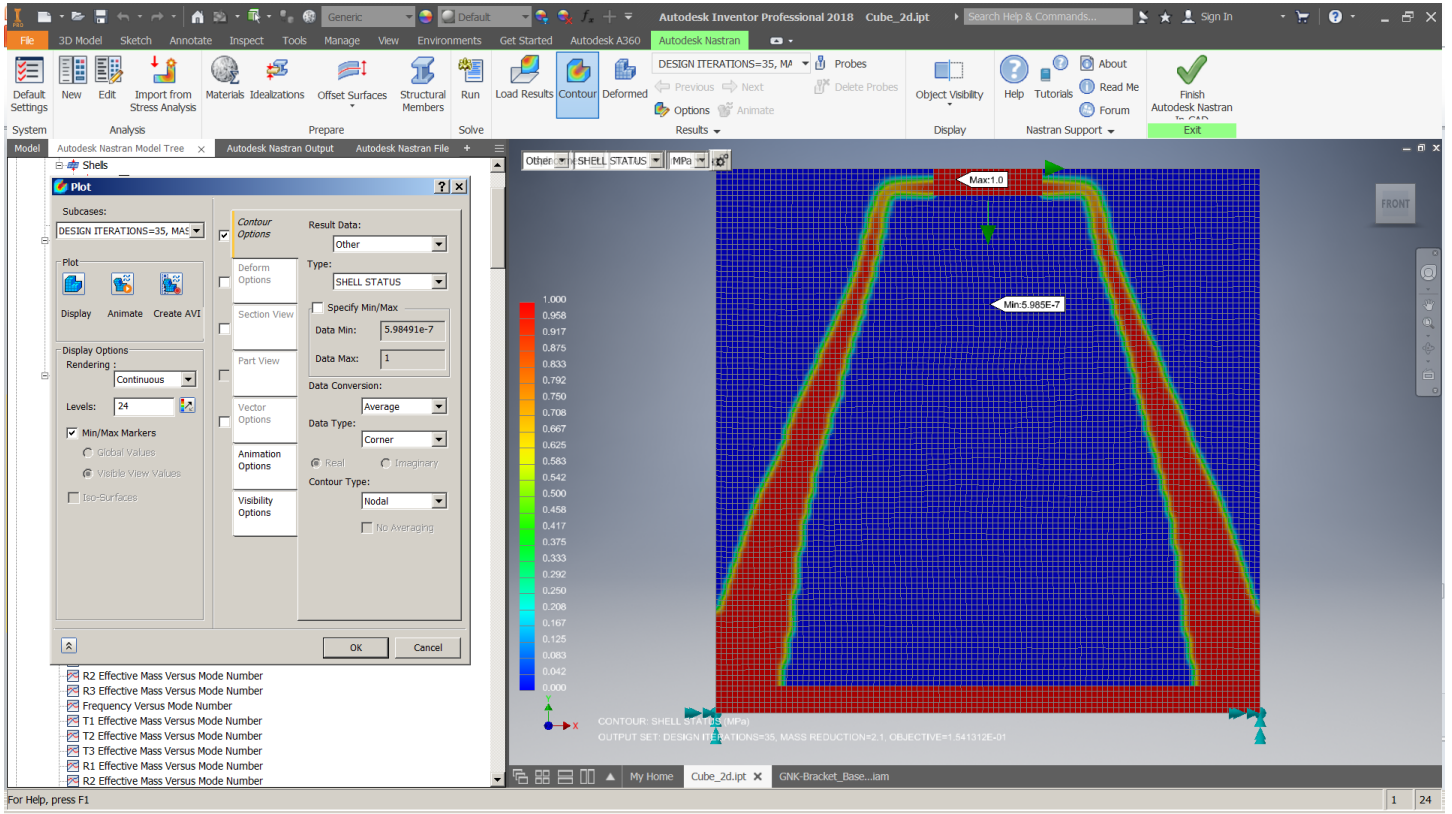


# Obj: Min. VF (mass), Constraint: Frequency



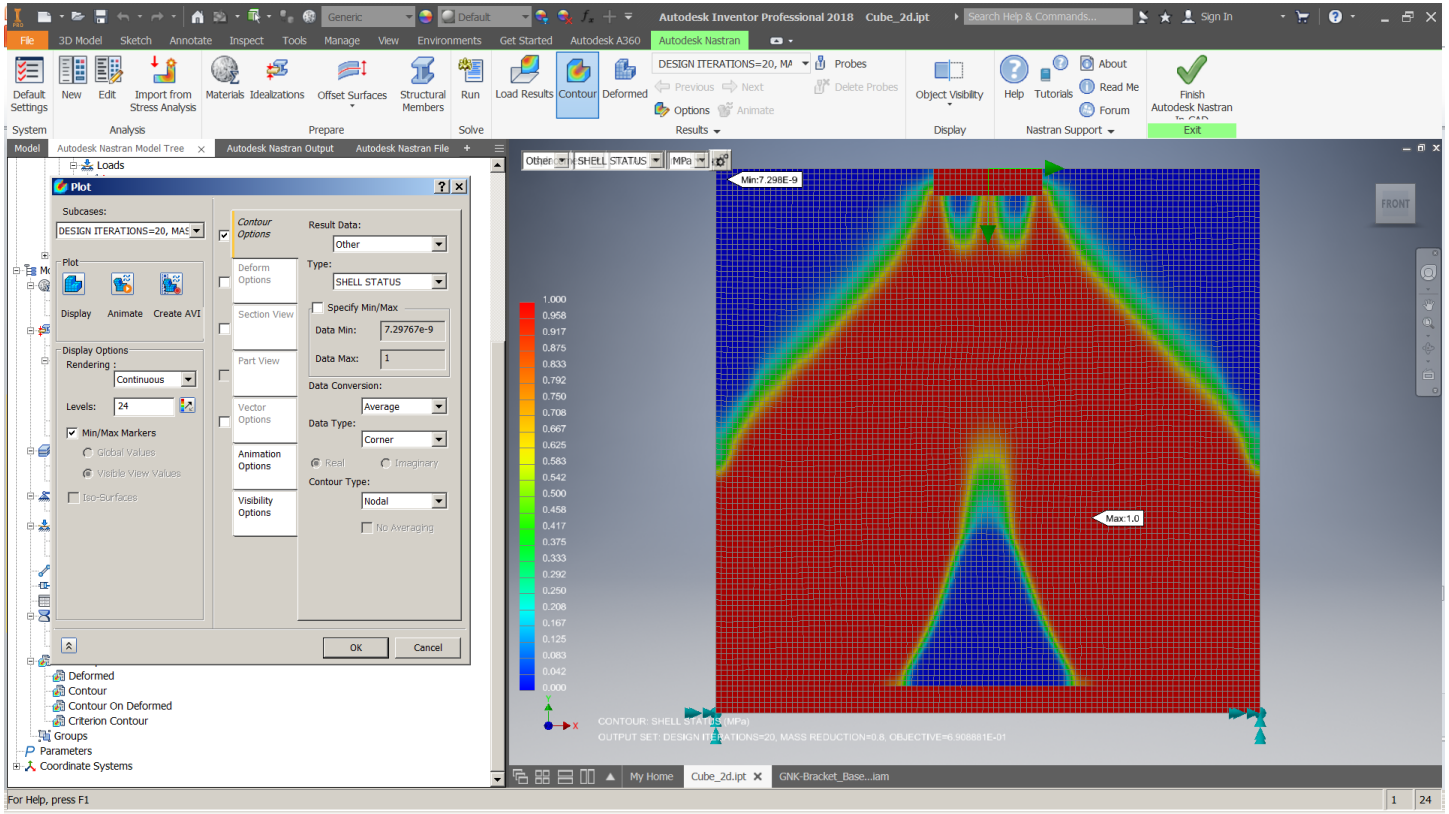
# Obj: Min. VF (mass), Constraint: Frequency > 10Hz

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



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

| | |
|---|---|
| NTOPTSTRESSDIV | 10 |
| TOPTCOMPINDEX | 5. |
| TOPTDESIGNCONSTR | 12. |
| TOPTDESIGNREGION | 1 |
| TOPTELEMEXTTOL | 1.0E-2 |
| TOPTELEMSYMTOL | 0.9 |
| TOPTGEN | VFFREQ |
| TOPTMANCONSTR | SYM |
| TOPTMANCORD | 0 |
| TOPTMANDIR | YZ |



# Commonly Used Topology Optimization Parameters

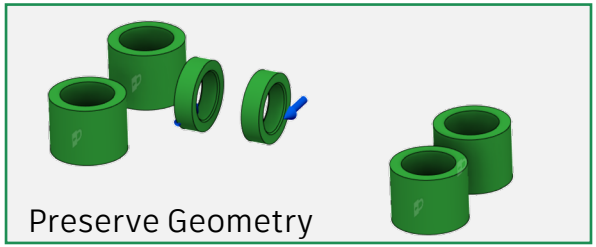
| Parameter | Description | Default | Suggested Range | Remarks |
|---|---|---|---|---|
| MAXTOPTITER | Limits the number of design iterations | 200 | 100 - 300 | Increase when iteration limit exceeded |
| TOPTELEMSYMTOL | Tolerance for symmetry manufacturing constraint | 1.0E-02 | < 1.0 | Increase if 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 |

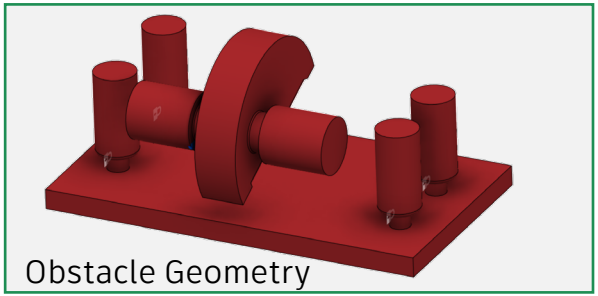
# Generative Design Versus Topology Optimization

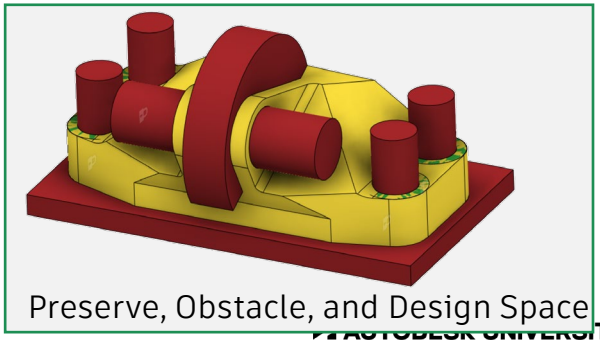
- Topology Optimization (TO) is a subset of Generative Design
- TO is a tool based on B-Rep geometries and the Finite Element Method that is used primarily **to remove unnecessary material** from an engineer's best guess at one initial design shape of a product
- Generative Design (GD) is a **design 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


Preserve Geometry


Obstacle Geometry


Preserve, Obstacle, and Design Space

# Fusion 360 Generative Design Outcomes