# Quasi-Static Finite Element Analysis (FEA) of an Automobile Seat Latch Using LS-DYNA

Song Chen, Yuehui Zhu

Fisher Dynamics Engineering
33300 Freeway Drive
St. Clair Shores, Michigan 48082, USA

Tel: (586) 296 – 1770 ext. 307, 278
E-mail: schen@fisherco.com
yzhu@fisherco.com

Abbreviations:

FEA: Finite Element Analysis
FOS: Factor of Safety

Keywords: FEA, LS-DYNA, Quasi-static, Latch, Design

## ABSTRACT

In the present automotive industry, all suppliers and OEMs are focusing on designing, developing and manufacturing products and automobiles with higher quality, lower cost and faster delivery to the customers. The automobile industry has placed a significant amount of efforts including time and funding into developing products that can meet these challenges.
This is probably one reason that in recent years Finite Element Analysis (FEA) has been widely used and become a mainstream design and developing process in automotive industry. This is especially true in the Noise, Vibration, Harshness (NVH) and safety fields.

This paper presents a project of Fisher Dynamics in the system level Quasi-static FEA using LS-DYNA effectively directed the design of an automotive seat latch to meet the stringent high load requirements. The analysis successfully predicted the results of physical tests including the ultimate load. Hence the company was able to deliver a design in conformance with the specifications of the customer on time.

## INTRODUCTION

Fisher Dynamics designs, develops and manufactures Seat Recliners including Power, Manual, Linear and Pawl/Sector, also Armrests, Latches, including Seat Back and Floor Latches, Headrests, and other safety related products. A large number of these designs are included in seat integrated systems of high strength with annual production of millions of units. Production of such a variety and such an enormous number means that any small improvement in the Research and Development phase can result in large degree of savings for the company in terms of both time and money. Performing FEA in the design and develop process has become a routine practice in Fisher Dynamics.

The design of a structural latch for an automotive seat system has become more and more complicated and challenging. A latch today not only needs to offer more functions but also has to meet stringent load specifications with strict packaging constraints.

A latch generally consists of an upper arm, a locking plate (Claw), a release cam and two supporting package plates. In an FMVSS 207/210 test which is the Federal Motor Vehicle Safety Standard, there are three areas that yield. They are the upper arm, claw and cam. In this paper, the key load-carrying components are the Claw, the Cam and the Plates. The load can be simplified as a straight pull downward force. Figure 1is a second row seat with a high latch.

By performing FEA analysis the load carrying-ability of the design can be assessed before releasing the design for prototype. If a design can not meet the specification, the FEA model combined with engineering calculations can provide insight into where additional design changes are required.

## APPROACH

### *Material Properties*

The materials used for the key parts in this program were all high strength steels. Some of them were heat-treated carbon steels. The property data of these steels were taken from Society of Automotive Engineers (SAE) J1397 and Fisher Dynamics own Material Standard.

### *Material Model*

At the ultimate load plastic yielding was desired, Piecewise Linear Plasticity Material Model was used for all the key components. This material is an elastic-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. Also, yielding based on a plastic strain or a minimum time step size can be defined. This material model is identified as Type 24 in the LS-DYNA code, and is available for beam, shell and solid elements.

### *Element Type Selection*

Fully integrated 8-node S/R solid element type was selected for all key components as shown in Figure 2.

### *Load*

Since concentrated node force load applied to a deformable material causes stress concentration. To simulate the reality more accurately, concentrated pressure loading was chosen. This was accomplished by applying the loads that represent pressures to the solid elements' faces that are supposed to contact the Load Bar.

### *Constraints*

The nodes at the periphery of the two mounting holes were placed constraints, i.e., all six degrees of freedom of these nodes were fixed, just like in an FMVSS 207/210 test set up.

### *Contact Set Up*

The interaction between disjoint parts was defined as Contact Automatic Single Surface. This was true except for the interaction between a pivot and its hole was defined using Contact Entity.

*Running Time*
After several runs with kinetic, internal and total energies of the assembly being examined, it was decided to attain the ultimate load in 12 milliseconds.

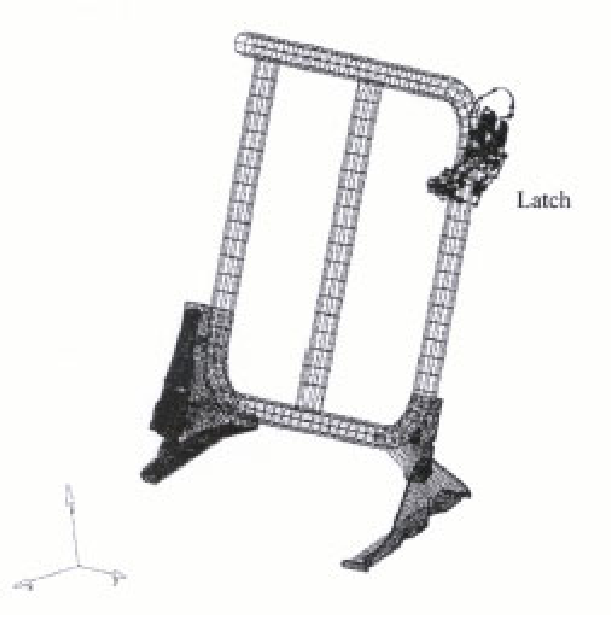

Figure 1. A 2nd Row Seat With High Latch

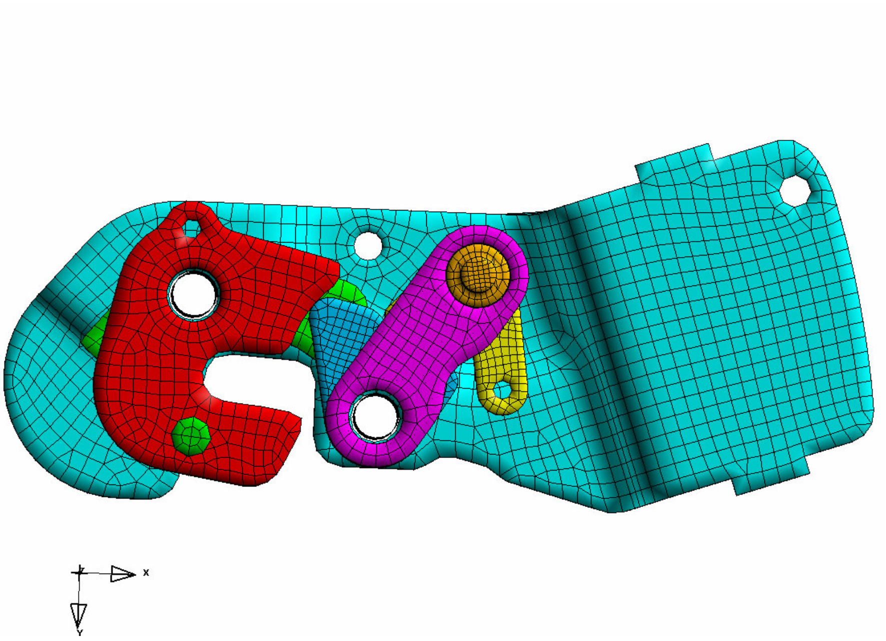

Figure 2. The FEA Model

## DISCUSSION OF RESULTS

### *The First Design*

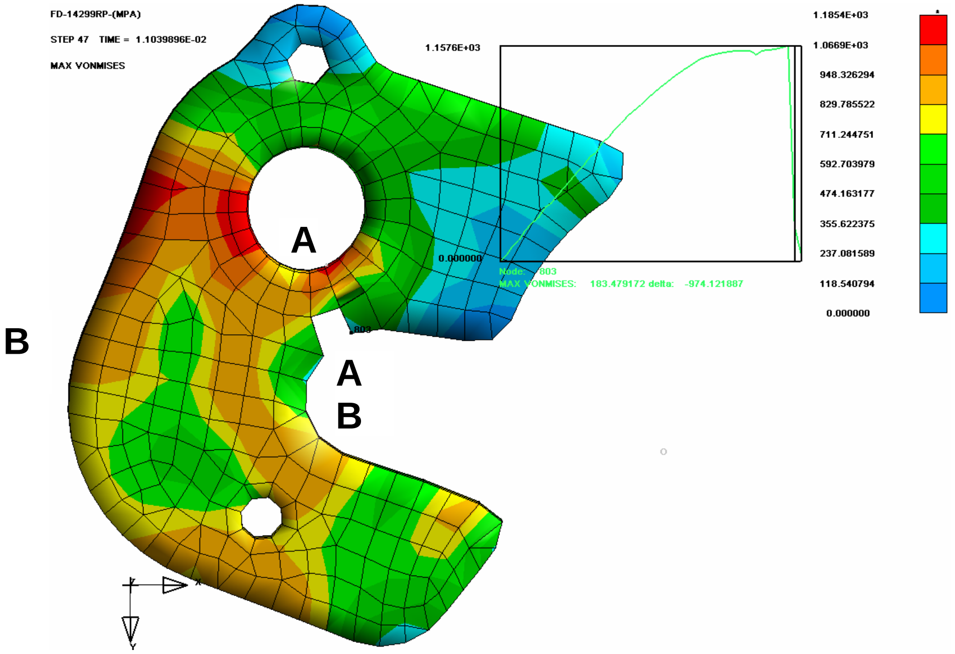

Figure 3. The FEA of the First Design

Figure 3 shows that the Claw would yield at 91.7% of the 35KN ultimate load applied, or 32.1KN, along section AA. It also indicates that the section BB had the second highest stress.
The physical test on the First Design found that the Claw yielded at the position the FEA had predicted and at 32.03 KN as shown in Figure 4 and Figure 5.

Figure 4. The Post - Test of the First Design

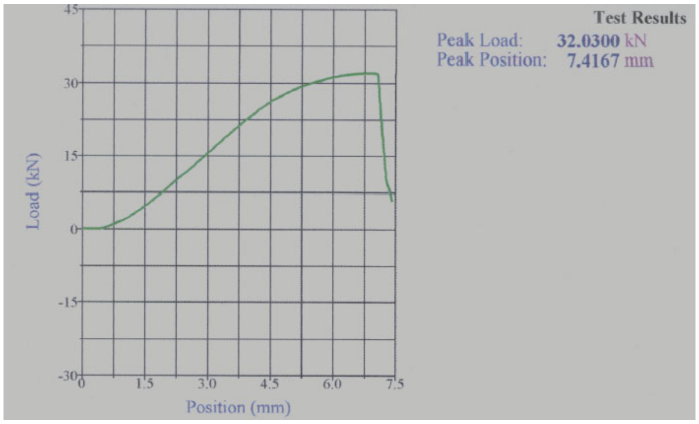

Figure 5. The Load Teat Data of the First Design

The First Design and FEA were a success, but a second design was needed to meet the increased load specification.

*The Second Design*
The FEA on the first design showed that the section modulus of section AA and BB of the Claw need to be increased to meet higher load specification. This means that either the thickness or the length of the sections needs to be increased. Since there was no room to increase the thickness and increasing length was more efficient. It was decided to increase the length of section AA to 8.26mm, BB to 28.64mm based on the following calculations.

$$I = b \cdot \frac{\left(H^3 - h^3\right)}{3}$$

This is the formula of the moment of inertia of a b x (H-h) square about an axis which is h away from the square.

$$\sigma = \frac{M \cdot Y}{I}$$

$$\frac{F2}{F1} = \frac{\dfrac{H2^3 - h^3}{H2}}{\dfrac{H1^3 - h^3}{H1}}$$

M - Moment
Y - Distance of the Point from the Axis, here Y = H
$\sigma$ - Tensile Stress
F - Load Applied

Figure 6 and 7 drawn in MathCad show the Factor of Safety (FOS) vs. section length. To reach 1.0 FOS, the lengths for section AA and BB are 8.26mm and 28.64mm.

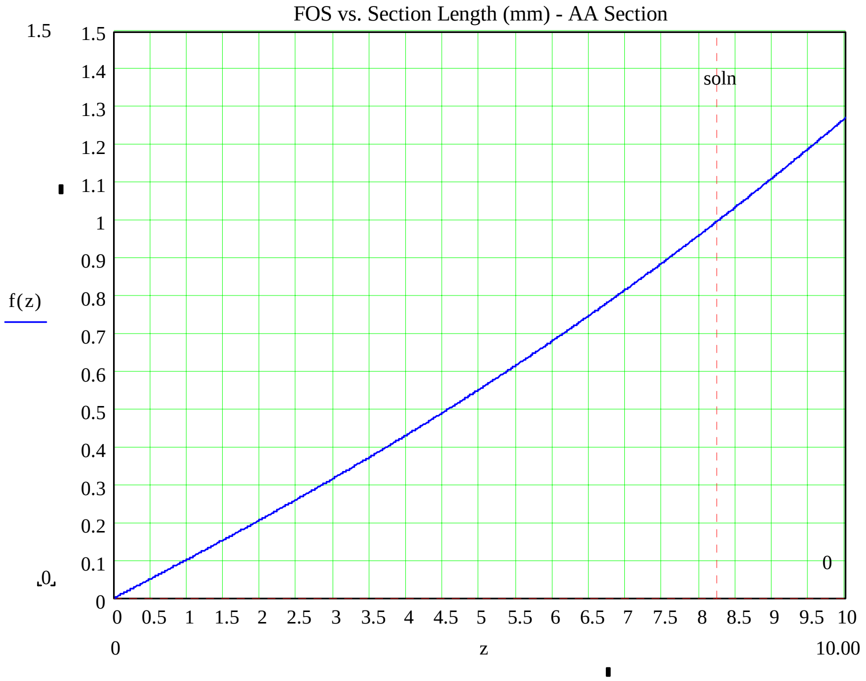

Figure 6. FOS vs. Section Length for Section AA

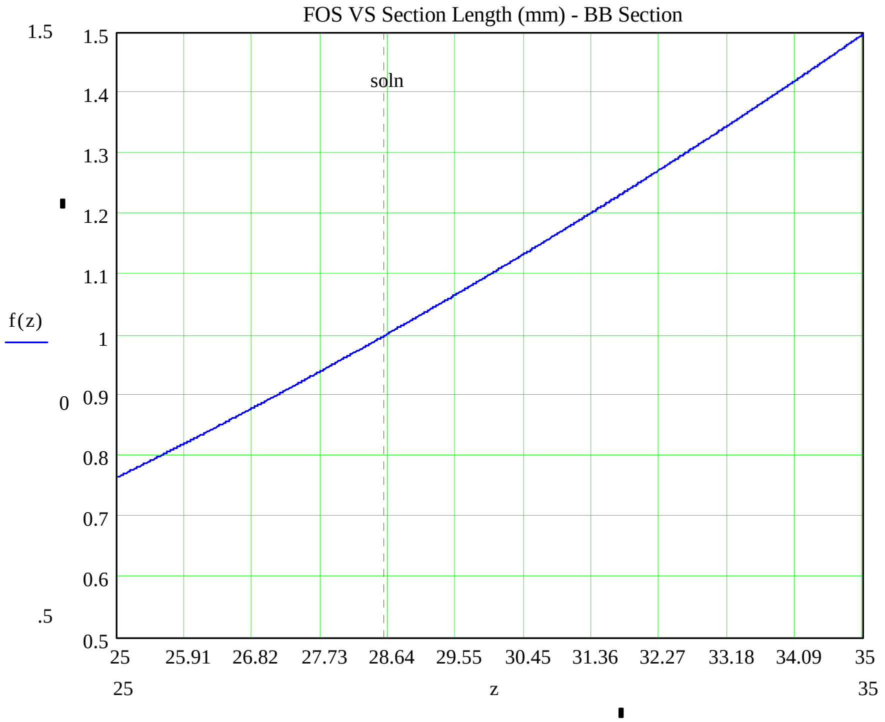

Figure 7. FOS vs. Section Length for Section BB

**The FEA on the second design shown on figure 8 indicated that the assembly would still yield at the Claw. However, the location has moved from section AA to section BB and the ultimate load would be 42 KN.**

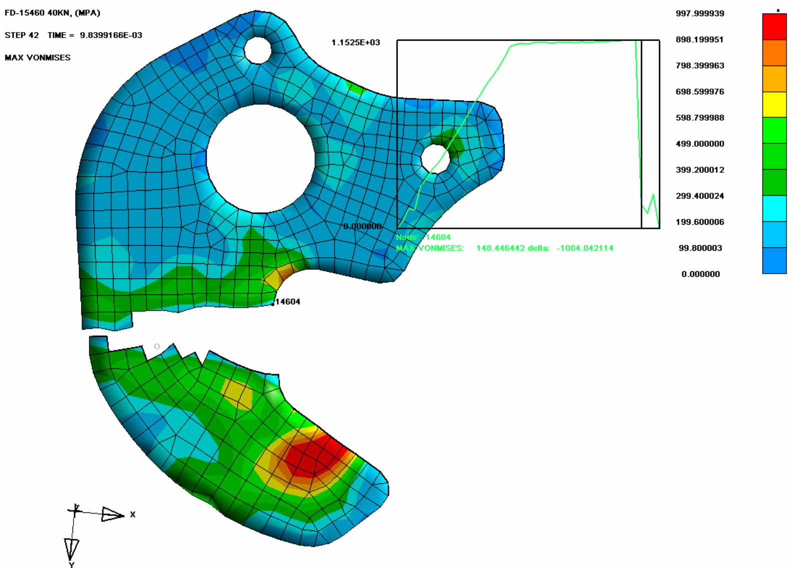

Figure 8. The FEA of the Second Design

The physical test conducted on the Second Design observed the same area yielding and ultimate load of 42.16 KN was the same as the FEA had determined it would be. (See Figure 9 and 10).

Figure 9. The Post - Test of the Second Design

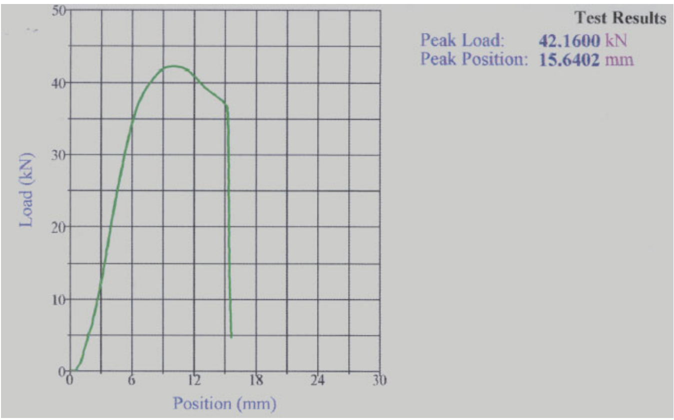

Figure 10. The Load Test Data of the Second Design

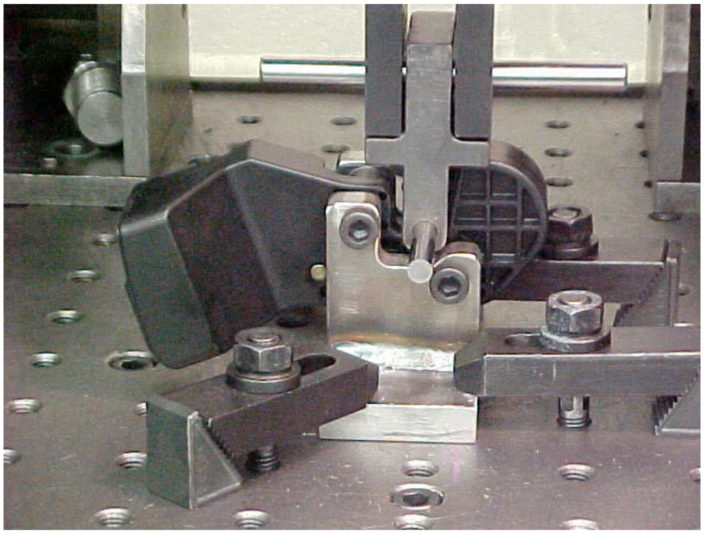

Figure 11. The Test Apparatus

## SUMMARY

An efficient Quasi-static FEA model was developed by Fisher Dynamics for use in predicting the ultimate load of an automotive seat latch. The FEA data was also used to direct the design layout. By means of engineering calculations and math application tools the intended design details could be chosen straightforwardly and graphically. Overall, the FEA model correlated well with the physical tests in terms of system yield location and peak load when LS-DYNA Quasi-static approach was used.

## ACKNOWLEDGMENTS

The authors would like to thank David Robinson of Fisher Dynamics's Engineering Department for the encouragement and modifying the manuscript , Craig Coughlin and Dmitriy Zelmanov of Fisher Dynamics's Engineering Department for the support in the design process and Steve Petrovski, Christopher Nader and James Grusser of Fisher Dynamics's Testing LAB for providing the physical testing parts and data.

## REFERENCES

ETA (2000). Proceedings of the 6th LS-DYNA User's Conference, Dearborn, MI, April 9-11, 2000.

HALLQUIST, J.O., LS-DYNA Theoretical Manual. Livemore Software Technology Corporation, May 1998.

LSTC (2001). LS-DYNA Keyword User's Manual, Version 960. Livemore Software Technology Corporation, March 2001.

OBERG, E. JONES, D.F. HORTON, L.H., and RYFFELL, H.H. (2000). 26th Edition Machinery's Handbook. Industrial Press Inc, New York, pp. 434-437, 518-521.