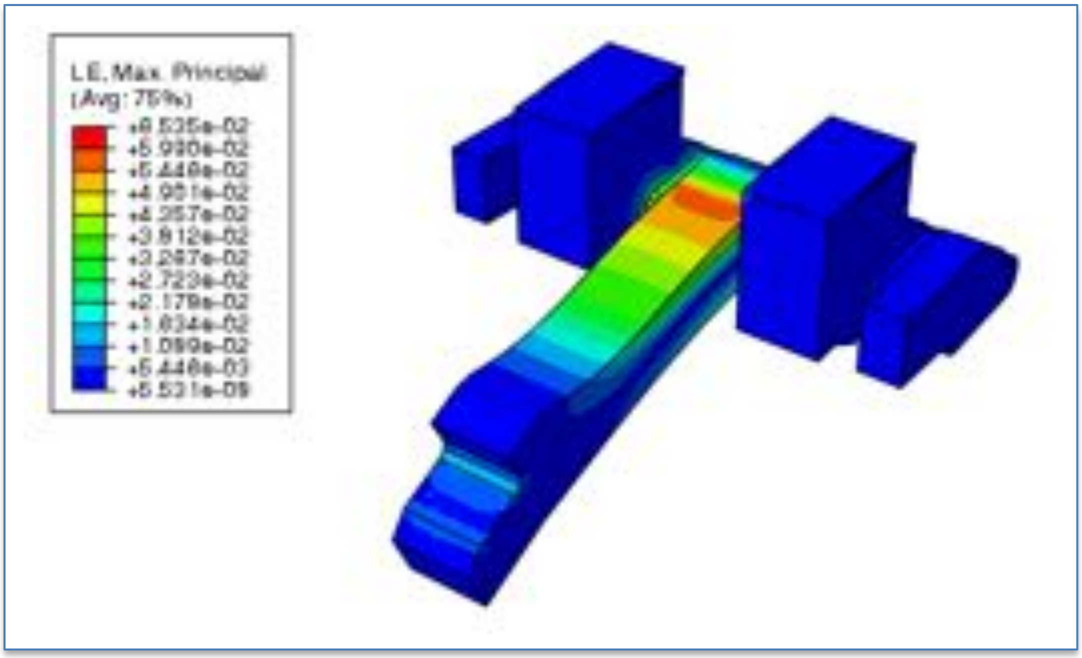


ME 461: Finite Element Analysis

Fall 2015

A Semester Report on:

The Development and Validation of Material Model for Use on Plastic Automotive Connectors

Group Members:

Reuben Kraft, Thomas Scully, Joseph Mentges, Patrick Cathrall, Sumit Zaver

PennState College of Engineering

Department of Mechanical and Nuclear Engineering, University Park, PA

Table of Contents

Executive Summary

Delphi produces electrical connectors made out of reinforced plastics for automobiles. Electrical connectors play a vital role in the safety of an automobile. The connectors complete the inner circuitry within a vehicle. If a connector disconnects or breaks while a vehicle is in use, systems of the vehicle can lose power, which can be very dangerous for the passengers of the vehicle and others on the road. A thorough analysis of the connector design is required to ensure that the connector does not break during insertion and that the connector does not come loose and disconnect the circuit.

A way to model the behavior of the plastic connectors is through finite element analysis within software such as Abaqus. The issue with the standard linear elastic finite element model is that it is more accurate for metals than plastics. An accurate representation of plastics in Abaqus requires a model that can successfully predict the behavior of plastics.

The goal of this project is to determine a model that successfully models the behaviour of the plastics used in the connectors. The model will be validated by replicating the results of tensile test data. Once the tensile tests have been replicated, the team will use that model to run a simulation on the electrical connector to determine data for the retention forces and stress strain behavior.

List of Figures

Section 1: Background and Project Plan

Background

Delphi Automotive produces electrical automotive connectors using various forms of re-enforced engineering plastics. These plastics behave differently at different strain rates. Currently, the FEA engineers at Delphi only model their parts using the linear elastic model within Abaqus, which does not capture any of the strain rate behavior or plastic strain of the material. The purpose of this project is to determine a material model within Abaqus that can more accurately model the behavior of their parts and then validate it based on actual test results.

For the connectors to properly function, a plastic response must not be initiated. The connectors deform during locking and snap back into place. If the part does plastically deform, the connector will not perform as intended to and the retention force will not be as strong. To determine if plastic deformation occurs, a plasticity model must be implemented.

Project Plan

Delphi's Penn State capstone project team completed tensile test data on the tree plastics at three crosshead speeds of 200 mm/min, 50 mm/min, and 5 mm/min. The team will use the Johnson-Cook Plasticity model within Abaqus to model the materials and capture the strain rate dependence of the plastics. In order to validate the model, the tensile tests conducted by the team will be simulated in Abaqus using the determined material parameters for the Johnson-Cook model. The axial stress and strain data from Abaqus will be compared with the test data from the tensile tests conducted by the Capstone team to validate the model. Once the team has validated the models, the team will begin to analyze Delphi's GT connectors using the material models. The tensile test data collected by the team is included below. It should be noted that due to an error with the injection molding process, accurate test data for M4629 was not collected, and Delphi only had data available on hand at one strain rate.

Figure 1: Test data for M6734

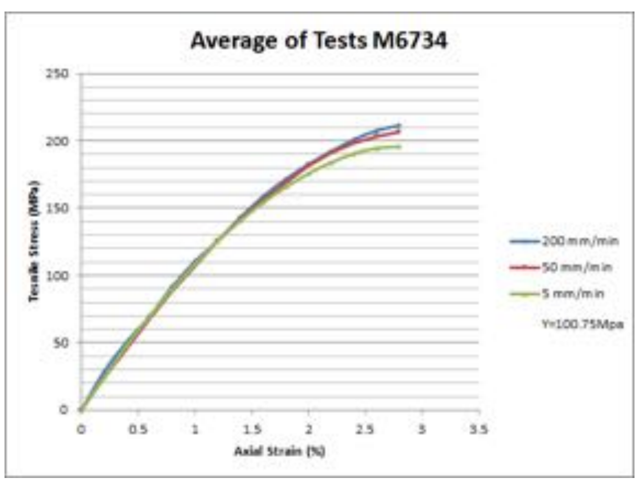


Figure 2: M4698 Test Data

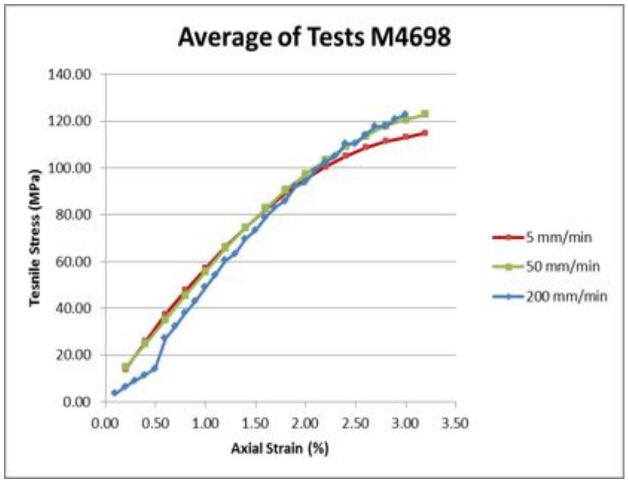
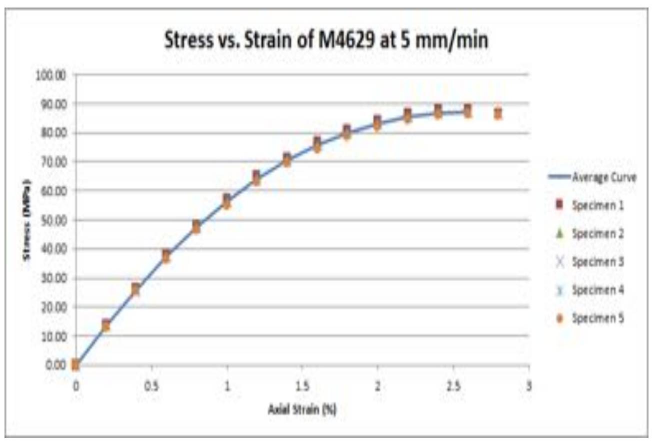


Figure 3:M4629 test data



The Johson-Cook Material Model

The Johnson-Cook Plasticity model was chosen too model the the plastics. According to the Abaqus 6.14 user manual, the Johnson-Cook model is a mises plasticity model with analytical forms of the hardening law and rate dependence. It is best used for high-strain rate deformation of various types of materials. The stress is expressed as:

$$\sigma = [A + B(\varepsilon^{pl})^n]\left[1 + Cln\left(\frac{\varepsilon^{pl}}{\varepsilon_o}\right)\right](1 - \Theta^m)$$

The testing data has been calibrated to the model and the material parameters are included in Table 1.

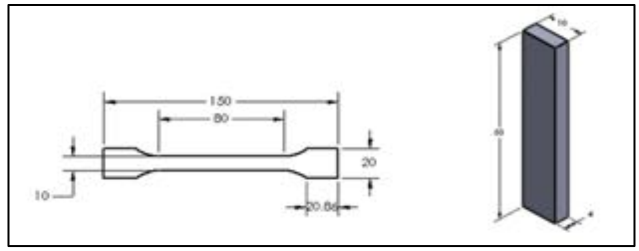
Table 1: Material Parameters for the Johnson Cook Model

Material	**E** (elastic modulus)	**Nu** (Poison's ratio)	**A** (Initial Yield Stress)	**B** (Strain hardening parameter)	**N** (Strain hardening parameter)	**M** (Strain hardening parameter)	**Theta Melt** (melting temp)	**Theta Trans** (Transition Temp)	**C** (Rate Factor)	**Eps dot** (Rate Factor)
M4698	6898	0.4	38	573	0.44	0.013	2000	1000	.044	.0013
M6734	11032	0.4	74.82	324.58	0.22	1.02	2000	1000	0.053	.00077
M4629	5986	0.4	29.06	142.89	0.19	0.99	2059	1000	-	-

Section 2: Development and Description of the CAD Geometry

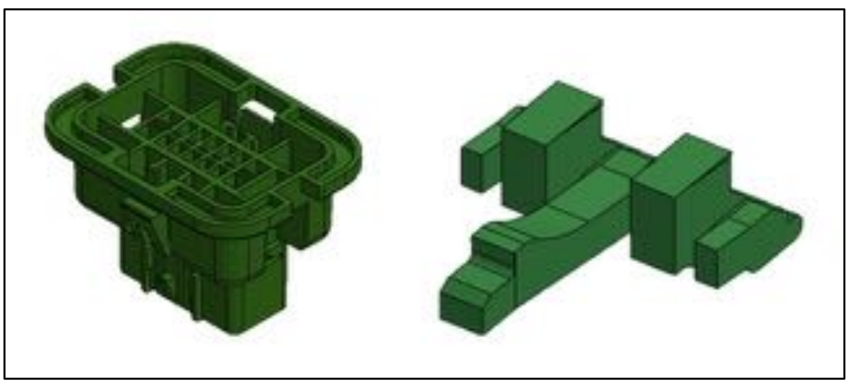
Simulations must be done on the tensile test specimen in order to validate the Johnson-Cook model and the chosen parameters. The tensile bar used in testing was the Type 1 Iso Bar. The dimensions for the Type 1 Iso bar can be seen on the left in Figure 1. However, since a 50 mm extensometer was used to collect data, the simulation can be simplified by only modeling the gauge length section. The gauge length part that will be created in Abaqus can be seen on the right in Figure 1.

Figure 4: Tesnile tesing parts (units of mm)



After the model is validated with the tensile test simulations, a simulation will be done on the GT Connector as seen in Figure 2. The area of focus is the locking arm mechanism that is inside of the body of the part. The part has been modified in solidworks to only include the relevant material for analysis. This locking arm as seen on the right in Figure 2 has been imported into Abaqus via an SAT file.

Figure 5: GT connector Part



Section 3: Development of Finite Element Meshes

Tensile Test Mesh

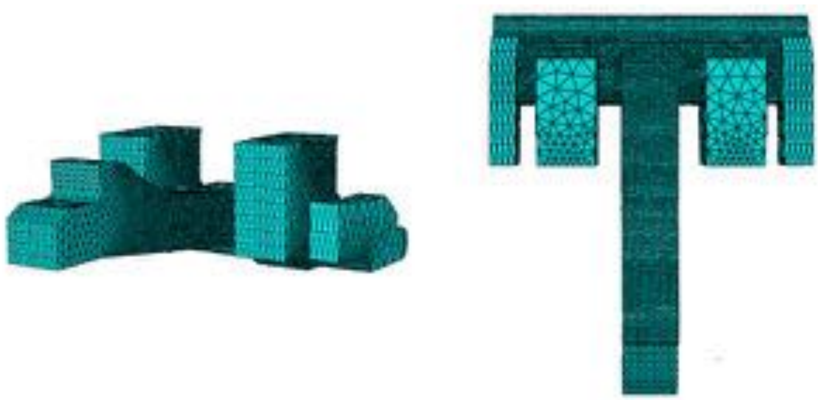
Figure 6: Gauge Length Mesh

Tetrahedral elements were used to comply with the FEA standards used at Delphi. Because the reduced gauge length is a very simple geometry, a very fine mesh was not needed, and the data converges with a coarse mesh.

GT Connector Mesh

Tetraheadral Elements were also used on the GT connector. Because the GT Connector has many faces and is a very small part, a very fine mesh is needed to get the data to converge on a value. To reduce computation time, the regions far away from the area of focus were assigned different edge seed sizes. The more critical regions feature a refined mesh. Section 6 includes the simulation results at different mesh sizes.

Figure 7: Mesh of GT Connector

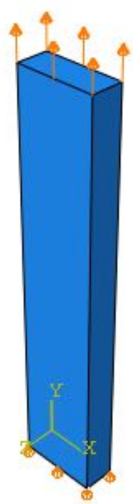


Section 4: Development and Description of the Model Assembly and Boundary Conditions

Tensile Test Boundary Conditions

The tensile test simulations only required two boundary conditions. The bottom end of the gauge length was fixed in the y direction only. This allows for the nodes to move in the x and z directions to allow for the the cross sectional area of the bar to change. Another boundary condition was required to pull on the bar. This was done using a displacement boundary condition in the y direction. The magnitude of this displacement was determined using the tensile test data. The final strain value was multiplied by the gauge length to determine the total elongation of the bar.

Figure 8: Boundary conditions on the 50 mm gauge length



GT Connector Simulation Boundary Conditions

The boundary conditions on the GT connector can be seen in Figure 6. The GT Connector required two boundary conditions. These boundary conditions can be seen in Figure 6. A zero rotational and translational boundary condition is applied to fix the body of the part in place. A displacement boundary condition is applied to simulate the displacement of the locking arm while another part is inserted into the connector. The values for these boundary conditions are determined by test data on the GT connector.

A test was done on the connector using an Instron machine where a wire was connected to the locking arm. The wire was pulled and the average displacement of the locking arm was recorded at fracture. This test setup is shown in Figure 7. The results of these tests are included in Table 2.

Figure 9: The boundary conditions as applied to the GT Connector

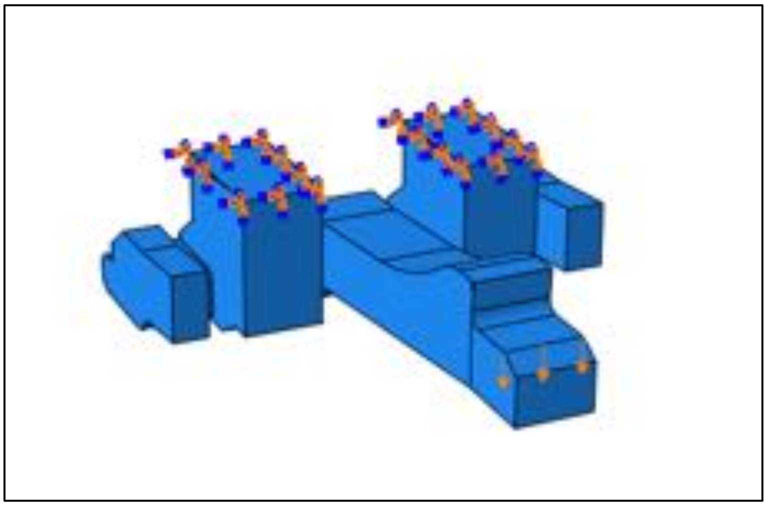


Figure 10: Test to determine the displacement of the locking arms at fracture

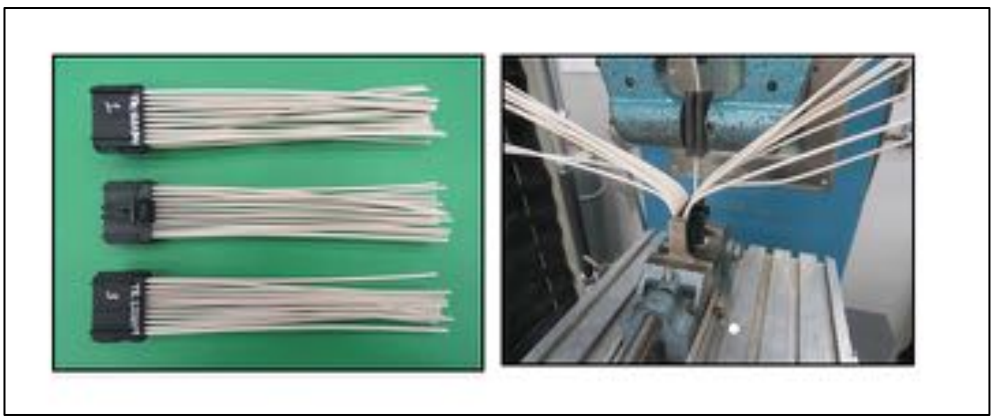


Table 2: Average displacement of locking arms at fracture

Material	Average Deflection of 5 Terminal Locks (mm)
M4698	2.3
M4629	2.8
M4692	4.6

Section 5: Development and Description of Model Interactions

No model interactions were required because the parts did not interact with any other parts in the simulation.

Section 6: Analysis of Finite Element Model

Tensile Test Simulation

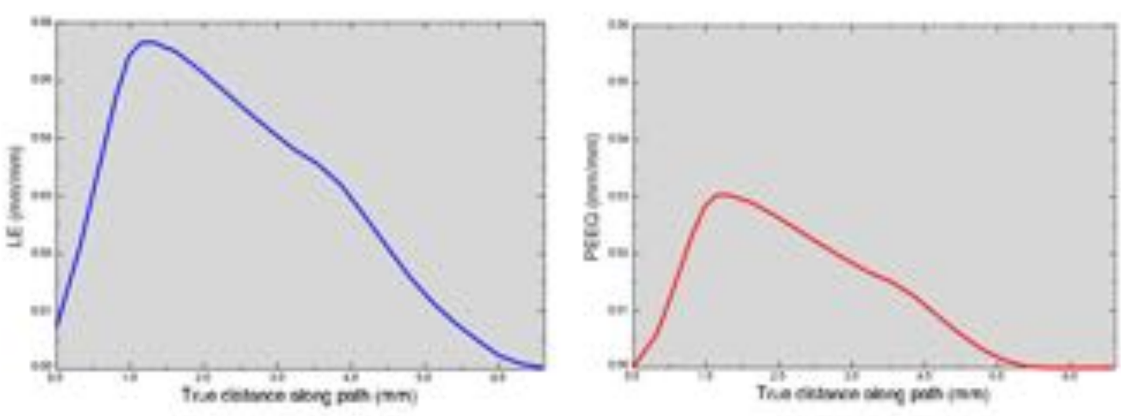
The tensile tests were run using a static general step. Due to the rate dependence of the model, the time increment of the steps needed to be correct in order to set the strain rate of the experiments. The time set for the step was same the same time where fracture occurred in the raw test data. Because of the coarse mesh, job only took 25 seconds to complete.

GT Connector

The GT connector also used a static general step. That was the only necessary step and both boundary conditions were applied in this step. Non-linear geometry was turned on for this step because of the complexity of the part and the plastic behavior of the material. The finest mesh used took 6036 seconds to complete. It is desired to simplify the mesh down in order to reduce the computation time.

<u>Plasticity</u>
The plastic strain along the connector flex arm was recorded for all three different materials. In the analysis, there was a considerable amount of plastic strain in all three materials. This plastic strain is expected because the boundary conditions that were applied to the parts replicated the displacement of the part at failure. To optimize the design of the connector flex arm, the displacement of the locking arm should produce zero plastic strain.

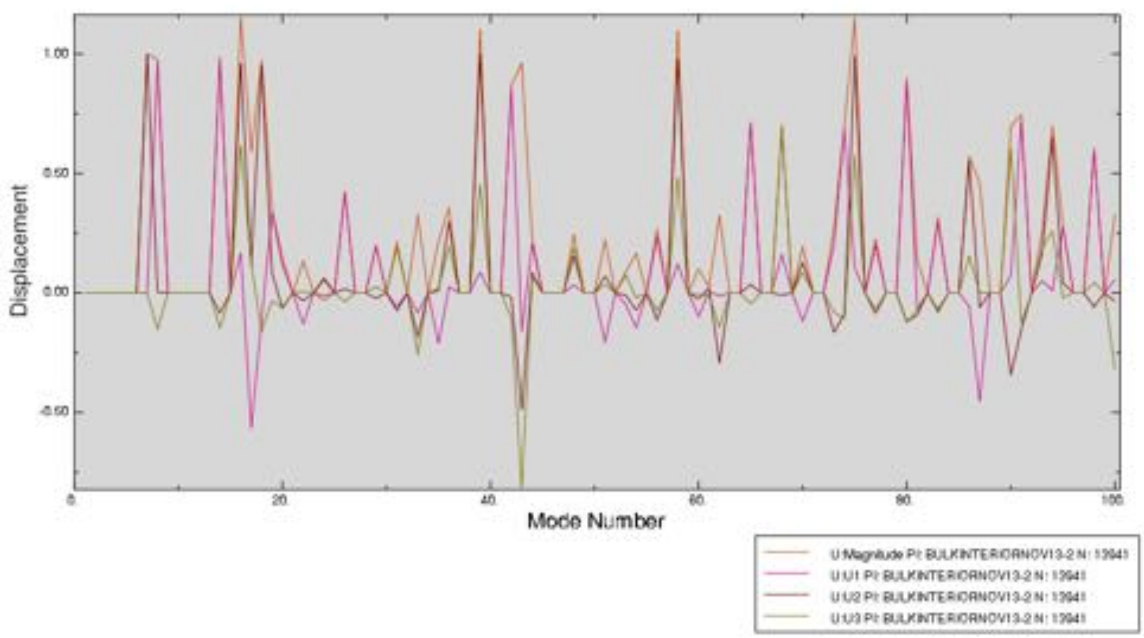
Figure 11: Total logarithmic strain and plastic strain in M4698



<u>Modal Analysis</u>

When running the modal analysis an element was selected at the tip of the arm for the connector to make sure the maximum results would be achieved. The analysis began by using all the eigenvalues, however this resulted in over 15,000 eigenvalues being generated for our model and this was too much to process resulting in an error. Reducing the number of eigenvalues allowed the job to run much faster and without any errors. The results generated show the displacement for each of the three directions as well as the magnitude of all the displacements summed up. The z-direction experienced the smallest max deflections, reaching a peak displacement of 1.00 mm fewer times than the other two directions. This is to be expected considering that this direction has the greatest thickness.

Figure 12: Mode shapes for M4698



The maximum deflection in any one direction is 1.00 mm which is relatively small magnitude. This shows us that the static load from inserting the wires in the connector is the greatest load this connector will experience and the vibrations it will experience are not expected to make it plastically deform or fracture.

Section 7: Summary of Major Findings

Tensile Test Results

Simulations were done for each material using the Johnson Cook Material models developed from the tensile test data. A simulation was done at all three strain rates. An output data field request was created to record the stress and strain data from an element in the reduced gauge length region. The comply with the standards used at Delphi; the strain value used is the logarithmic principle strain. The results of the simulation were close to the data collected in the physical experiments. At all three strain rates, the materials behaved the same in the linear region and behaved differently in the plastic region. Delphi was happy with the results of the simulation and requested that the team analyze the GT Connectors using the developed material models.

Figure 13: Comparison of simulation and tensile tests in the plastic region for M6734

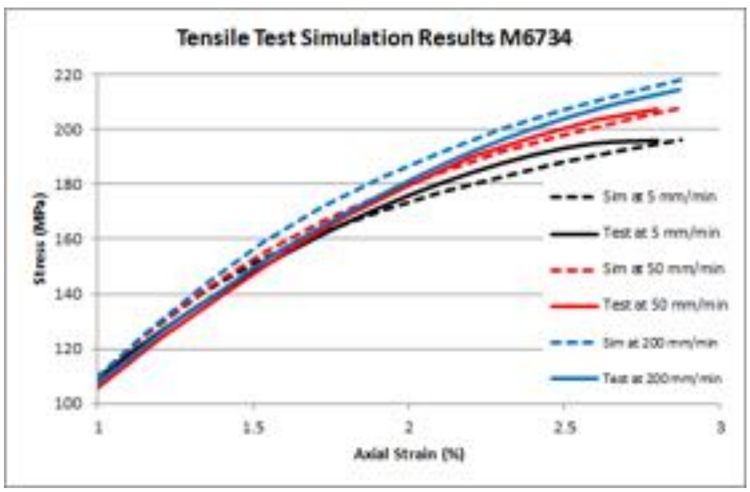


Figure 14: Comparison of simulation and tensile test results for M4698

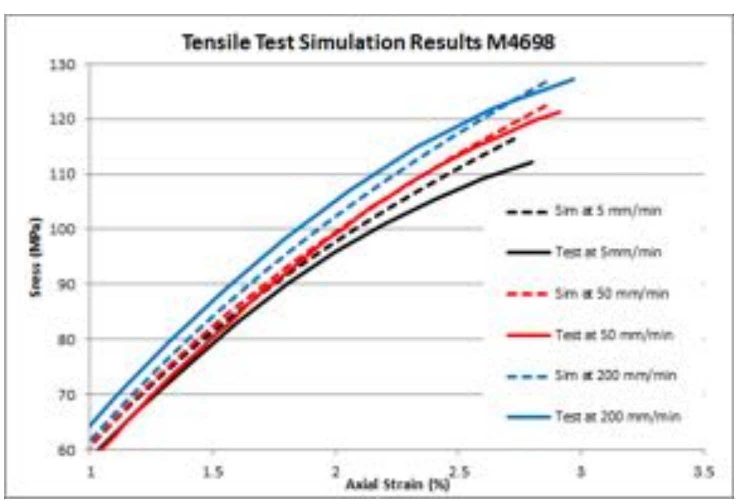
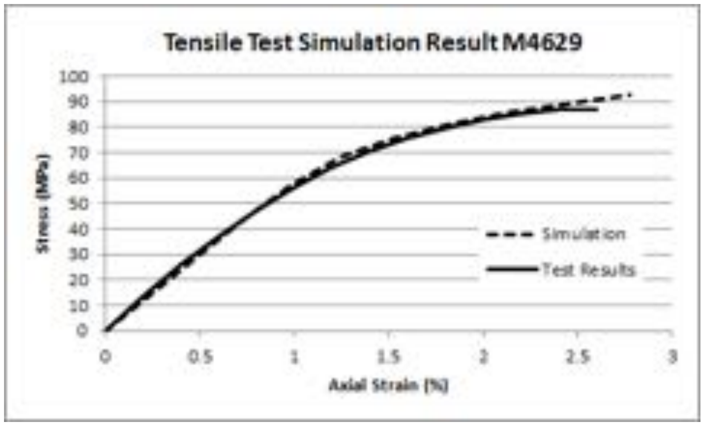


Figure 15: Comparison of simulation and tensile test results at 5 mm/min fo M4629



Simulation Results

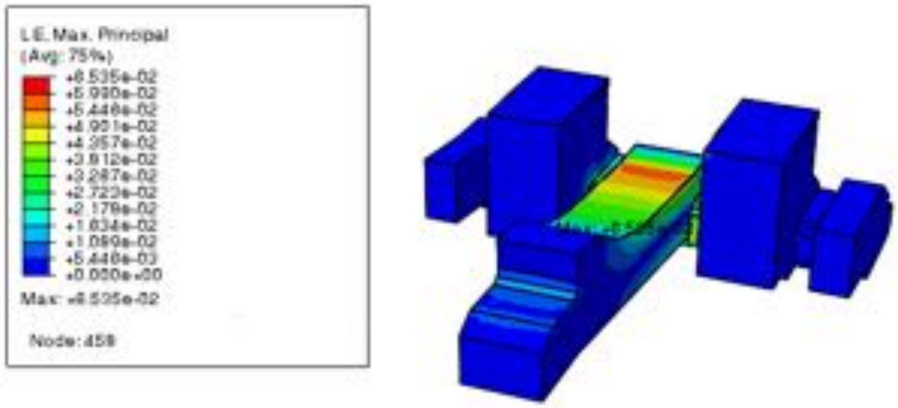
<u>M4698</u>
The M4698 arm was displaced 2.3 mm in the simulation, corresponding to the average deflection of the arm in physical GT connector tests. The time of this simulation was set to 0.69 seconds in order to use a displacement rate of 200 mm/min. A speed of 200 mm/min was chosen because the locking of the connector is a high speed process. The time period of the simulation was determined using the following method:

$$displacement\ rate * total\ displacement\ = simulation\ time$$

$$\frac{1\ min}{200\ mm} *\ 2.3mm * \frac{60s}{1min}\ = .69\ seconds$$

A maximum strain of 6.552% occurred in the part with a 2.3 mm deflection. This occurred at the sharp corner of the part where the lock arm connects with the body of the GT connector.

Figure 16: Strain contour for M4698 with global mesh seed size of 0.15



To test for convergence at different mesh sizes, the strain was extracted from node shown in Figure X. The values at different sizes were recorded in the the Mesh Convergence Table.

Figure 17: Node chosen to test for mesh convergence

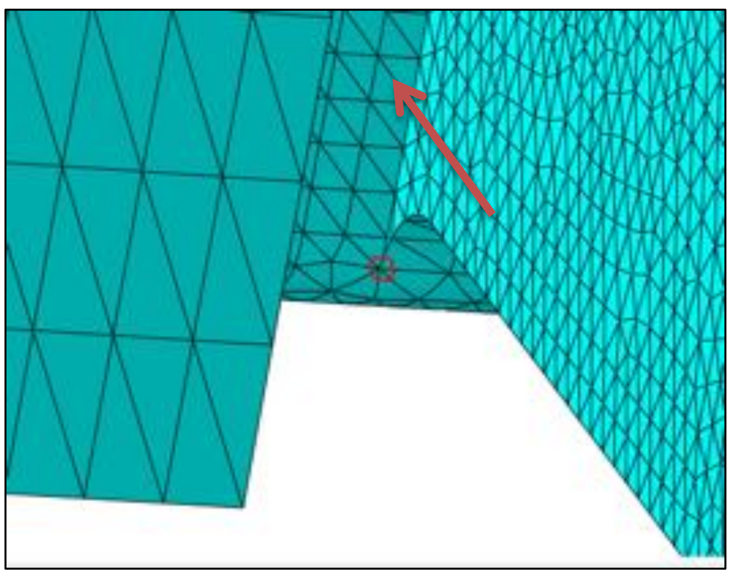


Table 3: Mesh Convergence Table

Global Mesh Size	Number of Elements	Logarithmic Strain at Corner Node (%)
0.3	34096	3.76
0.2	91182	3.83
0.15	199810	3.88

M6734

A maximum strain of 21.5% occurred in the part. For M6734, there is larger difference in the strain in the arm of the part compared to the strain in the corner where the arm connects to the body. A possible reason for this is because the the displacement boundary condition was set to 4.2 mm compared to the 2.3 mm displacement in M4698.

Figure 18: The strain contour for M6734 with a global mesh seed size of 0.08

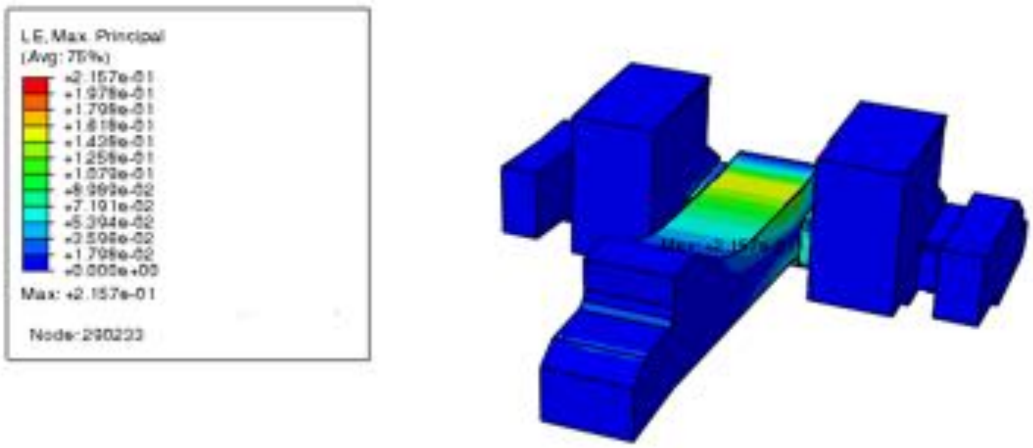


Table 3: Mesh Convergence Table for M6734

Global Seed Size	Number of Elements	Logarithmic Strain at Corner Node (%)
0.2	70850	6.64
0.12	201584	8.65
0.08	444810	8.88

M4629

The flex arm was displaced 2.8 mm, in accordance with the physical test conducted by delphi. This simulation may not be completely indicative of the true behavior of the GT connector due to model being calibrated only using data at 5 mm/min, which is too slow of a strain rate to model the locking of the GT connector. Using equation 2, this displacement rate implies that the time of the 2.8 mm displacement at a rate of 5 mm/min would occur over a time of 33.6 seconds.

Figure 19: The strain contour for M4629 with a globa mesh seed size of 0.10

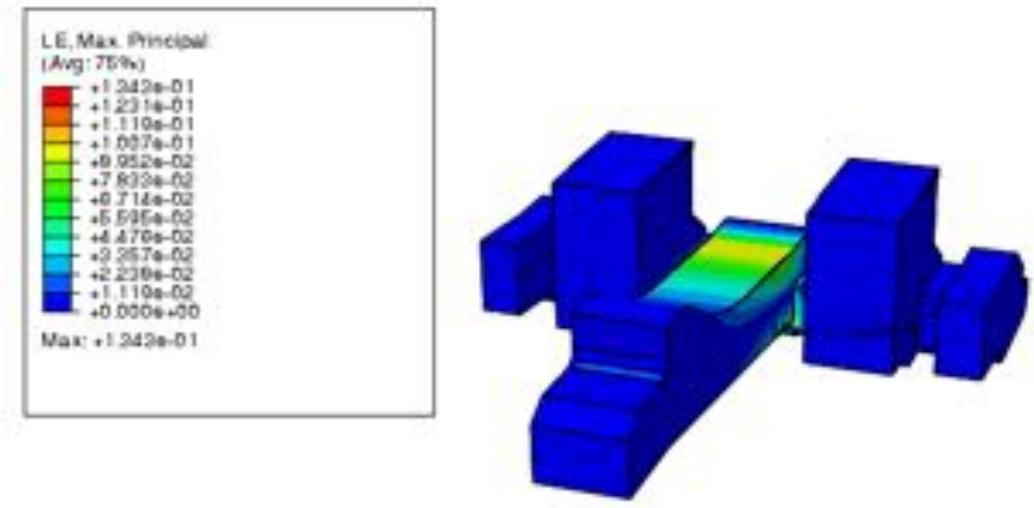


Table 4: Maximum Strain for M6734 based on mesh size

Global Mesh Seed Size	Number of Elements	Strain at Corner Node
0.2	57464	4.64
0.14	113900	4.76
0.10	229624	4.91

Section 8: Awknowledgements

The team thanks Delphi Automotive for giving us the opportunity to work on this project and learn about using finite element analysis and Abaqus. Delphi provided us with the CAD files in addition to help when we asked them questions.The team would also like to thank Dr. Reuben Kraft for providing help and support throughout the process.

Section 9: Works Cited

ABAQUS (2015) `ABAQUS 6.14 Analysis User's Manual'. Online Documentation Help: Dassault Systèmes.