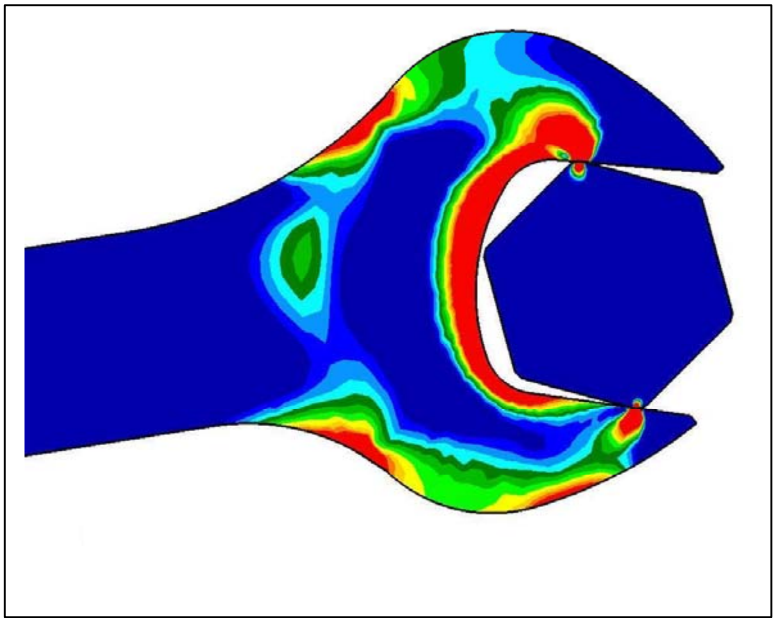
2004 Virtual Product Development Conference



Virtual DOE

Open-End Wrench Paper

Elasto-Plastic Simulation

Awad Gharib
Product Engineer/CAE
Danaher Tool Group
11011 McCormick Road
Hunt Valley, Maryland 21031
(410) 773-7892
awad.gharib@danaher.com

Tony Davenport
Sr. Application Engineer
MSC Software
1403 Saybrooke Court
Pasadena, Maryland 21122
(410) 255-9071
Tony.Davenport@mscsoftware.com

About the Authors

Mr. Awad Gharib has worked at Danaher Tool Group for seven years as a manufacturing engineer, design engineer, and currently as a full time analyst. At the present time Awad is a PhD student and a part time instructor at the University of Maryland Baltimore County (UMBC). The subject of his study is the plasticity of metal focusing in the stress flow of 4047 alloy steel as a function of temperature, strain, and strain rate. Awad has received BS in mechanical engineering science from the University of Hartford, CT.

Mr. Anthony Davenport has worked for MSC Software for three years as a Senior Application Engineer assisting customers in finite element analysis. Prior to his work at MSC Software, Mr. Davenport did computational analysis in the aerospace field for seven years at Northrop Grumman Corporation. He has presented at several conferences to include the MSC 1999 Aerospace Conference in Long Beach, CA, and at the NASA Goddard FEMCI conference. The majority of Mr. Davenport's work is in the field of dynamic analysis of satellite structures. He holds a Bachelor of Science in Mechanical Engineering at the University of Maryland, College Park and is a proud member of the Terrapin community.

Abstract

Correlation of test data to FEA simulation can be very useful to understand varied physical phenomena for a given design. However, lacking a clear goal, it can be hazardous and time consuming. A proposed method to overcome this predicament is to combine FEA simulation with Design of Experiment (DOE) analysis methods – Virtual DOE. One must first come up with a measurable output result, then set up DOE and simulate it using FEA. This method was found to be very time and cost effective in clarifying and analyzing an open-end wrench. Furthermore, it allows flexibility for future investigation and minimizes the time to achieve better results. Virtual DOE simulation may not completely eliminate test; however, it will minimize it. FEA helped Danaher Tool Group to achieve 50% reduction on testing cost and lead to a new innovative product.

The purpose of this paper, *Virtual DOE*, is to not only demonstrate the power of Virtual DOE versus Physical DOE, but to also explain the process that was used to understand Virtual DOE at Danaher Tool Group and its relationship to test and design improvements.

Table of Contents

Introduction

Background & Objective

Danaher Tool Group is a leading design and manufacturing company for hand tools. Danaher is the supplier of tools for Sears Craftsman®, which stands for made in the USA/Lifetime Warranty. Craftsman's tool specification require Danaher to meet 150% of the ASME loading standard, while matching the ASME standard for geometric and failure criteria. The failure criterion is described as loading and unloading an open-ended wrench and not exceeding a permanent deformation of 0.002" (2 mils) above the maximum allowable for the open-end id. A Design of Experiment[1] (DOE) study (both physical and virtual) was conducted to achieve that goal. In addition two more objectives were set:

1. Improve current product performance to better serve Danaher Tool Group customers, which will lead to increased profits for Danaher Tool Group.
2. Attempting to establish correlation between of the two approaches (physical and virtual DOE), in order to save money and time via a paradigm shift within Danaher; relying more on virtual DOE (MSC.AFEA) and less on physical DOE.

Method/Approach

A preliminary study was conducted to access the performance for the open-end wrenches. It was found that the 3/4th inch size had the worst performance, and therefore it was targeted for study and initial improvement. Design Of Experiment (DOE) is used to identify the means of improving the torque strength of 3/4th inch open-end wrench. Two types of DOE were conducted.

- Physical DOE using prototype specimens.
- Virtual DOE using Finite Element Analysis

A total of 8 factors were monitored in both DOEs. Thirty-two physical runs were conducted for each DOE variation. Virtual DOE was conducted in parallel to validate the use of Finite Element Analysis methods to reduce development time. Figure 1 shows the eight factors and the interaction that were studied indicated by "I" at the intersection between the column and the row.

[1] This paper is not meant as a discussion of Design of Experiment methods, but as a method to show that DOE methods can be used for both physical and virtual testing.

ID	Factors	Forge Coining H	Test Direction G	Throat Configuration F	Head Profile E	Hardness D	Thickness* C	Opening ID* B	Broach Depth* A	# Levels
A	Broach Depth*				I		I	I	M	2
B	Opening ID*						I	M		2
C	Thickness*			I			M			2
D	Hardness			I		M				3
E	Head Profile			I	M					2
F	Throat Configuration			M						3
G	Test Direction		M							2
H	Forge Coining	M								2

Figure 1: Design of Experiment Factor ID Table – Used to Interaction between the DOE Variables

Figure 2 is a liner graph is used to assist in sizing the experiment and to graphically represent the above table, and to show how many columns will be empty in a chosen experiment.

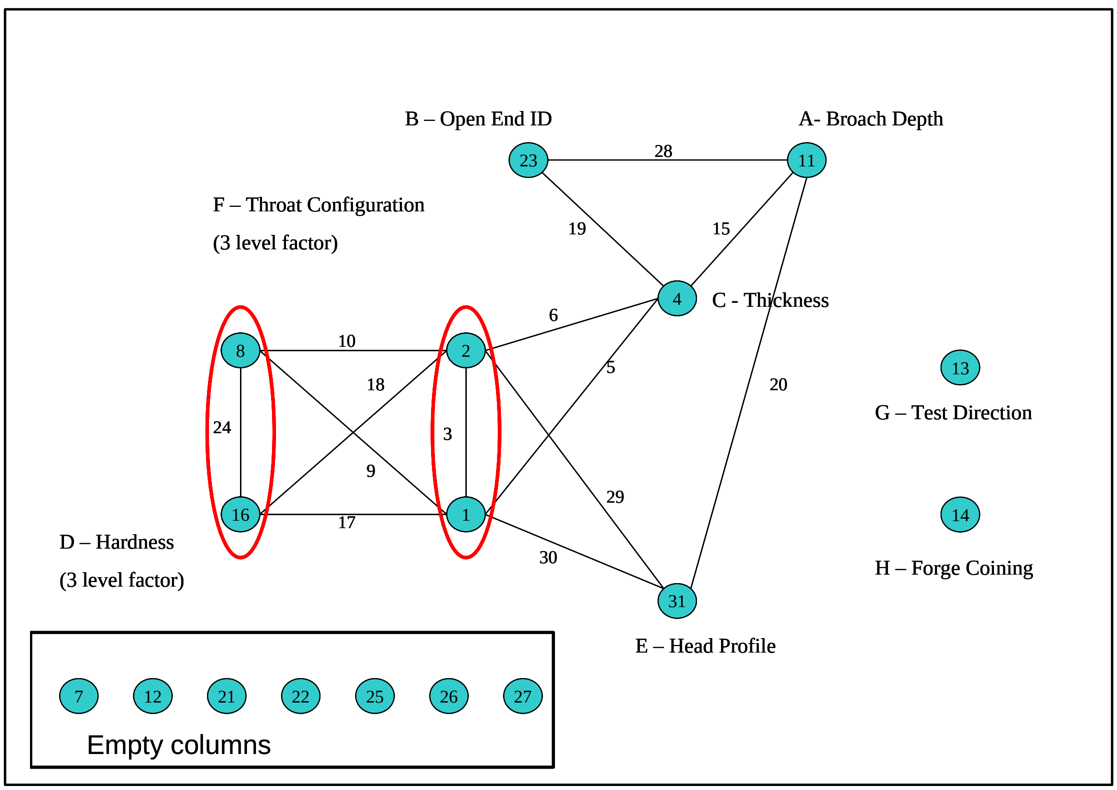


Figure 2: Design of Experiment Liner Graph - Used to describe the relationships between variables.

Factor setting levels and 32 run combination are shown in the following tables (see Table 1 and Table 2).

Table 1: Physical and Virtual DOE Variable Summary

ID	Factors	Levels	Low	Mid	High
A	Broach Depth	2	-0.020"	-	+0.020"
B	Opening ID	2	0.752"	-	0.763"
C	Thickness	2	0.343"	-	0.377"
D	Hardness	3	41	45	49
E	Head Profile	2	SD	-	K
F	Throat Configuration	3	Mat	Mod SD	K
G	Test Direction	2	Forward	-	Reverse
H	Coin	2	No Coin	-	Coin

Table 2: Physical and Virtual DOE Variable Definition Per Specimen

ID	D	C	G	A	B	E	F
1	HRc 41	0.343"	Forward	-0.020"	0.752"	SD	K
2	HRc 45	0.343"	Forward	-0.020"	0.763"	K	K
3	**HRc 49**	**0.343"**	**Reverse**	**+0.020"**	**0.752"**	**K**	**K**
4	HRc 45	0.343"	Reverse	+0.020"	0.763"	SD	K
5	HRc 41	0.377"	Reverse	-0.020"	0.763"	K	K
6	**HRc 45**	**0.377"**	**Reverse**	**-0.020"**	**0.752"**	**SD**	**K**
7	HRc 49	0.377"	Forward	+0.020"	0.763"	SD	K
8	HRc 45	0.377"	Forward	+0.020"	0.752"	K	K
9	**HRc 41**	**0.343"**	**Forward**	**+0.020"**	**0.763"**	**K**	**Mod SD**
10	HRc 45	0.343"	Forward	+0.020"	0.752"	SD	Mod SD
11	HRc 49	0.343"	Reverse	-0.020"	0.763"	SD	Mod SD
12	**HRc 45**	**0.343"**	**Reverse**	**-0.020"**	**0.752"**	**K**	**Mod SD**
13	HRc 41	0.377"	Reverse	+0.020"	0.752"	SD	Mod SD
14	HRc 45	0.377"	Reverse	+0.020"	0.763"	K	Mod SD
15	**HRc 49**	**0.377"**	**Forward**	**-0.020"**	**0.752"**	**K**	**Mod SD**
16	HRc 45	0.377"	Forward	-0.020"	0.763"	SD	Mod SD
17	HRc 41	0.343"	Reverse	+0.020"	0.763"	K	Mat
18	**HRc 45**	**0.343"**	**Reverse**	**+0.020"**	**0.752"**	**SD**	**Mat**
19	HRc 49	0.343"	Forward	-0.020"	0.763"	SD	Mat
20	HRc 45	0.343"	Forward	-0.020"	0.752"	K	Mat
21	**HRc 41**	**0.377"**	**Forward**	**+0.020"**	**0.752"**	**SD**	**Mat**
22	HRc 45	0.377"	Forward	+0.020"	0.763"	K	Mat
23	HRc 49	0.377"	Reverse	-0.020"	0.752"	K	Mat
24	**HRc 45**	**0.377"**	**Reverse**	**-0.020"**	**0.763"**	**SD**	**Mat**
25	HRc 41	0.343"	Reverse	-0.020"	0.752"	SD	Mod SD
26	HRc 45	0.343"	Reverse	-0.020"	0.763"	K	Mod SD
27	**HRc 49**	**0.343"**	**Forward**	**+0.020"**	**0.752"**	**K**	**Mod SD**
28	HRc 45	0.343"	Forward	+0.020"	0.763"	SD	Mod SD
29	HRc 41	0.377"	Forward	-0.020"	0.763"	K	Mod SD
30	**HRc 45**	**0.377"**	**Forward**	**-0.020"**	**0.752"**	**SD**	**Mod SD**
31	HRc 49	0.377"	Reverse	+0.020"	0.763"	SD	Mod SD
32	HRc 45	0.377"	Reverse	+0.020"	0.752"	K	Mod SD

Experimental/ Physical DOE

Physical DOE is conducting actual experiments with specimens and monitoring the changes in the number of concerned factors. Physical DOE is done to validate the design changes originally brought about by engineering/mathematics principles. Physical DOE is often accurate and if done properly can reduce the amount of testing required versus testing all combinations. However, even with its advantages, physical DOE is still very time consuming. Even with the reduction of test specimens, many days and weeks still may be spent manufacturing specimens, creating test fixtures, and performing the tests. Additionally, manpower costs can skyrocket, as tests need to be setup, and monitored. Tests lasting several weeks may run costs into the thousands of dollars. Finally, in most cases, physical tests may only tell the end-user of the data if the part has passed or failed. True, detailed stress may not be known. So even if a part does not fail, the end-user will never know how close the part came to failure, which makes engineering judgment for new designs very difficult.

On the other hand, an effective use of virtual DOE via finite element analysis (FEA) in combination with a reduced set of physical DOE can substantially reduce the time and money associated with improving designs. By combining Virtual DOE with Physical DOE Danaher Tool Group found that it could reduce both time and money associated with new designs.

Factors

Factors affecting the strength of the open end were related to the geometry, material and manufacturing process. As mentioned previously, a total of eight parameters were identified. The geometric factors are shown in Figure 3.

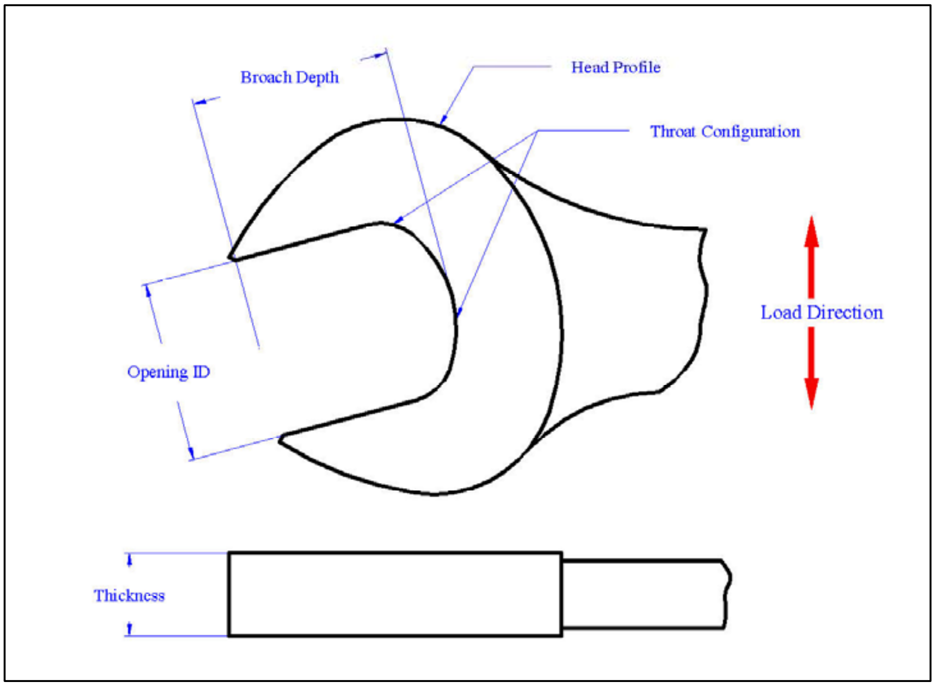


Figure 3: Description of Geometric Variables in Wrench

Hardness (HRc)

There was lots of variability observed in the hardness readings obtained in production. To achieve minimum hardness variation all the specimens were heat-treated by a third party contractor. In order to reduce the variations due to materials, tensile coupons were used to develop the material property models for their use in FEA. These were picked from the same lot as the wrenches in order to keep the variation to a minimum. Specimens were also heat-treated with the wrench blanks. Three levels of hardness were selected for this study 41, 45, and 49 HRc.

Open End ID

Open End ID is the distance between the two jaws as seen in the Figure 3. Production uses the nominal dimension as the lower specification limit. In experiments the Open End ID were set at the extremes of the ASME specification (0.752") as minimum.

Head Profile and Throat Configuration

Figure 4 shows various profile used for study. We discovered that there is an apparent interference between the mandrel/fastener and the Mat Throat configuration that prevents full insertion to the base of the throat. The interference has introduced some noise to the experiment and a possible source of error.

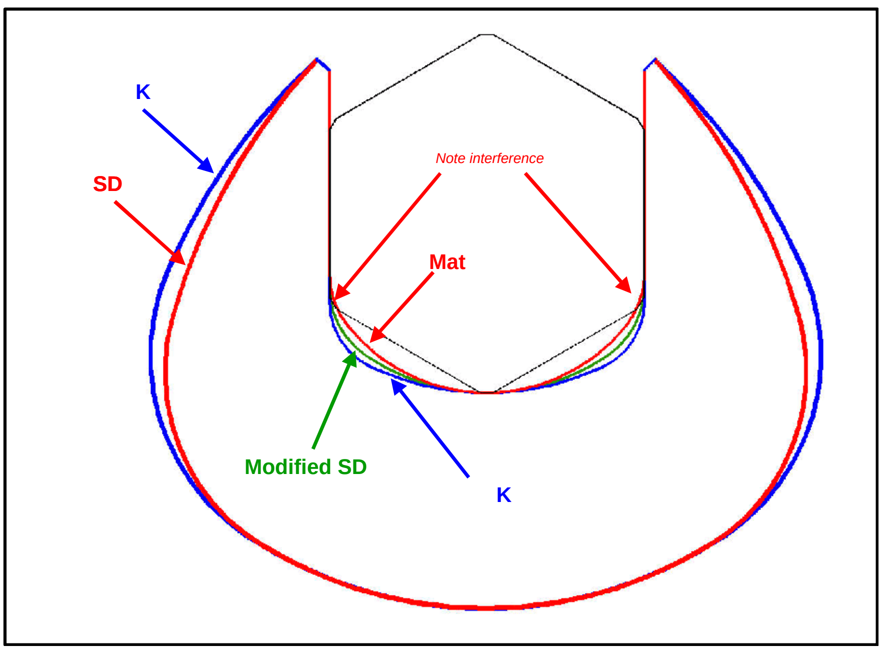


Figure 4: Head and Throat Configurations for the Wrench

Test Direction /Load direction

Load Direction is the direction in which torque is applied to the bolt head. Previous experiments have shown that the open-end is much weaker in reverse loading. This is due to the additional support the beam provides to the head as torque is applied in the "forward" direction. The difference in strengths of the open-end in forward and reverse loading is due to the 15°offset. As seen in Figure 5, the jaw of the open-end acts as a longer cantilever in reverse loading. This creates more bending moment at the section. Also, the thickness of the open end is same all around. The thickness makes the upper portion of open-end weaker than the lower portion.

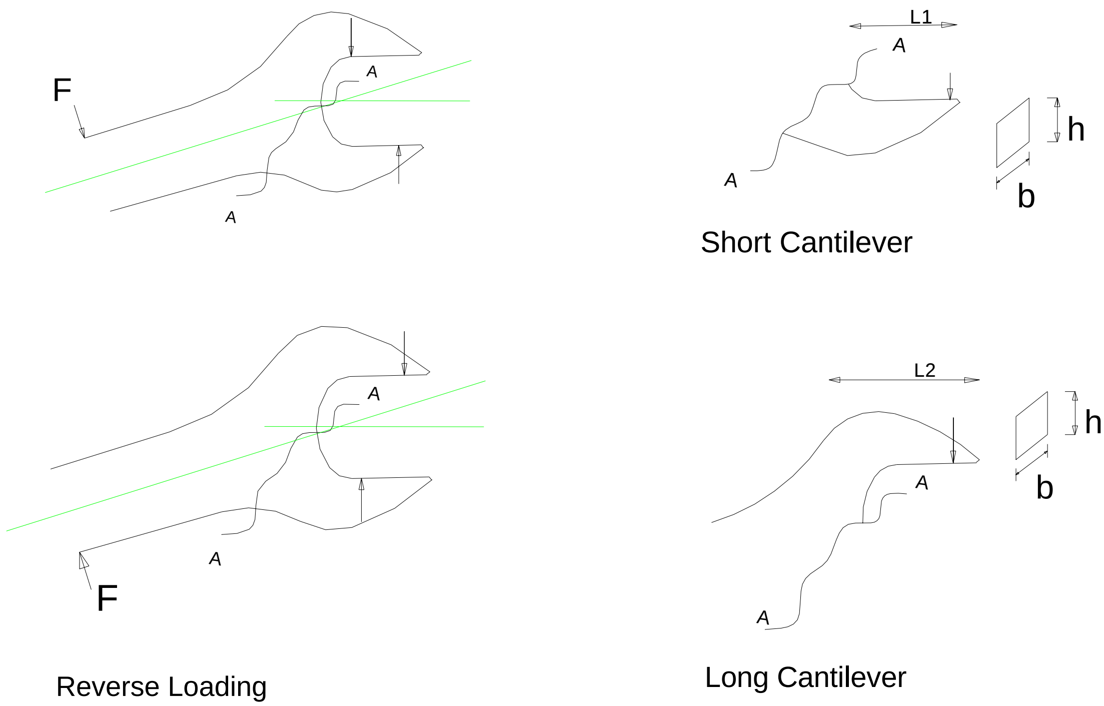


Figure 5: Loading Description of Various Sections of the Wrench

Wrench Specimen Preparations

The focus of this experiment was to identify the significant and important design parameters associated with open-end torque strength as measured by permanent tip spread observed after loading to 150% ASME. Consequently, we attempted to produce test pieces with minimum dimensional variation and did not attempt to account for manufacturing variability associated with the included factors.

Specimen were prepared by going through the following steps:

1. **Forging:** Forging is manufacturing process where metal is first preheated to a desired temperature 2200°F and then pressed to a desired shape.

2. **Annealing:** A process where steel is heated to high temperatures and held at this temperature for a sufficient period of time to allow the steel to become softer and more ductile. The steel is then slowly cooled to room temperature.

3. **Coining:** A method of squeezing metal while it is confined in a closed set of dies to get the required shapes. Very fine features can be obtained. Stress residuals are introduced to the parts due to the coining process.

4. **Milling:** Milling is a cutting operation to finalize the shape of the specimen.

5. **Heat Treatment:** Heat treatment is controlled heating and cooling of materials to obtain the desired strength properties. Our specimens are heat treated to obtain three levels of hardness 41,45, and 49 HRc.

6. **Milling to actual size:** Once the desired hardness is obtained, the specimen is milled to obtain the exact shape within the desired tolerances.

7. **Nickel Chrome Plating:** Nickel chrome plating is done to provide corrosion resistance. The chrome plating is exceptionally thin, measured in millionths of an inch. It gives high surface finish to the specimen.

Testing

The Actual wrench tests took place at the production facility to replicate the same tests that actual open-end products receive. The machine that was used is made in-house. The testing procedure was as follows:

1. Load specimen into test machine
2. Start loading slowly
3. Hold max load for 10 seconds
4. Unload
5. Measure and record open ID distance.

Holding the load for ten seconds may introduce a creep effect to the wrench, which was not simulated in the FEA analysis. This may contribute to sources of error between test and analysis data.

Open ID measurements from the physical DOE are shown in Figure 6 below.

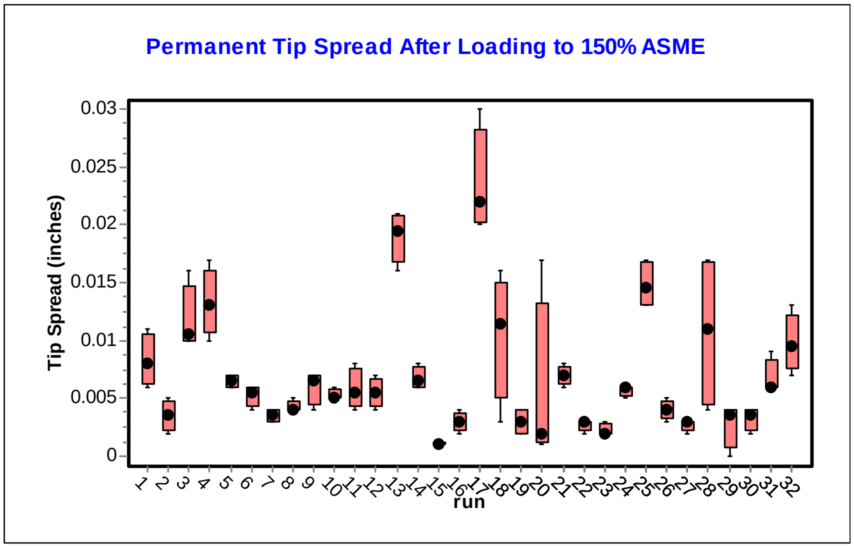


Figure 6: Permanent Deformation of Open-ID from Physical DOE

Analysis of data

The data was analyzed using SPSS software. Two ANOVA pie charts were developed to assist in analyzing the data. Figure 7 represents the relative importance of mean, while Figure 8 shows the relative importance of failure criteria.

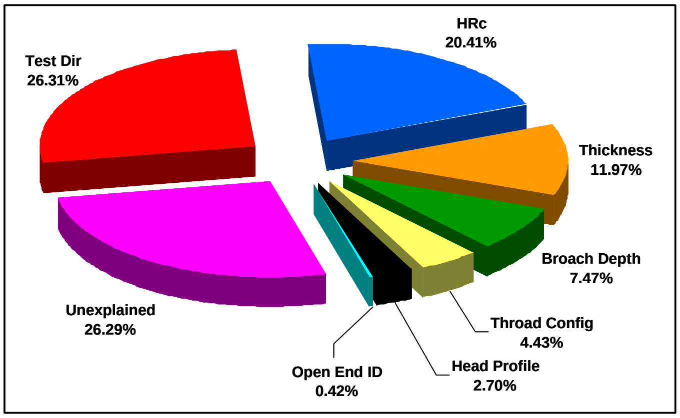


Figure 7: Relative Importance of Mean from Physical DOE

The following discussion describes the relative importance of various factors as found from the physical DOE data.

- Test Direction is considered a strong factor explaining 26% of the variation in the model. We believe this effect is associated with the additional support the beam provides to the head as torque is applied in the "forward" direction due to the 15° head offset.
- HRc and Thickness are considered moderate effects explaining 20% and 12% of the variation respectively. Both factors interact with Throat Configuration. The interaction between Thickness and Throat Configuration was unexpected in that the anticipated thickness effect was not present for the Modified SD profile.
- Broach Depth is considered a minor factor explaining 7.5% of variation. Broach Depth interacted with Head Profile. While shallower Broach Depths were stronger for both Head Profiles, the Kingsley (K) head profile had less deformation than did the SD profile at that setting.
- Throat Configuration, Head Profile and Open End ID are considered weak factors (explaining less than 5%).
- Throat Configuration was involved in three interactions probably due to the interference observed with the Mat profile that prevented positioning of the mandrel to the bottom of the throat.

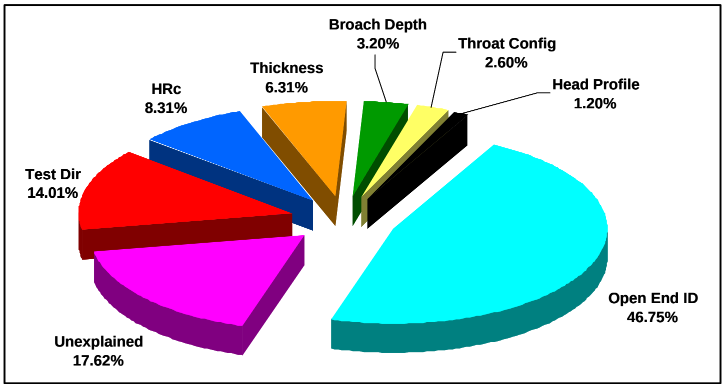


Figure 8: Importance of Failure Criteria for Physical DOE

The following discussion explains the relative importance of various failure criteria within the physical DOE data.

- This model looks at the absolute post load Open End ID (rather than the amount of deformation), which relates directly to the ASME failure criteria.
- Open End ID is considered a strong factor explaining 47% of the variation in the model. We believe this effect is associated with the additional support the beam provides to the head as torque is applied in the "forward" direction.
- Test Direction is considered a moderate effect explaining 14% of the variation.
- HRc and Thickness are considered minor factors explaining 8%and 6% respectively.
- The differences in relative importance between the two analyses discussed thus far illustrate the impact different test criteria can have on the corrective actions.
- In this case the previous analysis should be used to minimize the amount of deformation generated, while this analysis points to an additional opportunity to reduce failure as defined by ASME.

Given that these test specimens were fabricated by milling, coupled with the observation that coining had an apparent impact on residual stresses when we attempted to EDM the specimens, coining and broaching practices should be part of a Open End ID targeting and variation reduction effort if undertaken.

Physical DOE Variation - Results and Discussion

The curt discussion highlights important factors that were found in the physical DOE data.

- Overall, the factors studied explained all but ~25% of the run variability.
- Thickness is considered a moderate effect explaining 14% of the variation.
- Test Direction is considered a minor factor explaining 8%.
- As an interactive effect with Thickness, Throat Configuration is a weak factor explaining 4%.
- In general, the preferred settings for the significant factors coincide with the preferred settings for lower permanent deformation.

Source of Errors

The physical DOE is governed by the following sources of errors. It is possible that there are sources there were not discussed below.

- Configuration with internal throat did not permit a full insertion of the wrench onto the mandrel; therefore, some noise is introduced into the experiment.
- Although it is likely small, the coining process may have produced an error in the experiment by depositing a residual stress to the wrench.
- Heat-treat accuracy was within 2 points of the reported values 41, 45, 49 HRc ±1 HRc.
- The fraction coefficient between wrench and mandrel was not a fixed value for all runs. There were some small variations.
- Holding the max load for 10 seconds may introduces some creep to the wrench, which was not simulated at FEA analysis.

Virtual DOE

Initial Attempt At FEA

Finite Element Analysis models were completed using MSC.AFEA 2003. This is a combination of MSC.Patran (Graphical User Interface) and MSC.Marc (Analysis Solver). MSC.Marc is considered a "work-horse" for non-linear contact analysis, with approximately 20+ years of proven technology; therefore, Danaher Tool Group felt it would be the right solver for the task.

Analysis models and runs were completed on a DELL C800 laptop with 512 MB of RAM and an 800 MHz Intel Pentium III processor.

In the first finite element analysis (FEA) simulation of the Design of Experiment, the mandrel (bolt head) was fixed at the center and a vertical load was applied at the end of the wrench's handle. Below are the results using this approach.

- Generally, we observed good agreement between the FEA analysis and the physical samples in terms significance and relative importance.
  - A plot of FEA predictions of permanent deformation of the open-id vs. observed average deformation of the open-id for each run is shown in Figure 9.

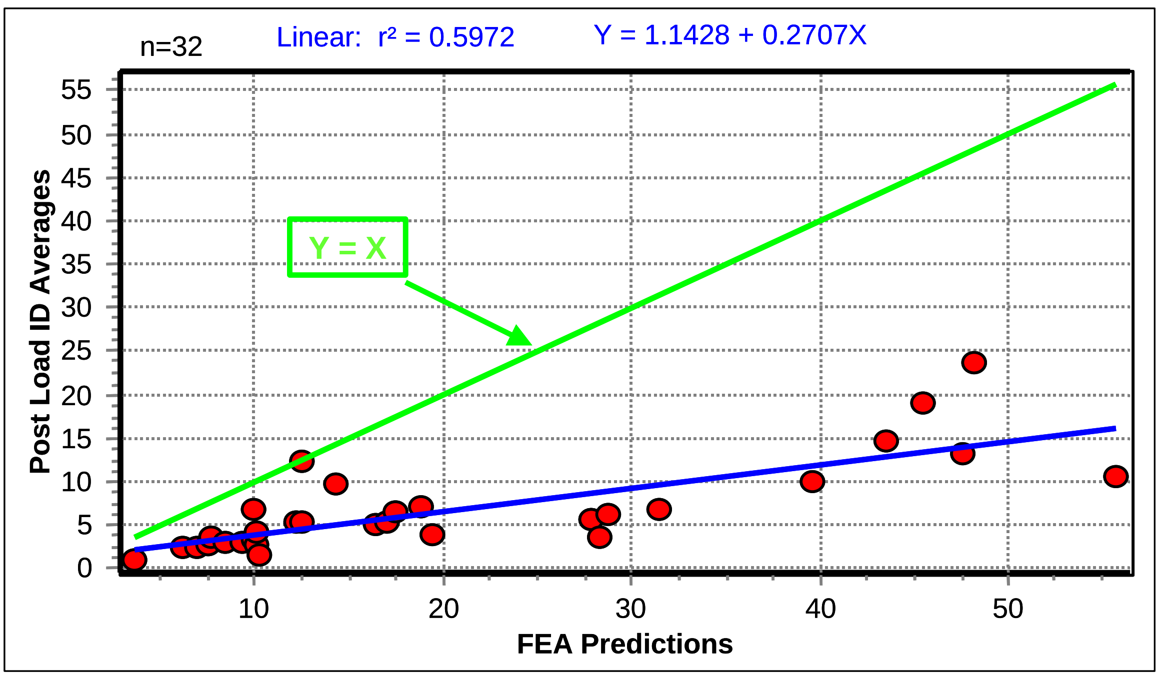


Figure 9: Initial Correlation Trend Lines - Physical vs. Virtural DOE

  - The correlation between the physical and virtual DOE is significant and explains 60% of the observed run-to-run variation.
  - Although the relationship appears linear, the FEA appears to over-estimate the deformation by a factor of three. This implies an overall error between the FEA and the physical test caused by one or more factors.
  - The rankings of significant factors by importance between the two analyses are highlighted in yellow in the following ANOVA table (Table 1).

Table 3: Rankings of Significant Factors

	Importance Rank	
	FEA	Physical DOE
Head Profile	4	6
Throat Config.	5	5
Hardness	1	2
Thickness	3	3
Test Direction	2	1

- HRc is considered a strong factor explaining 26% of the variation in the model.
  - In the FEA analysis HRc was not involved in the interaction unlike the Physical DOE analysis. This can be seen in Figure 10.

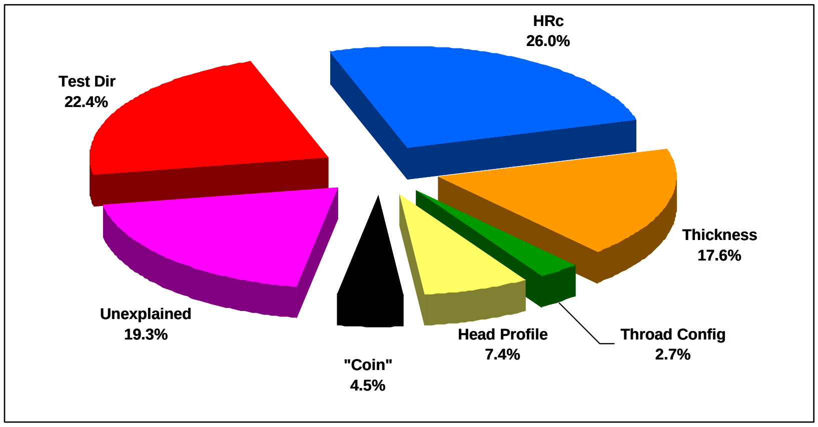


Figure 10: Contribution of Significant Factors for Initial Virtual DOE.

- Test Direction and Thickness are close behind and are considered moderate affects explaining 22.4% and 17.7% of the variation respectively.
  - Thickness interacts with Throat Configuration as discussed in the Physical DOE analysis.
  - Significantly the form of the interaction was virtually the same between the two analyses.
- Head Profile is considered a minor factor explaining 7.4%.
  - Broach Depth interacted with Head Profile. While shallower Broach Depths were stronger for both Head Profiles, the Kingsley head profile had less deformation than did the Springdale profile at that setting.
- Throat Configuration, is considered a weak factor explaining 2.6%.
- The factor "Coin" could not be modeled through FEA techniques. Never the less that "column" showed up as significant and suggests that there may be an interaction between Throat Configuration and Test Direction.

Note that coining showed up as a significant effect even though that factor could not be modeled in the FEA work. This suggests that there may be an interaction between Throat Configuration and Test Direction.

With all the source of error that we had the FEA, simulation results were satisfactory yet we needed to enhance the correlation and the predictability a little farther. Therefore, a more detailed FEA work was conducted to achieve that goal.

Detailed FEA Approach

The approach for creating the computation models was to use the same loading as used in the physical DOE method. This meant tracking 32 elasto-plastic analysis runs. Scripting methods using MSC.Patran session files and Unix batch commands were developed to ensure accuracy of model manipulation and consistency during the build, run and post-processing stages.

Analysis models were created in multiple steps. Continued use of products from MSC Software and Autodesk were used to create the geometry and finite element models. The steps and software packages involved were as follows:

1. Autodesk Autocad for creation of 2D geometry
2. MSC.Patran 2004r2 for pre-processing of analysis models. Pre-Processing included:
    a. Application of contact regions, boundary condition and loading
    b. Application of material properties
    c. Creation of Mesh
    d. Creation of Marc Run-Time parameters
    e. Exportation of Marc input deck
3. MSC.Marc 2003r2 for solving the multi-step finite element non-linear contact analysis problem.
4. MSC.Marc user subroutines, courtesy of Chris Tunney, MSC Software Application Engineer, for expediting pertinent result data.
5. MSC.Patran were used for graphical post-processing of results.

Boundary Conditions

Several methods for attacking the analytical model were explored. The most efficient, and most successful in terms of solution run times was to constrain the wrench at a single point (node 5002), and apply a torque to the bolt (node 5001). This method required the following additional constraints and contact regions (please reference Figure 11):

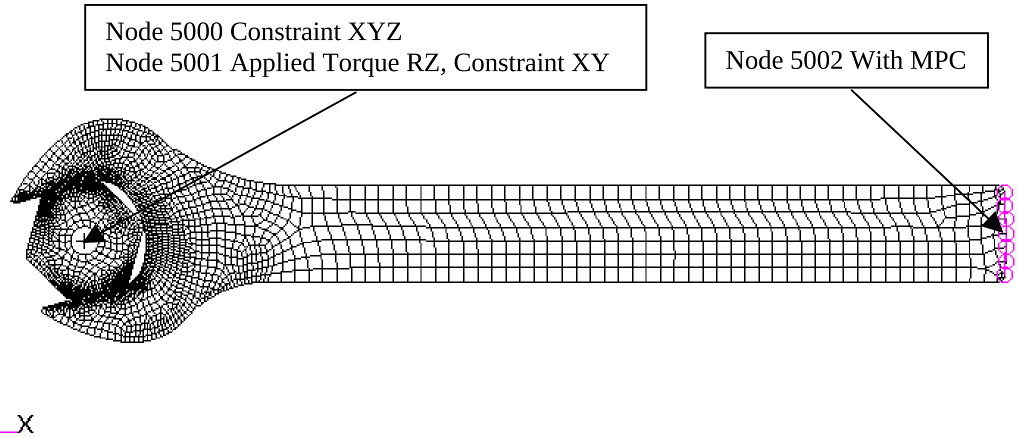


Figure 11: Refined Virtual DOE Model with Loading and Contrained Nodes Described

1. Node 5000 is constrained in translation for Degree of Freedom X and Y, and Rotation Z
2. Node 5000 is connected to 5001 via a rigid connection.
3. Node 5001 has a constraint in Degree of Freedom X and Y.
4. Node 5001 has a rigid body contact to the bolt inner circle.
5. Node 5001 has a torque of 2475 in-lb applied in the RZ direction.

Note: When a torque is applied to node 5001, it will torque the bolt through its contact connection to the inner surface of the bolt.

6. The Bolt Surface 1 and Wrench Surface 3 have a deformable body contact region between them. Friction is applied at the contact region between these two entities in both loading and unloading load steps.

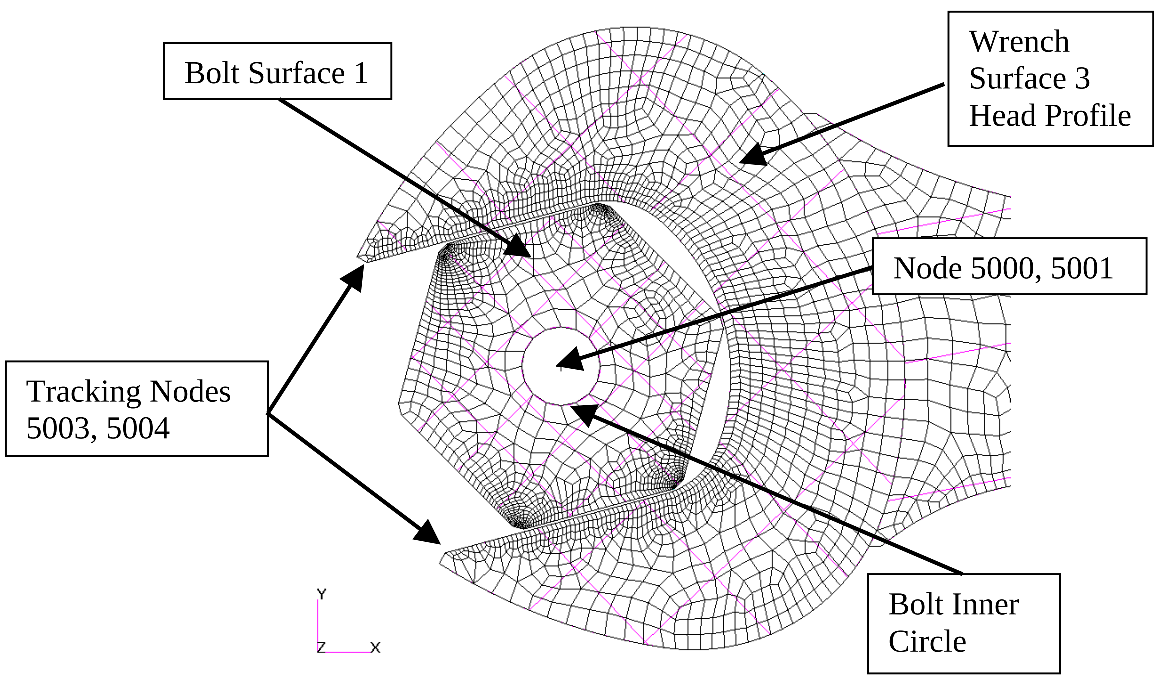


Figure 12: Detailed View of Virtual DOE Model

Material Properties

Material properties are determined by performing tension test on specimens taken from the same lot as the one from which the wrenches for physical DOE were created. The material specimens underwent the same process as the wrenches in order to maintain accuracy of material properties. Because of the attention to detail for creating the material properties, a complete description of the finite element analysis material model development for this analysis can be seen in Appendix A. Additionally, Appendix B describes the method to convert the material data from engineering stress/strain to true stress/strain, a requirement for MSC.Marc 2003r2. Highlights of the specimen material data collection and inputting into MSC.Patran are shown below. Figure 13: Material Property Definition from Test Specimens shows stress/strain plots for various materials used.

Material plasticity Using MSC.Patran

The following steps were used to properly enter the material true stress/strain data into Patran:

1. Create a Material Property field for the stress-strain data.
2. Create a Material with the elastic properties
   (Constitutive Mode: Elastic)
3. Using the same material as in STEP 2, add the plasticity properties
   Constitutive Mode: Plastic)

When complete, the material should have both the "Elastic' and "Plastic" options listed in the "Current Constitutive Models" option on the bottom of the "Input Properties' form.

The elastic region was described as follows:

Young's Modulus (E): 30 x 10^6 psi
Poisson's Ratio (υ): 0.29
Density (ρ): 0.278

The three materials were applied to the wrench in various runs.

The material property for the bolt is the same as the wrench except that it only covered the elastic region. This means that the bolt will not plastically deform and is three times as rigid as the wrench based on geometry (bolt thickness is 3x as great as the wrench). As loading was applied to the bolt via the torque and contact with the wrench, the material traverses the stress/strain curve as a slope of the modulus of elasticity. Because no visible permanent deformation occurred in the bolt during test, it is believed that this was an accurate analytical representation of the bolt.

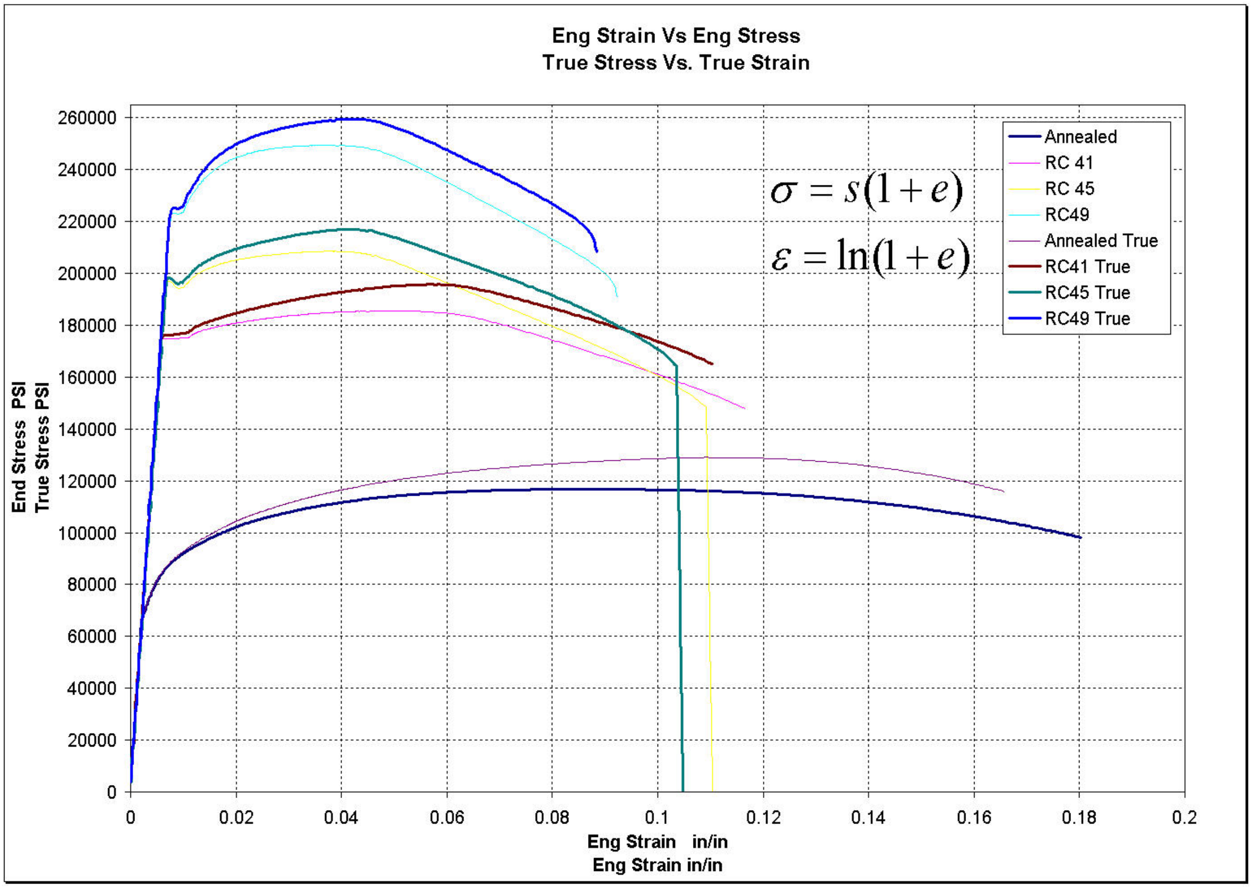


Figure 13: Material Property Definition from Test Specimens

Load Steps

For the analysis jobs, two load steps were used (Loading and Unloading).

1. ***Load Step One*** (Loading): Applied the 2475 in-lb torque at the bolt center. Contact was maintained between the bolt and the wrench.
2. ***Load Step Two*** (Unloading): Removed the torque and returned the bolt to its initial position via a rotational displacement in the RZ direction applied at node 5001. Additionally, in order to ensure that lock-up did not occur due to permanent deformation of the wrench, the contact region between the bolt and wrench was gradually reduced to zero contact by the final step. Gradual reduction of the contact was used add stability to the model by reducing the chance of rigid body movement within the model as contact separation occurred.

Additionally, a MSC.Marc user subroutine developed by Chris Tunney, MSC Application Engineer, was included in each run to track the job step, the distance between open-id (node 5003 and 5004), the torque applied to the bolt (node 5001), and the reaction load at the end of the wrench (node 5002). Details of the user subroutine can be seen in Appendix C.

Running the Analysis

Analysis runs were completed in blocks of 32 based on friction coefficients. The runs were completed on a Dell M60 (2GB of RAM, 1.7 MHz Centrino-M Intel Processor -- 2MB L2 Cache, Microsoft Windows 2000 Operating System, SP4). A DOS batch script was created to run all models. Wall clock time for models without friction completed in approximately 6 minutes. Wall clock time for models with friction completed in approximately 12 minutes. This includes time to compile and link the user subroutine used to gather result data.

Source of Errors - Analysis

Great care was taken to reduce the sources of error for Analysis. However, the following is a list of possible error sources:

1. Friction is probably the largest source of error for the analysis model. This is because it is difficult to track in test. Additionally, friction may vary from one run to another in test, which makes it very difficult to correlate with analysis where exact numbers are used. Therefore, all models were run with three different fiction coefficients (μ = 0.0, 0.1, 0.4). Initial studies were performed to narrow down the friction to the above. Studies varied friction of model configuration 10 from 0.0 (no friction) to 0.4. As expected, friction varied permanent deformation at the opening id.

   $\mu = 0.0, \Delta = 0.0009$ in

   $\mu = 0.1, \Delta = 0.0064$ in

   $\mu = 0.4, \Delta = 0.0191$ in

   To shed light on the variation in friction, consider an extreme case where the coefficient of friction of $\mu = 1.0$ would represent a glued connection between the bolt and wrench. Since we know this is not occurring, we examined lower values of friction.

   By comparing against test (Average Deformation at Opening ID = 0.005 in), it can be seen that for configuration 10, $\mu = 0.1$ is the closest friction value observed.

2. Mesh density of contact regions is another source of analysis error. Due to model times and a short study was performed to reduce the element size to the most appropriate for the contact region. Mesh density was increased in the bolt corners were contact forces were expected to be the highest (see Figure 14).

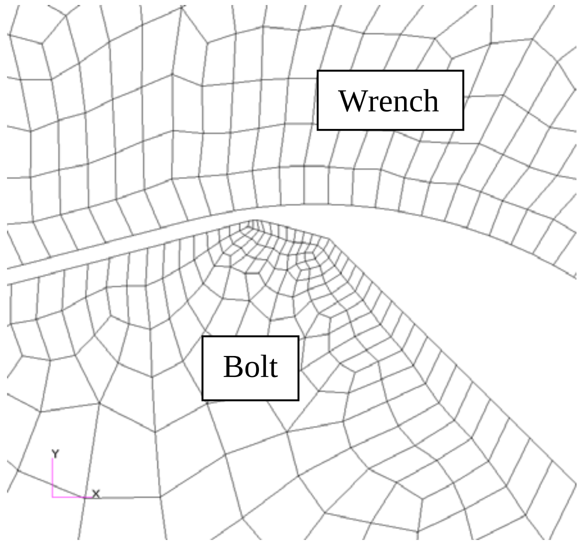


Figure 14: Mesh Density of Contact Region for the Bolt and Wrench

3. Other sources of error for the model can be related to run time variables and convergence criteria. Adaptive time stepping was used for both loads steps. Additionally, non-positive definite constraints were used to help stabilize the wrench as the contact region with the bolt was released. Generally, it is felt that run time variables were controlled and these offered little source of error.

Discussion

In order to satisfy the two objectives of this paper, we must first look at the second objective: correlation of FEA data to experimental data. By correlating FEA data to test data, Danaher Tool Group will be able to reduce the amount of physical tests required to certify a wrench in lieu of virtual (analytical tests).

The first check in the analysis is to visually inspect the rotation of the bolt to ensure that contact is occurring during Load Step 1 in the proper direction and that contact is being released during Load Step 2. An example of both of these objectives can be seen in Figure 15, which shows various steps from configuration nine[2]. Please note that increments 0 – 44 are related to loading the wrench (Load Step 1) via a torque applied the center of the bolt. Increments 45 – 104 are related to a rotational displacement (Load Step 2) to unload the wrench. By looking closely at increment 88, it can be seen (see Figure 16) that plastic deformation has occurred as the bolt has been removed from the wrench. This can also be seen in Figure 17 where the von mises stress (261 KSI) at step 44 is above the plastic region of the stress/strain curve for Steel 4047RC41 (200 KSI).

Now that we have proven that the FEA model is behaving as expected, the next step is to correlate the model to test. This is accomplished by running the 32 models through various friction coefficients to determine trends. Figure 18 shows the relative permanent deformation of the opening id for all 32 runs with 3 variations in friction ($\mu = 0.0$, $\mu = 0.1$, $\mu = 0.4$). From the graph, it can be seen that the coefficient of friction equal to 0.1 matches very closely to the Test values in both slope and coefficient of Determination ($R^2 = 0.78$). This is a significant improvement over the original FEA model which had a $R^2_{orig} = 0.60$.

[2] Configuration 9 is one of the 32 DOE analytical runs.

By using the new boundary condition approach of applying a torque at the bolt and fixing the end of the wrench, we were able to improve our correlation and predictability. The correlation enhanced from 60% to 80%, which is very significant, especially when correlating against test values. Furthermore, the predictability was enhanced by shifting the slope of the correlation line from 0.27 to 0.66 (goal was 1.0) for a coefficient of friction of 0.1.

Another trend that was discovered (see Figure 18), was that the stability of runs also increased as the friction increased (note there is less variation from the trend lines). This follows logic, as increased friction causes the bolt and wrench to behave as one unit.

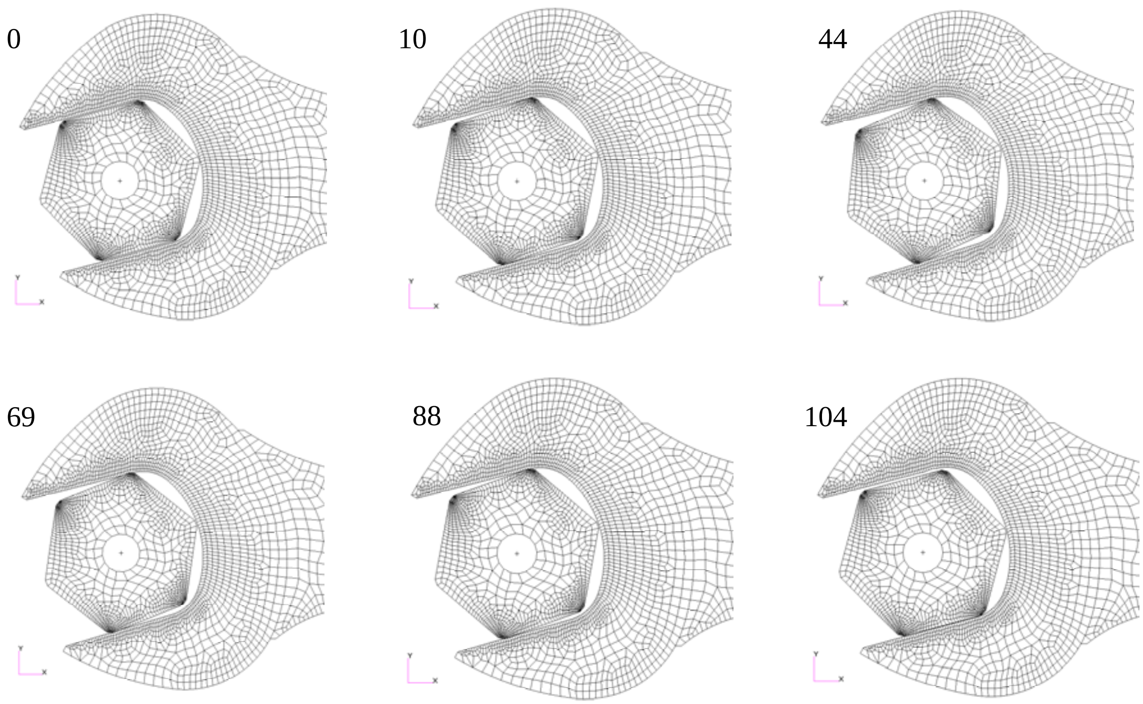


Figure 15: Deformation Views of the Analysis - Configuration 9
Load Step 1: 0 - 44, Load Step 2: 45 - 104

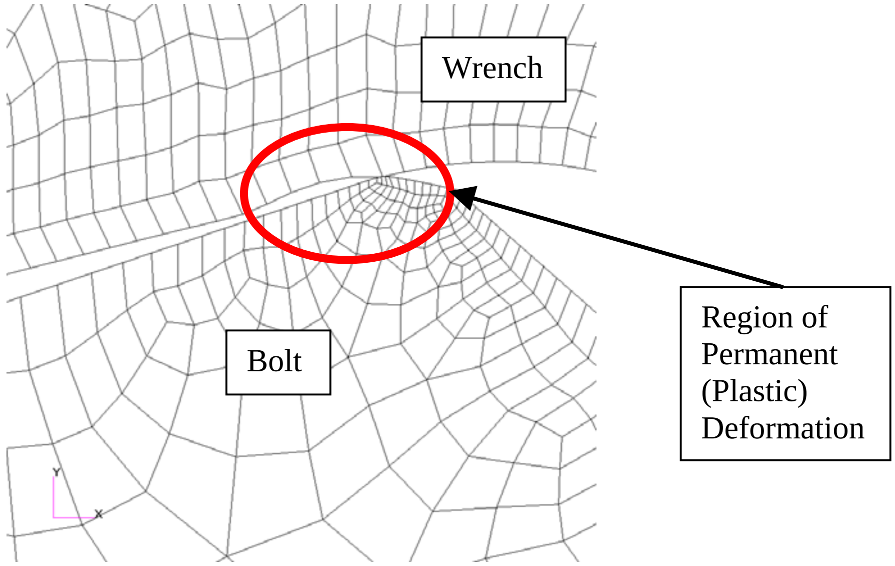


Figure 16: Permanent Deformation of Wrench for Configuration 9

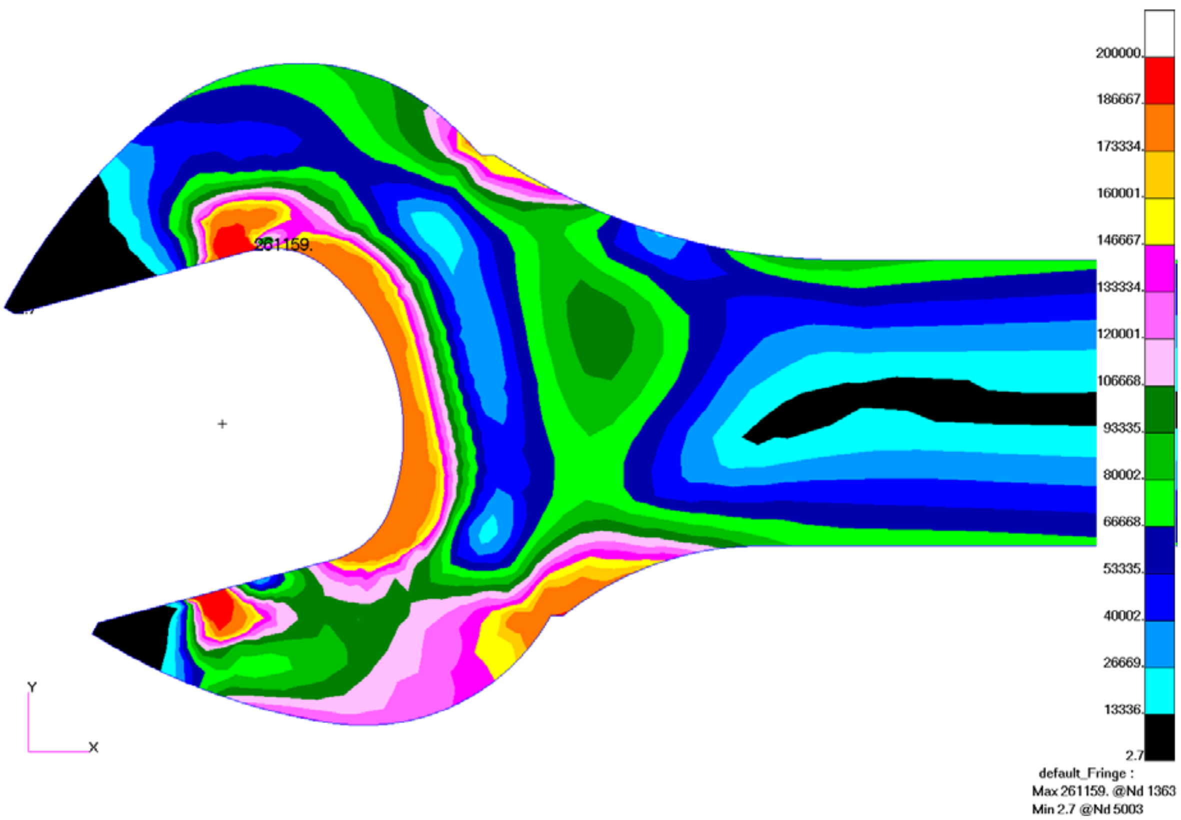


Figure 17: Von Mises Stress (Max Stress > Yield Stress) for Configuration 9

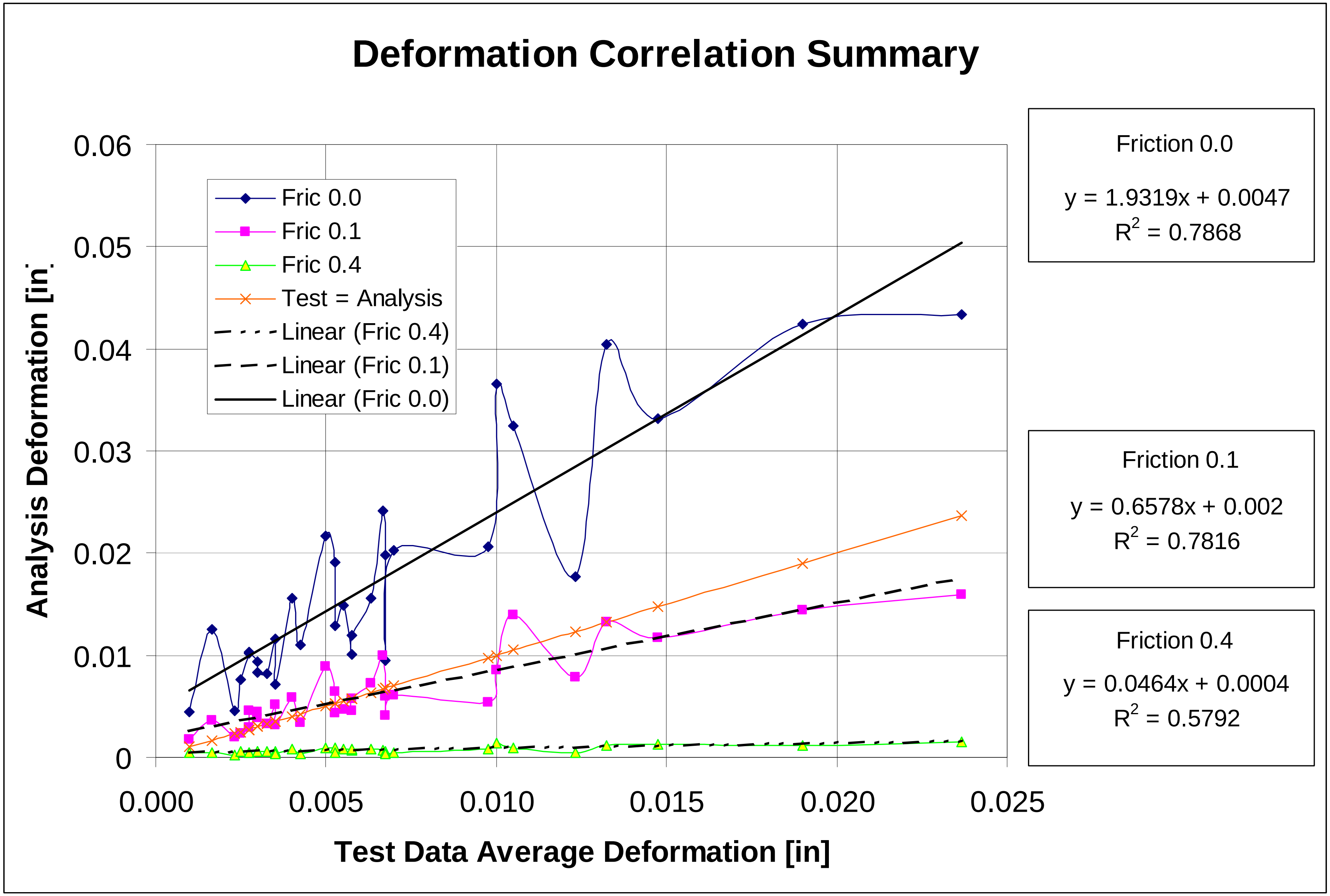


Figure 18: Correlation of Physical and Virtual Design of Experiment With Various Friction Coefficients

Conclusion

By conducting a physical DOE and an FEA DOE, the importance of ranking each factor within the design of the open-end wrench became relevant. Due to this study, the product performance was improved from 100% to 130% of the ASME standard for the product. In order to reach 150%, it was determined that other factors that were not studied may need to be examined. This includes, material type, heat treatment, and more drastic geometric shape changes. Additionally, the process for creating the opening id needs to be controlled to higher fidelity in order to reach the 150% goal.

With all of the error mentioned with both the physical and FEA DOE, we were very successful in establishing a very satisfactory correlation between the FEA and physical DOE. This has allowed Danaher Tool Group to gain higher trust in the Virtual Development Process (VPD) and the use of MSC.Patran and MSC.Marc, which will equate to lower costs in testing (specimen preparation, material costs, testing time, man-power, instrumentation, etc.,). Additionally, automation of FEA processes, which were developed during this exercise, has also reduced the amount of time required to perform FEA analysis – a further reduction in the cost of new product design.

Finally, by using the MSC.Marc solver and its robust contact algorithm, Danaher Tool Group was able to have a higher understanding of their open-end wrench interaction with the fastener (wrench to fastener contact interface). This has inspired a *break-through* in wrench design "The Claw", which moves the contact from the bolt tip, to the sidewall. This new design is currently under United States patent review, and will be available for sale by the October 2004. By using MSC.Marc, Danaher was able to reduce the conceptual design phase from 6 months to less then 2 months and assist in reduction of testing and costs required.

Traditional Open-End Wrench Design (see left configuration of Figure 19)

- Point loading at contact area causes rounding of fastener.
- Rounding of fastener reduces effectiveness of tool in applying torque to the fastener.

New Open-End Wrench Design (see right configuration of Figure 19)

- Offers greater contact area on fastener, which helps distribute torque.
- Higher torque helps in breaking fastener free or tightens.

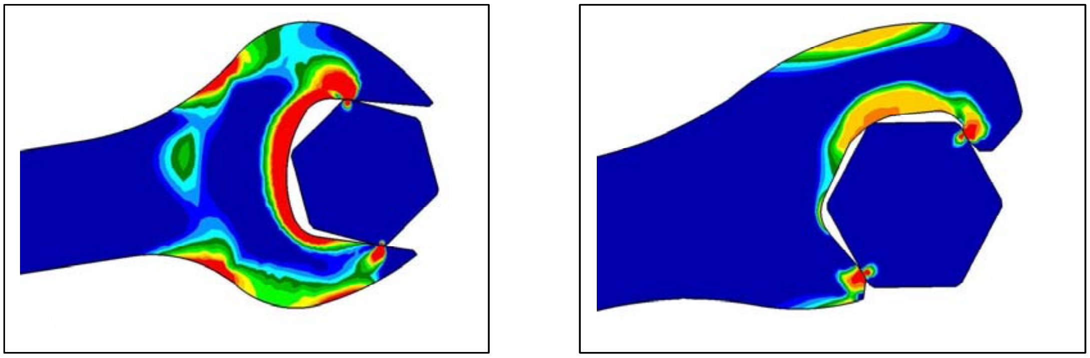


Figure 19: Traditional Open End Wrench vs. New Design "The Claw"

References

Avallone, Eugene A. & Baumeister, Theodore III ed., Mark's Standard Handbook for Mechanical Engineers 9th ed., McGraw-Hill, Inc., New York, 1987.

MSC.Marc Vol E Manual, Version 2004 On-Line, The MacNeal-Schwendler Corporation, Los Angeles, CA, 2004.

Acknowledgements

Awad and Tony would like to acknowledge all of the folks at Danaher Tool Group and MSC Software for giving us the time to work on this paper. Additionally, we would like to thank the following:

Edward Gillespie, Danaher Tool Group Quality Manager: Special thanks to Ed for helping us set up the Design of Experiment and the Statistical Analysis of the data.

Chris Tunney, MSC Application Engineer, for his contribution to the user subroutines and pushing us in directions not originally considered with MSC.Marc.

Muneer Baig, PhD Student University of Maryland Baltimore County, for helping with material testing of specimens.

Appendix A: Material Property Definition for FEA

The materials used for the physical prototypes were analyzed to verify physical composition and to determine properties for use in a finite element simulation. The material test specimens were taken from the same lot as the ones from which the physical prototypes were made.

Verification of material

The composition of the material was analyzed by Henry J.Yeager Laboratories and is provided in Table A1. The material was certified as type 4047 Steel.

Silicon	0.262
Manganese	0.733
Phosphorus	0.01
Sulfur	0.037
Total Carbon	0.499
Chromium	0.094
Nickel	0.064
Molybdenum	0.222
Copper	0.238
Aluminum	0.003

Table A1. Material composition.

Determination of physical properties.

The specimen was machined according to ASTM E8-01e1 standards as shown in Figure A1. The material is then heat-treated by same process as used for the physical testing. Again, three different heat treatments are done to obtain Rockwell hardness of 41HRc, 45HRc and 49HRc.

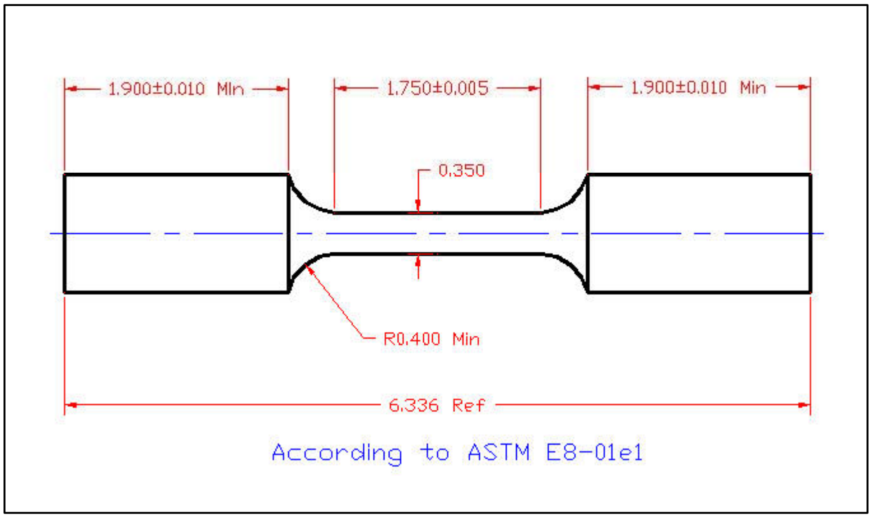


Figure A1: Tension Test Specimen

Uniaxial loading experiments were performed at UMBC lab using a MTS axial/tensional material test system. All experiments were done using quasi-static tensile loading; a low to medium strain-rate of 10^{-3} 1/sec.

The MTS machine was calibrated to increase the accuracy of results. Configuration files and test programs were fine tuned for every test. Load data was obtained directly from the MTS machine's load output. The stroke data of the MTS machine was found to be much greater than the actual deflection of the specimen; therefore strain gages were used to measure the deformations of the specimens.

The strain gages used in the material testing were from KFE-2-120-C1, KYOWA, Inc., Japan. All gages were bonded with M-Bond AE-15 adhesive (Measurement Group, Inc.) in the middle of the cylinder specimen. To reduce error, two strain gages were mounted on diametrically opposite sides of the specimen. The outputs of the strain gages were connected to a Model P-3500 strain indicator from Measurements Group Inc. using a Wheatstone bridge circuit. The strain gages were measured separately to ensure accuracy and to determine any data corrections. All of the information (time, load, and strain gages) was recorded using a digital acquisition system.

Several experiments were conducted at the same condition to verify repeatability. The test results were very repeatable and indicate that the machine response, testing procedure, and material response were consistent. A summary of the material properties from the tests is shown in Figure A2.

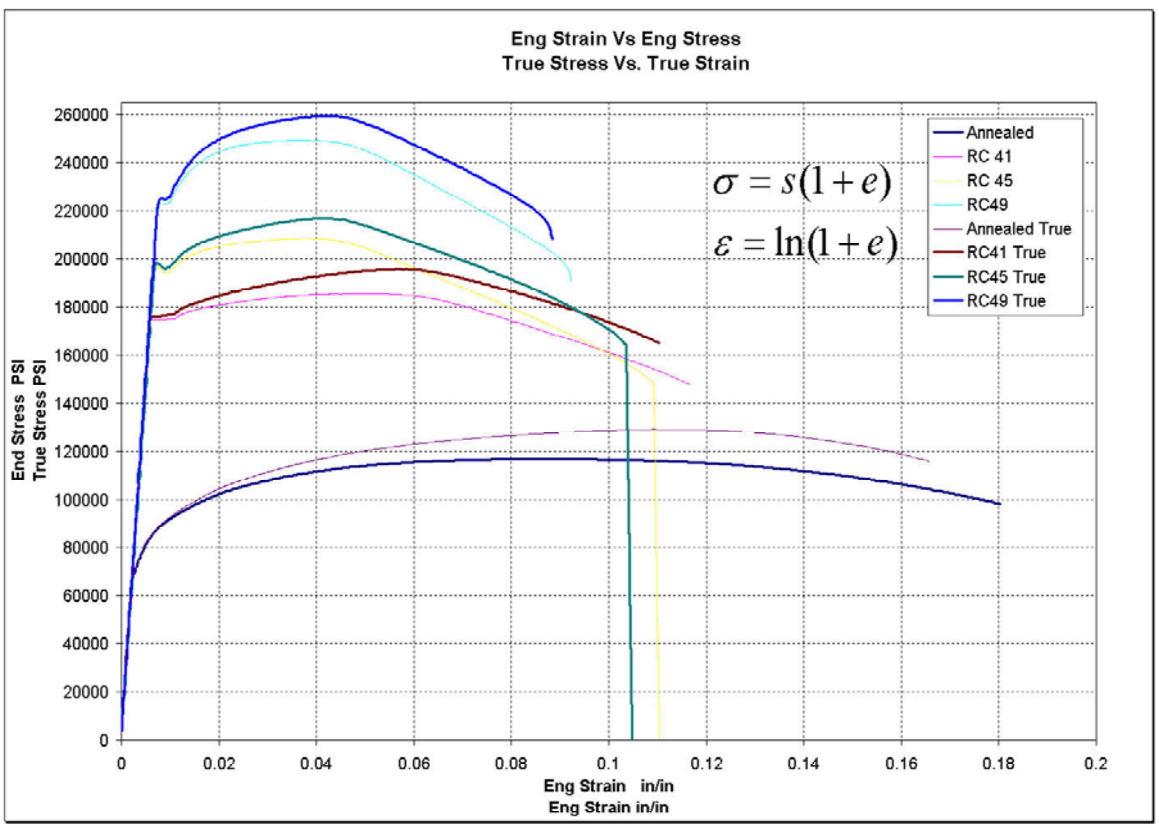


Figure A2: Engineering to True Stress/Strain Conversion

Based on the test data, the elastic region for all materials was described as follows:

Young's Modulus (E): 30×10^6 psi
Poisson's Ratio (υ): 0.29
Density (ρ): 0.278

Large strain nonlinear finite element simulations use true stress/strain and therefore the nonlinear material data needs to be converted from engineering stress/strain to true stress/strain. For the MSC.Marc finite element solver the material data is entered in 2 parts, the initial elastic material data (elastic modulus and poisons ration), and the nonlinear material data (true stress versus log plastic strain).[3]

[3]MSC.Software Corporation MSC.Marc 2003 documentation. MSC.Marc Volume A: Theory and User Information, Chapter 7 Material Library, Workhardening Rules, page 7-76 through 7-78.
MSC.Software Corporation MSC.Marc 2003 documentation. MSC.Marc Volume C: Program Input, Chapter 2 Parameters, "FINITE" keyword, page 2-34
MSC.Software Corporation MSC.Marc 2003 documentation. MSC.Marc Volume C: Program Input, Chapter 3 Model Definition Options, "Work Hard" keyword, page 3-367.

Appendix B: Material Conversion

The stress/strain values we get from the physical tests are typically reported using engineering stress/strain; these values are calculated using the original cross sectional area of the test specimen. However, as the load is applied to the specimen and it starts to strain the cross sectional area of the specimen changes. True stress values are based on the instantaneous area and therefore take into account the change in cross sectional area. Note that the true stress tensor is also referred to as Cauchy stress. The difference between engineering stress/strain and true stress/strain is typically small when in elastic region but it can be significant once the material strains into the plastic region.

Below is an excerpt from a MSC.Software training document summarizing the conversion of engineering stress/strain to the values required by MSC.Marc for large strain simulations. Christopher Tunney of MSC.Software provided the document and references.

Calculating True Stress & True Strain[4][5]:

A = Instantaneous/current cross section
A_o = Initial cross section
E = Elastic Modulus
L = Instantaneous/current length
L_o = Original Length
ε_o = Strain (engineering / nominal)
ε_{pl} = Strain (logarithmic plastic)
ε_{true} = Strain (true)
σ_o = Stress (engineering / nominal)
σ_{true} = Stress (true)

$$\sigma_o = \frac{Applied\ Load}{A_o} \qquad \varepsilon_o = \frac{\Delta L}{L_o}$$

In the elastic (linear) range:

$$\sigma_o = E\varepsilon_o$$

True Stress is based on the instantaneous cross section, A, such that

$$\sigma_{true} = \frac{Applied\ Load}{A}$$

[4] Michael R. Lindeburg, PE , Mechanical Engineering Reference Manual for the PE Exam, Tenth Edition, (Professional Publications Inc., Belmont CA. 1998), Chapter 46, Material Properties and Testing, pages 46-1 through 46-3

[5] Eugene A. Avallone and Theodore Baumeister III. Marks' STANDARD HANDBOOK FOR MECHANICAL ENGINEERS, Tenth Edition. (McGraw-Hill, 1996), Section 5.1, Mechanical Properties of Materials. Pages 5-3 through 5-4.

Prior to necking

$$\varepsilon_{true} = \int_{L_o}^{L} \frac{dL}{L_o} = \ln\left(\frac{L}{L_o}\right) = \ln(1+\varepsilon_o)$$

$$\sigma_{true} = \sigma_o(1+\varepsilon_o)$$

If necking has occurred the true strain must be calculated from the instantaneous cross sectional area and not the length.

$$\varepsilon_{true} = \ln\left(\frac{A_o}{A}\right) \quad \text{or} \quad \varepsilon_{true} = 2\ln\left(\frac{D_o}{D}\right) \text{ (circular cross section).}$$

$$\sigma_{true} = \frac{Applied\ Load}{A}$$

When inelastic material models are used in MSC.Marc, the total strain is composed of three parts.

$\varepsilon_{true} = \varepsilon_{elastic} + \varepsilon_{pl} + \varepsilon_{creep}$

Assuming no creep strains, the log plastic strain subtracts out the linear portion of strain.

$$\varepsilon_{pl} = \varepsilon_{true} - \varepsilon_{elastic} = \varepsilon_{true} - \frac{\sigma_{true}}{E}$$

Conversion of engineering stress/strain data

Typically the engineering stress/strain data is converted to true stress (σ_{true}) and log plastic strains (ε_{pl}) using a spreadsheet. The MSC.Marc data points should be based on a plot of the stress versus log plastic strain for a tensile test. The elastic strain components should not be included. The first log plastic strain value should equal 0.0 and the first stress should agree with the yield stress in the ISOTROPIC or ORTHOTROPIC options. Note that when entering the material data there only needs to be enough points to accurately capture the true stress and log plastic strain curve, entering large amounts of data points is typically not necessary.

Note that there can be difficulties with commonly reported values as often only the engineering stress/strain information is documented and the reduction in area is not provided. If significant necking of a test specimen has occurred, inaccuracies can occur if the reduction in area due to necking is not taken into account.

Entering material plasticity using MSC.Patran/MSC.AFEA

If using MSC.Patran to enter the material plasticity with a stress-strain curve the following steps are used.

1. Create a Material Property Field for the stress-strain data.
2. Create a Material with the elastic properties
   (Constitutive Model: Elastic)
3. Using the same material as in STEP 2, add the plasticity properties
   (Constitutive Model: Plastic).

When complete, the material should have both the “Elastic” and “Plastic” options listed in the “Current Constitutive Models” option on the bottom of the “Input Properties” form.

Appendix C: MSC.Marc User Subroutine

By Chris Tunney
Application Engineer
MSC Software Corp.

To aid in post processing, the MSC.Marc User Subroutine feature was utilized. The MSC.Marc user subroutine feature allows for customization of MSC.Marc with FORTRAN subroutines that are linked to the MSC.Marc solver. For the wrench simulation, a subroutine was created to assist in post processing of the results. The subroutine used for the wrench simulations was based on a subroutine provided on MSC.Software's on-line knowledge base, solution 1-17269801 (http://support.mscsoftware.com/kb/results_kb.cfm?S_ID=1-17269801). Information on using the subroutine with MSC.Marc is included in the knowledge base article and in the MSC.Marc documentation.

The original subroutine created a text file with information on the distance between 2 nodes during a nonlinear simulation. The knowledge base subroutine was modified to include additional output parameters; bolt rotation and moment applied to the bolt. To aid is using these subroutines all of the simulation models used the same node numbers for the wrench tip, and bolt center. The subroutine source code and sample output are attached in the subsequent pages.

Sample output from MSC.Marc User Subroutine created to assist in post processing:

Output from MSC.Marc User Subroutine UEDINC

Distance between nodes 5003 & 5004

Increment	Time	Initial Distance	Final Distance	Change in Distance	Nd 5001:Appld Moment	Rotation(rad)	Nd 5002:X Force	Y Force	Total
0	0.00000	0.75200	0.75200	0.00000	0.00	0.0000	0.00	0.00	0.00
1	0.01000	0.75200	0.75227	0.00027	-24.75	-0.0058	0.44	3.81	3.84
2	0.02500	0.75200	0.75270	0.00069	-61.88	-0.0069	1.09	9.53	9.59
3	0.04750	0.75200	0.75332	0.00131	-117.56	-0.0086	2.32	18.12	18.26
4	0.05000	0.75200	0.75336	0.00135	-123.75	-0.0087	2.61	19.06	19.24
5	0.07500	0.75200	0.75406	0.00206	-185.63	-0.0105	-1.39	28.55	28.59
6	0.10000	0.75200	0.75472	0.00272	-247.50	-0.0122	2.80	38.11	38.21
7	0.12500	0.75200	0.75540	0.00340	-309.38	-0.0140	3.44	47.64	47.76
8	0.15000	0.75200	0.75607	0.00406	-371.25	-0.0157	3.67	57.15	57.27
9	0.17500	0.75200	0.75676	0.00476	-433.13	-0.0175	3.70	66.68	66.78
10	0.20000	0.75200	0.75745	0.00545	-495.00	-0.0193	3.50	76.20	76.28
11	0.22500	0.75200	0.75813	0.00613	-556.88	-0.0210	3.06	85.71	85.76
12	0.25000	0.75200	0.75900	0.00700	-618.75	-0.0232	13.85	95.41	96.41
13	0.27500	0.75200	0.75958	0.00758	-680.62	-0.0248	0.40	104.76	104.76
14	0.30000	0.75200	0.76021	0.00821	-742.50	-0.0265	-4.18	114.18	114.25
15	0.32500	0.75200	0.76097	0.00896	-804.38	-0.0284	-14.48	123.67	124.52
16	0.35000	0.75200	0.76173	0.00973	-866.25	-0.0305	-9.69	133.17	133.53
17	0.37500	0.75200	0.76251	0.01050	-928.13	-0.0327	-27.88	142.49	145.19
18	0.40000	0.75200	0.76320	0.01120	-990.00	-0.0345	-30.08	151.99	154.94
19	0.42500	0.75200	0.76389	0.01188	-1051.88	-0.0364	-31.11	161.49	164.46
20	0.45000	0.75200	0.76416	0.01216	-1113.75	-0.0373	80.53	172.24	190.14
21	0.47500	0.75200	0.76507	0.01306	-1175.63	-0.0397	-55.44	180.27	188.60
22	0.50000	0.75200	0.76574	0.01373	-1237.50	-0.0415	-44.64	189.90	195.08
23	0.52500	0.75200	0.76629	0.01429	-1299.38	-0.0430	1.87	199.94	199.95
24	0.55000	0.75200	0.76694	0.01493	-1361.25	-0.0448	29.59	209.76	211.84
25	0.57500	0.75200	0.76734	0.01534	-1423.13	-0.0460	97.05	219.90	240.37
26	0.60000	0.75200	0.76770	0.01570	-1485.00	-0.0471	230.88	231.00	326.60
27	0.61250	0.75200	0.76743	0.01543	-1515.94	-0.0467	219.48	235.64	322.02
28	0.61251	0.75200	0.76769	0.01569	-1515.96	-0.0471	347.71	237.04	420.82
29	0.61252	0.75200	0.76871	0.01671	-1515.99	-0.0494	60.88	233.91	241.71
30	0.61253	0.75200	0.76871	0.01670	-1516.01	-0.0494	60.78	233.91	241.68
31	0.61255	0.75200	0.76871	0.01670	-1516.05	-0.0494	60.79	233.92	241.69
32	0.61257	0.75200	0.76871	0.01670	-1516.10	-0.0494	60.79	233.93	241.70
33	0.61260	0.75200	0.76871	0.01670	-1516.19	-0.0494	60.79	233.94	241.71
34	0.61265	0.75200	0.76871	0.01670	-1516.31	-0.0494	60.79	233.96	241.73
35	0.61273	0.75200	0.76871	0.01670	-1516.50	-0.0494	60.80	233.99	241.76
36	0.61284	0.75200	0.76871	0.01670	-1516.78	-0.0494	60.81	234.03	241.80
37	0.61301	0.75200	0.76871	0.01670	-1517.21	-0.0494	60.82	234.10	241.87
38	0.61327	0.75200	0.76871	0.01671	-1517.84	-0.0494	60.85	234.19	241.97
39	0.61365	0.75200	0.76871	0.01671	-1518.79	-0.0495	60.88	234.34	242.12
40	0.61423	0.75200	0.76871	0.01671	-1520.22	-0.0495	60.93	234.56	242.34
41	0.61509	0.75200	0.76871	0.01671	-1522.36	-0.0495	61.01	234.89	242.68
42	0.61639	0.75200	0.76872	0.01671	-1525.57	-0.0495	61.34	235.39	243.25
43	0.61834	0.75200	0.76872	0.01672	-1530.39	-0.0496	61.47	236.13	244.00
44	0.62126	0.75200	0.76894	0.01694	-1537.61	-0.0500	62.33	237.25	245.30
45	0.62564	0.75200	0.76906	0.01706	-1548.45	-0.0504	63.12	238.92	247.12
46	0.63221	0.75200	0.76924	0.01724	-1564.71	-0.0508	64.32	241.43	249.85
47	0.64206	0.75200	0.76953	0.01753	-1589.09	-0.0516	66.07	245.22	253.97
48	0.65000	0.75200	0.76976	0.01776	-1608.75	-0.0522	67.37	248.27	257.25
49	0.66478	0.75200	0.77018	0.01818	-1645.33	-0.0533	69.87	253.92	263.35
50	0.67956	0.75200	0.77068	0.01867	-1681.91	-0.0546	84.95	259.68	273.22
51	0.69434	0.75200	0.77117	0.01917	-1718.48	-0.0559	92.65	265.42	281.12
52	0.70000	0.75200	0.77135	0.01935	-1732.50	-0.0563	94.77	267.60	283.89
53	0.72217	0.75200	0.77185	0.01985	-1787.37	-0.0577	337.23	278.69	437.49
54	0.74434	0.75200	0.77211	0.02011	-1842.23	-0.0588	14.78	283.60	283.99
55	0.75000	0.75200	0.77281	0.02081	-1856.25	-0.0604	91.52	286.61	300.87

56	0.77217	0.75200	0.77290	0.02090	-1911.12	-0.0610	4.50	294.08	294.12
57	0.79434	0.75200	0.77417	0.02217	-1965.98	-0.0643	24.26	302.74	303.71
58	0.80000	0.75200	0.77435	0.02234	-1980.00	-0.0648	14.13	304.81	305.13
59	0.82217	0.75200	0.77455	0.02255	-2034.87	-0.0655	109.16	314.27	332.69
60	0.84434	0.75200	0.77528	0.02328	-2089.73	-0.0675	-75.07	320.69	329.36
61	0.85000	0.75200	0.77527	0.02327	-2103.75	-0.0676	-76.59	322.84	331.80
62	0.87217	0.75200	0.77521	0.02320	-2158.62	-0.0678	-95.02	331.08	344.44
63	0.89434	0.75200	0.77692	0.02492	-2213.48	-0.0719	-107.32	339.40	355.96
64	0.90000	0.75200	0.77712	0.02511	-2227.50	-0.0725	-109.41	341.54	358.63
65	0.92217	0.75200	0.77788	0.02588	-2282.37	-0.0747	-109.70	349.96	366.75
66	0.94434	0.75200	0.77862	0.02662	-2337.23	-0.0766	-80.88	358.69	367.70
67	0.95000	0.75200	0.77883	0.02683	-2351.25	-0.0771	-75.34	360.95	368.73
68	0.97217	0.75200	0.77833	0.02633	-2406.12	-0.0764	-27.14	369.90	370.89
69	0.99434	0.75200	0.77947	0.02747	-2460.98	-0.0790	113.49	379.88	396.47
70	1.00000	0.75200	0.77923	0.02722	-2475.00	-0.0787	49.44	381.33	384.53
71	1.01000	0.75200	0.77917	0.02717	0.00	-0.0779	-31.79	361.70	363.09
72	1.02000	0.75200	0.77900	0.02700	0.00	-0.0771	61.69	353.55	358.89
73	1.03000	0.75200	0.77889	0.02689	0.00	-0.0763	55.83	338.83	343.40
74	1.04500	0.75200	0.77873	0.02673	0.00	-0.0751	46.10	316.76	320.10
75	1.05000	0.75200	0.77868	0.02668	0.00	-0.0747	43.28	309.43	312.44
76	1.07250	0.75200	0.77844	0.02644	0.00	-0.0730	31.53	276.43	278.22
77	1.09750	0.75200	0.77818	0.02618	0.00	-0.0710	21.59	239.83	240.80
78	1.10000	0.75200	0.77816	0.02615	0.00	-0.0708	20.82	236.17	237.09
79	1.12500	0.75200	0.77789	0.02589	0.00	-0.0688	12.56	199.74	200.13
80	1.15000	0.75200	0.77763	0.02563	0.00	-0.0669	6.88	163.40	163.54
81	1.17500	0.75200	0.77737	0.02537	0.00	-0.0649	5.68	127.18	127.30
82	1.20000	0.75200	0.77710	0.02510	0.00	-0.0629	10.37	91.24	91.82
83	1.22500	0.75200	0.77682	0.02482	0.00	-0.0610	16.90	55.91	58.41
84	1.25000	0.75200	0.77642	0.02442	0.00	-0.0590	-52.46	25.29	58.24
85	1.27500	0.75200	0.77551	0.02351	0.00	-0.0570	-114.84	21.22	116.79
86	1.30000	0.75200	0.77464	0.02264	0.00	-0.0551	-126.72	17.28	127.89
87	1.32500	0.75200	0.77376	0.02175	0.00	-0.0531	-190.74	13.25	191.20
88	1.35000	0.75200	0.77284	0.02083	0.00	-0.0511	-120.54	14.70	121.43
89	1.37500	0.75200	0.77203	0.02002	0.00	-0.0492	-58.95	11.14	59.99
90	1.40000	0.75200	0.77117	0.01917	0.00	-0.0472	-36.26	9.88	37.58
91	1.42500	0.75200	0.77032	0.01831	0.00	-0.0452	-16.92	9.26	19.29
92	1.45000	0.75200	0.76946	0.01745	0.00	-0.0433	-5.42	8.71	10.26
93	1.47500	0.75200	0.76857	0.01657	0.00	-0.0413	12.01	9.71	15.44
94	1.50000	0.75200	0.76771	0.01571	0.00	-0.0393	2.41	8.32	8.66
95	1.52500	0.75200	0.76690	0.01490	0.00	-0.0374	-38.01	4.66	38.29
96	1.55000	0.75200	0.76609	0.01409	0.00	-0.0354	-98.23	-0.07	98.23
97	1.57500	0.75200	0.76524	0.01323	0.00	-0.0334	-98.65	-0.37	98.65
98	1.60000	0.75200	0.76434	0.01234	0.00	-0.0315	-30.71	3.68	30.93
99	1.62500	0.75200	0.76349	0.01149	0.00	-0.0295	-25.66	2.78	25.81
100	1.65000	0.75200	0.76260	0.01060	0.00	-0.0275	-25.40	4.39	25.78
101	1.67500	0.75200	0.76173	0.00973	0.00	-0.0256	-23.43	5.05	23.97
102	1.70000	0.75200	0.76085	0.00885	0.00	-0.0236	-23.67	5.03	24.19
103	1.72500	0.75200	0.76001	0.00800	0.00	-0.0216	-30.64	3.60	30.85
104	1.75000	0.75200	0.75920	0.00719	0.00	-0.0197	-62.37	1.10	62.38
105	1.77500	0.75200	0.75839	0.00639	0.00	-0.0177	-26.80	1.01	26.82
106	1.80000	0.75200	0.75756	0.00556	0.00	-0.0157	-22.64	1.10	22.67
107	1.82500	0.75200	0.75670	0.00470	0.00	-0.0138	23.36	3.17	23.58
108	1.85000	0.75200	0.75585	0.00385	0.00	-0.0118	24.76	2.30	24.87
109	1.87500	0.75200	0.75499	0.00298	0.00	-0.0098	17.80	1.83	17.90
110	1.90000	0.75200	0.75411	0.00210	0.00	-0.0079	7.37	1.01	7.44
111	1.92500	0.75200	0.75322	0.00122	0.00	-0.0059	0.62	0.10	0.62
112	1.95000	0.75200	0.75314	0.00114	0.00	-0.0039	0.00	0.00	0.00
113	1.97500	0.75200	0.75314	0.00114	0.00	-0.0020	0.00	0.00	0.00
114	2.00000	0.75200	0.75314	0.00114	0.00	0.0000	0.00	0.00	0.00

MSC.Marc user subroutine source code.

```
!  Subroutine UEDINC for MSC.Marc 2003r2
!
!-->  SUBROUTINE UPDATED BASED ON DANAHER BOLT-WRENCH MODELS
!
!  SUBROUTINE OVERVIEW:
!    This user subroutine calculates the change in distance between two nodes
!    as a simulation progresses.  The output from this is written to a text
!    file, "node_distance.out" and to the MSC.Marc .log and .out files.  A
!    sample of the output is below:
!       Output from MSC.Marc User Subroutine UEDINC
!       Distance between nodes   5003 &   5004
!       Increment   Time   Initial Distance   Final Distance   Change in
Distance
!           0     0.00000      0.76300           0.76300            0.00000
!           1     0.01000      0.76300           0.76338            0.00038
!
!  INPUT:
!    The 2 node numbers used.
!    These are entered below using the "DIST_NODE_1" and "DIST_NODE_2"
variables.
!
!  COMMENTS:
!    Written by C. Tunney
!    Email comments/questions to chris.tunney@mscsoftware.com
!
!  REVISION NOTES:
!      07/27/2004:  Updated <include 'CREEPS'> to be <include 'creeps'>
!                   to allow the subroutine to work on UNIX systems.
!      07/07/2004:  Initial subroutine created.
!
!  TESTING NOTES:
!    -Initially tested/compiled on Windows 2000 using Digital FORTRAN 6.0
!    -Tested using a nonlinear static solution.
!     (not tested for coupled simulaitons)
!
! MSC.Marc Volume D provided coding:
      SUBROUTINE UEDINC(INC,INCSUB)
      IMPLICIT REAL *8 (A-H, O-Z)
! INC =     the increment number.
! INCSUB =  the subincrement number.
!
! USER CODEING
      include 'creeps'
      !this is needed to obtain the "CPTIM" variable.

      INTEGER DIST_NODE_1, DIST_NODE_2
!  DIST_NODE_1 = Node to calculate distance from
!  DIST_NODE_2 = Node to calculate distance from
      REAL*8 NODE_1_LOCATION(3), NODE_2_LOCATION(3)
      REAL*8 NODE_1_DISP(3), NODE_2_DISP(3)
      REAL*8 NODE_1_FINAL_LOC(3), NODE_2_FINAL_LOC(3)
      REAL*8 INITIAL_DISTANCE, FINAL_DISTANCE, DISTANCE_CHANGE
      REAL*8 ND5001_MOMENT(3), ND5001_DISP(3)
      REAL*8 ND5002_FORCES(3), ND5002_TOTAL

!-->
! ENTER THE NODE NUMBERS OF THE 2 NODES TO BE USED
!-->
      DIST_NODE_1 = 5003
      DIST_NODE_2 = 5004


! Get node original locations
      CALL NODVAR(0, DIST_NODE_1, NODE_1_LOCATION, NQNCOMP,NQDATATYPE)
```

```
      CALL NODVAR(0, DIST_NODE_2, NODE_2_LOCATION, NQNCOMP,NQDATATYPE)
      IF (NQNCOMP.eq.2) THEN
        NODE_1_LOCATION(3) = 0.0
        NODE_2_LOCATION(3) = 0.0
      ENDIF

! Get nodal displacements at the end of the increment
      CALL NODVAR(1, DIST_NODE_1, NODE_1_DISP, NQNCOMP,NQDATATYPE)
      CALL NODVAR(1, DIST_NODE_2, NODE_2_DISP, NQNCOMP,NQDATATYPE)
      IF (NQNCOMP.eq.2) THEN
        NODE_1_DISP(3) = 0.0
        NODE_2_DISP(3) = 0.0
      ENDIF

! Calculate the deformed location of the nodes
      NODE_1_FINAL_LOC(1) = NODE_1_LOCATION(1) + NODE_1_DISP(1)
      NODE_1_FINAL_LOC(2) = NODE_1_LOCATION(2) + NODE_1_DISP(2)
      NODE_1_FINAL_LOC(3) = NODE_1_LOCATION(3) + NODE_1_DISP(3)
      NODE_2_FINAL_LOC(1) = NODE_2_LOCATION(1) + NODE_2_DISP(1)
      NODE_2_FINAL_LOC(2) = NODE_2_LOCATION(2) + NODE_2_DISP(2)
      NODE_2_FINAL_LOC(3) = NODE_2_LOCATION(3) + NODE_2_DISP(3)

! Calculate the initial and final distance between the nodes
      INITIAL_DISTANCE =((NODE_1_LOCATION(1)-NODE_2_LOCATION(1))**2+
     & (NODE_1_LOCATION(2)-NODE_2_LOCATION(2))**2 +
     & (NODE_1_LOCATION(3)-NODE_2_LOCATION(3))**2 )**0.5

      FINAL_DISTANCE=((NODE_1_FINAL_LOC(1)-NODE_2_FINAL_LOC(1))**2+
     & (NODE_1_FINAL_LOC(2)-NODE_2_FINAL_LOC(2))**2 +
     & (NODE_1_FINAL_LOC(3)-NODE_2_FINAL_LOC(3))**2 )**0.5

      DISTANCE_CHANGE = FINAL_DISTANCE - INITIAL_DISTANCE

!--> ADDED CALLS FOR DANAHER BOLT-WRENCH MODELS
      !GETTING APPLIED FORCE(MOMENT) AT NODE 5001
      CALL NODVAR(3, 5001, ND5001_MOMENT, NQNCOMP,NQDATATYPE)
      IF (CPTIM.gt.(1.00001)) THEN
!Setting the applied moment to zero for STEP 2 since it is displacement
controlled.
        ND5001_MOMENT(1) = 0.0
      ENDIF

      !GETTING DISPLACEMENT AT NODE 5001
      CALL NODVAR(1, 5001, ND5001_DISP, NQNCOMP,NQDATATYPE)

      !GET THE REACTION FORCE AT NODE 5002
      CALL NODVAR(5, 5002, ND5002_FORCES, NQNCOMP,NQDATATYPE)

      ND5002_TOTAL=(ND5002_FORCES(1)**2 + ND5002_FORCES(2)**2)**0.5


!Writing node distance information to a new file
      IF (inc.eq.0) THEN
      !If the first (zero) increment, open a new file.
        open(101,file='node_distance.out', status='UNKNOWN')
        write(101, 200)
        write(101, 205) DIST_NODE_1, DIST_NODE_2
        write(101, 210)
      ENDIF

      write(101, 215) INC, CPTIM, INITIAL_DISTANCE,
     &  FINAL_DISTANCE, DISTANCE_CHANGE,
     &  ND5001_MOMENT(1), ND5001_DISP(1),
     &  ND5002_FORCES(1), ND5002_FORCES(2), ND5002_TOTAL

!Writing node distance information to the ".log" file (FORTRAN UNIT = 0).
      write(0, 220)
```

```
      write(0, 200)
      write(0, 205) DIST_NODE_1, DIST_NODE_2
      write(0, 210)
      write(0, 215) INC, CPTIM, INITIAL_DISTANCE,
     &  FINAL_DISTANCE, DISTANCE_CHANGE,
     &  ND5001_MOMENT(1), ND5001_DISP(1),
     &  ND5002_FORCES(1), ND5002_FORCES(2), ND5002_TOTAL
      write(0, 220)

!Writing the node distance information to the ".out" file (FORTRAN UNIT = 6).
      write(6, 220)
      write(6, 200)
      write(6, 205) DIST_NODE_1, DIST_NODE_2
      write(6, 210)
      write(6, 215) INC, CPTIM, INITIAL_DISTANCE,
     &  FINAL_DISTANCE, DISTANCE_CHANGE,
     &  ND5001_MOMENT(1), ND5001_DISP(1),
     &  ND5002_FORCES(1), ND5002_FORCES(2), ND5002_TOTAL
      write(6, 220)

!Format statements for text output.
200     format('Output from MSC.Marc User Subroutine UEDINC')
205     format('Distance between nodes ', I6, ' & ', I6)
210     format(T1, 'Increment', T13, 'Time', T20, 'Initial Distance',
     &   T39, 'Final Distance', T56, 'Change in Distance', T76,
     &   'Nd 5001:Appld Moment Rotation(rad)    Nd 5002:',
     &   'X Force   Y Force    Total')
215     format(T1, I5, T10, F8.5, T22, F9.5, T40, F9.5, T59, F9.5,
     &   T85, F9.2, T97, F9.4, T119, F9.2, T129, F9.2, T139, F9.2)
220     format(' ')

      RETURN
      END
```