JÓVENES EN LA CIENCIA
REVISTA DE DIVULGACIÓN CIENTÍFICA

"Numerical Study of the Clamping Pressure of an Automotive Support"

Brady Jordan Enstad (1), Dr. José Angel Diosdado De la Peña (2)

1 Mechanical Engineering, Massachusetts Institute of Technology | Dirección de correo electrónico: benstad@mit.edu

2 Departamento de Ingeniería Mecánica, DICIS Sede Salamanca, Universidad de Guanajuato | Dirección de correo electrónico: jose.diosdado@ugto.mx

Resumen

Simulación en la industria del automóvil permite a los diseñadores para corregir defectos al principio del proceso de diseño o para encontrar la causa de un nuevo problema descubierto más tarde. Se realizó este análisis para descubrir por qué un soporte automotriz mantiene caerse después del montaje. La parte fue escaneado 3D y se convierte en un modelo que Ansys Workbench 14.0 podría analizar. Se descubrió el apoyo ve tensiones desiguales y por lo tanto tiene una distribución de presión desigual, que conduce a un sellado defectuoso con el adhesivo. Para solucionar este problema, la parte o bien debe estar diseñado de tal manera que la distribución de la presión es par o un método diferente de accesorio debe ser considerado. El trabajo futuro puede incluir el análisis de diferentes lugares de aplicación de presión, para ver si la colocación de adhesivo en un lugar diferente en la parte puede resolver el problema.

Abstract

Simulation in the automotive industry allows designers to correct flaws early in the design process or to find the cause of a new problem discovered later. This analysis was conducted to discover why an automotive support keeps falling off after assembly. The part was 3D-scanned and converted into a model which Ansys Workbench 14.0 could analyze. It was discovered the support sees uneven stresses and therefore has an uneven pressure distribution, leading to a faulty seal with the adhesive. To fix this problem, the part must either be designed such that the pressure distribution is even or a different method of fixture must be considered. Future work may include analyzing different locations of pressure application, to see if placing adhesive on a different place on the part may also solve the problem.

Palabras Clave

Stress; Finite Element Analysis; Simulation; Design; Modeling.

INTRODUCCIÓN

Simulation in the automotive industry plays an important role in the design of parts that go into millions of vehicles. It allows for flaws in construction to be found very early in the design process and corrected. In other cases, simulation is used to solve ongoing problems. For instance, this project exists as a result of a part on the inside of a vehicle that consistently falls off after assembly. The reason for this was unknown. However, using simulation in Ansys Workbench 14.0, the reason was found.

The theory behind this simulation is actually quite simple qualitatively. Ansys Workbench uses a type of analysis known as Finite Element Analysis in order to simulate results. What this analysis does is break apart a model into thousands of pieces, each of which is a shape that a computer knowns how to analyze. After that, it is a simple matter of applying constraints to these pieces and compiling the results from each piece, as found in Doctor Diosdado's training, "Taller fundamentos de ANSYS APDL y ANSYS Workbench" [1].

MATERIALES Y MÉTODOS

The part which was under investigation had a very complicated surface geometry. The simplest way to obtain an accurate model was to have the part 3-D laser scanned. The file which was generated by the laser scanner was then imported into Solidworks and converted into a Parasolids file. Ansys 14.0 accepts Parasolids files in Design Modeler, so the file could be further modified before analysis. The model was simplified in order to make a higher quality mesh.

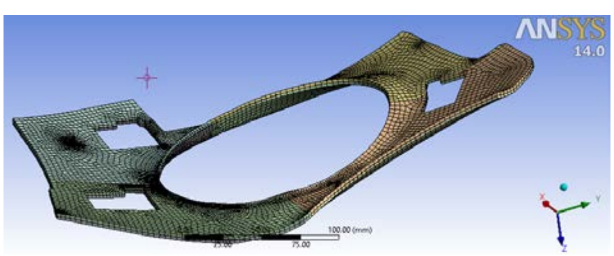

Figure 1: The automotive support shown after meshing in Ansys Workbench 14.0.

The geometry was then duplicated to provide a surface to push the part onto – the same way that the part is assembled on the assembly line. The duplicated part was treated as a fixed support in Ansys. The first part then had a pressure of 1,000 Pa applied to it into the fixed support to simulate the robotic arm that assembles the part.

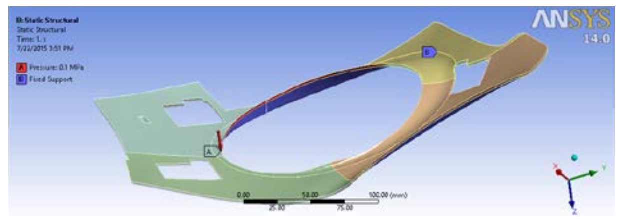

Figure 2: The automotive support with pressure and fixed support conditions shown.

RESULTADOS Y DISCUSIÓN

The results of the simulation are clear. The part sticks to the fixed support.

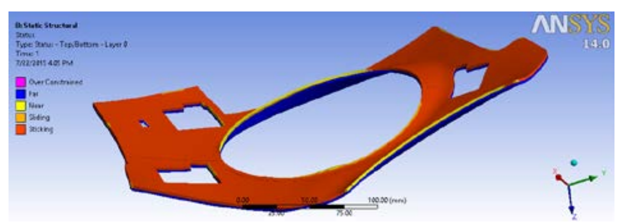

Figure 3: The automotive support clearly sticks to the fixed support, as it should and does after assembly.

However, the pressure distribution across the part is uneven. In the automobile construction process, glue is only placed on the surface around the headlight, so the pressure distribution needs to be even, or a good seal will not be achieved.

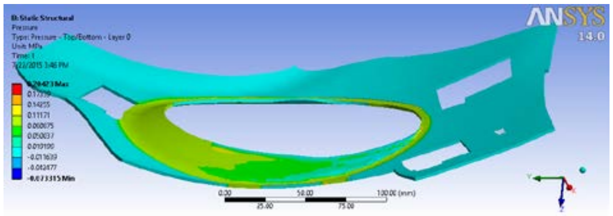

Figure 4: A clear indication that the pressure distribution across the automotive support is uneven. The light green color indicates an area of lesser pressure.

This pressure distribution indicates that the part also sees uneven stress distribution. Indeed, that is the case.

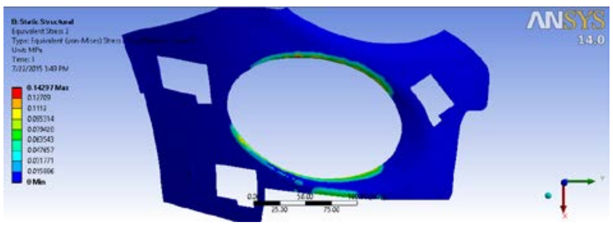

Figure 5: The stress distribution in the part is concentrated in just a few areas, instead of across the entire part in contact with the support.

As can be seen here, the part sees the most stress on the sides of the part – coincidentally, the sides are where the pressure distribution is uneven.

What this simulation has shown is the reason why the part fails to stay attached after the assembly process. The pressure distribution is uneven, leading to a poor seal with the glue holding the part together. The poor seal eventually fails, and the part falls off.

CONCLUSIONES

The part needs to be redesigned such that the pressure distribution across where the glue is applied is constant. If that flaw can be fixed, then the part will stay attached. Alternatively, a different method of fixture could be used instead of glue. A good idea for future study would be to change where the pressure is applied on the part. Perhaps applying glue and pressure in a different area would lead to a better seal.

AGRADECIMIENTOS

I would like to acknowledge Ing. Jesus Marañón Ruiz and CONMED Labmet S. de R.L. de C.V. for their invaluable assistance providing 3D laser scans of the part. I would also like to acknowledge the Massachusetts Institute of Technology for providing me with the Solidworks license I used for this project.

REFERENCIAS

[1] José Angel Diosdado De la Peña. "Taller fundamentos de ANSYS APDL y ANSYS Workbench". Meeting. Universidad de Guanajuato. Universidad de Guanajuato – Salamanca, 24 June 2015.