



AUTODESK UNIVERSITY 2015

IM10256

STRESSED – Take an Inventor Simulation Chill Pill

Wasim Younis
Symetri/CAD-Q, UK

Vince Adams
Autodesk Inc.

Learning Objectives

- Perform Dynamic Simulation
- Perform Stress Analysis
- Analyze and Interpret Results
- Create Innovative Designs

Description

This class will demonstrate workflows and processes to help you make better decisions enabling you to make innovative designs. Dynamic Simulation within Inventor Simulation allows you the designer to convert your static CAD models into a moving mechanism, allowing you to determine reaction forces, velocities, accelerations and much more. Whereas Stress Analysis within Inventor Simulation will allow you to determine whether your parts will fail in the real world, including the ability to optimize your designs enabling you reduce weight. This class is ideally aimed at the Inventor users who have very little knowledge of Inventor Simulation.

Your AU Experts

Wasim Younis is a Simulation Solutions manager at Symetri, with more than 20 years of experience in the manufacturing field, including working at Rolls Royce, British Aerospace, and Nuclear Electric. He has been teaching at AU for more than 5 years. Wasim has been involved with Simulation software since Autodesk first introduced it, and is well known throughout the Autodesk Simulation community. He has authored the *Up and Running with Autodesk Inventor Simulation* books, available worldwide. He also runs a dedicated forum for simulation users on LinkedIn: *Up and Running with Autodesk Inventor Simulation*.

Vince Adams has been a consultant, instructor, and advocate for finite element analysis (FEA) in product design for over 20 years. He has written 3 books on the subject, including the popular *Building Better Products with Finite Element Analysis*, as well as numerous articles for *Desktop Engineering* and *Design News* magazines. Vince has been an invited speaker and instructor with engagements all over the United States, as well as in Europe, South Africa, and Asia. He was nominated by peers to lead the founding of NAFEMS North America (www.nafems.org) and was selected by this organization to be on the Founding Member list for the international Professional Simulation Engineer program based on his career contributions to the simulation industry. Prior to focusing on simulation, Vince was a design engineer and engineering manager accumulating numerous patents, many due to novel uses of FEA (at the time) and learning firsthand the importance of properly applied simulation for improved innovation.

During a typical design process, you the designer go through a series of typical questions, such as: do the parts fit together? Do the parts move well together? Is there interference? Do the parts follow the right path? Even though most of these questions can be catered for by 3D CAD and rendering software, there may be other questions which cannot. For example, you may want to know the machinery time cycle. Is the actuator powerful enough? Is the link robust enough? Can we reduce weight? All these questions can only be answered by building a working prototype or a series of prototypes. The major issues with these methods are that they are timely and costly. An alternative cost- effective method is to create a working virtual prototype by using Inventor simulation. Inventor simulation allows you to convert assembly constraints automatically to mechanical joints, provides the capability to apply external forces including gravity, and allows the effects of contact friction, damping and inertia to be taken into account. As a result of this, inventor simulation provides reaction forces, velocities, acceleration and much more. With this information you can re-use reaction forces automatically to perform finite element analysis, hence reducing risks and assumptions. Ultimately all this information helps the designers to build an optimum product. In this class a real world example from British Waterways will be used to demonstrate inventor simulation workflows and processes.

Perform Dynamic Simulation

Dynamic Simulation within Inventor Simulation will be used to determine the maximum reaction loads acting on the **NEW** supporting structure underneath the bridge whilst opening and closing the canal bridge just under 30 seconds.



FIGURE 1: CANAL BRIDGE

The following is an illustration of a typical dynamic Simulation workflow

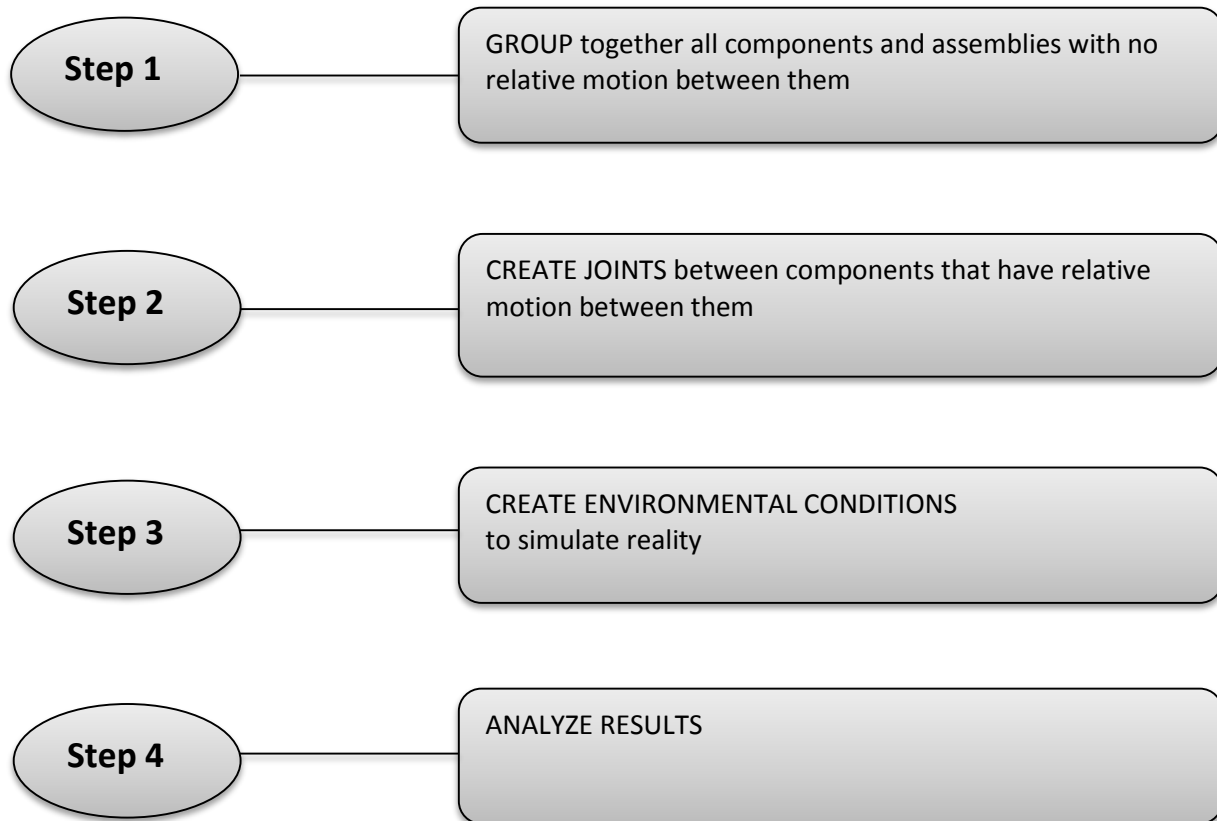


FIGURE 2: DYNAMIC SIMULATION WORKFLOW (REF UP AND RUNNING WITH AUTODESK INVENTOR SIMULATION)

Let's have a look at each step in more detail

Step 1.

Grouping components that do not have relevant motion between them significantly reduces the number of joints. This will ultimately make it easier to interrogate results with the added advantage of running simulations faster.

There are two options to group components together and both have their advantages and disadvantages.

Option 1 – Create subassemblies within Assembly environment. This method of restructuring your parts into subassemblies will affect your BOM database.

Option 2 – Weld components together within Simulation environment. This method will not alter your BOM database.



Step 2

The process of creating joints can be broken down into 2 stages

Stage 1 – Create standard joints

Stage 2 – Create nonstandard joints

Stage 1 - there are three options to create standard joints

Option 1 – Use Automatically Convert Constraints to Standard Joints. This is by far the quickest option to create joints by utilizing existing assembly constraints and joints.

Option 2 – Manually convert assembly constraints. This option although slower than option 1 gives you the designer full control on what assembly constraints are to be used when creating joints.

Option 3 – Create standard joints from scratch. This method is the slowest of all three options and more importantly does not make use of existing assembly constraints

Stage 2 – Comprises of creating nonstandard joints that do not make use of assembly constraints including Rolling, Sliding, 2D Contact and Force joints

Step 3

Once the appropriate joints have been created, the next step is to simulate reality. This can be achieved by applying any of the following:

- Joints – define starting position.
- Joints – apply friction to joints.
- Forces/torque – apply external loads.
- Imposed motion on predefined joints.
- Position, Velocity and Acceleration (constant values)
- Input Grapher – create specific motions (non-constant values)

Step 4

This is the final step, in which you can use the Output Grapher to analyze the results in joints, including velocities, accelerations and reaction forces

Once the reaction loads on the support structure in figure 3 have been determined you can then move into the stress analysis environment. It is here you can then analyze whether the structure will not fail during the opening or closing of the bridge.



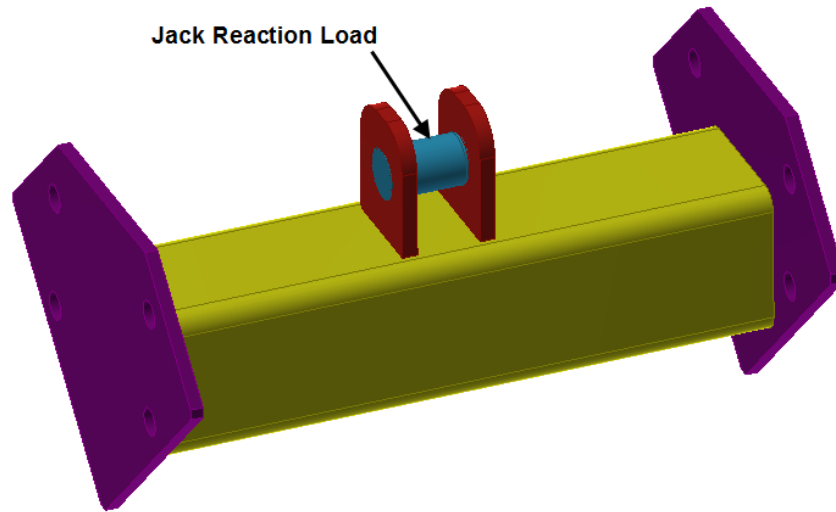


FIGURE 3: REACTION LOADS ACTING SUPPORTING STRUCTURE

Perform Stress Analysis

Stress analysis is an engineering discipline that determines the stress in materials and structures subjected to static or dynamic forces or loads. The aim of the analysis is usually to determine whether the element or collection of elements, usually referred to as a structure or component, can safely withstand the specified forces and loads. This is achieved when the determined stress from the applied force(s) is less than the yield strength the material is known to be able to withstand. This stress relationship is commonly referred to as factor of safety (FOS) and is used in stress analysis as an indicator of success or failure in analysis.

$$\text{Factor of Safety} = \frac{\text{Yield Stress}}{\text{Calculated Stress}} = \frac{\text{Ultimate Stress}}{\text{Calculated Stress}}$$

Factor of Safety can be based on either Yield or Ultimate stress limit of the material. The factor of safety on yield strength is to prevent detrimental deformations and the factor of safety on ultimate strength aims to prevent collapse, and can only be conducted by nonlinear analysis software. Autodesk Inventor can only perform linear analysis and hence FOS will more commonly be based on yield limit.



The process of creating a stress analysis involves four core steps:

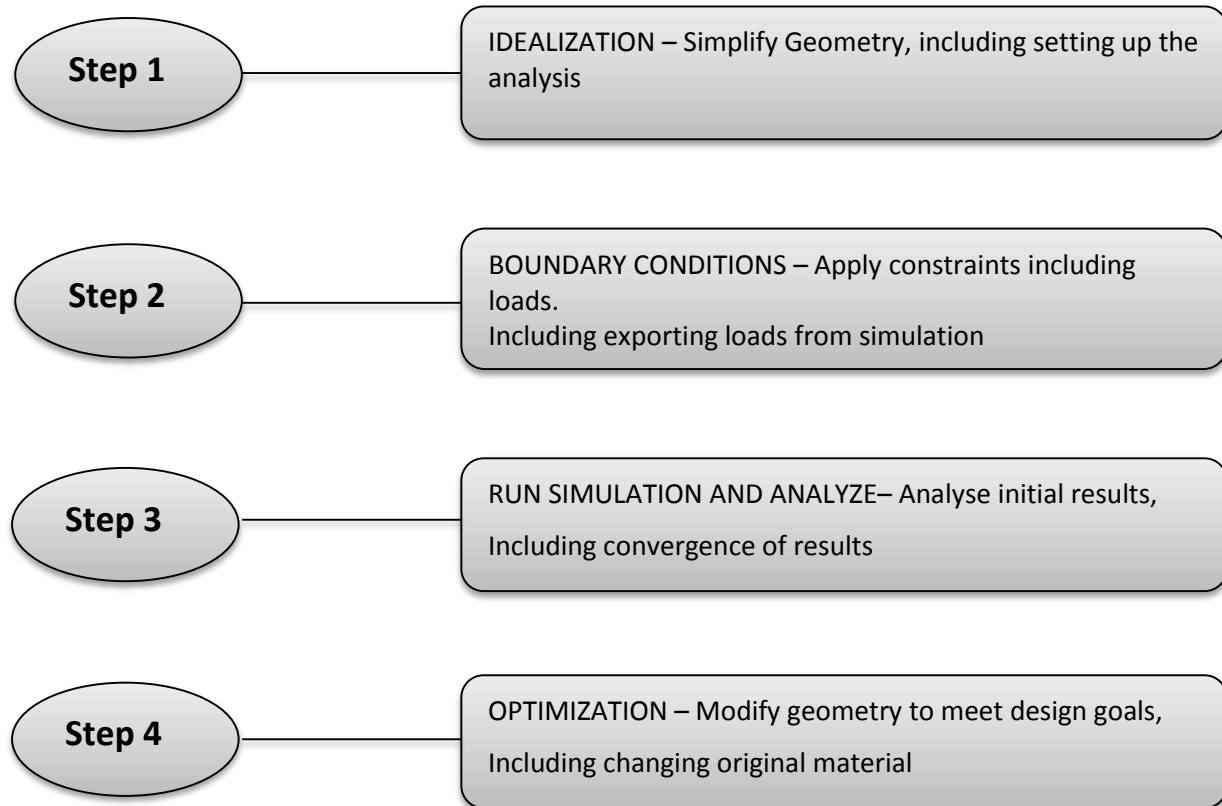


FIGURE 4: STRESS ANALYSIS WORKFLOW (REF UP AND RUNNING WITH AUTODESK INVENTOR SIMULATION)

Let's have a look at each step in more detail

Step 1

In Stress Analysis this is the most important step, in my opinion. This greatly has an impact on the speed and accuracy of the results. Assemblies and parts can be simplified and idealized within the modelling or Stress Analysis environment. Within Stress Analysis environment simplification is restricted to excluding non-structural components and features. A most common approach in idealization is to reduce the model into half a model, or even a quarter, that is if the geometry and loading are both symmetrical. In this class we will use a half model, although a quarter model could also be used.



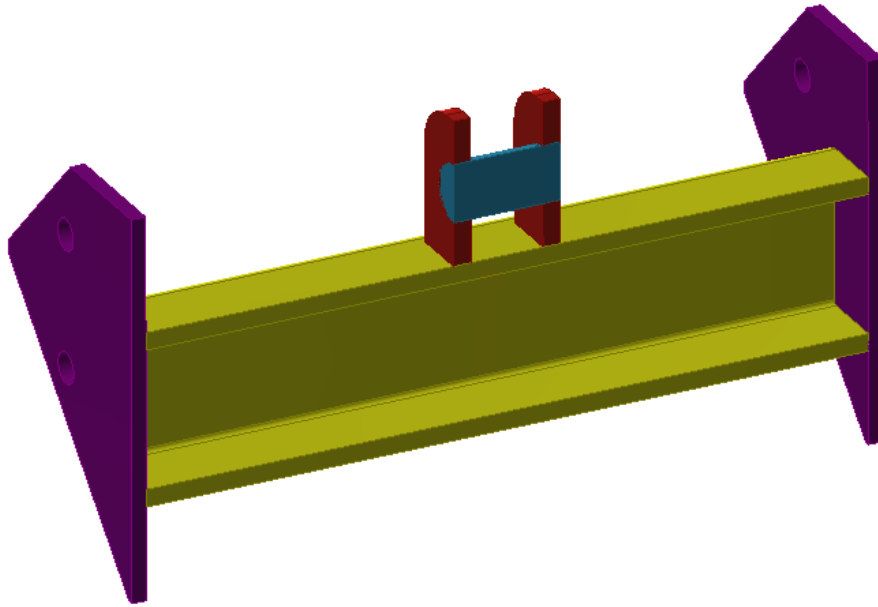


FIGURE 5: A HALF MODEL OF NEW SUPPORT STRUCTURE

Step 2

This step involves applying materials, constraints, loads and mesh. Stress Analysis uses the Inventor materials library and such the material properties are carried forward from the parts environment into the stress analysis environment. Regarding constraints there are only three types available and are fixed, pinned and frictionless. For example if the component is fixed using bolts or welds then you would use a fixed constraint. In terms of load there are various types available including force, pressure, bearing load etc. The load applied is the total load meaning if multiple faces are selected then the load will be divided by the number of faces selected. Stress analysis also allow the ability to control size of the mesh globally and locally. Mesh size will have little effect on displacement results but can have a significant impact on stress results. This will be discussed and presented in more detail in the class and is briefly mentioned in the Analyze and Interpret section.

Step 3

This step is primarily concerned with analyzing the results and making sure the design is fit for purpose. One of the most important question you may ask yourself at this stage of the process is how do I know the stress results are correct? One way to answer this question is to make sure the stress result convergences. One of the methods to help answer this question is to run simulation 3 times (minimum) with the mesh size getting finer in each simulation, in the area of interest. If the stress value remains within 10% difference between each simulation then we can assume the stress value has converged. It is normal to take the higher result from the last simulation for design verification purposes as shown in figure 7.



Step 4

This final step of the stress analysis process helps to make optimized designs. One example may be the reduce mass against certain design criteria's which can either or all be max operating stress, overall displacement etc. This is discussed more lately in the Create Innovative design sections

Analyze and Interpret Results

Analyzing and interpreting results are very important of the stress analysis process. Here one aspect of stress analysis we need to make sure of his whether the stress result has converged as mentioned earlier. This process is also referred to as manual convergence

Manual Convergence

1. Run analysis with default mesh settings - Average Element Size of 0.1

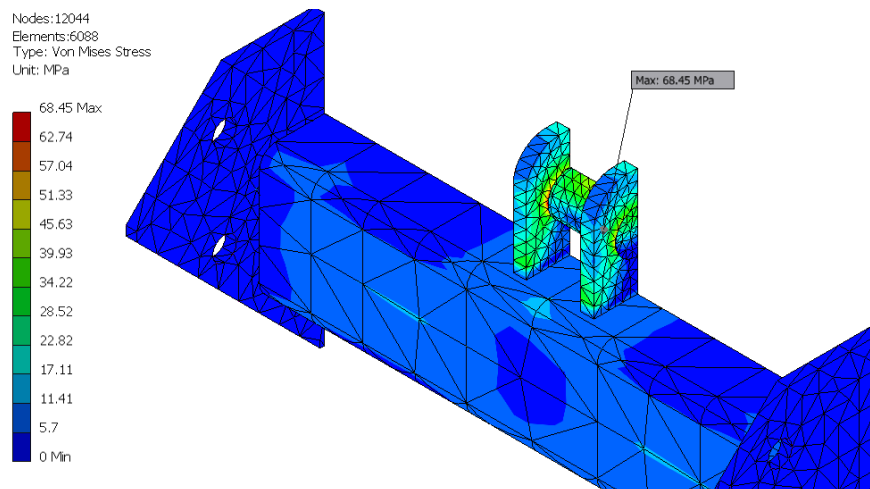


FIGURE 6: STRESS RESULTS

This will allow you to very quickly check if the structure is performing as expected.

2. Copy Simulation and rerun analysis with Average Element Size of 0.05



- Copy Simulation again and rerun analysis with Average Element Size 0.025. Here instead of changing the average element size you can also use local mesh control around the area of interest.

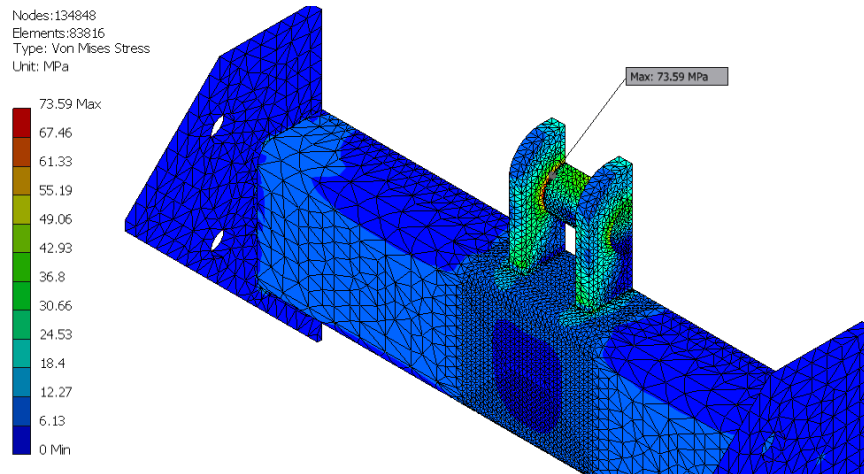


FIGURE 7: STRESS RESULTS WITH A FINER LOCAL MESH

To use local mesh control around area of interest it may be better to use split features to split faces of component.

Normal and Shear Stress results can also help to better understand the behavior of the structure by visually displaying tensile and compressive stress results as illustrated below in figure 8

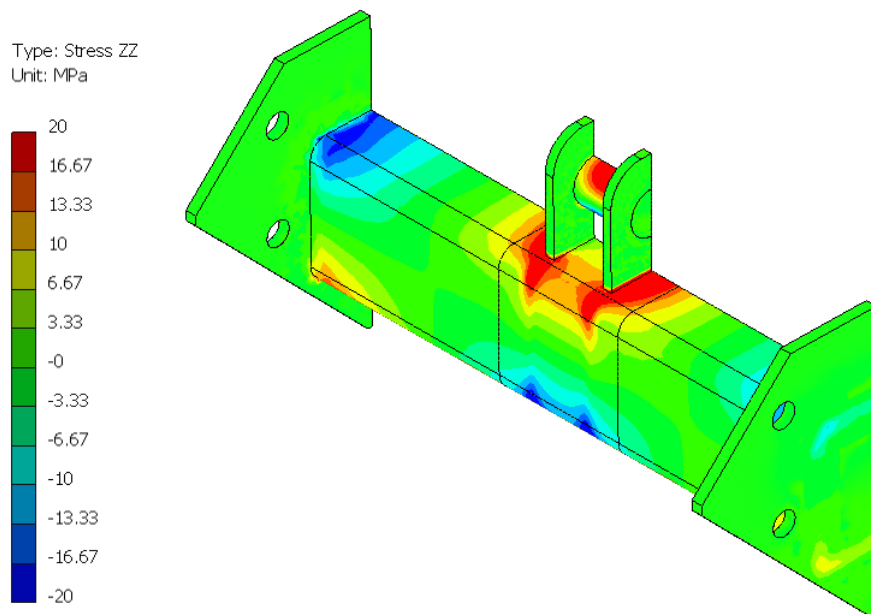


FIGURE 8: TENSILE AND COMPRESSIVE STRESS ALONG THE SUPPORTING STRUCTURE

Below are the available results within Inventor, with XX, YY, ZZ being directional or normal stresses and XY, XZ, YZ being the shear stresses.

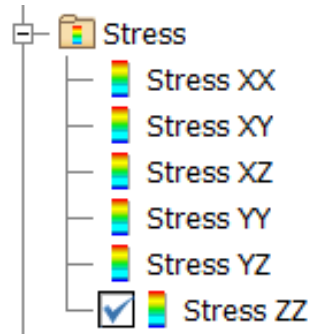


FIGURE 9: AVAILABLE STRESS RESULTS

Create Innovative designs

One of the unique and powerful features of stress analysis is the ability to perform parametric optimization studies. Model parameters can be accessed within Stress Analysis environment giving the flexibility to experiment with different design configurations as shown below in figure 10.

Parameters						
Component Name	Feature Name	Parameter Name	Values		Current Value	Unit
▶ 120 x 120 x 8 x 500		Boxsectionthickness	6,8,12		6	mm
Pivot Base Gusset		PivotBaseGussetThick	6,8,12		8	mm

FIGURE 10: DESIGN PARAMETERS

The different design configurations can also be validated against design constraints including stress, mass, safety factor etc. giving you the ability create innovative designs.

Design Constraints						
Constraint Name	Constraint Type	Limit	Safety Factor	Result Value	Unit	
▶ Max Von Mises Stress	Upper limit	207	2	78.1078	MPa	
Mass	Minimize			10.6151	kg	

FIGURE 11: DESIGN CONSTRAINTS

Once the designs have been verified with the new sizes you can then select Promote configuration to model which will update the assembly with the new values in one click,

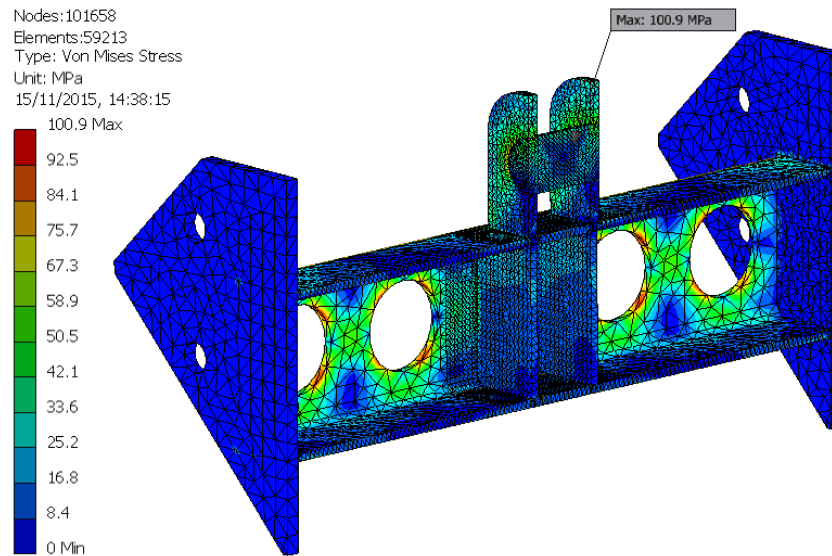


FIGURE 12: OPTIMIZED/INNOVATIVE DESIGN

Take Inventor Simulation to Next Level

Provided there is time in this class we will take an advanced dosage pill of Nastran IN-CAD. Autodesk® Nastran® In-CAD™ software, a CAD-embedded general purpose finite element analysis (FEA) tool powered by the Autodesk® Nastran® solver, offers a wide range of simulations above and beyond Inventor Simulation.

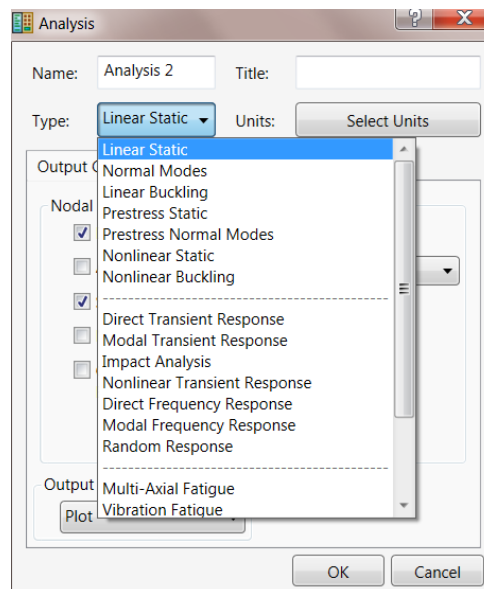


FIGURE 13: ANALYSIS TYPES AVAILABLE WITHIN NASTRAN IN-CAD



In this class we will expand on the analysis of the bridge structure to take account of bolted connections and see whether the initial suggested bolt sizes are strong enough? Nastran IN-CAD does not take into account bolted connections within assembly environment. The bolts are defined with Nastran IN-CAD using a simple dialogue including the ability to define bolt preloads and torque.

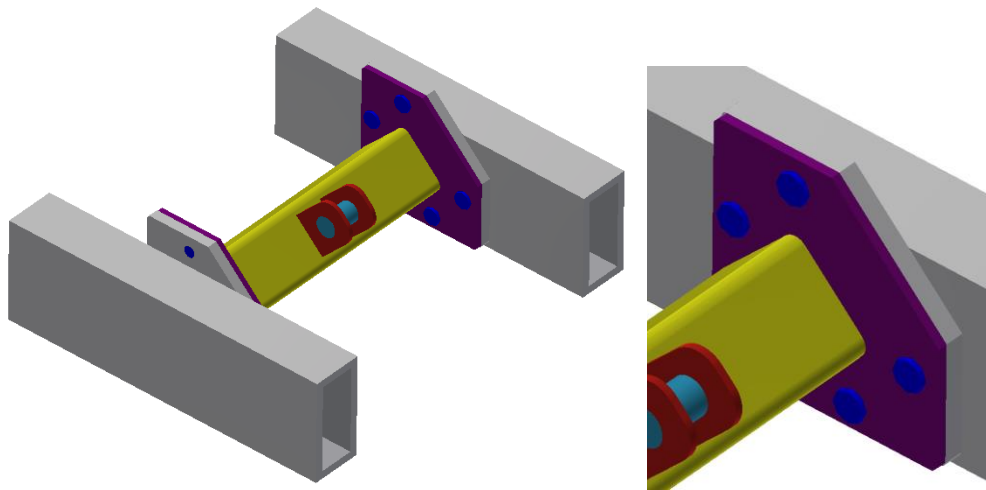


FIGURE 14: BOLT CONNECTIONS WITHIN NASTRAN IN-CAD

Another area where Nastran IN-CAD will be helpful is in its ability to perform fatigue analysis on the support structure to predict fatigue life and damage. Figure 15 below show some key extra data required in addition to a typical inventor stress analysis setup (plus the need to define a cyclic load).

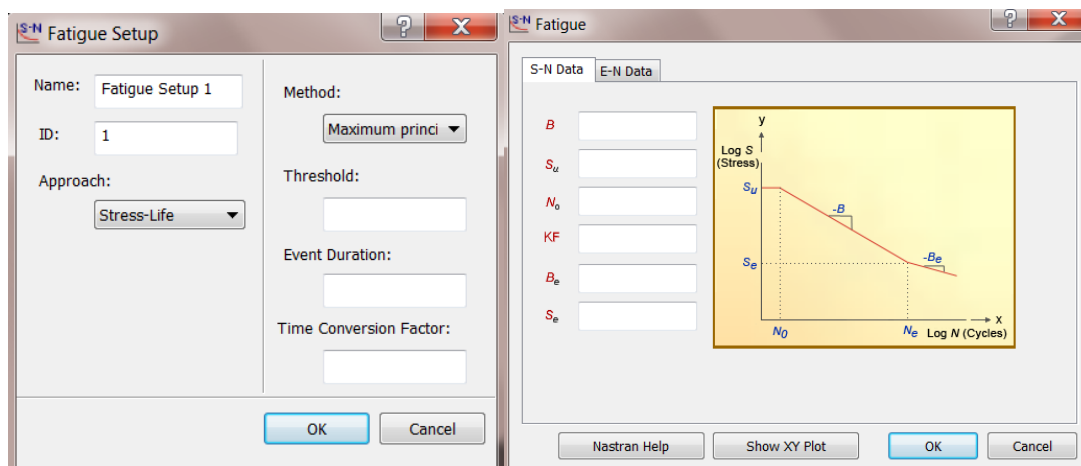


FIGURE 15: FATIGUE ANALYSIS GUI

Additional Resources

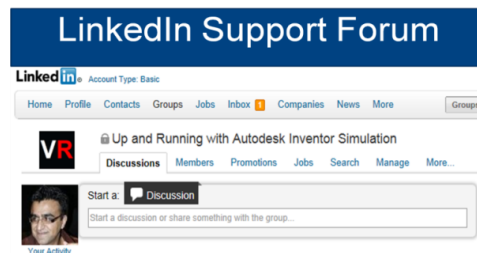
The material in this handout and lecture is based on my Up and Running with Autodesk Inventor Professional Series. The book's cover, stress analysis, frames analysis and dynamic simulation in a lot more depth with guidance and tips throughout the books. The books are only available through Amazon.com and all the local amazon sites.



In previous years at Autodesk University I have presented various aspects of Inventor Simulation which will be complimentary to this class. Below are a list of my classes.

- AU 2011 MA3425 – Real World: Real Autodesk Simulation Solutions
- AU 2012 MA2038 – Up and Running with Autodesk Inventor Professional Simulation in 90 Minutes
- AU 2014 SM5623 – Inventor Simulation Tips and Tricks

In addition to the class materials, on LinkedIn there is a dedicated support forum for Inventor Simulation Users



around the world. Here you can post any question on Inventor Simulation and get help from fellow peers from around the world, including myself. The Support forum is named Up and Running with Autodesk Inventor Simulation. To join the forum you first have to sign up to LinkedIn, which is free.

http://www.linkedin.com/groups?home=&gid=2061026&trk=anet Ug_hm

In addition to the support forum there is also a dedicated simulation blog called Virtual Reality again hosted by me. This blog is also one of the places where you can download the dataset to go with the books, mentioned earlier in the further reading section.

<http://vrblog.info>

