**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) 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.

**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.

**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.

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.

**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 in the model simulation and in real life would have been the surfaces contacting the testing bench.

**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.

**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.

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.

**FEA Results Summary Report**

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.

**Finite Element Analysis Application In High School Engineering CAD**

Daniel Stumpf, Thomas Scrace

Hutchinson Central Technical High School, UB