

MFG501490

# Up and Running with Inventor Nastran Nonlinear Analysis – Real World Examples

Wasim Younis  
Symetri

## Learning Objectives

- Learn about different types of nonlinear behavior
- Learn how to apply top Inventor Nastran nonlinear tips.
- Learn about performing analysis using both implicit and explicit solvers.
- Learn about designing products with less impact on the environment.

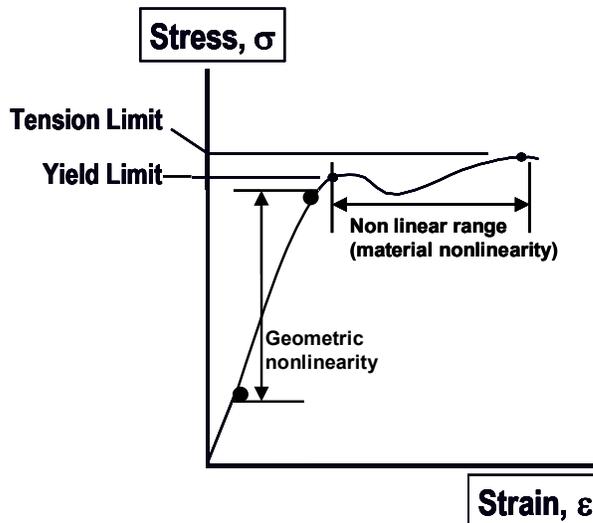
## Description

This session will start with real-life examples demonstrating how engineers and designers like you have greatly benefited from the advanced use of Inventor Nastran simulation technology within their companies. The software has helped them to make informed design decisions early on and enabled them to make cost-effective optimized designs with less impact on the environment, ultimately providing more time to explore “what if” scenarios. Real-life examples will include blast loads, drop tests, elastic/plastic analysis, and permanent deformation. We will then continue by explaining the process of applying nonlinear analysis using a straightforward, step-by-step approach, supported by industry best practices with explanations and tips. Our hope is to help make your Inventor Nastran adoption journey within your workplace successful. We want ultimately to help you simulate complex real-world scenarios early on, enabling the creation of sustainable designs faster and more cost-effectively.

## Speaker(s)

A passionate simulation solutions expert been involved with Autodesk simulation software from when it was first introduced, and is well-known throughout the Autodesk simulation community, worldwide. Also authored the Up and Running with Autodesk Inventor Professional books. He also manages a dedicated forum for simulation users on LinkedIn – Up and Running with Autodesk Simulation. Wasim has a bachelor’s degree in mechanical engineering from the University of Bradford and a master’s degree in computer- aided-engineering from Staffordshire University.

## Different types of nonlinear behaviour



Nonlinear analysis generically falls into the following three categories.

**Geometric nonlinearity** - Where a component experiences large deformations and as a result can cause the component to experience nonlinear behavior. A typical example is a fishing rod.

**Material nonlinearity** - When the component goes beyond the yield limit, the stress/strain relationship becomes nonlinear as the material starts to deform permanently.

**Contact** - Includes the effect of two components coming into contact; that is, they can experience an abrupt change in stiffness resulting in localised material deformation at region of contact.

While many practical problems can be solved using linear analysis, some or all its inherent assumptions may not be valid:

- Displacements and rotations may become large enough that equilibrium equations must be written for the deformed rather than the original configuration. Large rotations can cause pressure loads to change in direction, and to change in magnitude if there is a change in area to which they are applied.
- Elastic materials may become plastic, or the material may not have a linear stress-strain relation at any stress level.
- Part of the structure may lose stiffness because of buckling or material failure.
- Adjacent parts may make or break contact with the contact area changing as the loads change.

### Geometric Nonlinearity

The geometric nonlinearity becomes a concern when the part(s) deform such that the small displacement assumptions are no longer valid. The large displacement effects are a collection of different nonlinear properties, such as:

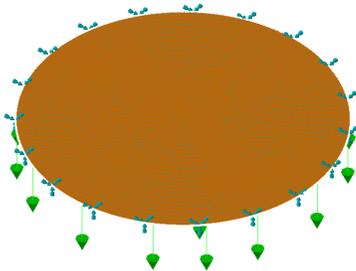
1. Large deflections.
2. Stress stiffening/softening.
3. Snap-thru.
4. Buckling.
5. Large strain.

#### Large Deflections

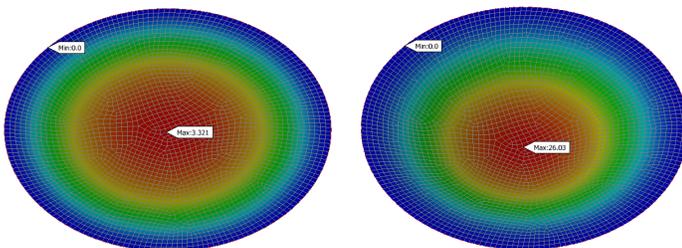
When your components or assemblies start to experience rotations of more than about 10 degrees you should start to consider nonlinear analysis. This is because linear analysis assumes small displacement theory in which  $\sin(\theta) \approx (\theta)$ .

#### Stress Stiffening

Stress stiffening (also known as geometric stiffening) only effects thin structures where the bending stiffness is very small compared to the axial stiffness. For instance, consider the following plate subjected to a load. The structure is fixed around the perimeter. This thin-walled structure will undergo significant stress stiffening as the part transitions from reacting the load in bending, to reacting the load in-plane.



The images below show two results of the plate. The first image is results from a nonlinear analysis (peak deflection 3.321mm). The second image is the results from a linear analysis (peak deflection is 26.03mm).



Stress stiffening effects are caused by tensile stresses which result from larger displacements, not by the displacements themselves. The actual displacement in the model is not a clear indication of the degree of nonlinearity, nor is the tensile stress magnitude. A similar tension in one geometry or load orientation may result in significantly less stress stiffening than in another.

### Snap-thru and Buckling

Other common geometric nonlinear situations involve snap-thru and buckling problems, often referred to as bi-stable or multi-stable systems. Many snap-thru problems behave nearly linearly until the point where a small amount of additional load causes a large amount of deflection where a secondary stable position is reached. Capturing this snap-thru is a very difficult numerical problem.

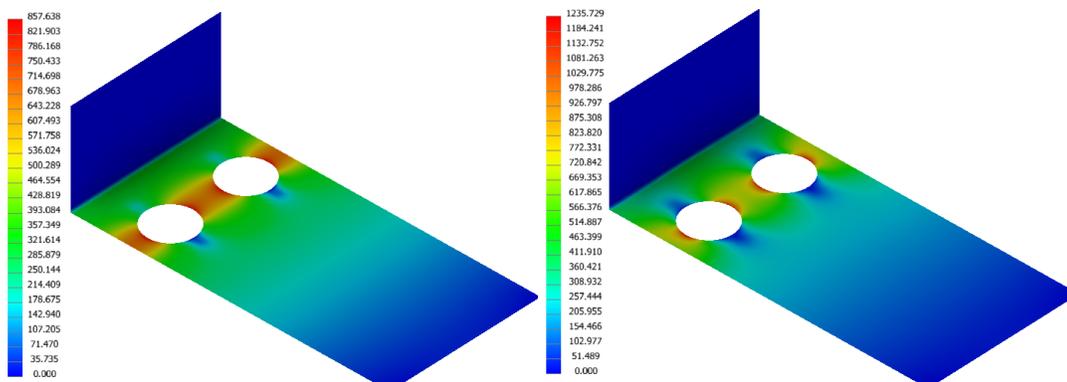
### Large Strain

Large strains are typically associated with large displacements causing permanent deformation as stresses above yield have been exceeded. Cold heading, rubber seal compression, and metal forming are good examples of large strain examples.

### Material Nonlinearity

When components experience stress above yield then the results obtained from linear analysis are not valid. In these cases, we need to define stress and strain behavior of materials above yield to get an accurate behavior. However, most materials and even metals have some amount of ductility. This ductility allows hot spots to locally yield thus reducing the stresses compared to what a linear analysis would predict.

The metal bracket from the image below shows very different stress distributions between linear and nonlinear materials. The right image contains linear analysis results and shows peak stresses well above yield. The nonlinear material analysis on the left shows a different contour due to the stress redistribution. Peak plastic strain was 5.7% in the nonlinear material analysis.



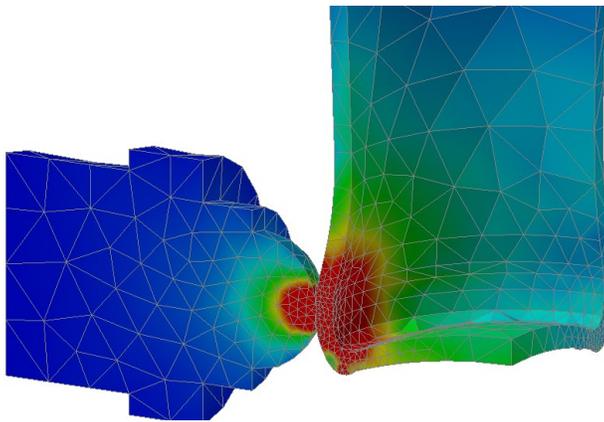
## Boundary Condition Nonlinearity

The most common boundary nonlinearities are:

1. Contact.
2. Follower forces.

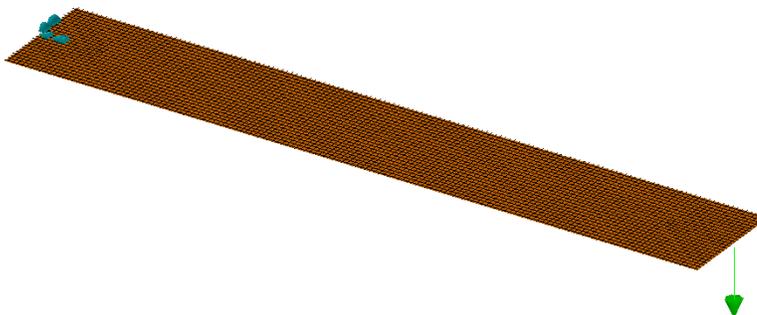
### Contact

Contact conditions model the interaction of two separate parts. Boundary conditions such as separation contacts are generally regarded as nonlinear, as the contact allows separation and sliding between components. This type of contact is typically used in bolted connections where two plates are held by the bolts and the plates allowed to slide and separate depending on the extent of the loading conditions. Another example is in impact type analyses as illustrated below.

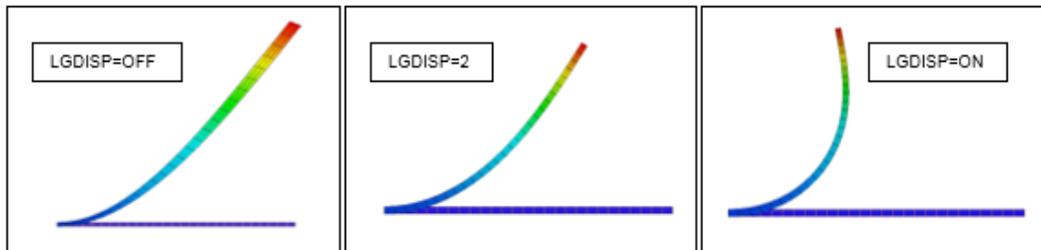


### Follower Forces

This nonlinear effect simply means that the direction of the force moves with deformation or movement of the part. This can be best demonstrated with the cantilevered strip shown below which is loaded with a force of 100N and three analyses are performed with different large displacement settings.



The first image shows the unrealistic "growth" that occurs when large displacement effects are turned off (LGDISP=OFF). The second image shows the results of large displacements turned on, but follower forces turned off (LGDISP=2). The final image uses large displacement effects with follower forces and is the most accurate (LGDISP=ON).



### Top Inventor Nastran nonlinear tips.

- 💡 Always run a linear analysis first to check if the yield limit has exceeded.
- 💡 Keep model simple and consider symmetry.
- 💡 Perform distortion checks to make sure there are no severely distorted elements.
- 💡 Only apply nonlinear materials in the areas of the model where you expect nonlinear or plastic behavior. This will help to speed up the analysis and can improve the convergence rate.
- 💡 Split faces at contact regions to reduce the number of generated contacts.
- 💡 Use Linear elements instead of Parabolic elements to help with achieving faster convergence in results.
- 💡 Use Continuous Meshing for Surface models to connect surfaces eliminating the need to create contacts. Contacts increase solution times.
- 💡 Leave the Number of Increments field blank in the Nonlinear Setup dialogue box. The software will calculate the optimum number of increments.
- 💡 Equivalent Stress Results follow the stress and strain curve data. Use this to analyse your stress and strain results.

-  Use the NPROCESSORS parameter to increase number of cores to help speed up analysis times.
-  Use explicit solvers if you are expecting high strains.
-  Run modal analysis to determine Dominant Frequency W3 required for Nonlinear Transient Response Analysis.
-  Use multiple subcases to determine permanent deformations in Nonlinear Static Analysis. With the first being loaded and second being unloaded.
-  Use multiple subcases to allow different time steps in Nonlinear Transient Response Analysis.

### Performing analysis using both implicit and explicit solvers.

The 1<sup>st</sup> Simatek example is based on the implicit solver and the following content is directly taken from my new Up and Running with Autodesk Inventor Nastran 2023 – Nonlinear Analysis book.



Available from Amazon worldwide.  
(Image hyperlink takes you to Amazon.com)

# DP4 – Inlet

(Design problem courtesy of Simatek A/S)

## Key features and workflows introduced in this design problem

Key Features/Workflows	
1	Material Nonlinearity
2	Nonlinear Static Analysis – Plastic Stress and Strain curve
3	Shell Elements - Continuous Mesh Connections
4	Multiple subcases – (Actual Permanent Deformation)

## Introduction

Simatek is a leading manufacturer of industrial emission and air pollution control solutions. Their high-value products and systems, optimise footprint, performance, powder recovery and maintenance for industrial plants worldwide. All at a low cost of ownership.



In this design problem we are going to analyse an inlet using the following design information and goal.

## Design Information

### ***Main Structure***

Material - **AISI Carbon Steel 304**

Youngs Modulus - **193GPa**

Yield Limit - **184MPa**

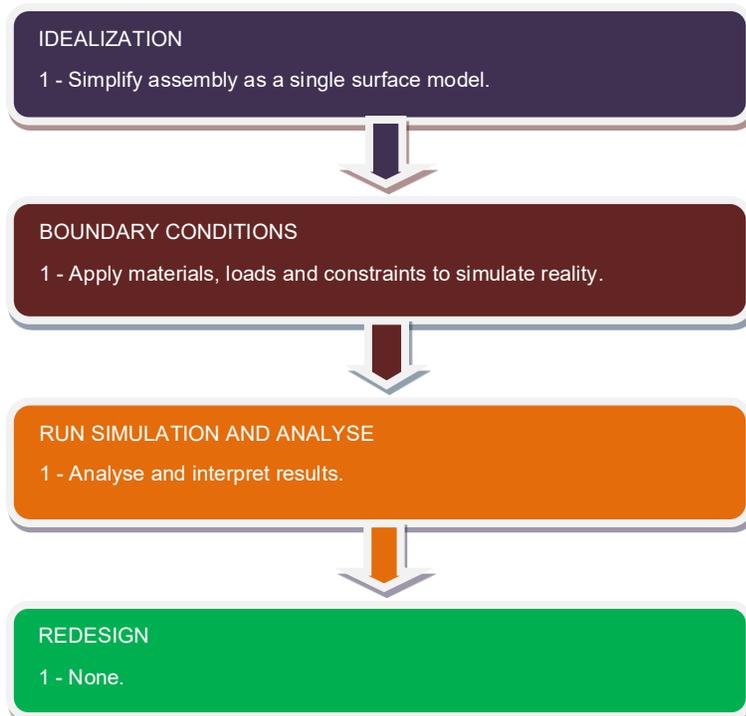
Poisson's Ratio - **0.27**

Blast Load – **0.082MPa**

### Design Goal

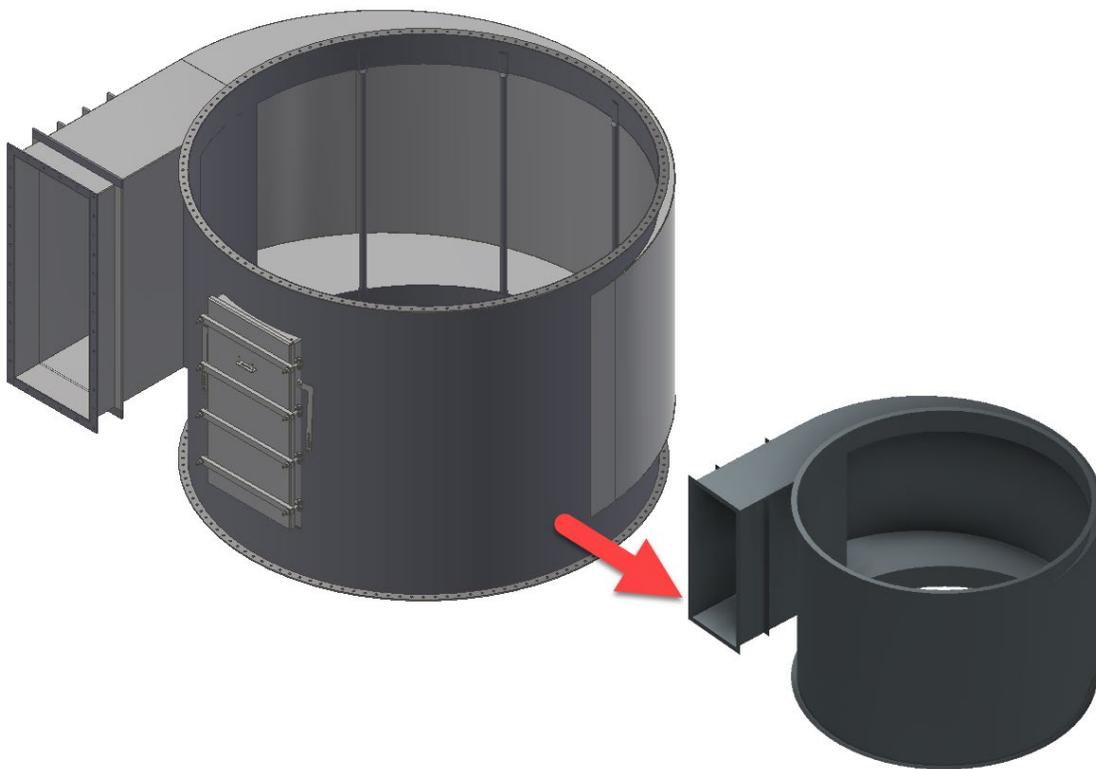
Strain to be less than 10%.

## Workflow of Design Problem 4

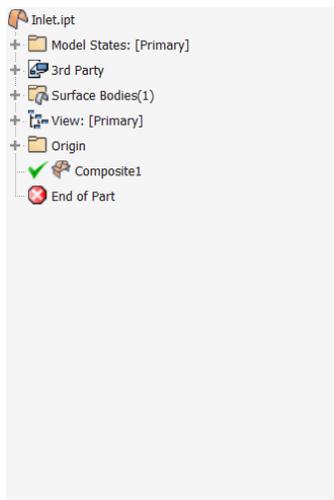


### Idealization

The assembly is remodeled as a single surface component to simplify the analysis and to speed up solution times. This includes removing small features like holes and non-structural components.



### 1. Open *Inlet.ipt*



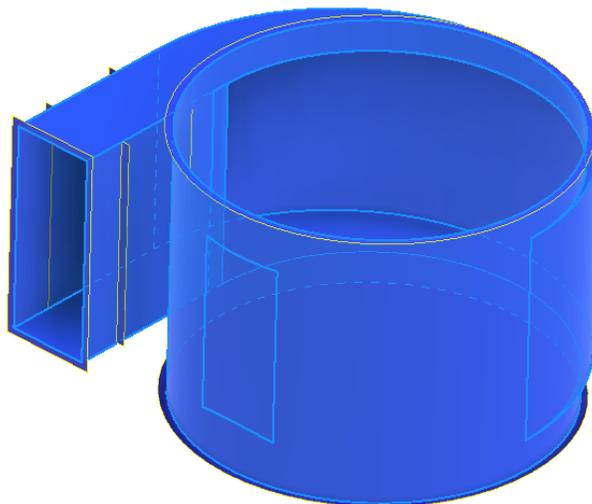
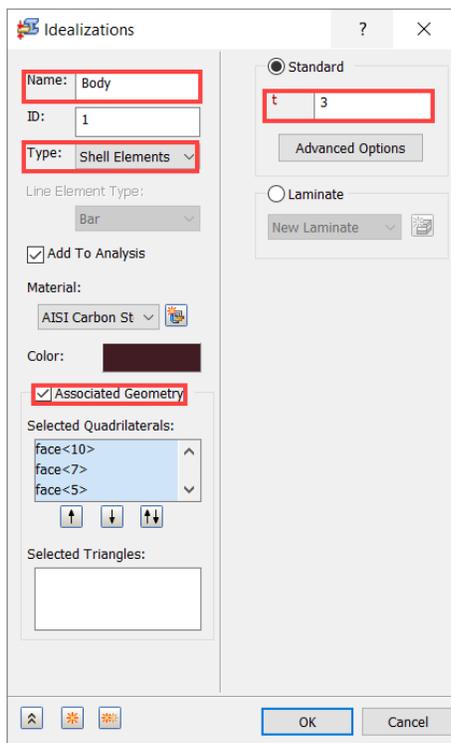
## Boundary conditions

2. Select **Environments** tab > Select **Autodesk Inventor Nastran**.
3. Double click **Generic** under Materials from the Model tree.
4. Select **Material** > Select **Load Database** > Open **ADSK\_materials.nasmat** > Select **AISI Carbon Steel 304** > Click **OK** > Specify **184** for Sy > Click **OK**.

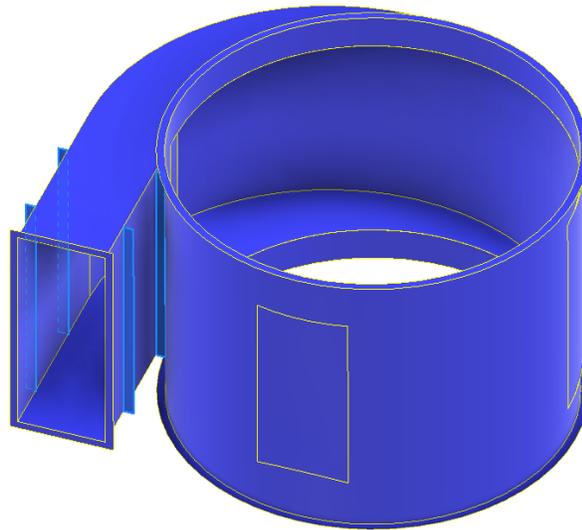
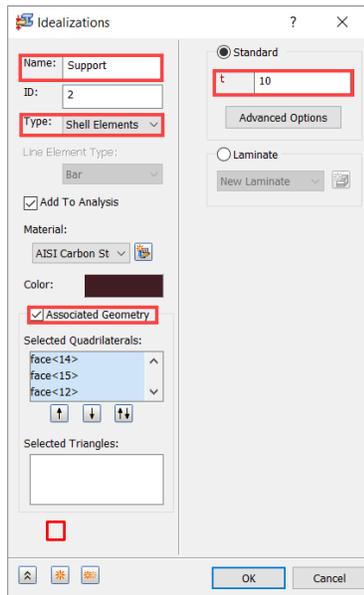


The default path to access library is C:\Program Files\Autodesk\Inventor Nastran 2023\In-CAD\Materials.

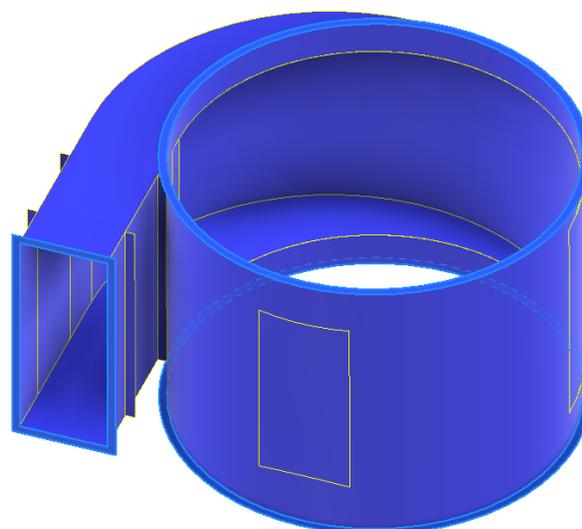
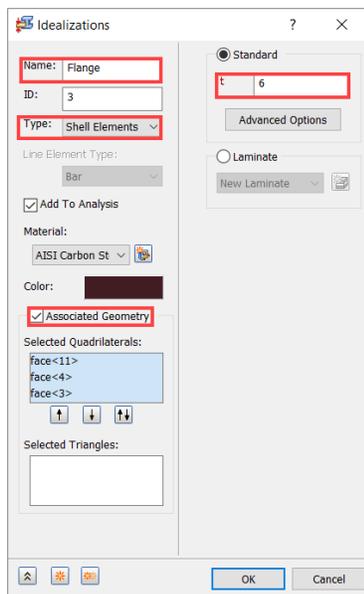
5. Select **Idealizations** tab > Select **Shell Elements** for Type of Idealizations > Specify **Body** for Name of Idealizations > Specify **3mm** for t > Select **Associated Geometry** > Right click in selection entities box > Select **Face Chain** > Select all faces making up body of the inlet.



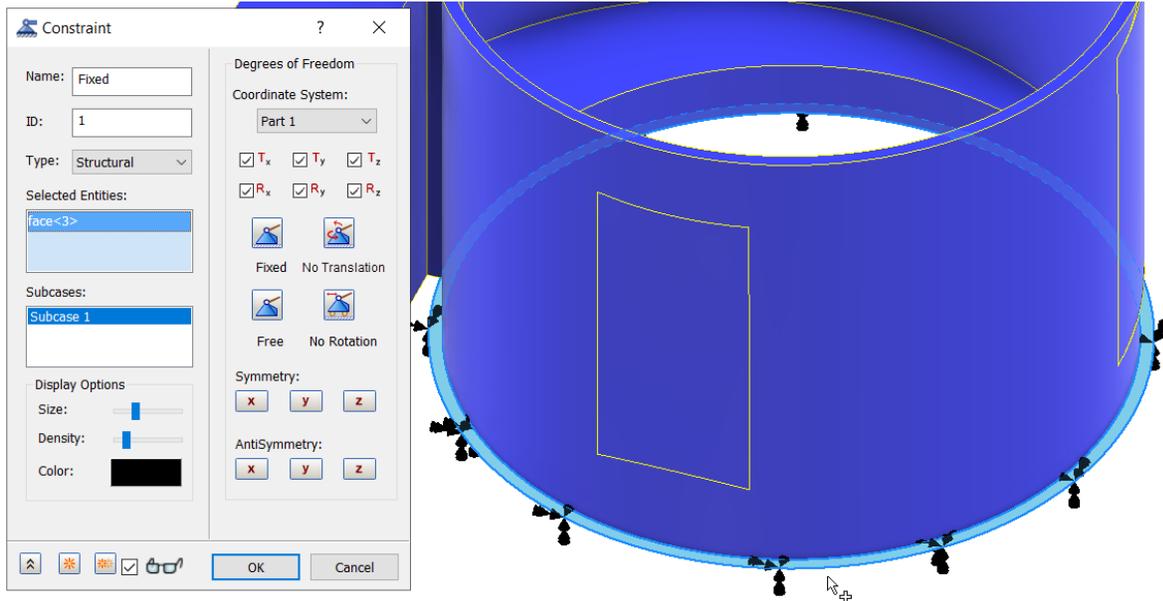
- Click **New** > Select **Shell Elements** for Type of Idealizations > Specify **Support** for Name of Idealizations > Specify **10mm** for t > Select **Associated Geometry** > Select the 4 highlighted faces as shown below.



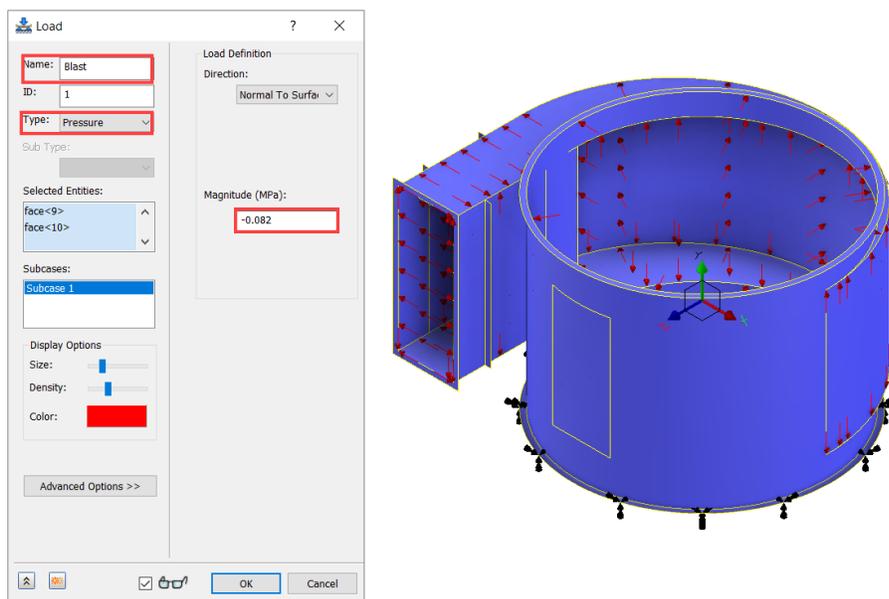
- Click **New** > Select **Shell Elements** for Type of Idealizations > Specify **Flange** for Name of Idealizations > Specify **6mm** for t > Select **Associated Geometry** > Select the 3 highlighted flange faces as shown below.



- Click **OK** > Select **Constraints** > Specify **Fixed Constraints** for Name > Select bottom flange as shown below > Select **Preview** so you can see constraint symbol. Adjust display options as desired.



- Click **OK** > Select **Loads** > Specify **Blast** for Name > Select **Pressure** for Load Type > Specify **-0.082** for Magnitude (MPa) > Select **Face Chain** option from Selected Entities box > Select all faces making up body of the inlet (No Support Plates and Flanges) > Select **Preview** so you can see load symbol. Adjust display options as desired.



10. Click **OK** > Select **Mesh Settings** > Specify **50** for Element Size (mm) > Select **Linear** for Element Order > Select **Continuous Meshing**.

11. Click **OK**. This will regenerate mesh.



Selecting linear elements will help to achieve results convergence quicker.



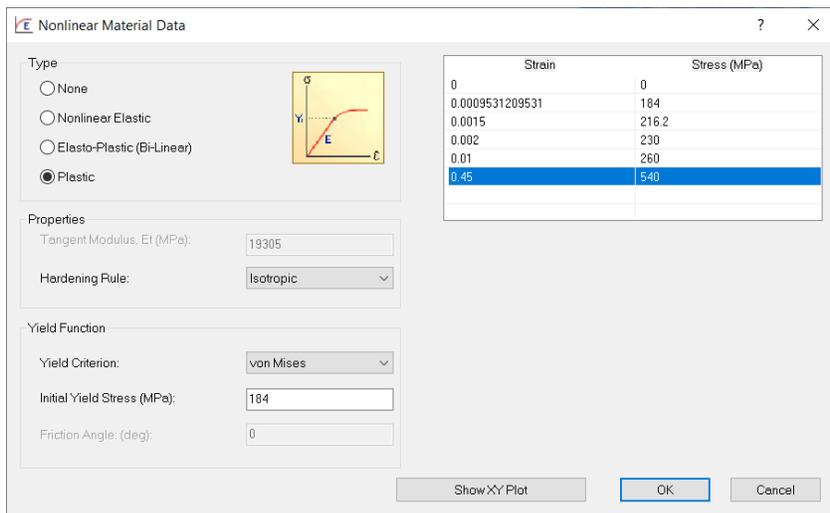
Selecting continuous meshing will connect nodes and elements at adjacent surface intersections avoiding the need to use contacts.



Continuous meshing will only work if surfaces have no gaps between them.

12. Double click **Analysis 1 [Linear Static]** > Select **Nonlinear Static** for Analysis Type > Click **OK**.

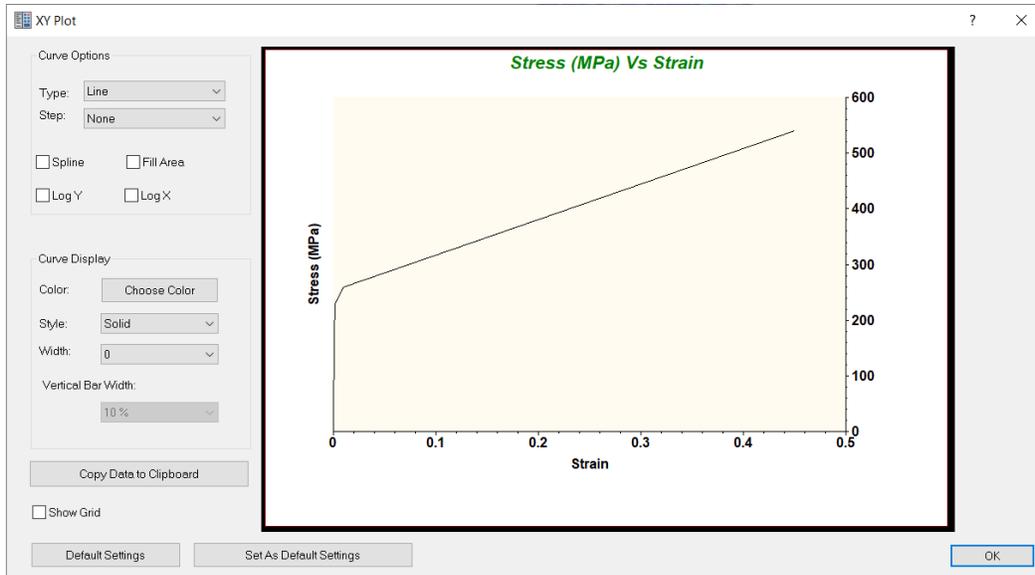
13. Right click **AISI Carbon Steel 304** material > Select **Edit** > Select **Nonlinear** > Select **Plastic** option > Specify the following values to define the stress and strain curve. First two rows already specified.



Strain	Stress (MPa)
0	0
0.0009531209531	184
0.0015	216.2
0.002	230
0.01	260
0.45	540

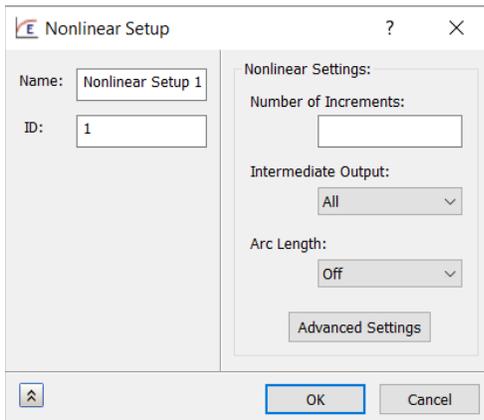
AISI 304	
Strain	Stress
0.000953	184
0.0015	216.2
0.002	230
0.01	260
0.45	540

14. Select **Show XY Plot**.



15. Click **OK** three times to exist all dialogue boxes.

16. Double click **Nonlinear Setup 1** > Select **All** option for Intermediate Output.



17. Click **OK**.



Selecting All will save all converged intermediate and bisected increments in the results file.



Nastran will calculate the number of increments automatically, if left blank. Typically, a run will complete after 10 iterations.

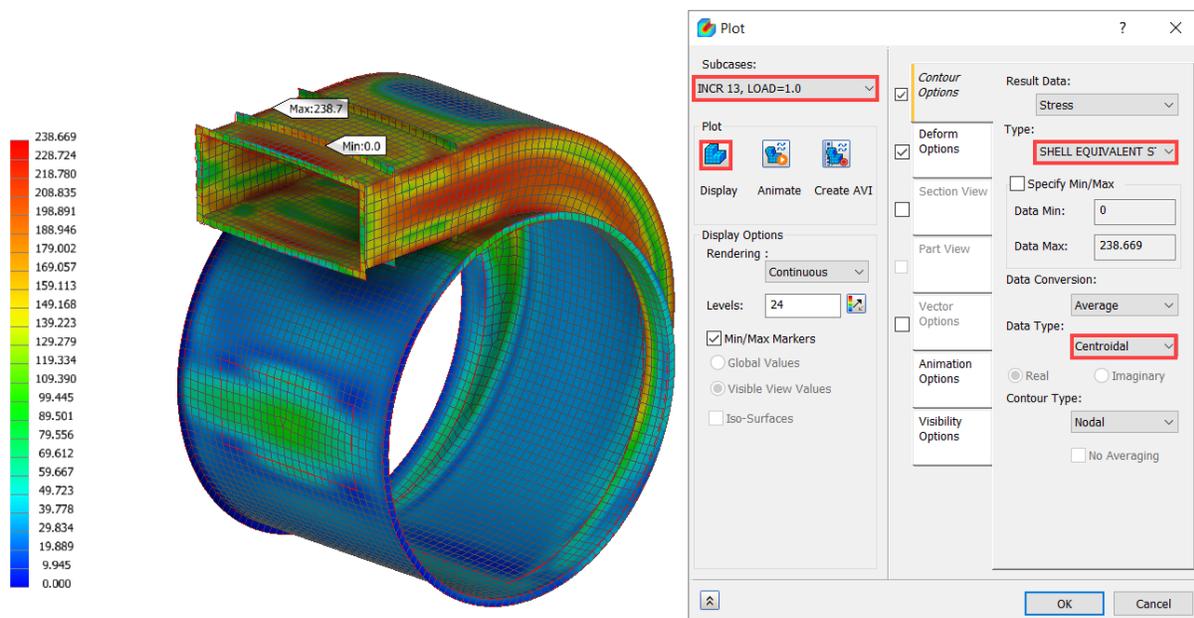
## Run simulation and analyse

18. Select **Run** > Click **OK** when run is complete.



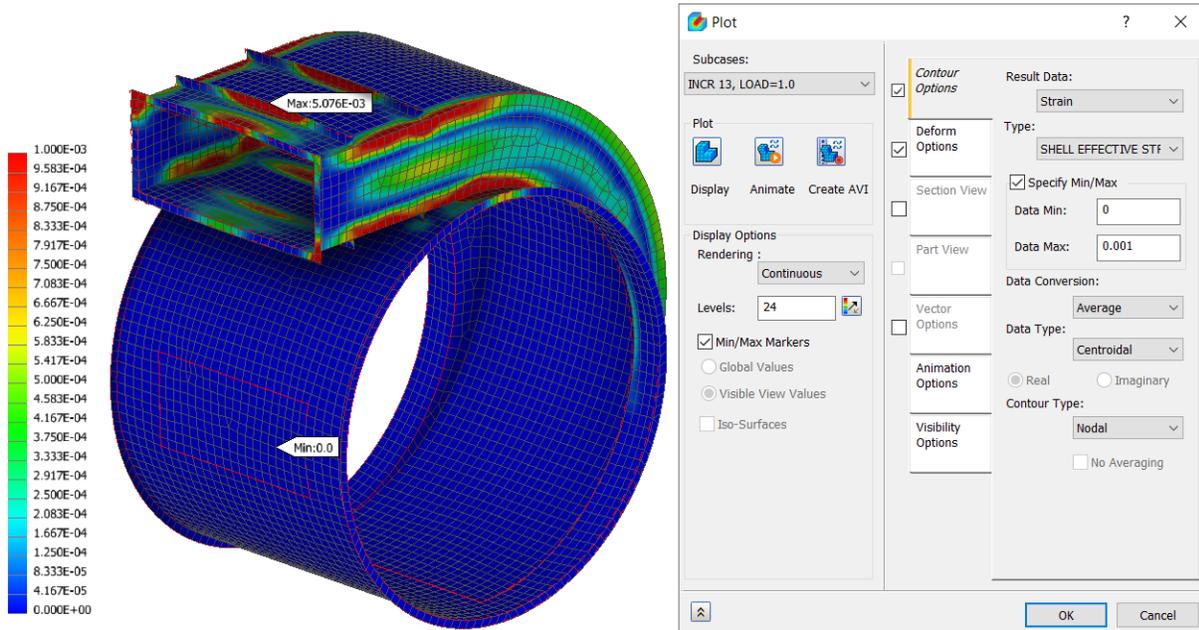
Depending on computer specification this can take up to 4mins.

19. Right click **Results** > Select **Edit** > Select increment showing **LOAD = 1.0** > Select **SHELL EQUIVALENT STRESS** > Select **Centroidal** for Data Type > Select **Visibility Options** > Switch visibility off for loads and constraints.



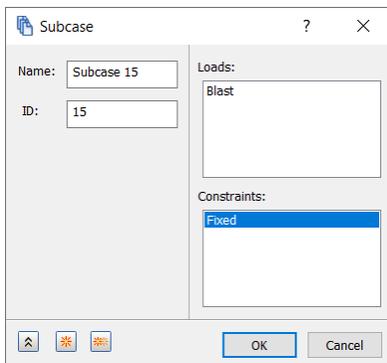
Equivalent Stress results in Nastran follow the stress and strain curve specified in the earlier steps.

20. Click **OK** > Select **Strain** from the results heads-up bar > Select **Options** from the Results panel > Select **Contour Options** from the Plot dialogue box > Select **Specify Min/Max** > Specify **0.001** for Data Max > Select **Display** to update results.



The component experiences up to 0.5% strain.

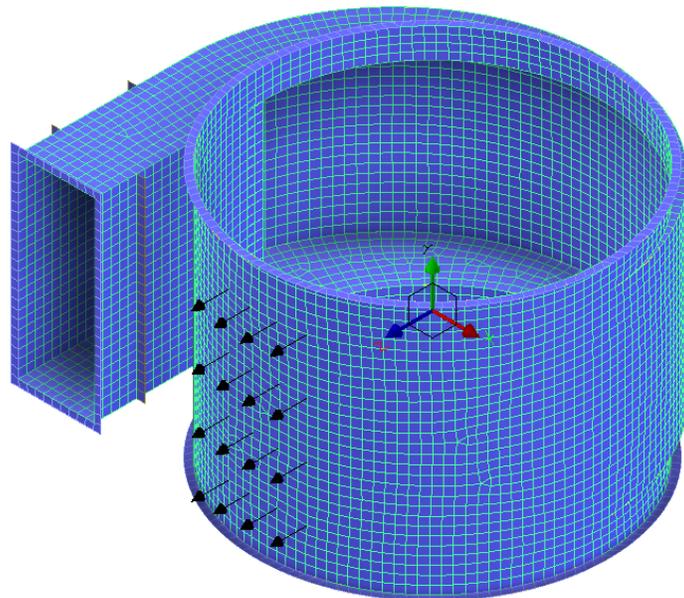
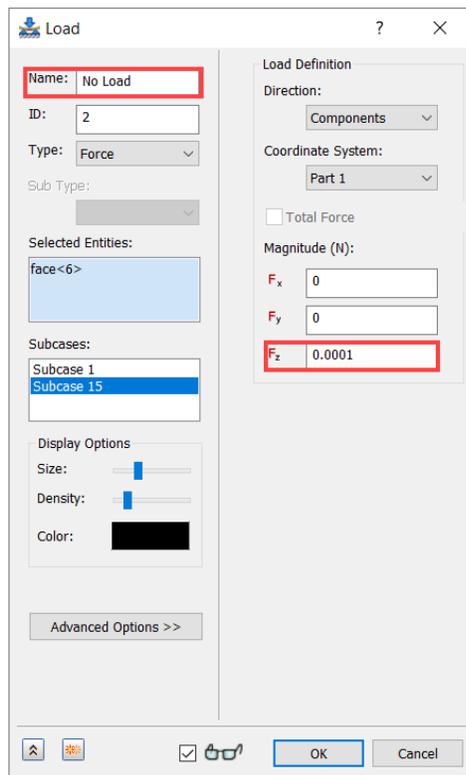
21. Click **OK** > Right click **Subcases** > Select **New** > Select **Fixed** constraint.



In Nonlinear analysis subcases are linked, unlike linear analysis where they are independent of one another.

This subcase will start from the previous deformed shape as a result of the blast. In this subcase no blast load will be specified, and we will be able to determine the permanent deformation after the blast load.

22. Click **OK** > Right click **Loads** in Subcases 15 (new subcase) > Select **New** > Select highlighted face > Specify **0.0001** for Magnitude (N) for **Fz** direction > Select **Preview** so you can see load symbol. Adjust display options as desired.

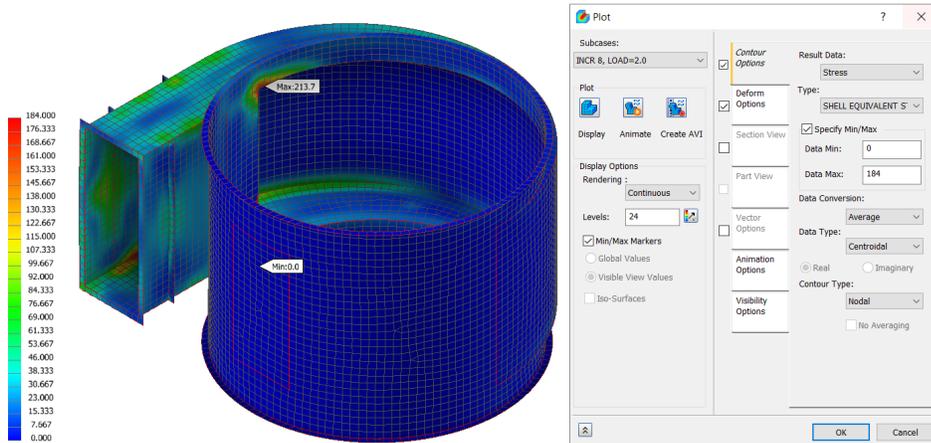


 For analysis to run we need to specify a negligible load. Location of the load is not important

23. Click **OK** > Select **Run**.

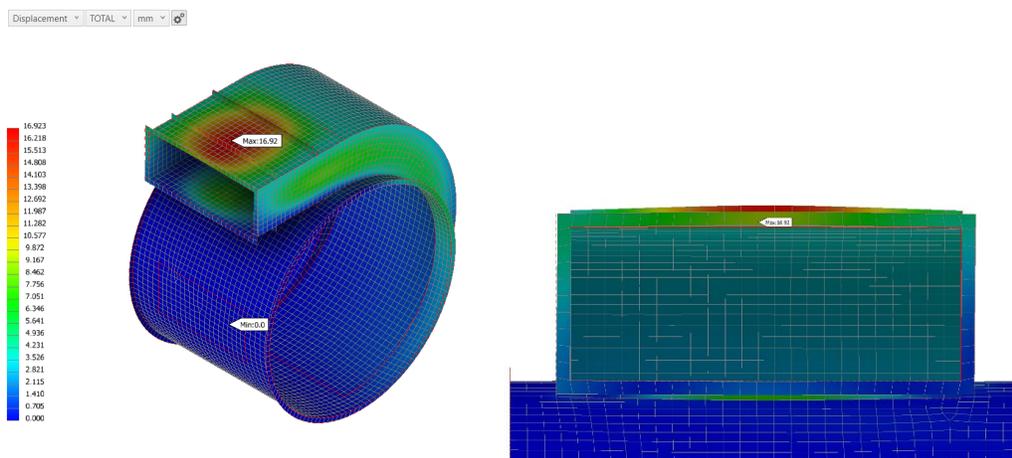
24. Click **OK** when run is complete.

25. Select **Shell Equivalent Stress** from the results heads-up bar > Select **Options** from the Results panel > Select increment showing **LOAD = 2.0 (No-load)** > Select **Contour Options** from the Plot dialogue box > Select **Centroidal** for Data Type > Select **Specify Min/Max** > Specify **184** for Data Max > Specify **0** for Data Min > Select **Display** to update results.



The contour plot is showing residual stresses in the component as a result of plastic deformation. So once the load is removed as in this subcase, the material tries to recover the elastic part of the deformation but is inhibited from full recovery due to the adjacent plastically deformed material. Residual stresses can affect fatigue life if the component is subjected to repetitive and cyclic loading. This is not the case in this example.

26. Click **OK** > Select **Displacement** from the results heads-up bar.

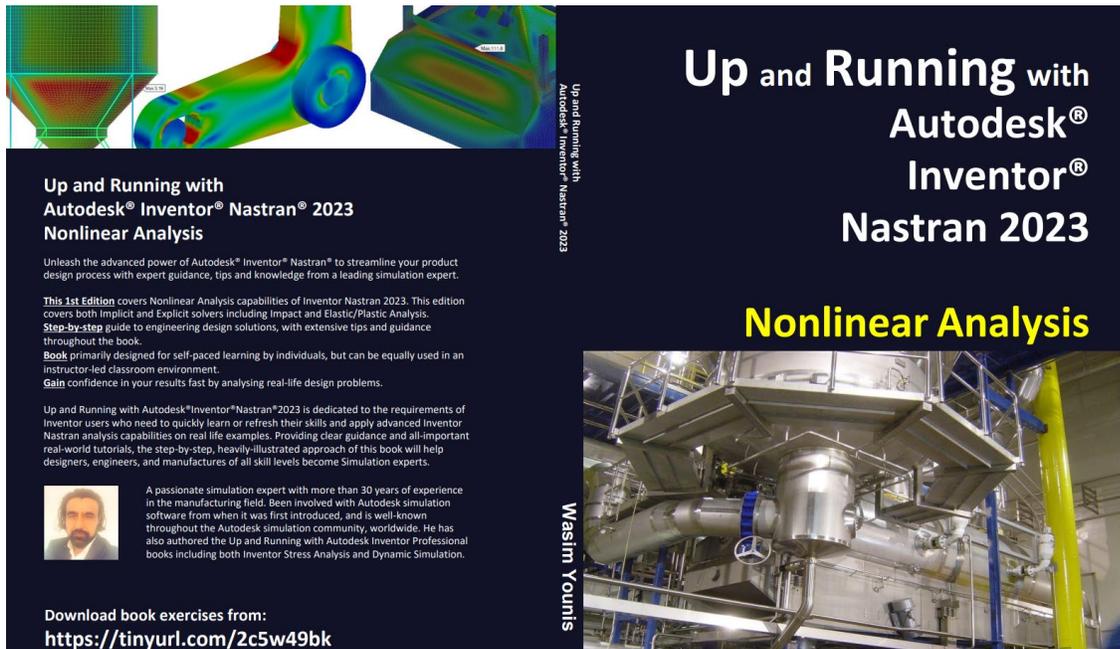


This shows permanent deformation of 16.9mm of the inlet as a result of 5% strain.

27. Close File.

The step-by-step workflow for Dellner and EMC example is in my new Up and Running with Autodesk Inventor Nastran 2023 – Nonlinear Analysis.

NB: Due to copyright issues could not include in this handout.



Available from Amazon worldwide. (Image hyperlink takes you to Amazon.com)

This book has been written using actual design problems, all of which have greatly benefited from the use of advance simulation technology. For each design problem, I have attempted to explain the process of applying nonlinear analysis using a straightforward, step by step approach, and have supported this approach with explanation and tips. At all times, I have tried to anticipate what questions a designer or development engineer would want to ask whilst he or she were performing the task using Inventor Nastran.

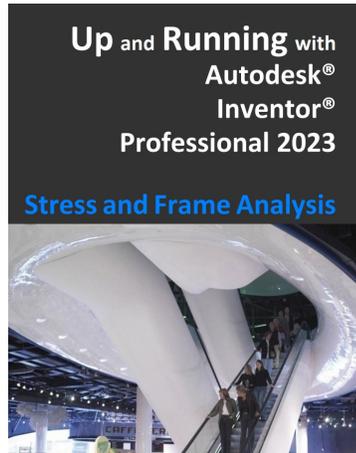
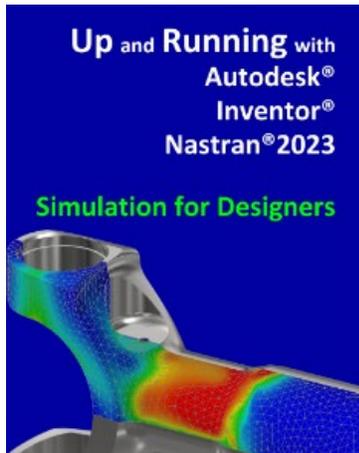
The design problems have been carefully chosen to cover the most popular nonlinear analysis capabilities of Inventor Nastran and their solutions are universal, so you should be able to apply the knowledge quickly to your own design problems with more confidence.

Chapter 1 provides an overview of Inventor Nastran Nonlinear and the user interface and features so that you are well-grounded in core concepts and the software's strengths, limitations, and work arounds. Each design problem illustrates a different unique approach and demonstrates different key aspects of the software, making it easier for you to pick and choose which design problem you want to cover first; therefore, having read chapter 1 it is not necessary to follow the rest of the book sequentially.

This book is primarily designed for self-paced learning by individuals but can also be used in an instructor-led classroom environment.

### Further Resources and Learning

The following books have also been authored by the speaker and are available from Amazon worldwide.



If you have any further questions, you can post them on my LinkedIn User group.

[Up and Running with Autodesk Simulation | Groups | LinkedIn](#)

My contact details if you have any further questions

Work email: [wasim.younis@symetri.com](mailto:wasim.younis@symetri.com)  
Personal email: [younis\\_wasim@hotmail.com](mailto:younis_wasim@hotmail.com)  
Mobile: +44(0)7980 735244  
LinkedIn: [www.linkedin.com/in/wasimyounis/](http://www.linkedin.com/in/wasimyounis/)