MFG225301

Up and Running with Autodesk Nastran In-CAD

Wasim Younis
Symetri

Learning Objectives

- Learn how to perform simulation using Autodesk Nastran In-CAD
- Learn how to interpret results using Autodesk Nastran In-CAD
- Learn how to apply industry best practices
- Learn top simulation tips for using Autodesk Nastran In-CAD

Description

Today Autodesk Nastran In-CAD software is available as part of Product Design & Manufacturing Collection, meaning many designers and engineers like you have the capability to adopt simulation early in the design process as this is where it has the most impact. Despite software availability and simulation continually becoming easier to use, there remains a fundamental question on the mind of every designer and engineer: How do I know my results are correct? This is the main reason for the lack of Autodesk Nastran In-CAD adoption within workplace. Autodesk Nastran In-CAD, although easy to use, has a comprehensive set of tools to help you answer this question and make sense of your results. These include mesh convergence, numerous result plots, section views, and much more. The session will include workflows and tips based on industry best practices. The goal of this session is to make you more confident in using Autodesk Nastran In-CAD software, and thus help you make better and more-innovative products within your workplace.

Speaker

An Autodesk simulation solutions manager with more than 30 years of experience in the manufacturing field, including working at Rolls Royce and British Aerospace. Has been involved with Autodesk simulation software from when it was first introduced, and is well-known throughout the Autodesk simulation community, worldwide.

He has also authored the Up and Running with Autodesk Inventor Professional and Nastran In-CAD books. He also runs a dedicated forum for simulation users on LinkedIn – Up and Running with Autodesk Simulation.

Currently he is employed @ Symetri (http://www.symetri.com) – an Autodesk value added services partner across UK and Northern Europe.

Introduction

The aim of this handout along with the presentation is to provide you with information to help you become more confident in the use of Nastran In-CAD. Since September 2018 I have been regularly publishing short "Simulation for Designers" articles on Linkedin.com. With the view that not everyone will be active on Linkedin.com I have decided to collate all the articles here in this handout with some additional information. In the hope you can quickly become Up and Running with Nastran In-CAD using single document.

Article 1 - Designers have a good idea of where they expect highest stress results.

Simulation has come a long way from the days when it was used by a few select experts. Today the technology, like Autodesk Nastran In-CAD, has become more accessible and easier to use. This in turn makes it easier for designers and engineers like you to adopt simulation early on within the design process as this is where it has the most impact. Despite simulation technology becoming easier to use there remains one fundamental question on the mind of every designer and engineer: How do I know my results are correct? Nastran In-CAD, although easy to use, has a comprehensive set of tools to help you answer this question and make sense of your results. These include mesh convergence, nodal and elemental result plots, section views, and much more including workflows and tips based on industry best practices. Over the coming weeks I will be sharing some of my experiences in the hope to make you more comfortable in using Autodesk Nastran In-CAD to help you make innovative products.

Today I would like to highlight that all engineers designing products typically have a very good idea of where they expect their design to experience high stresses without using any commercial simulation package. So, let's take the following simple example. Where would you expect the highest stress to be?

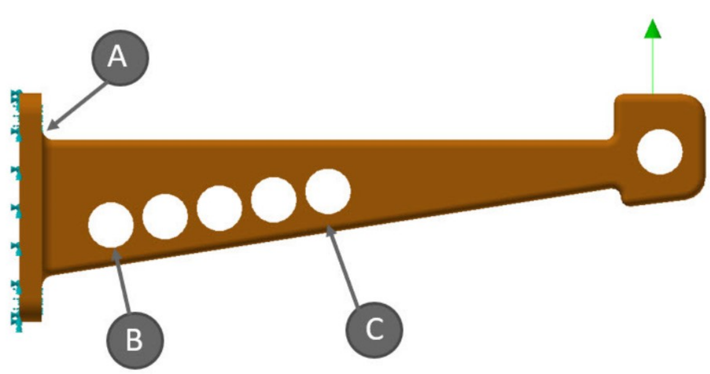


FIGURE 1 - CANTILEVER EXAMPLE

The answer is B. As the model gets more complex it becomes extremely difficult and tedious to determine the stress and safety factors of your design with just hand calculations. This is where simulation can help and provide more insight into the model in addition to being able to calculate stress results. This then leads to my next section on how do I know? my stress results are correct (or converged-meaning will not change if I change the mesh). Stay tuned till next week.

Article 2 - Never rely on one set of Analysis Results!

The only exception to this is displacements results. However, in most cases we are usually interested in stress results which are required to calculate the factor of safety of our designs. When analysing results, we need to make sure the stress value is not affected by changing the mesh size. I usually get asked a lot on "What element size should I use to get an accurate result?" In response to the question I usually say, "I honestly don’t know" but I also say, "Is analyse your results 3 times with a different mesh size in each analysis". This is more formally known as manual mesh convergence (sometimes also referred to as mesh sensitivity study). Below is my suggested workflow*.

Step 1 - Analyse with default mesh size.

Step 2 - Analyse again by reducing the mesh element size by half.

Step 3 - Analyse again by further reducing the Step 2 mesh size by half.

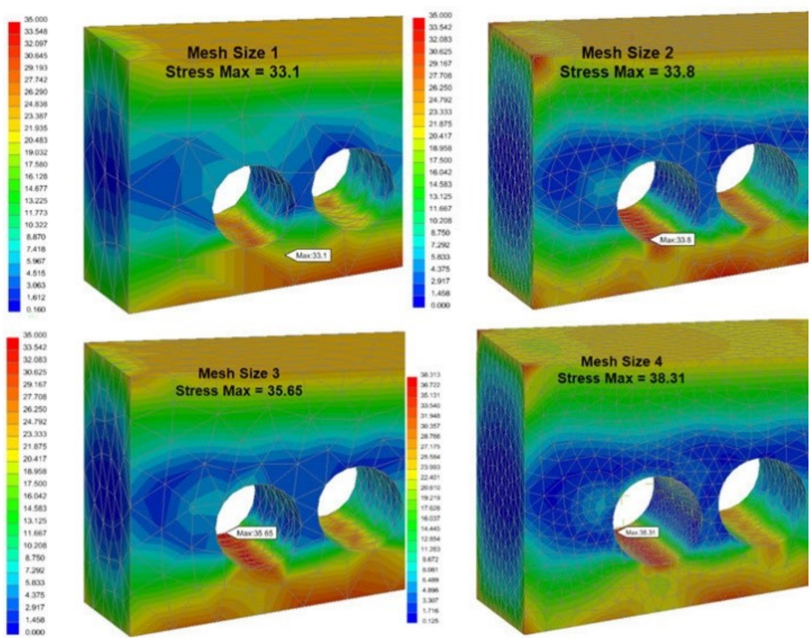


FIGURE 2 – MESH SENSITIVITY

Sometimes you may need to run an analysis fourth time in the case when stress comparison is either very close to 10% or above it. If this is the case I suggest you use local mesh control at the area of interest rather than changing the global mesh size. This will help to reduce the file size in addition to quicker run times.

However, in some cases the stress value will keep on rising as you make the mesh finer. This phenomenon is referred to as stress singularities (or hotspots) which leads to my next article on stress singularities and how to avoid them. Stay tuned till next week

* Some software's have the ability to run a mesh sensitivity study.

Article 3 – Stress Singularities

Are a major headache when analysing results as they considerably distort results? They are also a main cause for non-convergence of results. Which can further lead to lack of confidence in results and in extreme cases not using simulation at all.

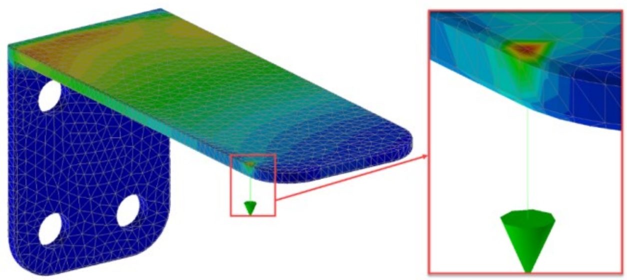


FIGURE 3 – STRESS SINGULARITIES AT POINT LOADS

So, what is stress singularity and why does it occur even in the simplest of designs and models? This can be best explained by the above simple bracket example which shows a high localised stress around the force applied on a point. This stress can be considerably higher than the operational stress and applying a denser mesh around this simply leads to a much higher stress. This phenomenon is known as stress singularity where the stress can become infinite, as illustrated by the following formula;

$$\textbf{Stress (infinite)} = \frac{\text{Force}}{\text{Area of point (almost} = 0)}$$

Therefore, to avoid stress singularities when applying loads, it is recommended not to apply loads at points and small edges. Stress Singularities can also occur by applying constraints on points and small edges – even faces with sharp corners as illustrated here.

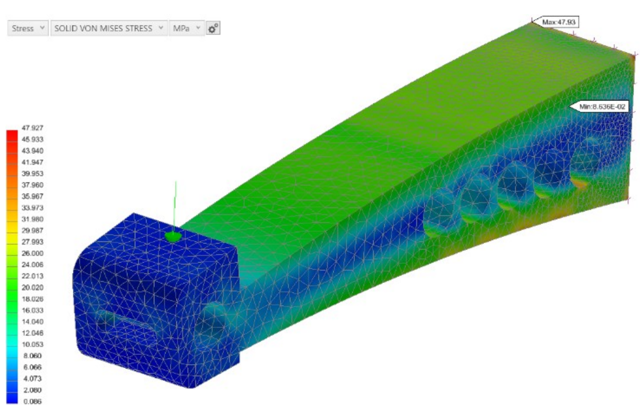


FIGURE 4 – STRESS SINGULARITIES AT BOUNDARY CONDITIONS

Finally, another cause of stress singularity is over-simplification of components as illustrated below.

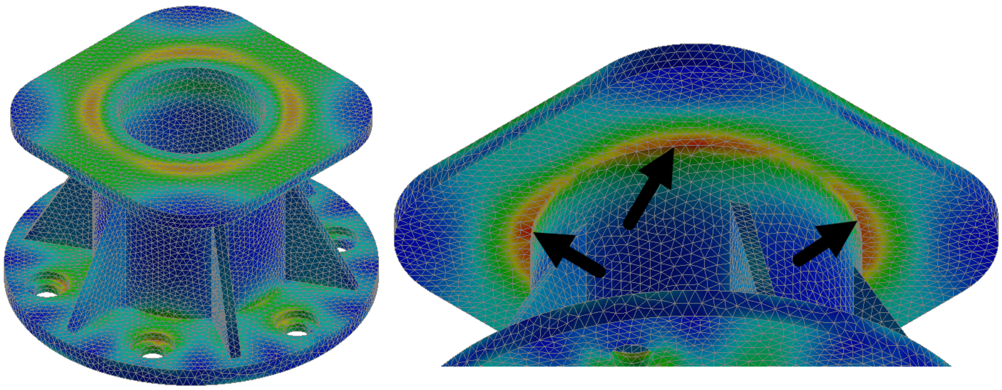


FIGURE 5 – STRESS SINGULARITY DUE TO MODEL SIMPLIFICATION

In this example, the fillets have been removed to simplify the analysis; however, when we keep refining the mesh, the maximum stress value does not converge as all the stress is concentrated around the edge, as shown in the above picture. In this scenario it would be advisable to un-suppress the fillets to help distribute the load more uniformly as shown below.

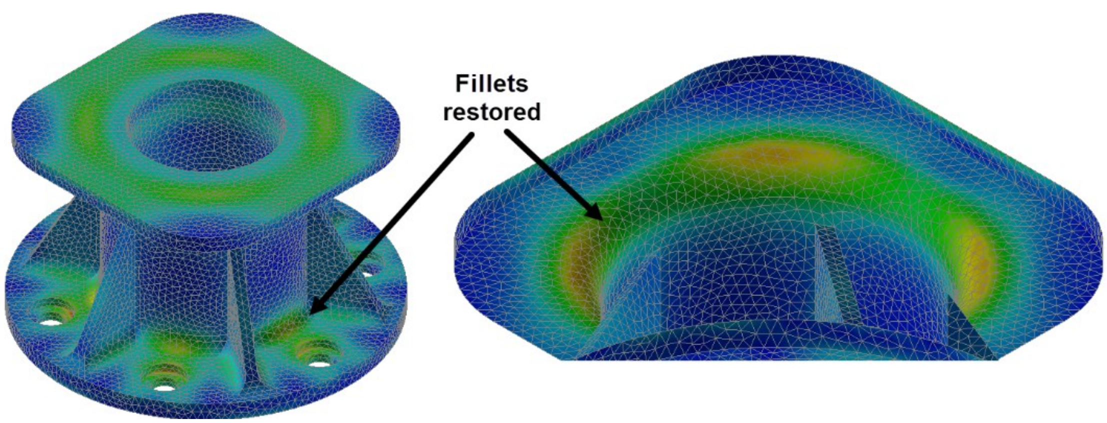


FIGURE 6 - STRESS SINGULARITIES REMOVED BY INTRODUCING FILLETS

Over-time and with experience it will become easier and easier to interrogate results of models and designs with stress singularities present.

Nastran In-CAD provides a comprehensive set of post-processing results that can help you gain more confidence when analysing results. Stay tuned till next week.

Article 4 - Fringe Display Stress Plots

As mentioned in the previous article, based on best practice, there is a need to analyse results at least three times, to check whether the stress results have converged. In most cases, hopefully, the difference in stress values will become minimal. But however due to stress singularities the stress values will not converge. In these scenarios Nastran In-CAD's comprehensive set of post processing tools can proof to be helpful. As there are many post-processing tools available within Nastran In-CAD here I will initially discuss about Fringe Display plots.

The main benefit of using fringe results is that once we alter the maximum value of the legend bar any value higher than the new maximum will be displayed as grey. This can proof to be very useful as we can use this technique to isolate high stresses due to singularities and concentrate in the area of interest.

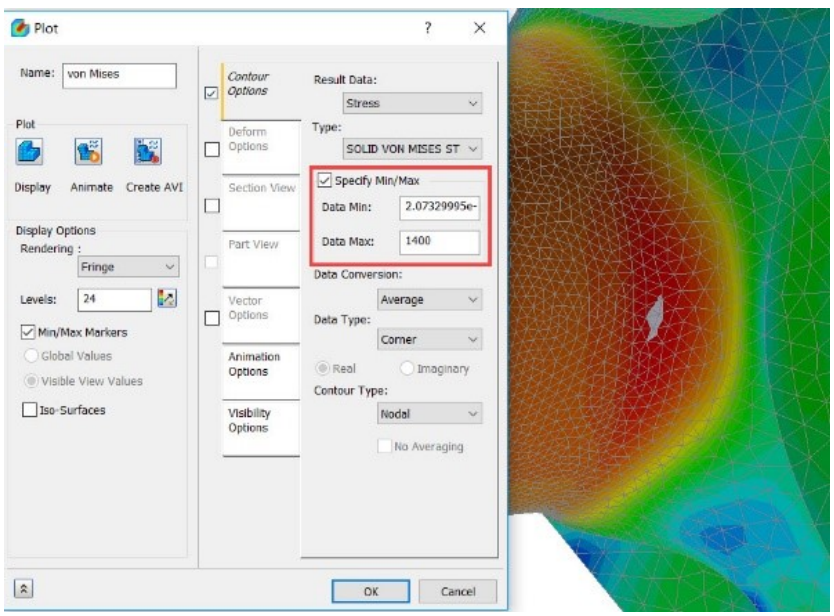


FIGURE 7 – FRINGE STRESS PLOT

It is important to note that altering the maximum value is not about fiddling (non-technical term) with results but is merely about presenting results which are meaningful. Once we have a minimal grey area we can use the probe tool available within Nastran In-CAD to display the maximum value.

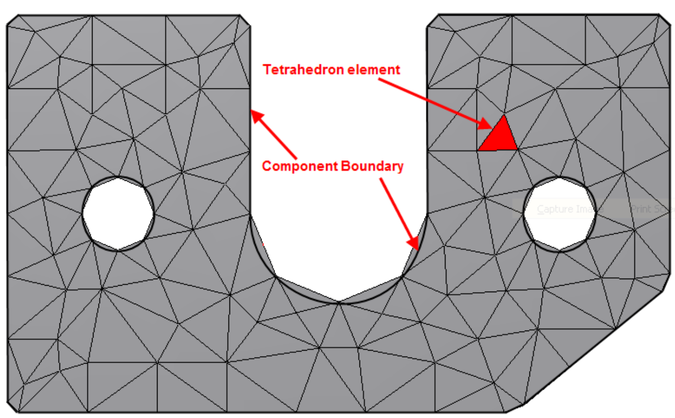
The benefit of using fringe display and probe like this is to help locate the maximum value location with ease in the area of interest within the model.

FIGURE 8 – RESULT PROBES

I hope you find this article helpful and stayed tuned till next week for more post processing options.

Article 5 – Corner (Nodal) Stress Results

Nastran In-CAD provides the ability to display stress results using either corner or centroidal results. It is important to note this does not apply to displacement results. In this article we will concentrate on corner stress plots with average being the default stress plot. Stress Analysis by its very nature provides an approximation of the real-world problem by breaking it down into finite elements as shown below



Discretization of a component into a number of Tetrahedron finite elements

FIGURE 9 – FINITE ELEMENT MODEL

At each element corner (junction) there is more than result from adjoining elements as illustrated below @ node B. Within Nastran In-CAD you can select whether you want to see the maximum value from all the adjoining elements or the average of all the values at the node.

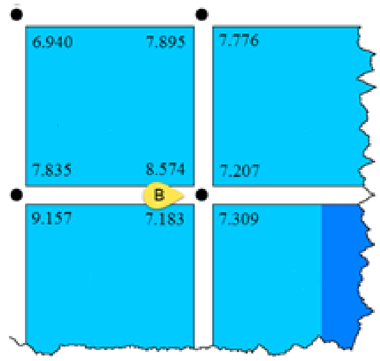


FIGURE 10 – CORNER NODAL RESULTS

So, if we look @ Node B we can have any of the following values displayed.

Maximum option	8.574
Minimum option	7.183
Average option	(8.574+7.207+7.183+7.309)/4 = 7.568

In an ideal world to obtain convergence in results, in addition to running analyses 3 times, we should expect the difference between minimum and maximum results to also be within 10% in the area of interest. The reason I am mention area of interest is that there may be scenarios when the location of the maximum or minimum value may not be in the same location most likely due to stress singularities. The difference between maximum and minimum results will reduce as the mesh size gets smaller and smaller. I hope you find this article useful stay tuned till next week when I will talk about centroidal results.

Article 6 – Centroidal (nodal) results

Carrying on from the previous article I will continue discussing about centroidal results within Nastran In-CAD. Below is an example of centroidal results from different elements.

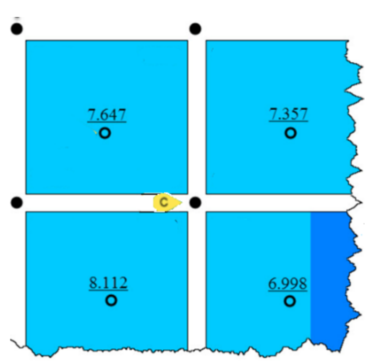


FIGURE 11 – CENTROIDAL NODAL RESULTS

So, if we look @ Node C we can have any of the following values displayed.

Maximum option	8.112
Minimum option	6.998
Average option	(7.647+7.357+8.112+6.998)/4 = 7.529

Again, in an ideal world we should also look for less than 10% difference between corner average and centroidal average results in the area of interest as another sanity check for convergence of results.

In the picture below, we can see the stress results are above 10% due to stress singularities present in the model so do take care interpreting results. In other words, the maximum and minimum values are not in the same location. In these situations, using fringe display and probes can proof to be useful as mentioned in the previous articles.

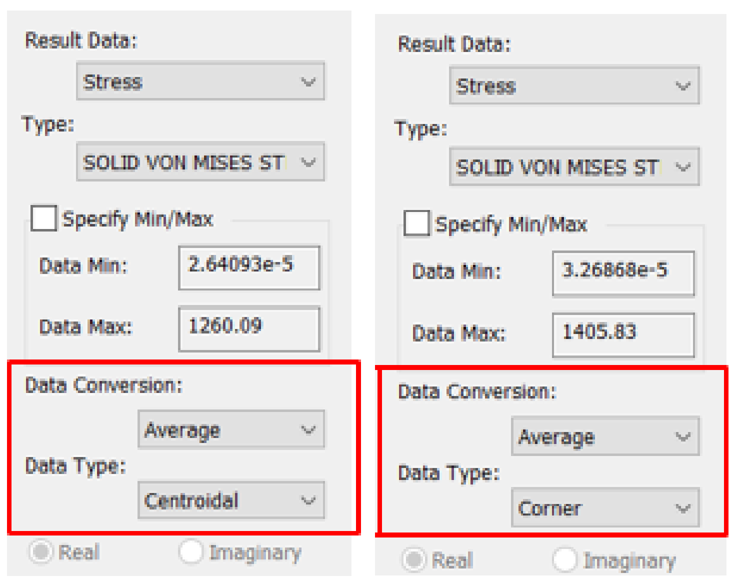


FIGURE 12 – AVERAGE STRESS RESULTS

There is also a great article on "Nastran In-CAD: Understanding Data Conversion, Data Type, and Contour Type" by John Holtz from Autodesk which you may find very useful. (https://goo.gl/45uesu).

I hope you found this article useful stay tuned till next week when I will discuss Safety Factor plots within Nastran In-CAD.

Article 7 – Safety Factor Plots

I frequently get asked that safety factor plots results within Nastran In-CAD are not correct. The reason for this is that safety factor values are calculated in a slightly different way. This is best explained by the following example. I will use 200MPa for yield limit for comparison purposes.

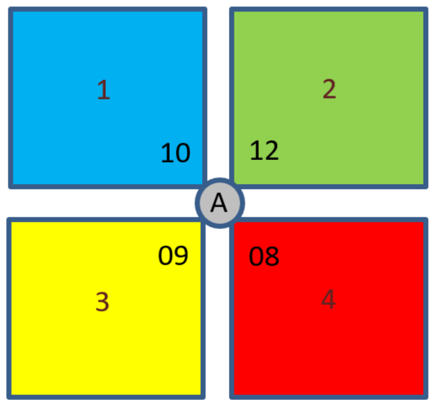


FIGURE 13 – ELEMENTS

Safety factor calculation based on average stress at location A is 20.51.

Average Stress value = (10+12+9+8)/4 = 9.75

Safety Factor = 200/9.75 = 20.51

Safety factor calculation based on how Nastran In-CAD displays plots at Location A is 20.97.

Safety Factor for Element 1 = 200/10 = 20

Safety Factor for Element 2 = 200/12 = 16.667

Safety Factor for Element 3 = 200/9 = 22.222

Safety Factor for Element 4 = 200/8 = 25

Average Safety Factor = (20+16.667+22.222+25)/4 = 20.97

So, this is the reason why there is slight difference in the safety factor results. Both results are correct as they have been calculated in slightly different ways. It is entirely up to you which value you take.

Below is a safety factor display showing a minimum value of 21.11.

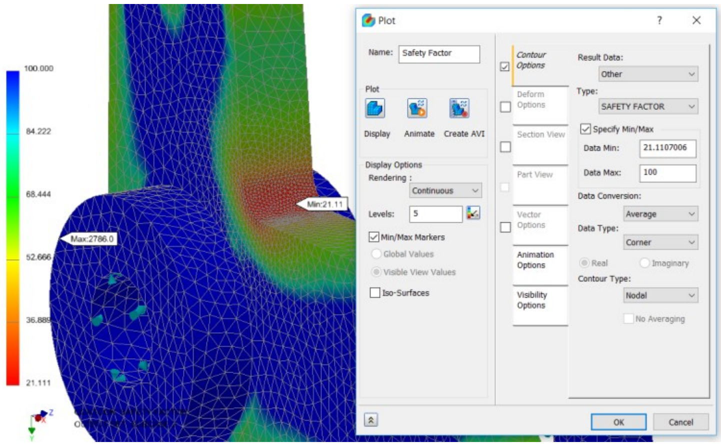


FIGURE 14 – SAFETY FACTOR RESULT PLOTS

Note changing Data Type to Corner or Centroidal will yield the same safety factor result. So, for the same model and results if we display the average centroidal results the maximum stress value becomes 9.4381MPa.

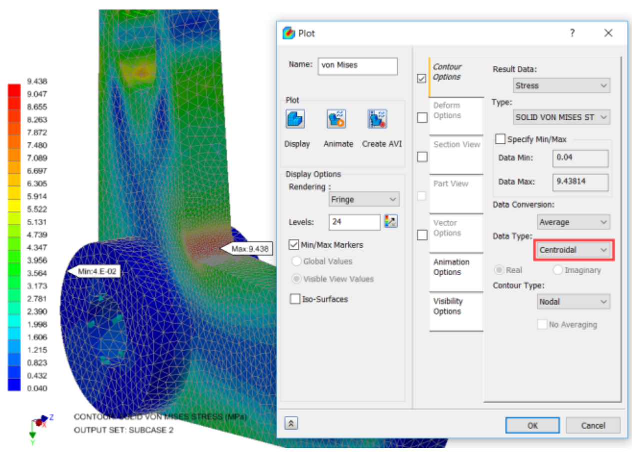


FIGURE 15 – CENTROIDAL AVERAGE STRESS PLOTS

So, based on this value the factor of safety (hand calculation) is 21.19.

$$\boldsymbol{Factor\ of\ Safety} = \frac{200}{9.4381} = 21.19$$

This is very close to the Safety Factor plot value of 21.11 with a difference of less than 0.5%. I hope you found this article useful.

There is also a great article on "Safety factor is not correctly reconciled in Autodesk Nastran In-CAD" by John Holtz from Autodesk which you may find very useful. (https://goo.gl/MKXRcv)

Article 7 was my last article before AU Las Vegas 2018. It is my intention to author more articles on social media including LinkedIn.

Useful Nastran In-CAD Parameters

There are thousands of parameters available for you to manually configure if you wish. Most of the parameter values are set by using the Nastran In-CAD user. The parameters are accessible by right clicking Parameters and selecting Edit from the bottom of the Nastran In-CAD browser.

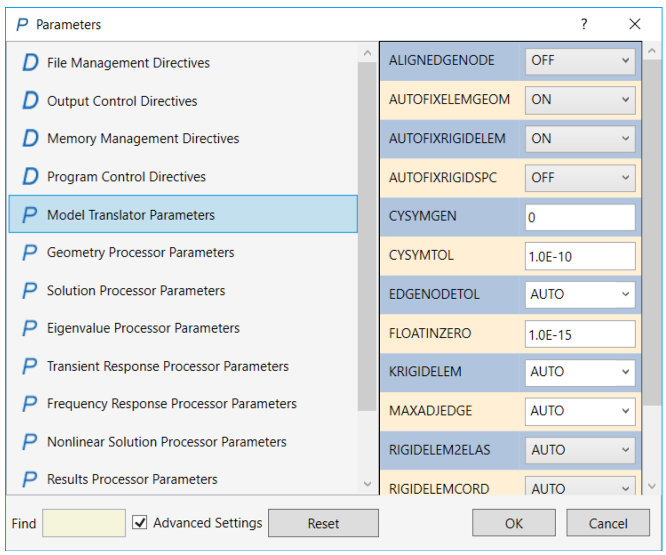


FIGURE 16 – NASTRAN PARAMETERS

Here I would like to mention three parameters which I think you will find useful.

1. NPROCESSORS
2. WARNING
3. TETFACEMINIATOL

NPROCESSORS

By changing the default value of 2 to 8 in the example below will allow Nastran In-CAD solver to use all 8 processors of the machine. This will help to solve analyses quicker.

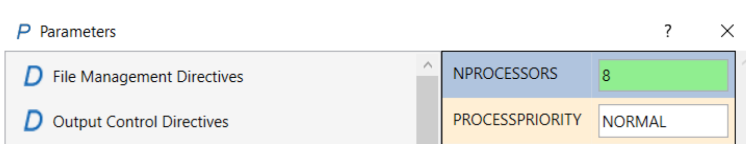


FIGURE 17 – NASTRAN ELEMENT WARNINGS

WARNING PARAMETER

Have you ever encountered millions of warnings (well a bit exaggerated) when you run an analysis within Nastran In-CAD.

```
WARNING G3015: TET ELEMENT 1847 HAS AN INTERIOR ANGLE LESS THAN 10.0

OPERATION RESUMED
PERCENT COMPLETE:    0

WARNING G3015: TET ELEMENT 1849 HAS AN INTERIOR ANGLE LESS THAN 10.0

OPERATION RESUMED
PERCENT COMPLETE:    0

WARNING G3015: TET ELEMENT 2005 HAS AN INTERIOR ANGLE LESS THAN 10.0

OPERATION RESUMED
PERCENT COMPLETE:    0

WARNING G3015: TET ELEMENT 2007 HAS AN INTERIOR ANGLE LESS THAN 10.0

OPERATION RESUMED
PERCENT COMPLETE:    0

WARNING G3015: TET ELEMENT 2287 HAS AN INTERIOR ANGLE LESS THAN 10.0

OPERATION RESUMED
PERCENT COMPLETE:    0

WARNING G3015: TET ELEMENT 2289 HAS AN INTERIOR ANGLE LESS THAN 10.0

OPERATION RESUMED
PERCENT COMPLETE:    0

WARNING G3015: TET ELEMENT 2565 HAS AN INTERIOR ANGLE LESS THAN 10.0

OPERATION RESUMED
PERCENT COMPLETE:    0

WARNING G3015: TET ELEMENT 2567 HAS AN INTERIOR ANGLE LESS THAN 10.0

OPERATION RESUMED
PERCENT COMPLETE:    0

WARNING G3015: TET ELEMENT 2754 HAS AN INTERIOR ANGLE LESS THAN 10.0

OPERATION RESUMED
PERCENT COMPLETE:    0
```

FIGURE 18 – NASTRAN ELEMENT WARNINGS

Although the warning numbers will reduce with a finer mesh but trying to get 0 warnings may not become possible without specifying a super very fine mesh (or alternatively by further simplifying the model). This in turn makes the run times very slow and file sizes excessive. The warnings do not necessarily mean your results are not correct as in practice we are defining a fine mesh locally in the area of interest to get results convergence. Meaning we are not interested in the coarse mesh regions of the model the probable cause of the remaining warnings. So therefore, changing the default value from ON to OFF will not mean your results are incorrect but hopefully will make you feel less stressed when you do not see all the warnings popping up during the analysis run.

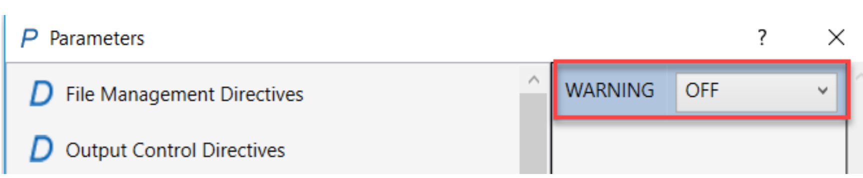


FIGURE 19 – WARNING PARAMETER

The WARNING parameter set to OFF will stop the warnings appearing during the solve but it will not remove the total warnings listed at the end of the output file as shown below.

```
GENERATING RESULTS NEUTRAL FILE
PERCENT COMPLETE:   100

DELETING FILE:  dfdbybvw4.ECD

MODEL ANALYSIS TIME SUMMARY

TOTAL CPU TIME = 12.7 SECONDS
WALLCLOCK TIME = 12.9 SECONDS

EXECUTION TERMINATED NORMALLY

TOTAL WARNINGS     = 26
TOTAL FATAL ERRORS = 0
```

FIGURE 20 – TOTAL WARNINGS MESSAGE

The good news is that we can remove total warning list by further relaxing the default settings of TETFACEMINIATOL parameter.

TETFACEMINIATOL

Below information is taken from Autodesk Nastran Online help.

TETFACEMINIATOL SHARE

Description	Type	Default
Tet element face minimum interior angle tolerance. A warning message will be given if a tet element has a face interior angle less than or equal to the specified value.	Real	5.0

FIGURE 21 – TETFACEMINIATOL PARAMETER DESCRIPTION

As the warnings are related to angles of less than 10%. We can remove all the warnings including the listing at the end of the output by relaxing the settings of TETFACEMINIATOL parameter as shown below.

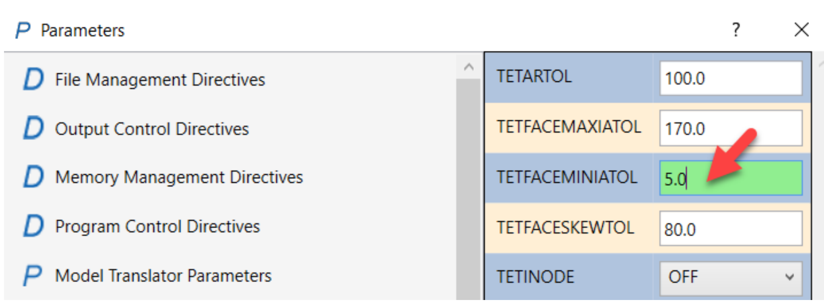


FIGURE 22 – TETFACEMINIATOL PARAMETER

By changing the default of 10 to 5 will remove the warnings and listing as shown below.

```
DELETING FILE:  dfdbybvw4.NDB

GENERATING RESULTS NEUTRAL FILE
PERCENT COMPLETE:  100

DELETING FILE:  dfdbybvw4.ECD

MODEL ANALYSIS TIME SUMMARY

TOTAL CPU TIME = 13.8 SECONDS
WALLCLOCK TIME = 14.1 SECONDS

EXECUTION TERMINATED NORMALLY

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

FIGURE 23 – ZERO WARNINGS MESSAGE

Ilogic and Nastran In-CAD

In the current version of Nastran In-CAD 2019 you have the ability to automate some of the standard commands that are required for all analyses. At the moment the following commands are supported by iLogic commands for Linear Static analyses (information taken from online help).

Delete Material - Use the DeleteMaterial command to delete a single material.
Add Material - Use the NewMaterial command to add a new material.
Delete Idealization - Use the DeleteIdealization command to delete a single idealization.
Add Idealization - Use the NewIdealization command to add a new idealization.
Mesh Model - Use the MeshModel command to define a mesh without midside nodes.
Generate Mesh - Use the GenerateMesh command to create the mesh.
Create Load - Use the Load command to create a new load.
Create Constraint - Use the strCmdForConstraint command to create a new constraint.
Run Analysis - Use the strCmdForSolve command to run the analysis.
Display Contour Plot - Use the strCmdForDisplayContour command to create a contour plot after the solution is complete.

As I am not an ilogic person I don't have much to say here apart from suggesting the following on line tutorials.

- iLogic Tutorial: Linear Static Analysis (https://goo.gl/K1TKyb)
- iLogic Tutorial: Gravity Load on an Aluminum Frame (https://goo.gl/2AiBkU)

In the presentation I will show you an example of how ilogic can help automate standard workflows within Nastran In-CAD. Here I would like to thank my colleague Chris Atherton from Symetri (chris.atherton@symetri.com) for helping with the ilogic part of the presentation.

Demonstrations.

In this lab I will be going through two real-life examples. The 1st one is a yoke part of a motorsport prop shaft assembly and the 2nd one is a lever part of a high security door system.

Yoke Example

GKN Motorsport is sub-sector of GKN Land Systems and manufactures driveshafts, prop shafts and CV joints for applications such as world rally and touring cars. This also includes driveline components and assemblies for high performance road cars. Motorsport parts are often highly stressed; however, weight is also an important aspect, and parts must be lightweight, without compromising performance

FIGURE 24 – MOTORSPORT PROP SHAFT

This demonstration will go through step by step analysis, including tips and workflows mentioned earlier, of a high strength yoke used on a WRC prop shaft as shown below

FIGURE 25 – YOKE

Ilogic Demonstrations

In this part of the demonstrations volunteers from the class will create or reuse existing models and then use ilogic form to setup and run Nastran In-CAD with the view to then look at results,

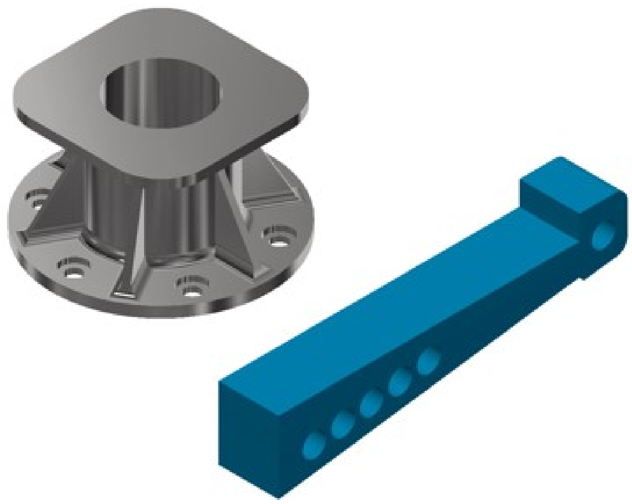


FIGURE 26 – SAMPLE PARTS TO BE ANALYSED USING ILOGIC FORM

Further Resources

The content in this handout and the presentation, including demonstration examples have been taken from my Up and Running with Autodesk Nastran In-CAD 2019 book published world-wide this year in August. The book has been written using actual design problems, all of which have greatly benefited from the use of simulation technology. For each design problem, I have attempted to explain the process of applying stress 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 Nastran In-CAD.

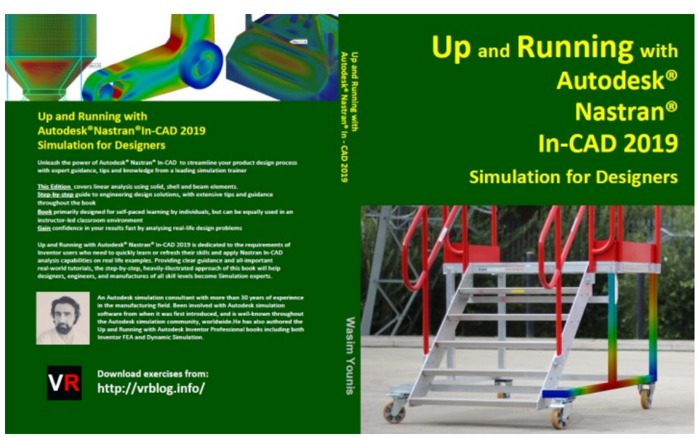


FIGURE 27 – UP AND RUNNING WITH AUTODESK NASTRAN IN-CAD BOOK COVER

The book is available worldwide through Amazon sites. Below are further links to get extra help and guidance to get you up and running with Nastran In-CAD.

Nastran In-CAD Forum -

https://forums.autodesk.com/t5/nastran-in-cad-forum/bd-p/75

Nastran In-CAD Online Tutorials –

http://help.autodesk.com/view/NINCAD/2019/ENU/?guid=GUID-DB7160BE-0C72-47B9-B5EF-FC4925B455CE

CONTACT DETAILS

- younis_wasim@hotmail.com
- wasim.younis@symetri.com
- https://www.linkedin.com/in/wasimyounis/
- +44 (0) 7980 735244