Analytical Frame Design for Commercial ZTR Mowers

Aleysha Kobiske, Mathew Weglarz, and Joshua Wilson

The Ariens Company

Abstract*: Computed simulation can be a powerful tool when utilized to predict the behavior and life of a structure. Difficulties in properly utilizing finite element software arise when determining how to implement the loads and boundary conditions into a model so that it correlates with the real world.*

This paper will discuss how the engineers at The Ariens Company utilize Abaqus/CAE and fe-safe software throughout the design process of a zero-turn mower. These two programs, in collaboration with Wolf Star Technologies True-Load software, are used to determine the actual loads acting and predicted life of a current zero-turn mower frame. Once these loads are known, the capabilities of Abaqus/CAE allow for the model of the current frame design to be easily and confidently constructed. The analysis of that frame is then used to drive the design of the new frame which is then analyzed with the loads from True-Load in Abaqus/CAE. When the correct loading is applied, meaningful and representative data of how the frame will respond in real-world conditions result. This results in a more efficient product design due to a deeper understanding of structural behavior.

Keywords*: ZTR mower, Finite Element Analysis, Fatigue, Load Identification, Abaqus, fe-safe, True-Load, Strain, Strain Gauges, Inverse Load Identification*

1. Introduction

The integration of fe-safe and True-Load with Abaqus/CAE as the FEA preprocessor, solver, and postprocessor is a valuable combination that allows for a confident analysis of a zero-turn mower frame. A data-driven design of the new mower frame is accomplished due to the ability for Abaqus to communicate and interact with other software. The simulation capabilities of Abaqus provide insight into structural behavior under specified loading and boundary conditions. In cases where loading and boundary conditions are unknown, the event being simulated will not present meaningful results. This is the case for the analysis of a new frame design - the loads driving the response of the frame are unknown. To determine the loading to be used during the analysis of a new frame, True-Load is implemented for its inverse load-identification method. Abaqus provides True-Load with simulated strain response from applying unit loads on the structure, and then True-Load provides Abaqus with the loading functions, allowing for Abaqus to provide a meaningful simulation. The loading functions found to be acting on the current frame are then used in the analysis of the new frame, driving its design.

The process used to find the actual loads acting on the mower frame starts by modeling the current frame in Abaqus/CAE where unit loads are applied to known locations. Once the model is solved, the strain data output from Abaqus is used by True-Load to determine optimum gauge locations. With these gauges, data is collected by operating the mower on a test track. The strain data collected is then compared to the simulated strain values from the unit loads applied to the model in Abaqus. By analyzing the strains from the simulated unit loading and the field strain data, loading functions can be determined by the True-Load software. Using these loading functions derived from the measured strains in Abaqus, the model is re-solved to analyze the response of the structure when it is in use. The new results then illustrate the frame's response and behavior exhibited during testing. The model of the frame with the determined loading functions is passed into fe-safe to determine its fatigue life. Fe-safe processes the Abaqus results file and, together with the loading time histories, analyzes every degree of freedom for each element to determine the crack-initiation time frame. When finished, a file is generated that predicts the fatigue life of the frame. Next, the new frame is modeled in Abaqus/CAE and is given the same reconstructed loading functions as the current frame. Using the loads found acting on the current frame to drive the analysis of the new frame allows for direct comparison of behavior and fatigue between the two frames under the same loading and boundary conditions. This comparison is a powerful tool that guides the design of the new frame to better improve upon the current frame.

Creating FEA models and investigating loads and boundary conditions on the structure results in a model that is representative of the actual structure. Abaqus, when combined with True-Load, provides engineers at Ariens with results that are nearly identical to field test events. The engineers are then able to determine areas of concern and life expectancy of the frame through simulation. This allows for any area of concern to be identified and addressed analytically before production. This enables Ariens to design a great, structurally sound product in a more cost-effective manner that meets the demands of the market.

2. Project Motivation

The mower will be used by both commercial and residential customers of Ariens. It is designed to meet a variety of needs and hold up to commercial demands. The goal of the redesign is to produce a superior product that addresses any areas of concern in the current design and incorporates enhancements based on customer feedback.

2.1 Areas of Improvement in the Current Design

The first steps of the redesign are to examine areas of the current mower that can be improved, such as geometry, material thickness or maintenance access. One area examined for the new design is the loading path of the structure. The stresses around the welds were closely examined in the post processor of Abaqus to identify any areas of concern and to ensure the load was distributed through the machine as expected.

Another area of focus was the front-end appearance of the current mower. A great deal of effort went into designing the front-end piece so that it functioned as a stiffening member between the frame rails while still being visually appealing to customers. The responsibility of the front end is to pull the customer's eye towards the machine while enhancing the overall structure of the frame. Abaqus post-processor played a vital role in consultation for the design of the front end to achieve an enhanced structure with the desired finished look. These areas were the major focus for the design of the new frame that both benefitted greatly from the use of Abaqus/CAE and fe-safe.

2.2 Customer Expectations for the New Design

To truly make a better product, the voice of the customer must be heard. The Ariens marketing team set out to determine what customers have to say about the current design and what they would like to see in the new one. From a customer standpoint, the biggest demand was for larger tires on the new frame. In order to produce a frame able to support a larger tire, many of the components on the frame had to be modified. Customer feedback was incorporated as often as possible into the design so that structurally a better product was designed that still appeals to customer demands.

3. Procedural Overview

As discussed, the first step taken in analyzing the new frame design was to analyze the current frame. In order to have a true data-driven design, the loads of the structure had to be known. To determine these loads acting on the frame, the integration of Abaqus and Wolf Star Technologies True-Load program was used extensively. The relationship between Abaqus and True-Load allows the engineer to determine gauge placement on a model based on the strain results of unit loads acting on the structure as determined by Abaqus. Then a field test is run on the structure with gauges positioned as determined by True-Load. The strain data gathered during the field test are read into True-Load which interfaces with Abaqus to get the simulated strains resulting from unit loads. Through load-recovery techniques, True-Load returns the loading functions from the field data to Abaqus. These loads can then be utilized on the structure for fatigue analysis. To illustrate this process used for analysis of the current frame, a simple cantilever beam example is considered as seen below in Figure 1.

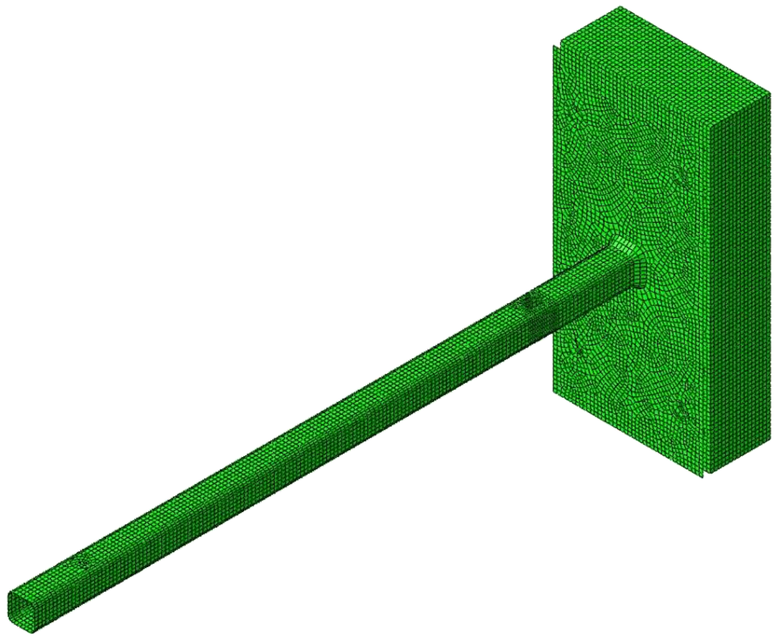


Figure 1: Cantilever Beam Example

This beam is modeled using shell elements with a mesh weld at the end. The plate the beam is welded to is then bolted to a steel block for testing, creating a contact constraint between the plate and the block to model the behavior exhibited by the structure. A finer mesh is developed around the hole in the beam to provide a more detailed look at how the structure is behaving around the hole. Shell elements must be utilized because of the surface strain values required by True-Load. A unit load of -1 LBF is applied to the reference point which is tied into a half-inch diameter area 2 inches from the end of the beam. The block of the beam is then fixed; Figure 2 illustrates the actual setup used in testing and Figure 3 shows the FEA model in Abaqus/CAE representing the setup.

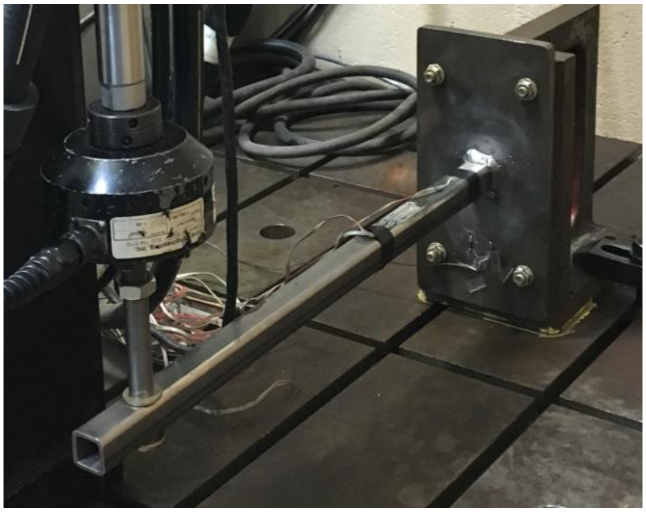


Figure 2: Cantilever Beam Setup

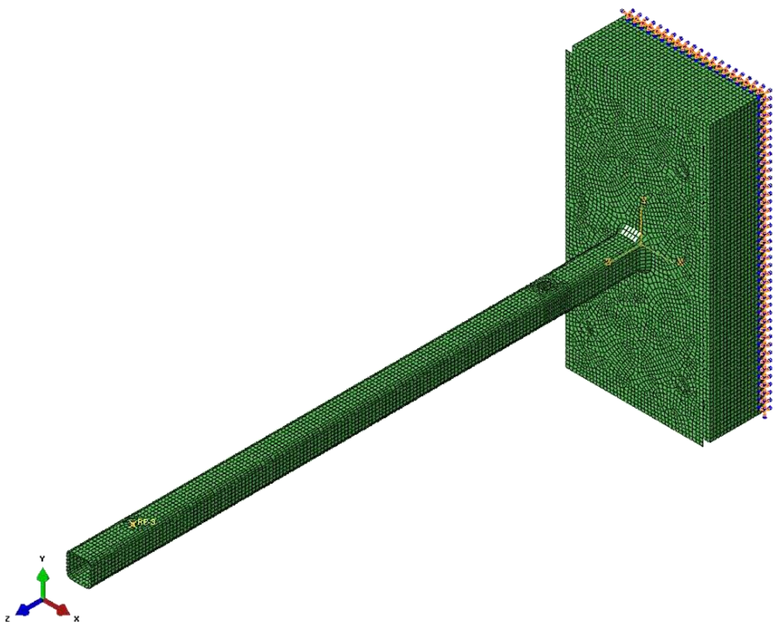


Figure 3: Cantilever Beam Assembly Model

The plate that the beam is welded to is joined to the fixed block through TIE constraints. The surfaces of the plate and the block also have a contact constraint. From the unit load in the y-direction to this model the resulting maximum principal stresses shown below in Figure 4.

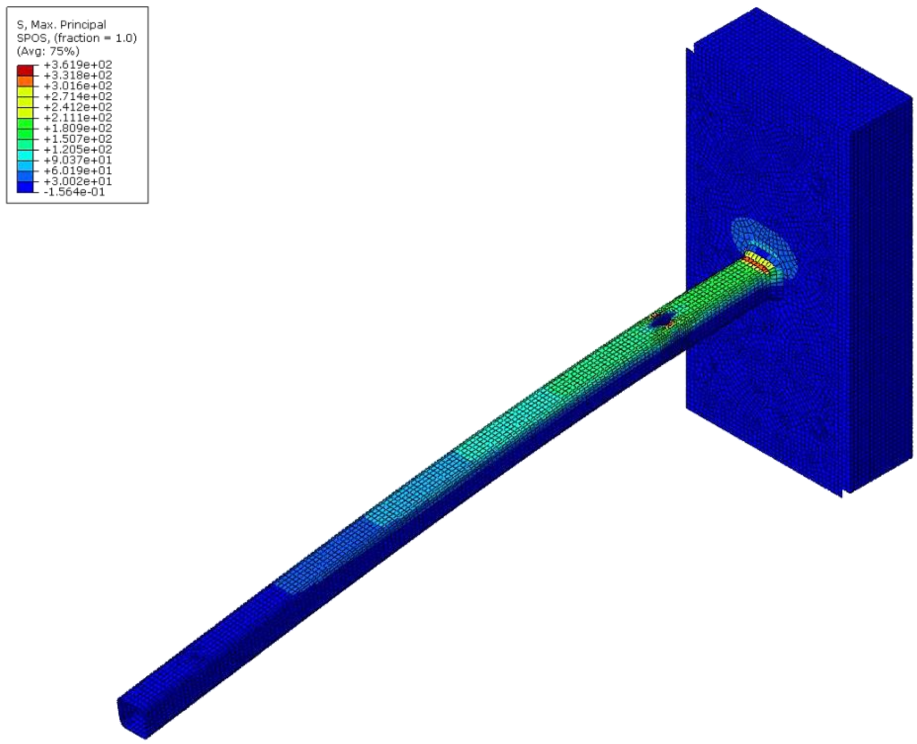


Figure 4: Cantilever Beam Unit Load Stress Result

As expected, high-stress areas occur at the weld and around the edges of the hole. Using True-Load the following gauge locations were determined as shown in Figure 5 below.

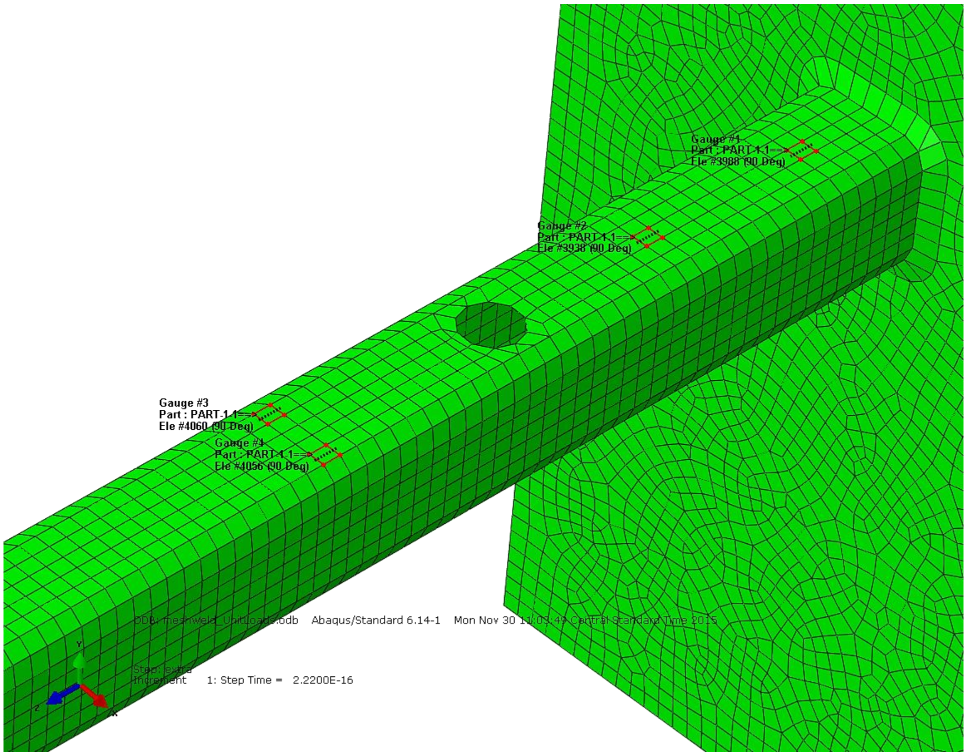


Figure 5: Cantilever Beam Gauge Placement

These gauges were then laid on the physical beam at these locations in order to collect strain data during test and compare it to the strain from each respective virtual gauge. Using this data, True-Load recovered the loading function within 5% error, as seen in Figure 6. In the figure below the load applied on the beam during test is shown in green, while the load recovered by True-Load is shown in blue.

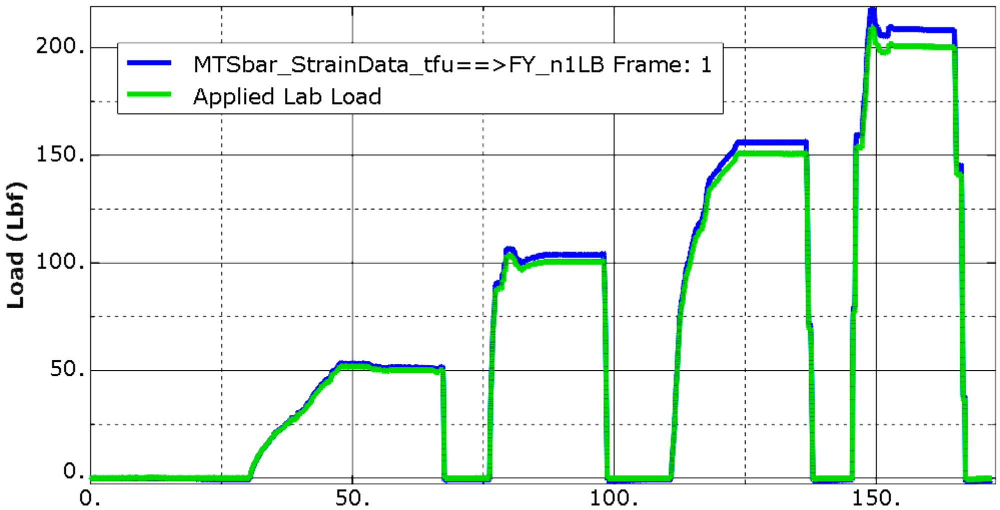


Figure 6: Cantilever Beam Loading Function Applied (Green) and Recovered (Blue)

Figure 6 shows how the profile and magnitude of a loading function can be successfully recovered through the use of one test opposed to guessing what the loading function is, designing to it, testing it, and then finding out it was wrong. The plot in Figure 7 shows a cross plot of how the measured load compares to the simulated load. As noted, it is a straight line on a 45º angle signifying that the load recovered matches the load measured in test.

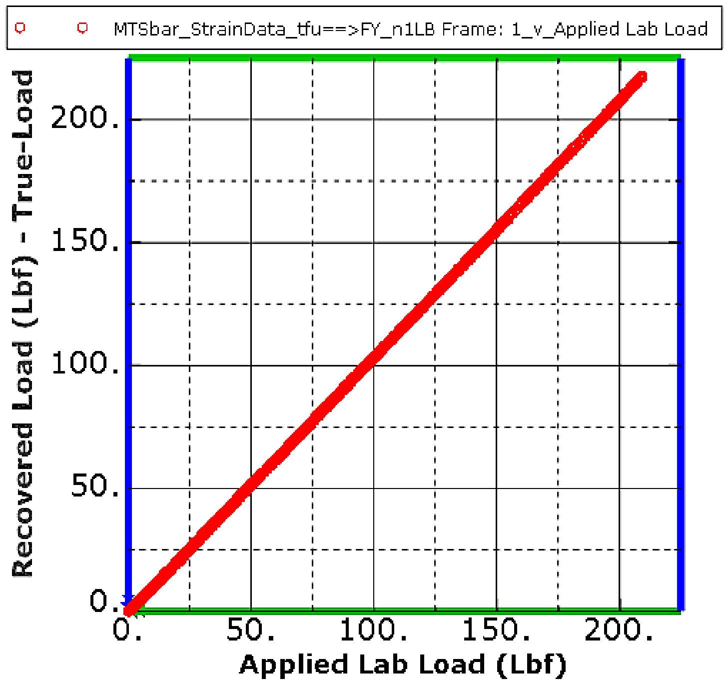


Figure 7: Cantilever Beam Simulated Strain and Measured Strain Correlation

With this correct loading function, the ability of Abaqus/CAE to simulate structural behavior reaches its full potential. It also becomes possible to now run a fe-safe fatigue study on the beam to determine the life of the beam, under this known loading function. The result is shown below in Figure 8.

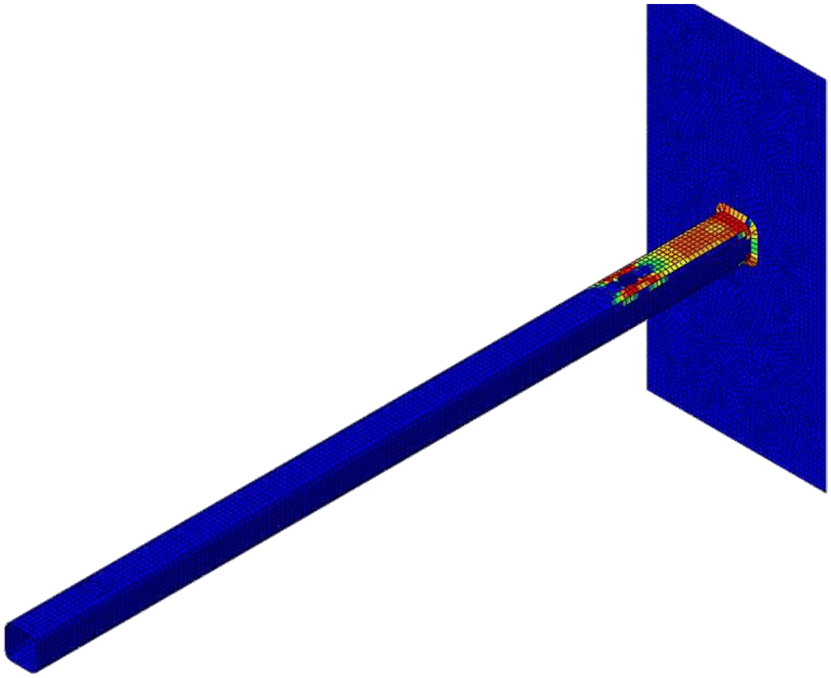


Figure 8: Cantilever Beam Predicted Fatigue Results

With the predicted fatigue life from fe-safe being 40,110 cycles the beam was then put to test to see how accurate the predication was. Results from fatigue tests show positive correlation with the predicted life of the cantilever beam.

Through this example this method has been proven effective when used correctly at predicting the life of a structure, given the correct loading. While there are several factors, such as location of loads, type of loads, and boundary conditions that must be known, the results from Abaqus, analyzed by an experienced FEA user, can ensure that those factors are as accurately represented as possible. Using the simulation power of Abaqus/CAE, the load predicting capability of True-Load, and the life prediction of fe-safe, the new mower frame is designed.

4. Modeling of Current Frame

The frame model starts with a SolidWorks step file. The part is imported into Abaqus/CAE and is mid-surfaced for the use of shell elements. Shell elements are used to determine surface-strain values from the structure, which are compared to the strain values read by strain gauges during testing. Appropriate shell thicknesses are assigned to each section along with steel property values. TIE constraints are utilized to act as the welds between partitioned faces on the model. To represent the engine, transaxles, seat, and deck masses acting on the frame; reference points at each respective center of gravity are coupled to appropriate attachment points along the frame and are assigned masses and moment of inertia values. An eighth inch element size is used for the meshing of the frame. Figure 9 below shows the current mower's finite element model.

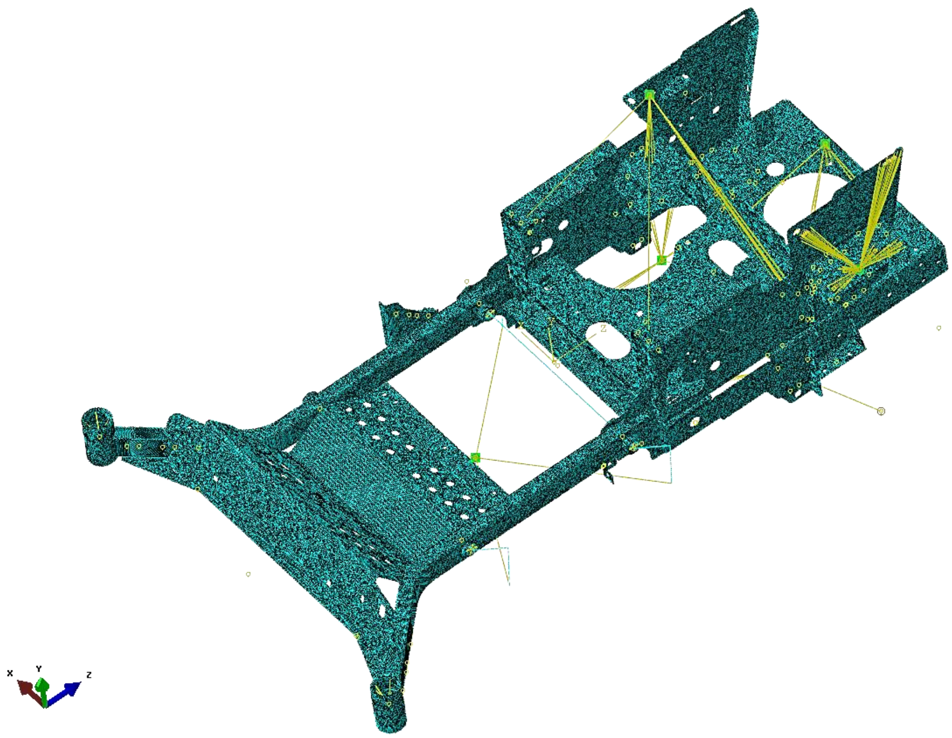


Figure 9: Mower Frame Model

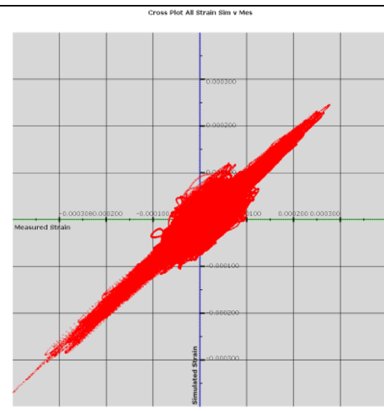
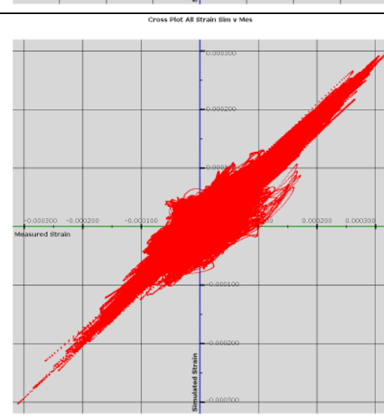
4.1 Modeling of Unit Loads

To determine the loading on the frame, a set of unit loads are defined that represent the actual loading seen by the machine when in use. The loads acting on this machine are all driven through the wheels. Therefore, unit loads are set up in individual steps for X, Y, and Z directions on each wheel. A loading point is placed in the middle of each caster in the front and at the center of each of the rear wheels; resulting in 12 unit load steps. Inertial relief must be applied in all DOFs since the machine is not constrained at any point during test.

4.2 Results of Unit Loads

The strains output from the unit loads are used to determine gauge locations with True-Load. The user friendly graphics capability of True-Load as implemented in Abaqus allowed for gauges to be moved around as seen fit by the engineer. A total of 22 gauges had to be placed on the mower frame. With the gauges laid, the machine was put through testing to collect actual strain values during operation. The strain data collected was read back into True-Load, which output the loading functions found. The loading functions were applied to the model so that the simulated strain signals could be compared to the measured strain signals. Table 1 below shows this correlation of the measured vs. simulated strain for each model using the strain simulated in Abaqus and True-Load and the strains measured on a unit during testing.

Table 1: Strain Correlation for Current Frame

	Correlation for Strain from Loads acting on Front Casters
Test Track Study 1	
Test Track Study 2	

As shown in the table, the simulated strain data, which was derived from applying scaling functions determined by True-Load to the unit loads applied to the model in Abaqus/CAE, correlated well with the strain that was measured from the test track. The loading functions found by True-Load show a favorable correlation so they are implemented into Abaqus/CAE in order to determine the behavior and life expectancy of the current frame. This data is then used as a comparison to the new frame design.

4.3 Fatigue of Current Frame

With the correct loads acting on the frame, the model is exported to fe-safe in order to determine its life expectancy and eventually compare it to the life expectancy of the new frame. Figure 10 below shows an overall plot of the life-repeats found from fe-safe on the frame model.

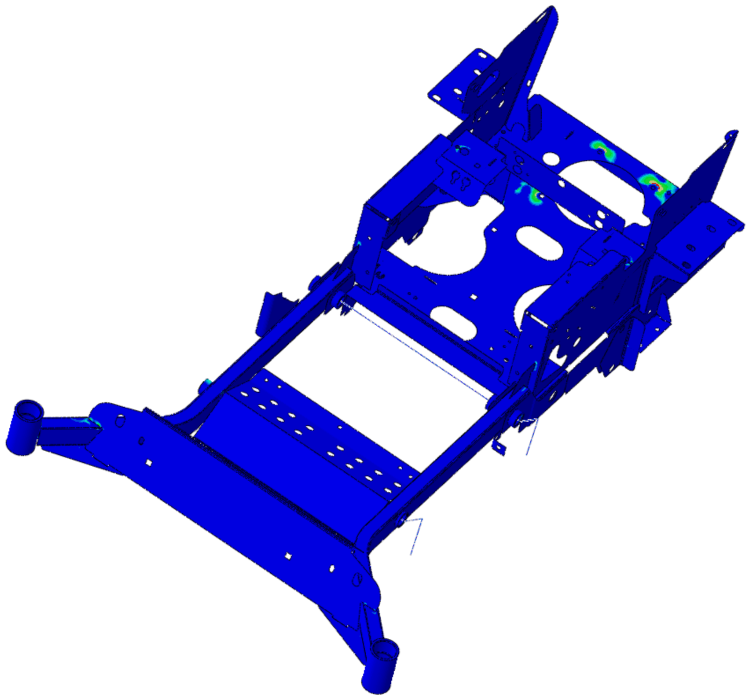


Figure 10: Predicted Fatigue Life Results

The areas of low life are found to be the result of TIE constraints acting too rigidly around the bolt holes for the engine. Besides these areas, the rest of the frame was found to meet the required life expectancy.

5. Modeling of New Frame

This process of determining loading functions and fatigue life of the current frame continued as the new frame was beginning to be designed. Once a preliminary design was completed, the loading functions found to be acting on the current frame were integrated into the analysis of the new frame. The new frame was modeled in Abaqus/CAE in a similar fashion to the current frame. When it came to creating boundary conditions and loads on the new frame, the loading functions True-Load found on the current frame were implemented with similar boundary conditions. With these loading functions, Abaqus/CAE was able to give valuable feedback during post processing on the direction to take in order to achieve the desired results from the new design. Fe-safe was used to look at fatigue life and areas of concern when the new unit was running under the simulated test track loading. With the results from Abaqus/CAE and fe-safe, the model was able to be redesigned to present more favorable structural responses on the first try without the need to produce a prototype, test, redesign and repeat. Once the analysis presented a structural response of the new frame with no concerns, a prototype was built for testing.

6. Conclusion

Abaqus/CAE was used as the primary analysis software for the new design of the mower because of its ability to communicate easily with other software. The ability to make changes in the pre-processor and easily view them in the post-processor, earned the confidence of engineers at Ariens when the correct loading and boundary conditions are given by the True-Load software. Having the ability to simulate a new product lends many advantages, but if the simulation is not representative of what is happening in reality then confidence and time is lost. With correct boundary conditions and loading functions implemented on a structure, engineers are able to see how the structure behaves with the help from Abaqus post processor to animate, highlight, and exaggerate deformations. With Abaqus and the load reconstruction capabilities of True-Load, the analysis and test community at Ariens can deliver new products with high confidence to meet the voice of the customer.