2015-01-15SS-0175

Accounting for Thermal and Gravity Force Effects on Automotive Components Using 3D Simulation Software

Donald Jasurda, Vice President
Dimensional Control Systems, Inc.

Presented at SAE World Congress 2015

Abstract

The effects of thermal expansion and gravity on assembly processes in automotive manufacturing can and often do cause unexpected variation. Not only do these effects cause assembly issues, they can also create non-conformance and warranty problems later in the product lifecycle. Using 3D CAD models, advances in simulation allow engineers to design out these influences through a combination of tooling, process and tolerance changes to reduce costs.

This whitepaper examines the process of simulating the effect of both thermal expansion and gravity on automotive structures. Using real life examples, a number of solutions were determined and tested in a simulated environment to reduce product variation and account for unavoidable environmental variation.

Introduction

Automotive manufacturers frequently use rigid body CAD models to simulate variation. These models account for part and process deformation—the variation inherent in the manufacture of parts, and in their total assembled location. A detailed model can also account for tooling and assembly processes by incorporating additional tolerances into the overall simulation analysis. [1]

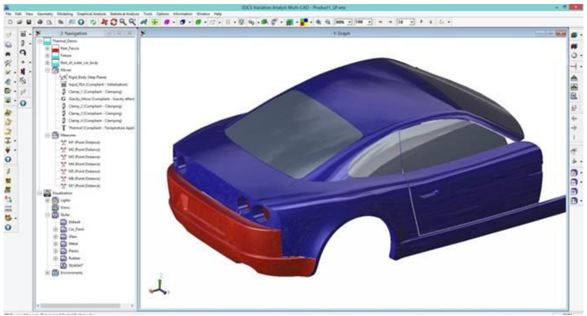

Figure 1. Automotive Body Structure Model. The model was originally created in NX, a CAD modeling software, and imported into 3DCS Variation Analyst Multi-CAD, a dimensional analysis tool.

During these analyses, the additional variation caused by natural forces is often missed. The effect of these natural forces, including gravity and thermal influences, negatively impact body components during production. As a result, components in the plant released from tooling or attached to the chassis sag from the stress of their own weight. This may cause additional fit and functionality issues during final assembly that were not accounted for in the original analysis. Besides the effects of gravity on components, it's important to consider material composition and properties. In this scenario, the plastic material of the rear fascia exhibits thermal expansion from environmental temperature change, compounding further on the variation caused by gravity. [2]

This project was completed as a proof of concept for use in a dimensional analysis process to determine the amount and placement of attachments between the vehicle body and the rear fascia, as shown in Figure 1, to reduce the added variation from both thermal and gravity. The analysis included:

- Both the weight of the part as influenced by gravity, and
- Expansion from temperature changes that may occur when operating the vehicle in different regions

3DCS Variation Analyst was used to complete the study using Monte Carlo simulation. As rigid body models fail to include additional forces, Finite Element Analysis, along with an FEA Solver and an add-on module were also needed to accurately simulate the effects of gravity and thermal change on the rear fascia of the automotive assembly.

To complete the study, FEA mesh files were generated using Abaqus software in SIMULIA provided by Dassault Systemes. The mesh files were applied to the rear fascia component, which was then simulated in 3DCS Variation Analyst using 3DCS FEA Compliant Modeler, the add-on module for adding Finite Element Analysis to the software suite. [3, 4]

Problem Definition

The primary issue to be resolved is the deformation of the rear fascia caused by environmental temperature changes and sag from gravity. This unexpected variation results in displaced components on the final vehicle requiring further inline rework.

This analysis is used to answer three questions:

1. Are the FEA simulation results accurate in 3DCS Variation Analyst, allowing the users to conduct the entire analysis in a single software system?
2. Is the deformation from gravity and thermal expansion large enough to validate the additional analysis?

3. Are the results from the tool accurate enough to substitute physical prototyping for iterative studies of possible solutions to build issues?

The end goal is to reduce the need for on-site rework by creating solutions through design changes and testing using simulation in a single 3D CAD environment.

Simulation

Using 3DCS Variation Analyst, Monte Carlo simulations were run of the rear fascia of an automotive body structure originally created in NX CAD software. To analyze force deformation, Finite Element Analysis was added to the model through mesh files created in SIMULIA, an FEA solver. Both gravity and thermal simulation were applied separately from the original simulation, isolating the variation from these two sources from all other influences. This was done to determine if the variation was significant enough to justify the additional analysis of these forces in addition to standard variation analysis.

The goal is to combine both analyses in a single tool, accounting for all sources of variation in the same CAD environment. This reduces the need for multiple models, as well as increases the accuracy of iterative studies to test solutions by including all influence factors in the same study.

Two separate ‘moves’ were used to complete the simulation, one for gravity and one for the thermal scenario. The gravity move used a mass matrix, which is a mesh file generated from Abaqus that plots the material properties onto the feature part. The thermal scenario uses an FEA mesh for the material properties, and was set for a change in temperature of 35 degrees Celsius.

The fascia was attached to the body structure in three directions as shown in Figure 2. The gravity move resulted in a variation in the –Z direction, which is the sag of the fascia when released from assembly tooling.

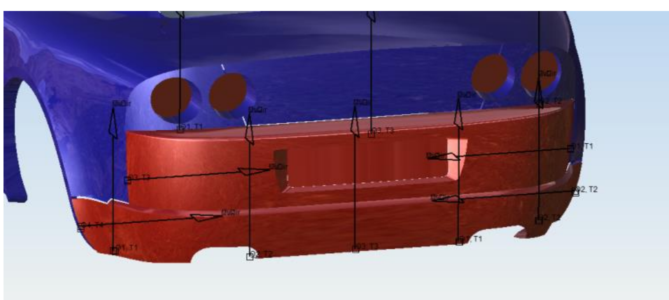

Figure 2. Direction of Assembly Moves. The move to attach the rear fascia to the body structure.

As shown in Figure 3, the variation from both effects caused a change in the gap from the rear fascia to the body structure.

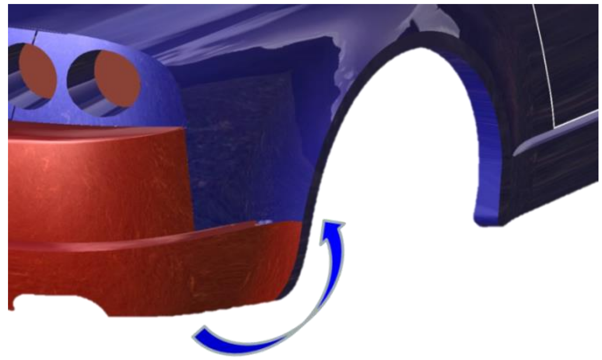

Figure 3. Location of Variation. The variation from the thermal and gravity move will cause this gap, between the rear fascia and the body structure, to increase.

The gravity move used a finite element analysis mesh file laid over the entire rear fascia. The mass properties were tied to nodes across the entire surface, which then deformed individually based on the influence of surrounding nodes, causing larger areas to sag more than thinner areas due to overall part weight.

Measurement points along the gap between the rear fascia and body structure were used for measuring the change from the two moves. Gap size was found to increase based on the two forms of deformation. From here, different clamping and attachment scenarios can be tested to account for this variation in the study, and theoretically, on the actual product. Figure 4 shows the locations of the measurements used for the study.

Figure 4. Measurement Points for Study. These seven locations are the measurements used to view the amount of variation from the gravity and thermal effects.

Monte Carlo Simulation Influencers

The original Monte Carlo simulation incorporated variation and tolerances from parts across the body of the vehicle in a 3D stack-up analysis. To isolate the change caused by gravity and thermal expansion, and to give a more accurate understanding of the variation added from only these two force effects, additional influencers were removed from this study. Any significant variation could then be re-incorporated later into the 3D tolerance stack and final simulations.

Mesh

There were two mesh files used in this study, both generated using Abaqus software. The first is a mesh overlay that was tied to nodes on the affected part. This mesh contained the material properties of the rear fascia and determined the deformation of the part for the thermal move. The second mesh file was a mass matrix, which determined the effect of the gravity move on the part.

Both mesh files originated in Abaqus, were imported into 3DCS and mapped onto the surface of the feature part, the rear fascia, then linked to nodes distributed throughout the part. Changes in part temperature compounded the variation across nodes, causing an expansion at the edges that consisted of the stack-up of all of the nodes' individual deviations. Figure 5 shows the material properties used in generating the mesh file. [5]

MATERIAL, NAME=Plastic
DENSITY = 1200.0 kg/m3
Young's Modulus = 2.2 e+09 Pascal
Poisson's Ratio = 0.38
Coefficient of thermal Expansion = 6.84 e-05

Figure 5. Material Properties of Rear Fascia. The material properties for the rear fascia were a basic plastic material.

Finite Element Analysis Methodology

The final result required using a single platform for the analysis, in this case, 3DCS Variation Analyst. The Finite Element Analysis and variation analysis could not be completed in separate software systems, so the FEA deformation results in 3DCS Variation Analyst had to be accurate. A validation study comparing the effects of a series of welds on a part in Figure 6 was completed using both Abaqus and 3DCS FEA Compliant Modeler separately to test the accuracy of 3DCS Variation Analyst's FEA capability.

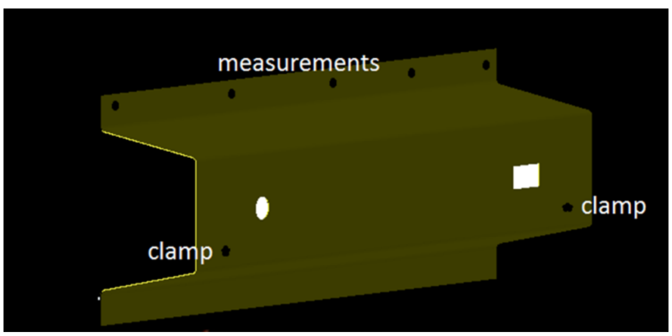

Figure 6. Validation Model for Accuracy Test. The above model, a rail component with clamps and welds was used in the accuracy comparison test between 3DCS Variation Analyst as a dimensional tool and Abaqus as an FEA tool.

The accuracy of the data, shown in Table 1, collected by 3DCS Variation Analyst demonstrates that a study once requiring both a dimensional analysis tool and FEA software could now be completed in a dimensional analysis tool alone to be viable in determining a future process. Table 1 shows the results of the deformation of the node points in 3DCS Variation Analyst compared to Abaqus.

As shown in Table 1, the results were not significantly different. This was used as demonstrative proof of the accuracy of the tools for this analysis. [6]

Table 1. Weld Deformation Points in 3DCS Variation Analyst versus FEA Node Deformation in Abaqus.

3DCS Point (node)	3DCS Deformation	Abaqus Deformation	FEA Node
Weld 1	0.6321	0.6321	41
Weld 4	0.65654	0.65654	46
Weld 5	0.15128	0.15128	42
Weld 6	0.15924	0.15924	45
Weld 7	0.27453	0.27453	44

As shown in Table 1, the results between collection methods were not significantly different. This demonstrates that the FEA solver's results are comparable to the dimensional analysis tool's results. With comparable results, the analysis data from FEA in the dimensional tool was determined to be accurate for this study.

Mathematical Results

The final results of the study are shown in Table 2. As demonstrated, there was a significant change in the gap between the rear fascia and the body structure.

Table 2. Total Change Measured in MM from Additional Effects. Refer to Figure 3 for the measurement locations.

Measurement	Effect of Gravity (mm)	Effect of Thermal (mm)	Total Variation (mm)
M1	2.71	5.80	8.51
M2	2.28	4.60	6.88
M3	1.83	3.58	5.41
M4	1.79	3.40	5.19
M5	2.64	4.26	6.90
M6	2.25	3.52	5.77
M7	1.52	2.21	3.73

Discussion

Addressing physical prototyping and dimensional issue repair with rework and warranty claims can be expensive. Creating reliable and accurate 3D simulations to determine the size and context of the dimensional issues before production and modifying the design in order to solve the variation, can significantly reduce quality costs. [7]

This study was conducted to determine if the Monte Carlo simulation could include the thermal and gravity effects needed to create a single robust model for iterative design changes. Abaqus was used in a comparative study to determine the accuracy of the dimensional analysis tool, which proved accurate for this use case. Employing this kind of model early in the design and concept phases of a product lifecycle can reduce the costs associated with rework and warranty claims later in the product lifecycle. Using simulation to find root causes gives designers and engineers the opportunity to modify a design and optimize tolerances before production to reduce or remove these costs altogether.

The team was also interested in discovering ways to accurately predict unexpected sources of variation, in this case those caused by moving the vehicle to a warmer climate, or found when the components of the body structure are released from tooling and support in assembly. In this way, engineers can incorporate all of the influences on a product during simulation, reducing the need for as many physical prototypes and tests.

Figure 7 shows the cost associated with variation in the product life cycle. Early in product design, simple changes to CAD models cost very little when compared to the cost of handling warranty claims and recalls when the product reaches the customer site. Therefore, it is most cost effective to find and solve issues as early as possible. [8]

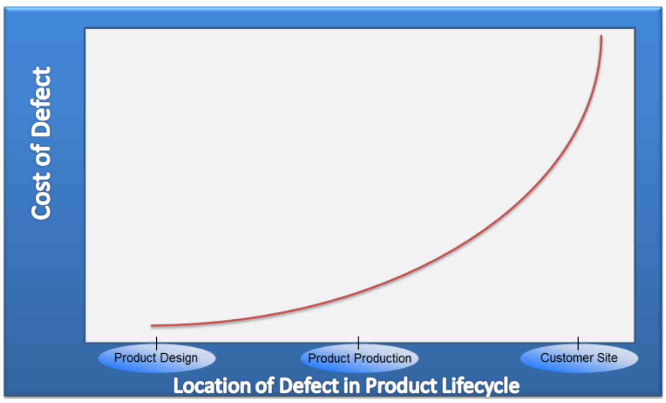

Figure 7. The Cost of Defects and Variation Based on Location in Product Lifecycle. As the defect is found closer to the customer site, the cost of fixing it becomes exponentially higher.

Conclusions

This study was conducted to answer three specific questions.

1. Could the effects of environmental influences (thermal, gravity) be accurately simulated in a 3D CAD dimensional analysis tool?
2. Was the influence of those forces significant enough to warrant the additional analysis?
3. Finally, is it possible to use the simulation to test solutions, reducing the need for physical prototypes and testing?

The study of the rail component in Figure 6 demonstrates the accuracy of the dimensional analysis tool compared to the FEA solver. A Finite Element Analysis could be conducted in either software accurately, allowing this portion of the process to be performed in the dimensional analysis tool.

In Table 2, the variation from the thermal and gravity force effects equated to more than half a centimeter in variation, which is significant enough to warrant additional analysis. Three to ten millimeters of variation can change the position on the rear fascia enough to require rework or cause issues elsewhere in the assembly when incorporated into a 3D tolerance stack. Therefore, design changes and additional attachments are necessary to counteract the additional variation.

With the simulation results of the force effects, and the Finite Element Analysis being conducted in the same software as the dimensional analysis, it is an easy next step to alter the design, and rerun the same simulation, verifying whether the design changes will reduce the variation in the measured gap. In this case, additional attachments are recommended along the gap.

Contact Information

Donald Jasurda
Dimensional Control Systems
djasurda@3dcs.com
248-269-9777

References

1. Kakar, S., Prentice, E., "Three Methods for Modeling Compliant Parts" presented at SAE AMAF 2012, USA, Sept 18-20, 2012.

2. Thornton, Anna C.; Donnelly, Stephen; and Ertan, Basak. “More than Contend with Variation and its Impact on Quality.” Research in Engineering Design, Volume 12, Number 3, 127-143. Springer-Verlag: London Limited, 2000.

3. 3DCS Variation Analyst CAA V5 Based (Version 7.2.2.0), Computer Software, DCS (Dimensional Control Systems), Troy, MI, 2014.

4. Abaqus FEA (Version 6.14), Computer Software, Dassault Systemes, SIMULIA, Paris, France, 2014.

5. Cook, Robert Davis., "Concepts and applications of finite element analysis." Wiley, 1989, New York, ISBN-10: 0471356050.

6. Kakar, S., Quinlan, B., "3DCS Compliant Modeler and Advanced Analyzer Optimizer " presented in *3DCS Variation Analyst Version 7.3, the Exciting New Release from DCS Part 2*, online webinar, USA, Aug 28, 2014.

7. Jasurda, D., "Dimensional Engineering and Tolerance Analysis: Do It Right the First Time," SAE Technical Paper 2012-01-0949, 2012, doi:10.4271/2012-01-0949.

8. Jasurda, D., "The Correlation of As-Manufactured Products to As-Designed Specifications: Closing the Loop on Dimensional Quality Results to Engineering Predictions," SAE Technical Paper 2011-01-2569, 2011, doi:10.4271/2011-01-2569.