ACTA TECHNICA CORVINIENSIS
– Bulletin of Engineering
Tome IX [2016], Fascicule 4 [October – December]
ISSN: 2067 – 3809

[1.]Peter SZUCHY

BASIC FAILURE POSSIBILITIES USING FINITE ELEMENT METHOD OF AUTODESK INVENTOR 2012 STRESS ANALYSIS

[1.]Department of Technology, Faculty of Engineering, University of Szeged, Mars tér7, Szeged, HUNGARY

Abstract: Teaching Finite Element Method (FEM) with Autodesk Inventor 2012, Statics and Strength of Materials we have collected a lot of sample how the lack of Statics knowledge and/or accurate FEM knowledge leads to incorrect results during stress analysis of Inventor. Our students use the 3D model part of the software really well but the application of Stress Analysis brings very often mistakes. Wrew are going to introduce the two most common problems that we could meet recently during the students' practice: choosing false constraints and leaving out of consideration the buckling.
Keywords: Finite Element Method, constraint, buckling

INTRODUCTION

Coaching of Autodesk Inventor, three dimensional model designing software has been executed for years on the Faculty of Engineering, University of Szeged. We have worked with Inventor 2012, thus our experiments are the widest with this version. Most of the student enjoy the creative job, discovering this playful way of self-realization.

It is good to see as their curiosity pursues them ahead on the self-supporting development. The playfulness recoils and the first mistakes are made as they reach the Stress simulation and its necessary knowledge from Statics or Strength of Materials.

The general steps of a FEM software are the following:

1. Preparing of 3D geometric model.
2. Characterising of the raw material.
3. Determination of constraints and loads.
4. Mesh settings, calculation, and valuation of the results.

Though mesh setting knowledge is a key skill in the process, in a beginner's work the most problems occur at the last two steps: determining the constraints and valuating the results.

INTRODUCTION OF Autodesk Inventor 2012 Stress Analysis

Autodesk Inventor 2012 is a user-friendly software for 3D simulation, really suitable for self-learning. It has a Stress Analysis module that usesFinite Element Method (FEM). For its use the user does not need to have deep knowledge in the math of FEM, but it is essential to have and use wellthe knowledge of Statics and Strength of Materials. The followings are recommended by the Manual [1.]: Inventor 2012 Stress Analysis is designed for estimation of deformation, stress, natural frequencies in linear static problems. It does not substitute physical testing, only identifies areas of the highest stress and deformation reducing the numbers of required physical tests. Its application is limited in the following situations:

» Non-linear material features.
» Non-linear effects (e.c.: buckling)
» Dynamic loading effects.
» Thermal influence.
» Large deformation compared to the part's dimensions.

When somebody has these circumstances further analysis is recommended.

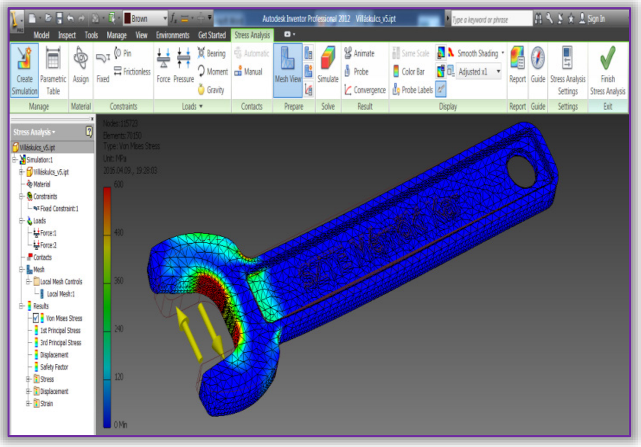

Figure 1. Stress Analysis environment of Autodesk Inventor 2012

The Autodesk Inventor 2012 has a really useable environment for Stress Analysis [Figure 1]. The Panels lead the user through the steps of analysis in logical way: Managing the simulation, setup of Materials, Constraints, Loads, Contacts, preparing Mesh and calculation, Simulation, visualizing and analysing of Result.

The beginning two steps are easy to do - even if setting Material well needs advancedknowledge. The first problem occurs at setting the constraints.

CONSTRAINT SETTING FAILURES

For setting the constraints the possibilities are the following in Inventor 2012:

1. Fixed constraint, k=6 or less, as custom needs, ("k" is the number of constraints, how many degree of freedom is tied down.)
2. Pin constraint, k=5 or 4.
3. Frictionless Constraint, k=1.

The first problem can occur if the user does not constrain fully the model and the simulation cannot be run. Often happens it when Pin and Frictionless constraints were used in combination. In this case there is a chance to set the Detect and eliminate rigid body modes when weak springs are automatically added without influencing the result [Figure 2]

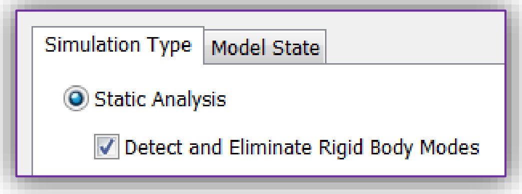

Figure 2. Preventing under constrain

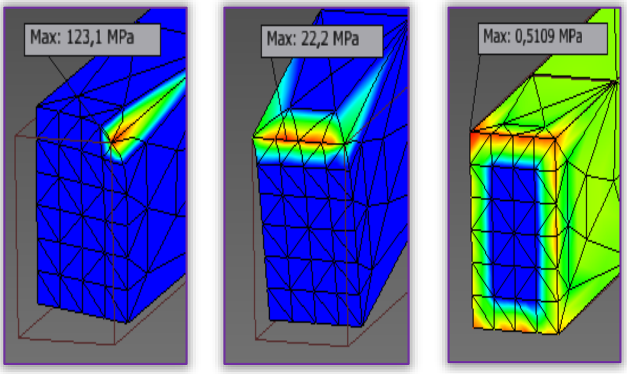

Figure 3. Pulled bar with a fixed constraint applied on: 3.a. upper right point (left), 3.b. upper edge (middle), 3.c. closing surface (right)

The second problem comes forward if the user adds the constraint to a geometry that has no surface (vertex, edge). In this case the stress in the surroundings of applied constrain will be extremely high because of the force/surface rate. Figure 3. shows a pulled bar, where fix constrain is applied on the upper right corner [3.a.], then on the upper edge [3.b.] and in the end on the whole closing surface [3.c.]. The probe labels show the maximum stress values. (Mesh settings also influencethe results.)

The previous problems are almost childish, easy to find and avoid them, but the following example isnot so obvious. Over-constraining the model can show lower stresses than they are in truth, thus it can lead to undersizing. The following example is a stress analysis of a home-made wrench [Figure 4].

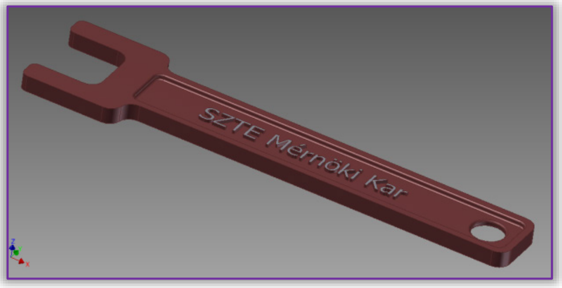

Figure 4. Home-made wrench

During the simulation two fixed constraints are put on the inner surfaces of the wrench [Figure 5]. 300 N Force is added to the end of wrench, Embossed text is excluded, than we run the simulation and analyze the result [Figure 6]. It is visible that the maximum stress occurs on both side of the neck.

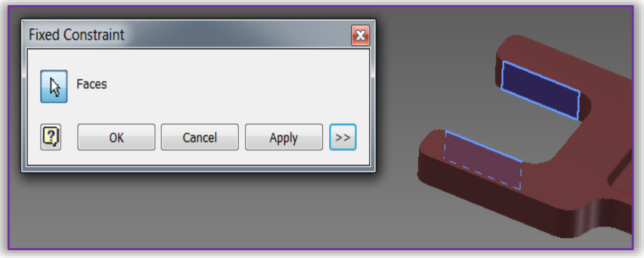

Figure 5. Fixed constraints added on the inner surfaces of fork

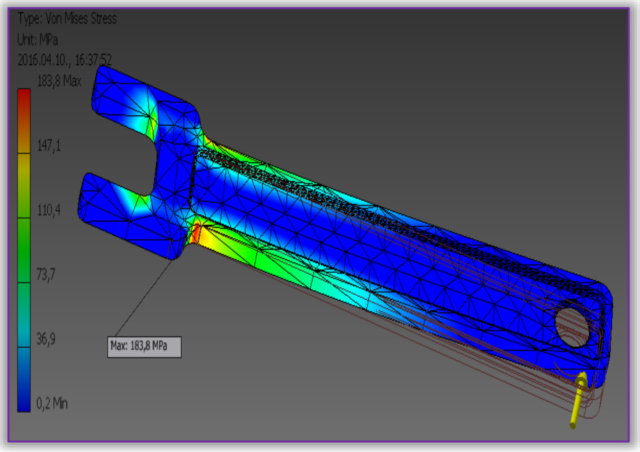

Figure 6. Maximum stresses are on the neck fixing the forks together

As we analyze the result it turns out that on the inner side of the fork's tines the image of stress is not too high, its escalation is narrow and does not correspond to the theoretical stress distribution learned in Strength of Materials [4]. Running a Local Mesh Control the result is not better. What is the problem? As we put two fixed constraints on both tines of the fork, we fixed the two tines to each other, so they could not move independently, they strengthen each other. With these constraints we can analyze the stress only in the neck of the wrench. For better result in the tines there are several way to analyze, for example we can change the places of the constraint and the load [Figure 1.], or we can put a screw-nut between the tines, using fixed constraint to the nut.. The problem is similar to a simple bar that is constrained on both ends and loaded on the middle. If two fixed constraints are used on both ends of the bar, there will occur normal and shearing forces in the constraints and in the bar as well [Figure 7]. If one of the fixed constraints is replaced to a Frictionless constraint, the normal forces disappear [Figure 8]. The structure is more rigid in the first case.

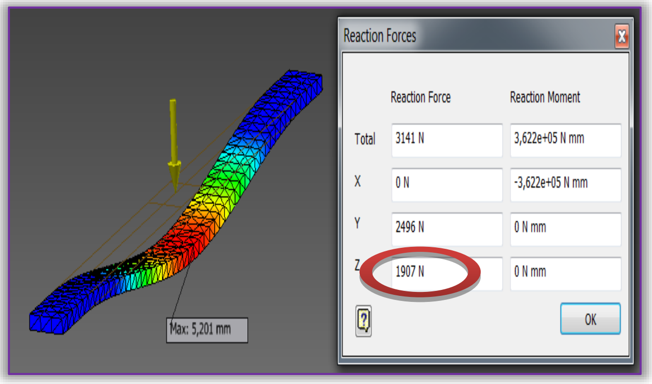

Figure 7. Bended bar with two Fixed constraints on both ends. Normal and shearing forces appear

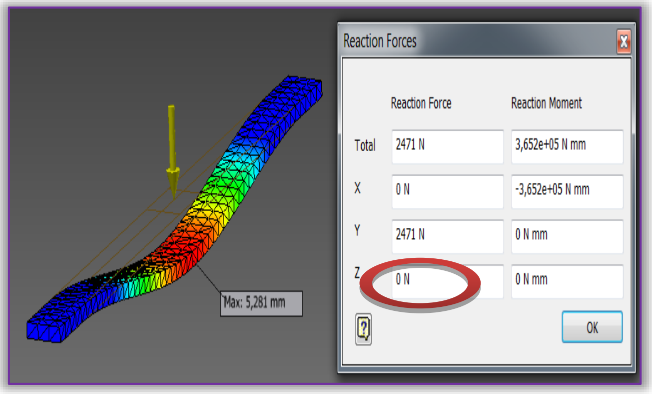

Figure 8. Bended bar with one Fixed and one Frictionless constraint. Only shearing forces appear

The same failure can be made at stress analysis of vehicle under-carriage. If somebody adds only the combination of Fixed or Pin constraints to the connecting points of wheel suspension without any Frictionless constraint, the structure will be over-constrained and it gets more than real rigidity.

BUCKLING

A bar compressed theoretically exactly in the center of mass axis has no buckling. For producing it we need some moment from unpunctuality or from the surroundings. Let's make a calculation for critical compressing force for buckling in case of a 40x100x3000 mm bar, fixed on one ending section, pressed on the other end (β=2) [3]. (The equations are well-known, I concentrate on the calculation):

$$A = 40\ \mathrm{mm} \cdot 100\ \mathrm{mm} = 4000\ \mathrm{mm}^2 \quad (1)$$

$$I_2 = \frac{(40\ \mathrm{mm})^3 \cdot 100\ \mathrm{mm}}{12} = 533333\ \mathrm{mm}^4 \quad (2)$$

$$i_2 = \sqrt{\frac{533333\ \mathrm{mm}^4}{4000\ \mathrm{mm}^2}} = 11{,}547\ \mathrm{mm} \quad (3)$$

$$\lambda = \frac{2 \cdot 3000\ \mathrm{mm}}{11{,}547\ \mathrm{mm}} = 519.6\ \ (\beta = 2) \quad (4)$$

$$\sigma_{critical} = \frac{\pi^2 \cdot E}{\lambda^2} = \frac{\pi^2 \cdot 210 \cdot 10^3\mathrm{MPa}}{519{,}6^2} \quad (5)$$

$$\sigma_{critical} = 7{,}68\ \mathrm{MPa} \quad (6)$$

$$F_{critical} = 7{,}68\ \mathrm{MPa} \cdot 4000\mathrm{mm}^2 \quad (7)$$

$$F_{critical} = 30{,}7\ \mathrm{kN} \quad (8)$$

Without using coefficient of safety!

If the effect of buckling is not considered, the supposed allowable pressing force ($F_{supposed}$) from the permissible stress ($\sigma_{perm} := 150MPa$) is:

$$F_{supposed} = 150\mathrm{MPa} \cdot 4000\mathrm{mm}^2 = 600\mathrm{kN} \quad (9)$$

What a difference! The Manual of Inventor 2012 [1] declares that it does not handle the buckling, but many students forget it or even do not know it. Let us see what happens, if we make a stress analysis for the compression of the above mentioned bar.

After modelling the bar we put a fixed constraint on one of the ends, 200kN Force on the other one. Figure 9.a. shows the replacement of the bar. If we double the load, the strain doubles as well, without any sign of buckling. Perhaps an above mentioned moment is missing. Let us add M=1 Nm moment to the loaded section. Figure 9.b. shows the result, and it is visible that replacement does not changed.

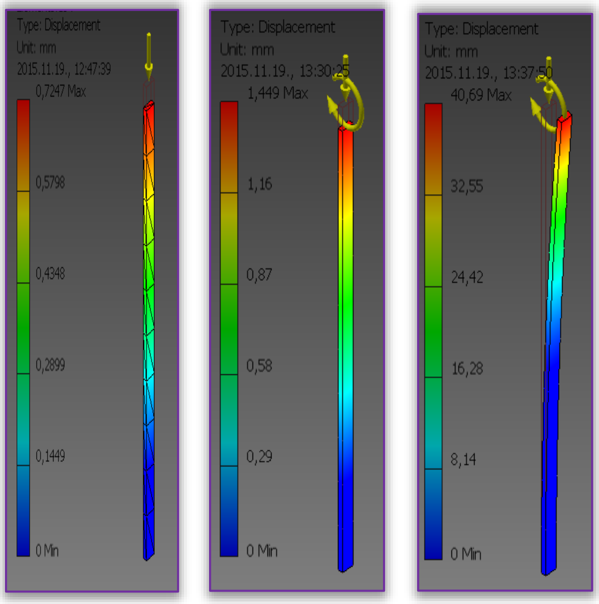

Figure 9. a. (left) Load: $F_{normal} = 200\mathrm{kN}$, b. (middle) Load: $F_{normal} = 400\mathrm{kN}, M = 1\mathrm{kNm}$,
c. (right) Load: $F_{normal} = 400\mathrm{kN}, M = 1000\mathrm{kNm}$

Increased the Moment to 1000 Nm Figure 9.c. shows the replacements. The loaded end section has moved lateral around 40 mm from its original position. The loading force is more than 10 times higher than the theoretical critical force, beside it there is inducing moment, but there is no collapse. The Manual was right. I heard this problem from students designing truss (compressed members) and driving (bearer-bar).

CONCLUSIONS

I have introduced the basic failure possibilities using Autodesk Inventor 2012 Stress Simulation. Their emergences come rather from the moderate knowledge of Statics and Strength of Material then Finite Element Method. The easiest problems, like forces, constraints without surfaces applied on, are simply to be avoided. The developer of the software declares that buckling is not considered during the FEM simulation, but people can forget it and do not calculate it plus on pushed elements. The hardest problem to find is the false constraining. Over-constraining can show more rigidity and less stress than it is indeed. False constraints can cause false results as well. How can we find the false results of FEM? Not accepting the result at once, always being suspicious and using our engineering mind: can it be true? The real answers arrive after the execution at the first tests, but with good practice we can reduce the number of the false tests.

Bibliography

[1.] Autodesk Inventor Professional 2010, Simulation, Autodesk, Inc., 462B1-050000-CM06A, June 2009.
[2.] OrbánFerenc, FEM Lectures 2014/15, Univerity of Pécs, Technical and Information Faculty, Machine construction Institute
[3.] BíróIstván: Strenth of materials, Lecture notes, University of Szeged, Technical Faculty, Szeged, 2009.
[4.] M. CsizmadiaBéla – NádoriErnő: Statics, National Coursebook Publisher, Budapest, ISBN 978-963-19-2850-1,
[5.] KovácsÁdám – UjJózsef: Basics of Finite Element Methods, Publisher of Technical University, 2007.
[6.] PáczeltIstván – SzabóTamás – Baksa Attila: Basics of Finite Element Methods, HEFOP lecture notes (electronic form, supported by: HEFOP 3.3.1-P.-2004-09-0102/1.0 tender), 2007.
[7.] KovácsÁdám - MoharosIstván - OldalIstván – SzekrényesAndrás: FINITE ELEMENT METHOD, University notes (electronic form), 2011. ISBN 978-963-279-539-3, TÁMOP-4.1.2-08/2/A/KMR-2009-0029 tender

ACTA Technica CORVINIENSIS
BULLETIN OF ENGINEERING
ISSN:2067-3809