REDESIGN OF THE TORSION BRACKET

Sara Cinnamon
Research and Design Engineer
Ford Motor Company

ABSTRACT
Given an initial design for the bracket, our group aimed to make the part manufacturable, while minimizing process costs and mass of the bracket. ANSYS 5.5 was used for finite element analysis of the redesign.

INTRODUCTION
Ford trucks and SUVs have a new suspension design to improve over the previous I-beam design. The new suspension consists of an upper and a lower control arm, where the lower arm is connected to a torsion bar that controls vertical damping. This torsion bar must be connected to the frame of the vehicle via an isolation bracket. Given a proposed design of the bracket our group was given the task of optimizing said design, for ease of manufacture and cost-savings.

RESULTS
The Research and Design Group in charge of optimizing this bracket came up with a preliminary design for the greatest stability and modified that design for compliance with manufacturability and cost. The part will be constructed of 3 mm thick cold rolled steel. An image of the first design, showing its geometry, deflection, and Von Mises stress under typical loading is shown in Figure 1, below. Welds are applied along the five straight edges that make up the L-shaped region of the bracket, where it will mate with the frame of the vehicle. The torsion rod runs through the holes as requested by the original design, and effectively apply a pressure of 51.3 MPa on the inner surfaces of the holes. We see from the graph that the maximum deflection the part will undergo as it is subjected to loading is 0.044 mm, a scant amount compared to the original design. The maximum stress on the part is 266 MPa, which only occurs at the inner corners of the bracket. In production, there will in reality be a small fillet at this location, which will act to reduce the stress felt at this point. Elsewhere in the body of the bracket, the maximum stress encountered is 147 MPa, a tolerable amount considering the loading.

Unfortunately, this particular design would be quite expensive to manufacture. We would need to start with a large size stock of aluminum to be able to punch out the necessary flap to close the boundary. A figure of the necessary initial sheet can be seen in Figure 2. The bends are denoted by dashed lines.

Figure 1: Stress and Deflection of Preliminary Design
As you can see, there will be a lot of wasted material if we use this design. There will also necessitate at least two punch processes, three bend processes, and six welds to make this bracket viable. This will require a lot of time, effort, and manpower to complete, and will thus cost the company too much to be a reasonable solution. Additionally, the bracket would have a mass of 0.44 kg, which is more material than we would like, raising costs, especially considering the wasted material.

Instead, we look to other designs that will reduce the mass and manufacturing cost for the bracket. One way to do this would be to just remove the flap on the front of the previous design, making the sheet look like Figure 3.

It is readily seen that there will be less wasted material with this design, one less bend process and two less welds. However, now we must ensure that the behavior of the bracket under loading is satisfactory. To do this, we model the design in ANSYS and run the finite element analysis package to output the stress and deflection of the bracket. The results of the simulation are seen in Figure 4.

We see that the maximum stress has gone up almost three-fold, but this time is concentrated on the front corners, where we do not have a fillet planned, nor would one help. The stress here is caused by moment induced by the torsion bar twisting the welds and flexing the flanges of the bracket. This is not a healthy design and would tend to wear the welds, causing them to fatigue with vibration. The mass has decreased, as desired, to 0.36 kg, a non-trivial savings over Option 1.

Despite the added savings in cost of materials and manufacture, this iteration of the design did not perform up to expectations. We must determine another design that will satisfy all of these requirements. Instead of removing the front panel, our third iteration removed the back panel from the preliminary design. Now the initial sheet looks like Figure 5.

This sheet can likely be created in one punch, maybe two, depending on whether adequate grip can be maintained to punch through the holes. There will only be one bending process, and five welds. However, this additional weld is quite worthwhile, as it stabilizes the first and second modes of shearing, which attempt to make the front flanges skew from a rectangle to either a parallelogram or a trapezoid, as in Figure 6.
Next, we study the behavior of the bracket under the same loads as before, and see the results of the ANSYS analysis in Figure 7 below.

![Figure 7: Stress and Deflection of Third-Stage Design](image)

We see that the maximum stress is 356 MPa, a moderate increase over the preliminary design, but this stress is again located at the inner corners where a fillet will be made in production to alleviate stress concentrations at this location. The maximum deflection is only 0.11 mm, well within standard, and occurs in a non-critical position on the bracket. Also, the mass of the bracket is only 0.32 kg, lower than our second iteration, and a 27% decrease in weight from our preliminary design, with comparable performance and lower manufacturing costs.

**CONCLUSION**

The torsion bracket design that resulted from the third iteration is the best available, in terms of both performance and cost of manufacture. The Research and Design group recommends implementation of this design into all Ford trucks and SUVs for use until technology improves to give a better solution.