Contents

Introduction

Since our migration to 64 bit OS we have noticed that some of our simulations in Autodesk Inventor 2014 assemblies have had erroneous results. This has led us to conduct multiple trials in order to determine why and what causes the simulation to yield erroneous results. While examining multiple designs that have had erroneous results we have determined that some contacts hinder the mesh of the simulation. In this document we will explore one of these possibilities with a contact between a tangent face and a flat one. The 3 analysis that will be conducted are:
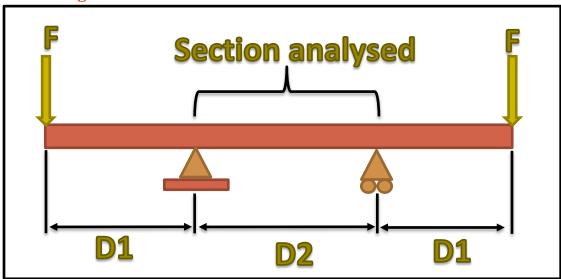
1. Section properties (the classical method)
2. FEA analysis in in Autodesk Inventor 2013 32bit
3. FEA analysis in in Autodesk Inventor 2014 64bit

The classical method will be considered to be the benchmark in your evaluation.

Section properties

This method is the classical way of calculating the stress in a section of a member. All the calculations are listed below.

Evaluating the force on the section

Where:

- F= force applied = 2000 lbf
- D1= Distance 1 = 30 inches
- D2= Distance 2 = 40 inches

$$M = F * D1 = 2000 * 30 = 60000 \; lbf * in$$

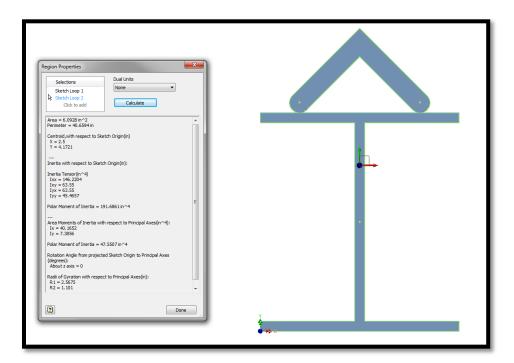Pure bending of 60000 lbf * in is acting on the section.

Calculating the Stress in the Section

Stress can be determined in a section with the following equation

$$\sigma_1 = \frac{y_1 * M}{I}$$

Where:

- y_1, y_2 = Distance to centroid from point to be analyzed (top or bottom of the beam)
- M = bending moment
- I = Moment if inertia

From the section properties in Inventor:

- I=40.165inches^4
- Y=4.172 inches

y_1 = Total height – Y = 7.6213-4.172 = 3.4493 inches

y_2 = 4.172 inches

Stress at the top of the beam

$$\sigma_1 = \frac{y_1 * M}{I} = \frac{3.449 * 60000}{I40.165} = 5.15x10^3 PSI$$
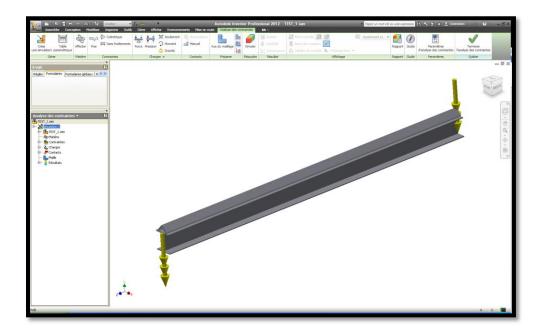
Stress at the bottom of the beam

$$\sigma_1 = \frac{y_1 * M}{I} = \frac{-4.172 * 60000}{I40.165} = -6.23x10^3 PSI$$

The stresses in the section of the beam are expected to reach $5.15x10^3 PSI$ (5.15 ksi) tension on the top and $6.23x10^3 PSI$ (-6.23 ksi) compression on the bottom.

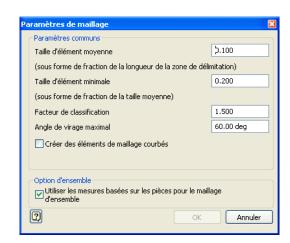# FEA analysis in in Autodesk Inventor 2013 32bit

Depicted below is the beam in the simulation environment with all of the forces and constrains shown in the first section.
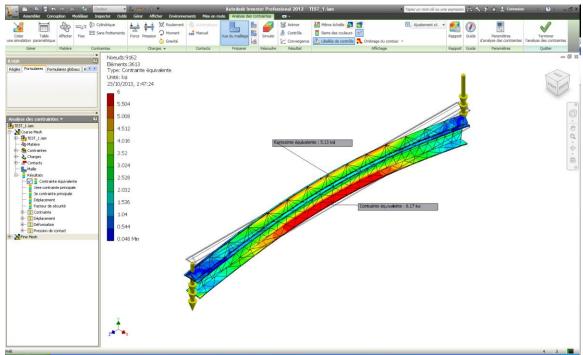

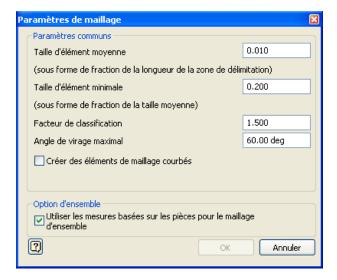## Parameters

Simulation and mesh parameters

Simulation with coarse mesh

Refined Mesh

The simulation was copied and ran with a finer mesh shown below are the new mesh parameters:

Simulation with fine mesh

Results

The coarse mesh has yielded the following results:

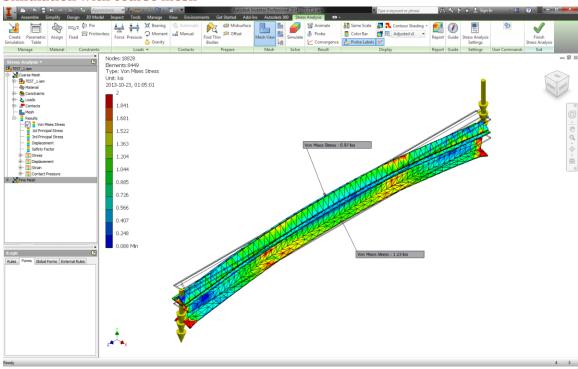- 5.13 Ksi tension
- 6.17 Ksi compression

The fine mesh has yield the following results:

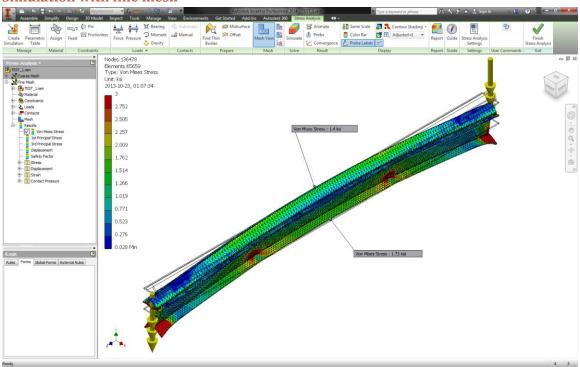- 5.21 Ksi tension
- 6.22 Ksi compression

FEA analysis in in Autodesk Inventor 2014 64bit

The same model was then loaded and ran into a 64 bit version of inventor. The model was re-meshed and simulated. There were no modifications between the first FEA and the second.

Simulation with coarse mesh

Simulation with fine mesh

Results

The coarse mesh has yielded the following results:

- 0.97 ksi tension
- 1.23 ksi compression

The fine mesh has yield the following results:

- 1.4 ksi tension
- 1.73 ksi compression

Analysis of the results

Below are all of the results obtained from the analysis

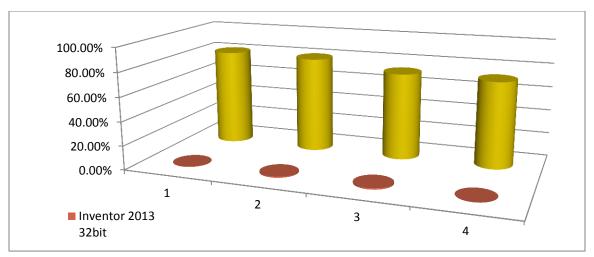|	top	Bottom	
Section analysis	5.15	6.23	in ksi

	Coarse mesh Stress		Fine mesh Stress		
Inventor 2013 32bit	5.13	6.17	5.21	6.22	in ksi
Inventor 2014 64bit	0.97	1.23	1.4	1.73	

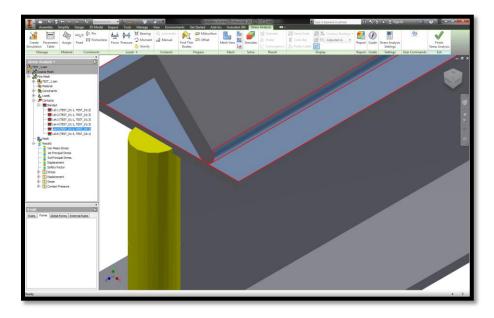| % error compared to section analysis				
	Coarse mesh Stress		Fine mesh Stress	
Inventor 2013 32bit	0.39%	0.96%	1.17%	0.16%
Inventor 2014 64bit	81.17%	80.26%	72.82%	72.23%

The classical method is considered to be "correct". This means that the 64bit Inventor FEA has an average error of 76.62%.
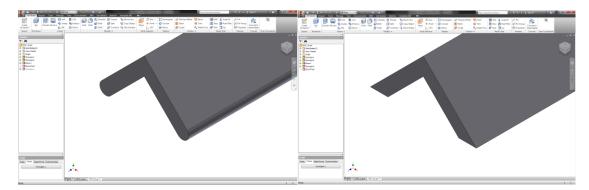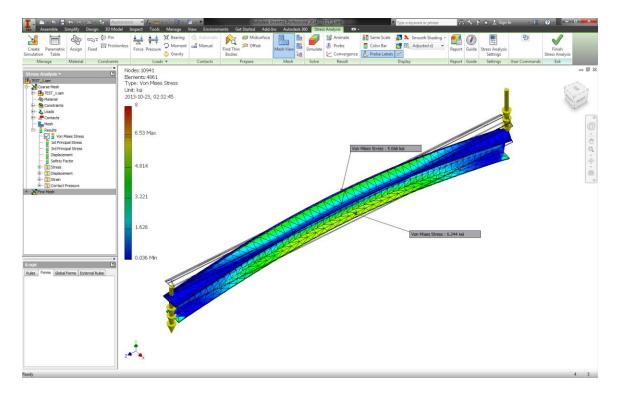
%Error Graphed

Suspected Cause

In our experiment we have found that the contacts are the issue. In the case of our beam and angle, the tangential contact is the culprit (picture below)

We can virtually eliminate the error by extruding a profile that would create a flat surface to flat surface contact. Below we can see the original model and the changed one.

Below is the result of the new assembly ran in the Inventor 2014 64bit environment.

Further test have also shown that if you add manual contacts the result may also be skewed considerably. Therefore it is not only tangent contacts but also the doubling of contacts for a given surface or edge.

Conclusion

The new 64 bit version of inventor has trouble managing contacts within the analysis environment. It seems that the contacts add infinitely rigid elements that interfere with the normal elements of the mesh.

This is a severe problem because the analysis gives the impression that the model is strong enough to withstand the prescribed loading. This means that the engineer may pass the results without knowing that their model analysis has been skewed by the new contacts.