Finite Element Analysis Application in High School Engineering CAD

Daniel Stumpf, Thomas Scrace

Hutchinson Central Technical High School, UB

Introduction

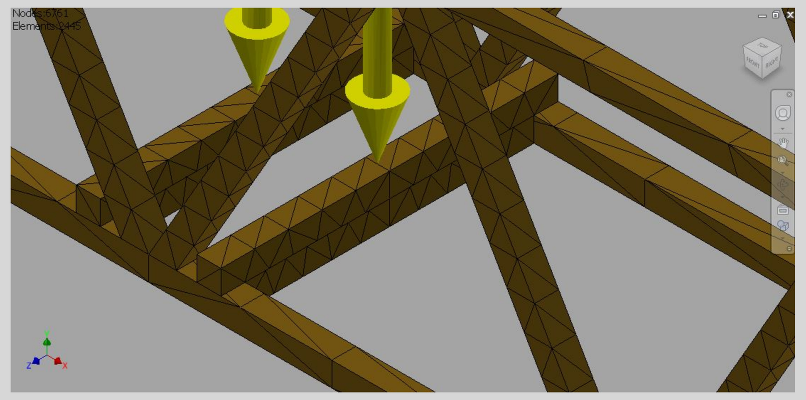
Students at Hutchinson Central Technical High School that are enrolled in the Engineering Technology major use a variety of different engineering based software to engage in a problem solving, project based curriculum. One of the software the students use in their Engineering CAD class is Autodesk Inventor. Inventor is a 3D modeling computer aided design software used to create three dimensional objects, assemblies and two dimensional technical drawings and other technical document. The software is also equipped with Finite Element Analysis (FEA) simulation modules that have the ability to test specific design and engineering attributes of the 3D models in a virtual environment. This researcher, while highly skilled with modeling aspects of the software wished to increase understanding of the analysis abilities of Autodesk Inventor. The purpose of this research project was to gain a better understanding of FEA, learn to use the simulators properly and test the application potential of FEA in the High School classroom environment.

Finite Element Analysis

Finite Element Analysis (FEA)

Finite Element Analysis, or FEA, as described by Autodesk the software manufacture of Inventor is a computerized method for predicting how a product reacts to real-world forces, vibration, heat, fluid flow, and other physical effects. Finite element analysis shows whether a product will break, wear out, or work the way it was designed. It is called analysis, but in the product development process, it is used to predict what is going to happen when the product is used.

FEA works by breaking down a real object into a large number (thousands to hundreds of thousands) of finite elements, such as little cubes. Mathematical equations help predict the behavior of each element. A computer then adds up all the individual behaviors to predict the behavior of the actual object.
Finite element analysis helps predict the behavior of products affected by many physical effects

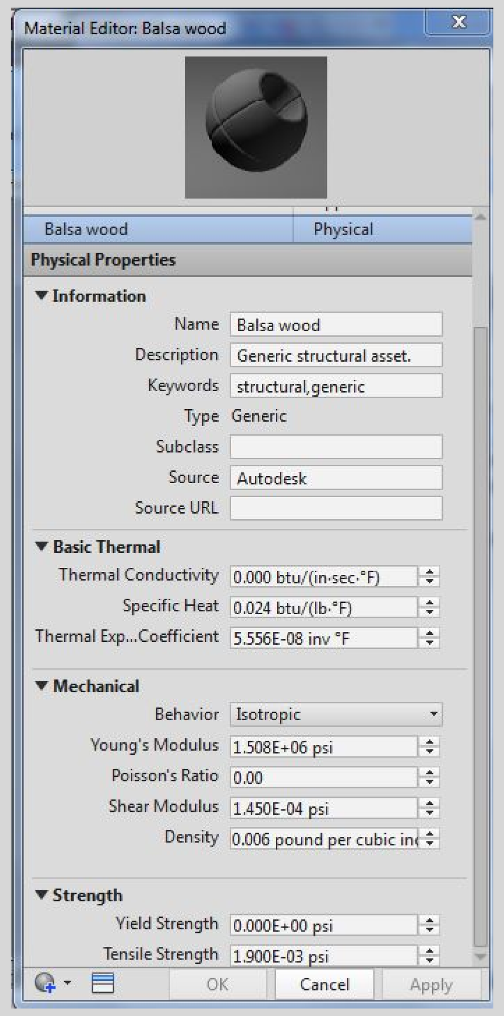


Application Project Selection

Each school year the Engineering Technology students at Hutchinson Central Technical H.S. compete in the Annual Tech Wars competition at Erie Community College. The competition consists of various events where students from area districts compete against each other to see who has the best design. Our students participate each year in the Balsa wood bridge event. The idea is to have the best strength to weight ratio. In the past we have always designed our bridges in 2D CAD. With the ability to use the 3D modeling software Inventor the opportunity to analyze the structures using FEA is presented. This is a project is an excellent candidate but we had to explore proper application requirements to ensure that virtual results would mimic real world testing.

Simulation Model Requirements

In order to research the analysis itself it was required for us to ensure every element for the simulation was satisfied. That included creating and assigning material properties for the Balsa wood and then building and constraining the model in the simulator.



The analysis itself is run through a simulation module. The simulation needed various inputs. The material properties of Balsa wood, as we discovered, varied greatly based on density.

Material(s)

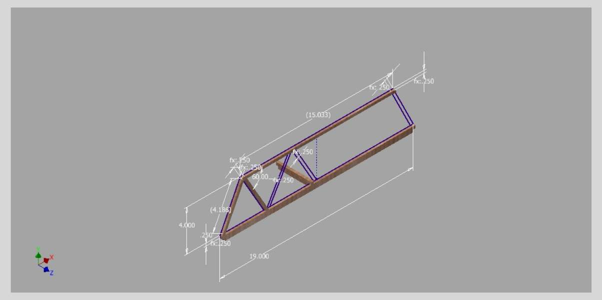
Name	Balsa wood	
General	Mass Density	0.00588875 lbmass/in^3
	Yield Strength	0 psi
	Ultimate Tensile Strength	0.00190131 psi
Stress	Young's Modulus	1509.43 ksi
	Poisson's Ratio	0 ul
	Shear Modulus	754.717 ksi
Stress Thermal	Expansion Coefficient	0.00000018 ul/f
	Thermal Conductivity	0.00187285 btu/(ft hr f)
	Specific Heat	0.0774195 btu/(lbmass f)
Part Name(s)	FEAB2-LOWER-MAIN-1 FEAB2--UPPER-MAIN-1 FEAB2-END-ANGLE-1 FEAB2-END-ANGLE-1 FEAB2-INNER-ANGLE-1 FEAB2-INNER-ANGLE-1 FEAB2-INNER-ANGLE-1 FEAB2-INNER-ANGLE-1 FEAB2-INNER-ANGLE-1 FEAB2-INNER-ANGLE-1 FEAB2-LOWER-MAIN-1 FEAB2--UPPER-MAIN-1 FEAB2-END-ANGLE-1 FEAB2-END-ANGLE-1 FEAB2-INNER-ANGLE-1 FEAB2-INNER-ANGLE-1 FEAB2-INNER-ANGLE-1 FEAB2-INNER-ANGLE-1 FEAB2-INNER-ANGLE-1 FEAB2-INNER-ANGLE-1 FEAB2-CROSS-1 FEAB2-CROSS-1 FEAB2-CROSS-2 FEAB2-CROSS-2 FEAB2-CROSS-2 FEAB2-CROSS-2 FEAB2-CROSS-2 FEAB2-CROSS-2 FEAB2-CROSS-1 FEAB2-CROSS-1 FEAB2-LOWER-MAIN-2 FEAB2-LOWER-MAIN-2	

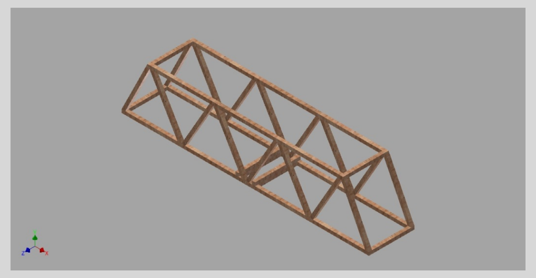
An average material was derived based from typical properties found through distributors.

Once the material properties were defined they could be applied to the various parts that made up the assembly of the structure.

Model Creation

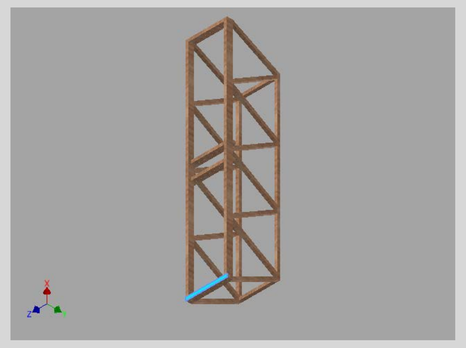
The model itself was designed using an advanced skeleton design method that undertakes creating different parts all in one model for ease of geometric constraint association. Individual parts are then derived and assembled to create a finalized assembly model.





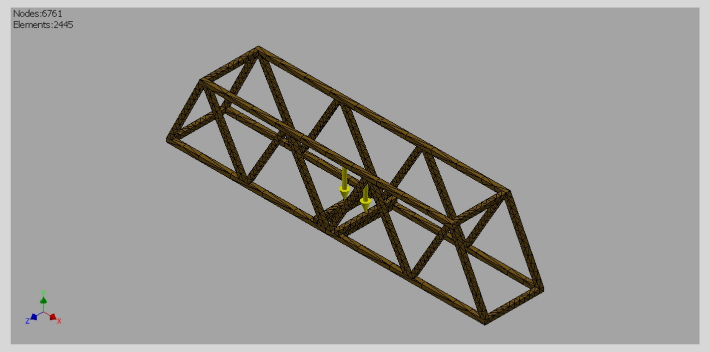
Identical frame members were only created once and then assembled multiple times in order to reduce errors. The final assembly has physical constraints assigned to each member defining joint attachments.

Simulation Set Up



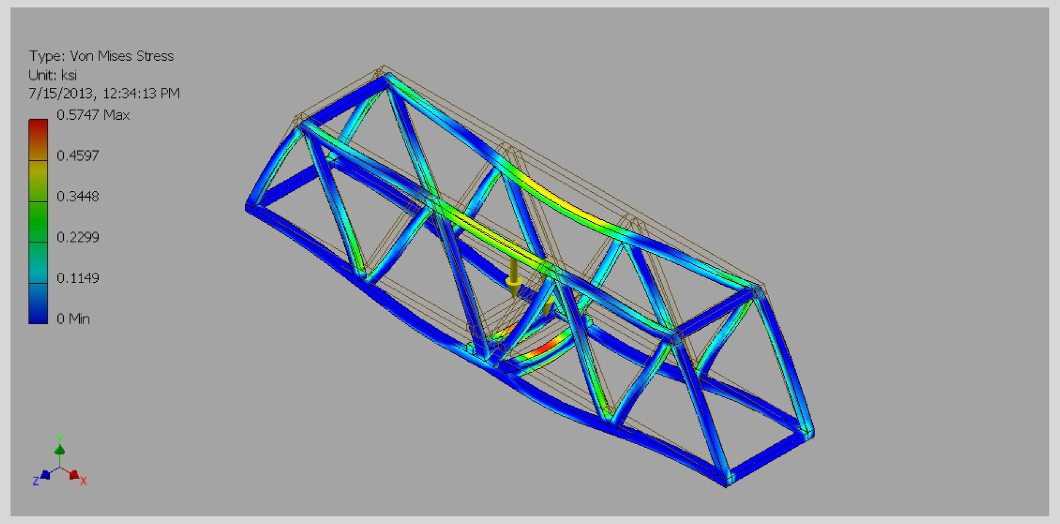
With the model assembly created with proper joint definitions the simulation could be set up. First requirement was setting up contact points. These were the surfaces that were defined as being stationary during the model simulation and in real life would have been the surfaces contacting the testing bench.

The next element that needed to be set up to run the simulation was to define the forces that would be acting on the model and where those forces would be placed. A mesh was then calculated by the computer, breaking down the overall shape into the small finite element nodes.



Static Analysis Simulation Results

The FEA simulation results are quite remarkable to look at. A visual graph showcases areas of high stress where the model would see extreme deflection and structure failure.



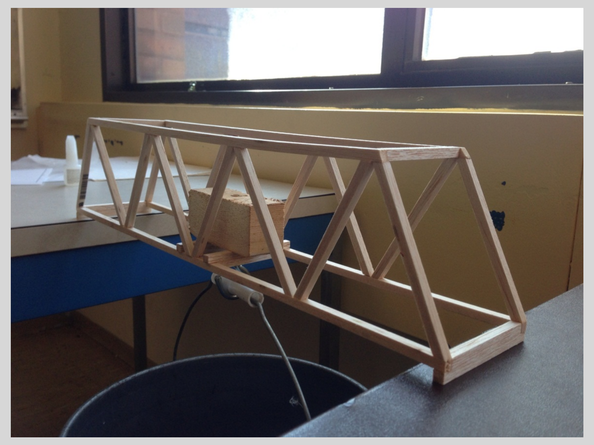
FEA Results Summary Report

Result Summary

Name	Minimum	Maximum
Volume	12.8761 in^3	
Mass	0.0758242 lbmass	
Von Mises Stress	0.000587114 ksi	0.555201 ksi
1st Principal Stress	-0.00448934 ksi	0.555384 ksi
3rd Principal Stress	-0.552965 ksi	0.0244087 ksi
Displacement	0 in	0.00626225 in
Safety Factor	0.0000341765 ul	0.0028916 ul
Stress XX	-0.463332 ksi	0.180668 ksi
Stress XY	-0.169029 ksi	0.162052 ksi
Stress XZ	-0.0321903 ksi	0.0269793 ksi
Stress YY	-0.267215 ksi	0.268446 ksi
Stress YZ	-0.141263 ksi	0.119139 ksi
Stress ZZ	-0.552963 ksi	0.555384 ksi
X Displacement	-0.00129418 in	0.00129705 in
Y Displacement	-0.00626159 in	0.0000689831 in
Z Displacement	-0.00279911 in	0.0022525 in
Equivalent Strain	0.000000349848 ul	0.000300632 ul
1st Principal Strain	-0.0000297624 ul	0.000368196 ul
3rd Principal Strain	-0.000366592 ul	0.0000161819 ul
Strain XX	-0.000307169 ul	0.000119775 ul
Strain XY	-0.000112059 ul	0.000107434 ul
Strain XZ	-0.0000213408 ul	0.0000178861 ul
Strain YY	-0.000177152 ul	0.000177968 ul
Strain YZ	-0.0000936517 ul	0.0000789843 ul
Strain ZZ	-0.000366591 ul	0.000368196 ul
Contact Pressure	0 ksi	0.782443 ksi
Contact Pressure X	-0.744245 ksi	0.73538 ksi
Contact Pressure Y	-0.321981 ksi	0.494518 ksi
Contact Pressure Z	-0.484602 ksi	0.515379 ksi

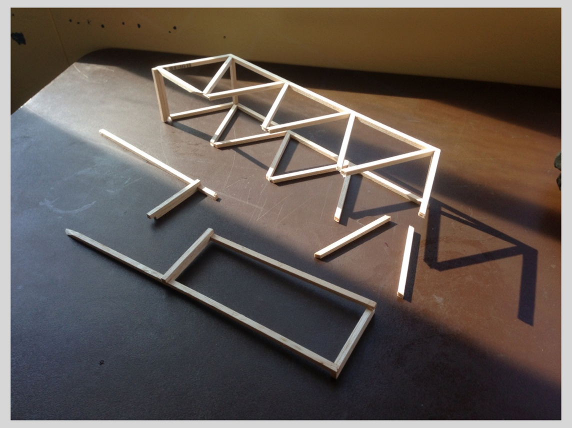
The results summary report then computes and outputs of the desired information into a general table. The engineering student would use this information to find a design that would have the lowest maximum Von Mises stress and the highest minimum Safety Factor. Armed with those numbers the students must compare designs to find the structure with the lowest mass and the highest minimum safety factor and that should be the design the student chooses to build for competition.

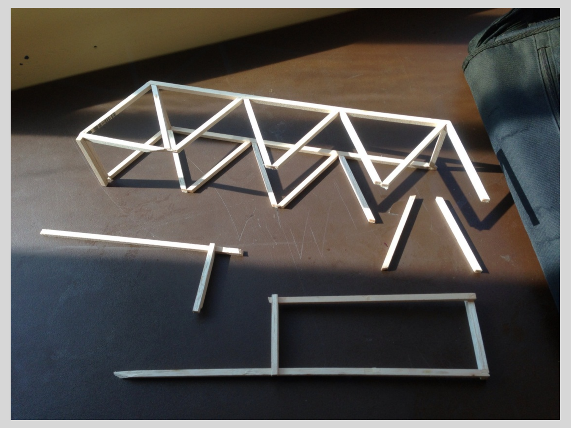
Real World Testing Results



To verify the FEA virtual model simulation a real balsa structure based on one of the models was created to test to failure.

With the test complete it showed that the model failed where indicated on the simulation results.





More extensive damage did occur, however the catastrophic nature of the failure was initially generated from the shown location.

Application Conclusion

The research done on this topic shows tremendous opportunity for application in the Engineering CAD classroom. The real structure test proved the analysis data to be valid. The data can be analyzed and students would be able to then “test” their structures virtually without building it first to maximize their design intent and success rate.