AUTODESK UNIVERSITY

PD20690

Simulation for Everyone

Wasim Younis
Symetri

David Truyens
Autodesk

Learning Objectives

- Learn about innovation
- Learn how to design with more confidence
- Learn how other designers have successfully utilised analysis capabilities within Inventor
- Learn how to apply best practices

Description

In today's difficult times, all around the world engineers and designers like you are constantly facing challenges, including reducing costs, reducing weight, making innovative product designs faster, and on and on. In this session, we will take your designs to the next level by utilizing the phenomenal Shape Generator and Stress Analysis available within Inventor Professional software. The session will go through workflows and tips based on industry best practices. The class will also demonstrate various examples of how designers like you have made effective use of Shape Generator and Stress Analysis. This session features Inventor Professional.

Your AU Expert(s)

Wasim Younis is a simulation solutions manager at Symetri, with more than 20 years of experience in the manufacturing field, including having worked at Rolls-Royce, British Aerospace, and Nuclear Electric. Younis has been teaching at Autodesk University for more than 6 years, and has been involved with Simulation software since Autodesk, Inc., first introduced it. He is well known throughout the Simulation community, and has authored the Up and Running with Autodesk Inventor Simulation books, available worldwide. He also manages and runs a dedicated forum for simulation users on LinkedIn called Up and Running with Autodesk Inventor Simulation.

David Truyens has buildup a strong overview on the Autodesk manufacturing products in the 10 years he has been working for distribution in the Benelux. Always had a passion for simulation, but also multi-disciplinary projects. He studied at the University of Antwerp as an electro-mechanical engineer, with a final project on the optimization of a wing box in a collaboration with Dassault. Later on he moved from heavy lifting to product design to find his passion in CAE. Other passions are tinkering with Arduino and CNC (creating a self-balancing robot for example). In rare occasions when he can’t be found behind his computer or at a customer he seeks complete disconnection while sailing at sea.

In today's difficult times, all around the world engineers and designers are constantly facing challenges, including reducing costs, reducing weight, and making innovative product designs faster etc. As a consequence of this more and more companies are now looking to adapt some form of digital prototyping to help reduce design to market lead times. Shape Generator and Inventor Stress Analysis are a key part of this digital prototyping process enabling the designers to analyse and validate designs very early with the design and manufacturing process of the product as shown below.

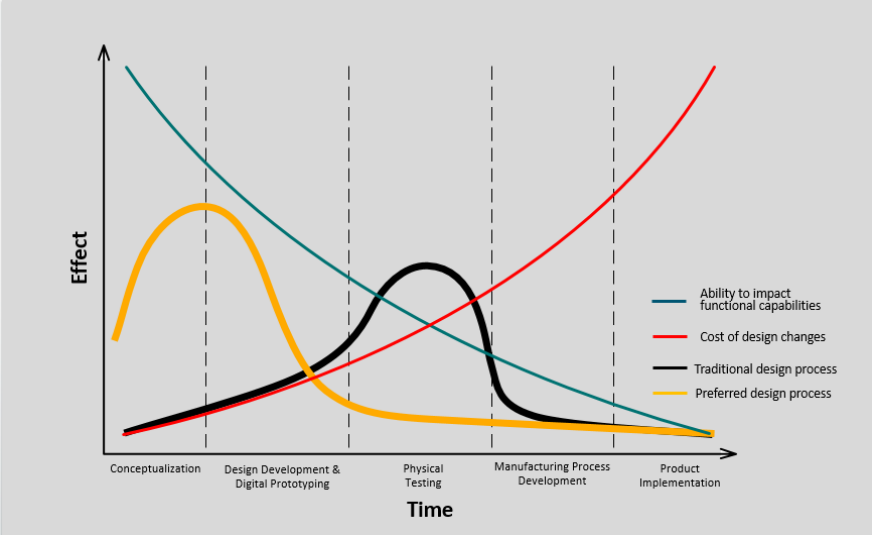

FIGURE 1 – IMPACT OF SIMULATION ON PRODUCT DESIGN TO MARKET PROCESS

Shape Generator and Inventor Stress Analysis can also help to reduce the number of physical prototypes required to validate designs.

Shape Generator

Is a conceptual design tool which allows you the designer to create structurally efficient parts? Shape Generator is not available for assemblies and can only be accessed from the part modelling environment.

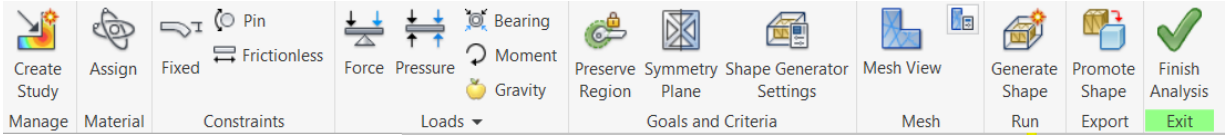

FIGURE 2 – SHAPE GENERATOR RIBBON/PANEL

Within the shape generator environment you can apply materials, constraints and loads just like stress analysis. For further information on constraints and loads refer to the relevant sections. Density is the main material property needed for Shape generator as this is the value used to determine the initial mass of the component, which is the basis for setting mass target reductions within shape generator.

Preserve Region

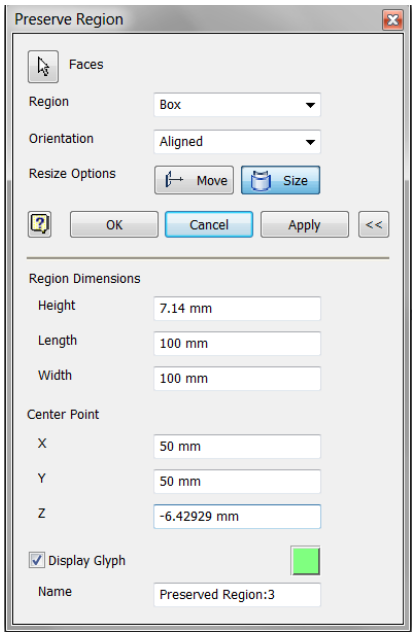

FIGURE 3 – PRESERVE REGION DIALOGUE BOX

Only single faces can be selected to define cylindrical, rectangular or square volumes to be preserved when removing elements during the simulation, to meet mass targets. If a planar face is selected whether it's horizontal or angled the resize dimensions will be aligned to that face. At present the orientation can only be defined by aligned or global XYZ.

Region and Orientation
Will define shape of the model geometry you wish to keep. Currently you can only define a box or cylindrical shape. Switching between the two region types will adjust the region dimension inputs. If you choose a flat face, region will be a box. If you choose a cylindrical face, region will adjust and change to cylinder. If this assumption is not what you want, you can change to either.

Region Dimensions
Controls the size of the preserving shape. The box region option will use values defined by the X, Y and Z vectors whereas the cylinder region option will use radius and length dimensions.

Center Point
Will control the region position with respect to Global XYZ coordinate systems or aligned with the geometry you have selected.

Display Glyph
You can also display and change the color of the transparent preserved region volume.

Faces selected for constraints and loads are automatically preserved for at least one layer of mesh elements.

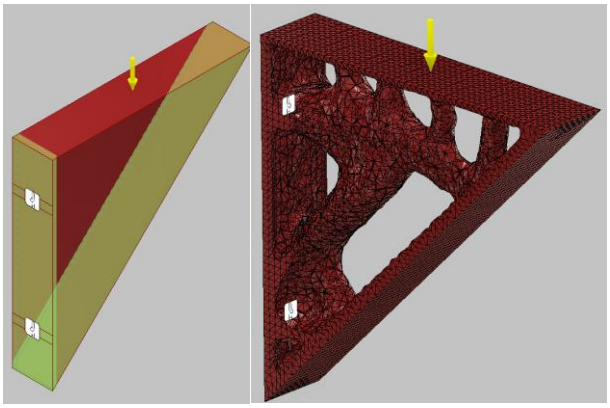

FIGURE 4 – EXAMPLE ILLUSTRATING RESULTS WITH PRESERVE REGIONS

Symmetry Plane

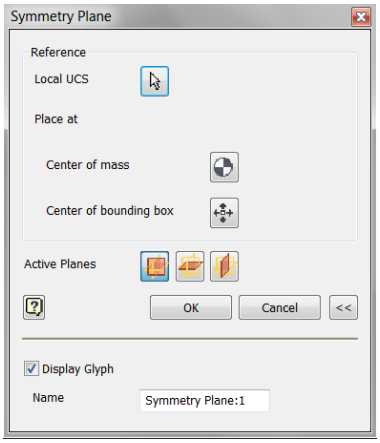

FIGURE 5 – SYMMETRY PLANE DIALOGUE BOX

Shape Generator will use symmetry planes to remove material (elements) such that they are symmetrical about the plane. You can define three symmetrical planes within shape generator.

Local UCS
The symmetry planes can be oriented using the default global axis or any user defined local user coordinate system illustrated by the following example.

Center of mass
This allows you to place the UCS and symmetry plane at the center of mass of the part.

Center of bounding box
This allows you to place the UCS and symmetry plane at the center of the bounding box of the part.

Active Planes

Allows you to add a symmetry plane in one of the other orthogonal directions

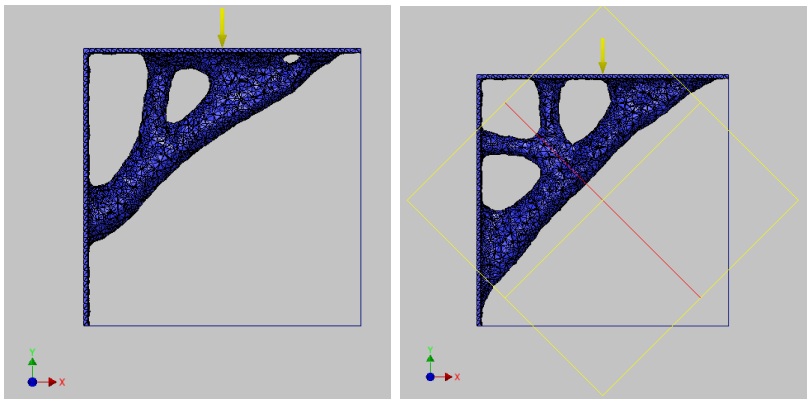

FIGURE 6 – RESULTS BY DEFINING SYMMETRY PLANES

Shape Generator Settings

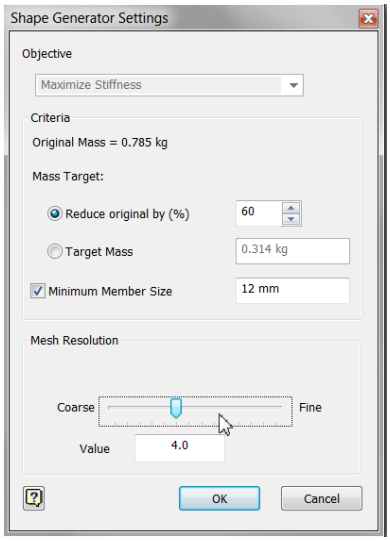

FIGURE 7 – SHAPE GENERATOR SETTINGS DIALOGUE BOX

Objective

Will define the criteria for the solver. The solution is currently limited to maximizing stiffness of the structure.

Mass Target

Currently you can define reduced mass target as a percentage or actual value.

Mesh Resolution

This provides a simplified way of deciding the density of the mesh. This is controlled by a scale of 10 (coarse mesh) to 0.5 (fine mesh). You can select mesh Vvew to visually see the density of the mesh. The values defined are not exact size of the mesh but a ratio. For example if you specify 2 and the largest edge of the bounding box of the component is 100mm. Then the element size will be 20.

Minimum Member Size

This allows you to specify the minimum member thickness value which shape generator tries to retain during the simulation. A warning will appear if the thickness is not more than 3 times the average element size. However this does not mean you cannot run a simulation with a member thickness value twice the size of an average element size.

Promote Shape

Allows you to save the optimized generated shape as an stl file within the current part or as a new file.

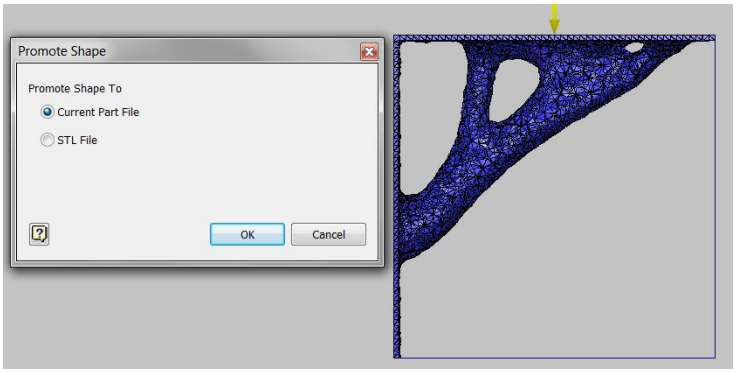

FIGURE 8 – PROMOTE SHAPE DIALOGUE BOX

Once saved you can then redesign the original component around the generated shape as shown below.

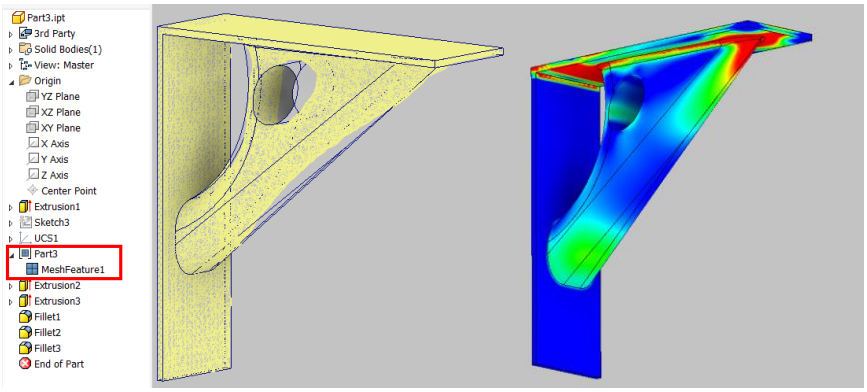

FIGURE 9 – OPTIMISED DESIGN BASED ON SHAPE GENERATOR RESULTS

Once the changes have been made the shape generator study can be modified to stress analysis. This will transfer the loads, constraints, materials and mesh data to the stress analysis environment ready to be solved.

Additional Information – Refer to the following documents

- Up and Running with Autodesk Inventor Professional 2017 Page 24-30

Inventor 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 of the material. This stress relationship is commonly referred to as factor of safety (FOS) and is used in many analyses as an indicator of success or failure in analysis.

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

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.

Inventor Stress Analysis allows Inventor users to very quickly verify that their designs are fit for purpose.

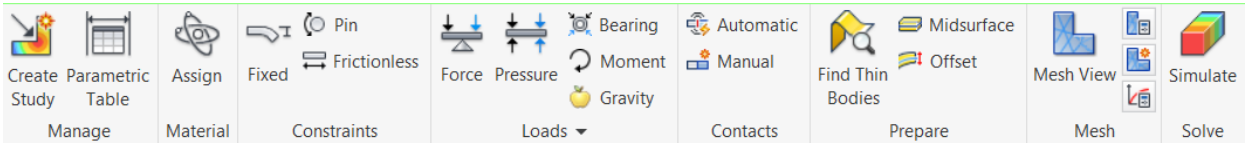

FIGURE 10 – STRESS ANALYSIS RIBBON/PANEL

Manage

Within Inventor Stress analysis you can only carry out linear stress and modal analysis. One of its unique and powerful functionalities is its abilities to experiment with various parameters enabling you to optimise designs.

Material

Inventor Stress Analysis makes use of the Inventor's defaults material library. Usually material are defined within the part environment and are carried forward into the stress analysis environment.

Constraints

There are currently only three types of constraints available. Frictionless constraint is also used to define symmetry conditions when analysing a part of the design, for example a quarter model.

Additional Information – Refer to the following documents

- AU 2010 MA222-1V – Autodesk Inventor Simulation 2011: Tips & Tricks Page 6
- AU 2011 MA3425 – Real World: Real Autodesk Simulation Solutions Page 3-4
- Up and Running with Autodesk Inventor Professional 2017 Page 37-38

Loads

Inventor analysis only provides the ability to define structural loads including bearing, force and pressure loads which typically are present in most real-life applications.

Additional Information – Refer to the following documents

- AU 2011 MA3425 – Real World: Real Autodesk Simulation Solutions Page 4
- Up and Running with Autodesk Inventor Professional 2017 Page 38-40

Contacts

Currently there are seven types of contacts available within Inventor Stress Analysis which can be defined to simulate behaviour between components.

1. Bonded
2. Separation
3. Sliding/No Separation
4. Separation/No Sliding
5. Shrink Fit/Sliding
6. Shrink Fit/No Sliding
7. Spring

Bonded and Sliding/No Separation are the most commonly used contacts.

Additional Information – Refer to the following documents

- AU 2014 SM5623 – Inventor Simulation Tips and Tricks Page 3-4
- Up and Running with Autodesk Inventor Professional 2017 Page 40-42

Prepare

Here you can automatically or manually convert sheet metal parts or any thin parts into surfaces. This will then allow Inventor Stress Analysis to use thin elements (shells) which will significantly speed up solve times. You will normally get a warning if you have a part with length to thickness ratio of greater than 250:1, if you try to solve using default solid elements.

Additional Information – Refer to the following documents

- Up and Running with Autodesk Inventor Professional 2017 Page 43-46

Mesh

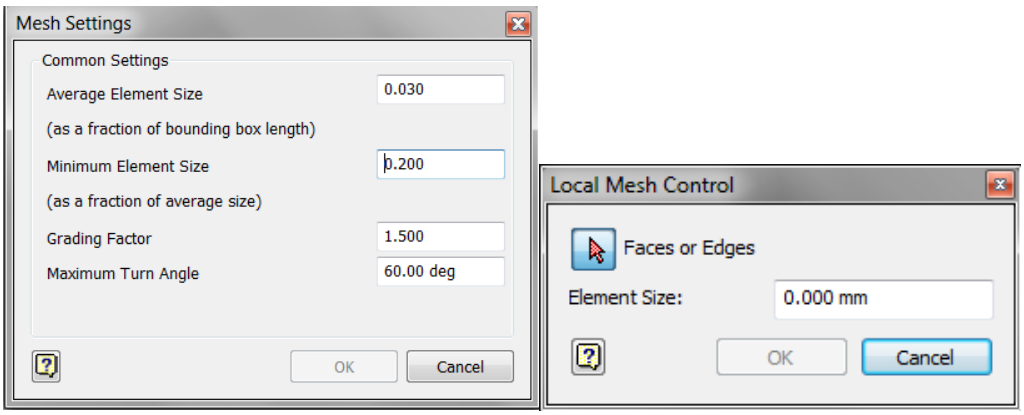

FIGURE 12 – GLOBAL AND LOCAL MESH SETTINGS DIALOGUE BOX'S

Currently Inventor Stress Analysis allows you to control the mesh globally and locally. However you can only specify maximum mesh size within the local mesh control dialogue box. Within the Mesh Settings dialogue Average Element Size is typically the only value you need to change, the others are just optional.

Additional Information – Refer to the following documents

- AU 2014 SM5623 – Inventor Simulation Tips and Tricks Page 12-13
- Up and Running with Autodesk Inventor Professional 2017 Page 40-42

Solve and Analyse Results

Once all boundary conditions including contacts and mesh have been defined the next stage is run and analyse the results. It is best practice to run the analysis a couple of times to see if the stress results in particular are not sensitive to the mesh. You can refer to the following documents for more detailed info.

Additional Information – Refer to the following documents

- AU 2011 MA3425 – Real World: Real Autodesk Simulation Solutions 5-6
- AU 2012 MA2038 – Up and Running with Autodesk Inventor Professional Simulation in 90 Minutes Page 6-7
- AU 2014 SM5623 – Inventor Simulation Tips and Tricks Page 7-8 and 12-13
- Up and Running with Autodesk Inventor Professional 2017 Page 51-54

The session will include customer examples that have successfully utilised Inventor Stress Analysis. Inventor Stress Analysis is a linear software analysis and has certain limitations. This session will conclude with my next class SIM20692 - Simulation for Designers – Take your Inventor analysis to the next level. The examples used in this class will be used again to demonstrate what more you can achieve with Nastran In-CAD including

- More comprehensive preprocessing
- More comprehensive results processing
- More analysis capabilities including buckling, drop test, fatigue and more

Additional Resources

The material in this handout and lecture is extensively based on my Up and Running with Autodesk Inventor Professional book's. 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 available through Amazon.com and all the local amazon sites, worldwide.

FIGURE 13 – LATEST EDITIONS OF THE BOOKS

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

AU 2010	MA222-1V	– Autodesk Inventor Simulation 2011: Tips & Tricks
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
AU 2015	IM10256	– Stressed Take an Inventor Simulation Chill Pill

In addition you may also find the following lab on Shape generator by Andrew Sears very useful

AU 2015	AT12113-L	– Save the Money, Lose the Weight with 100% Grade A (Autodesk) Lightweighting

In addition to the above resources, on LinkedIn there is a dedicated support forum for Inventor simulation users around the world. Here you can post any questions 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/2061026

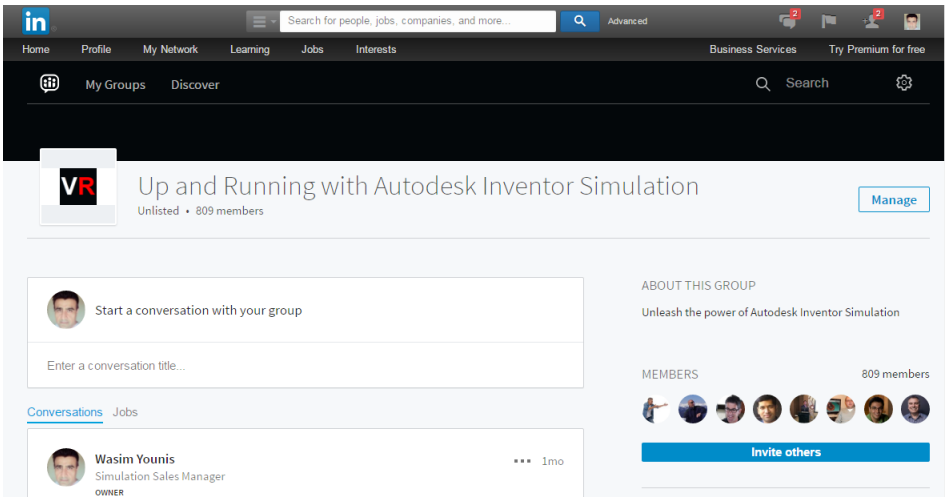

FIGURE 14 – LINKED FORUM FOR INVENTOR STRESS ANALYSIS USERS

If you have any questions before the class do not hesitate to contact me via email
Email: wasim.younis@symetri.com