FEA Analysis of an Extruded Flange

Prepared by
Scott McIntosh

for
Fabco

December 2004

ALDEN RESEARCH LABORATORY, INC.
30 Shrewsbury Street
Holden, MA 01520

Model Creation

Alden conducted an FEA analysis of an extruded flange for Fabco's storm basin unit. The solid model for the simulation was provided by Fabco as an SAT file. This file was then processed into the COSMOS / DesignStar software package, and a mesh was generated using a global cell size constraint of 0.585 in. with a tolerance of 0.0292 in. There was also a mesh control (average cell size at model boundaries) of 0.195 in., with a growth rate ratio (size increase from adjacent cell) of 1.25. Only one section of the flange assembly was modeled, to save on computer time and mesh complexity; as seen in Figure 1.

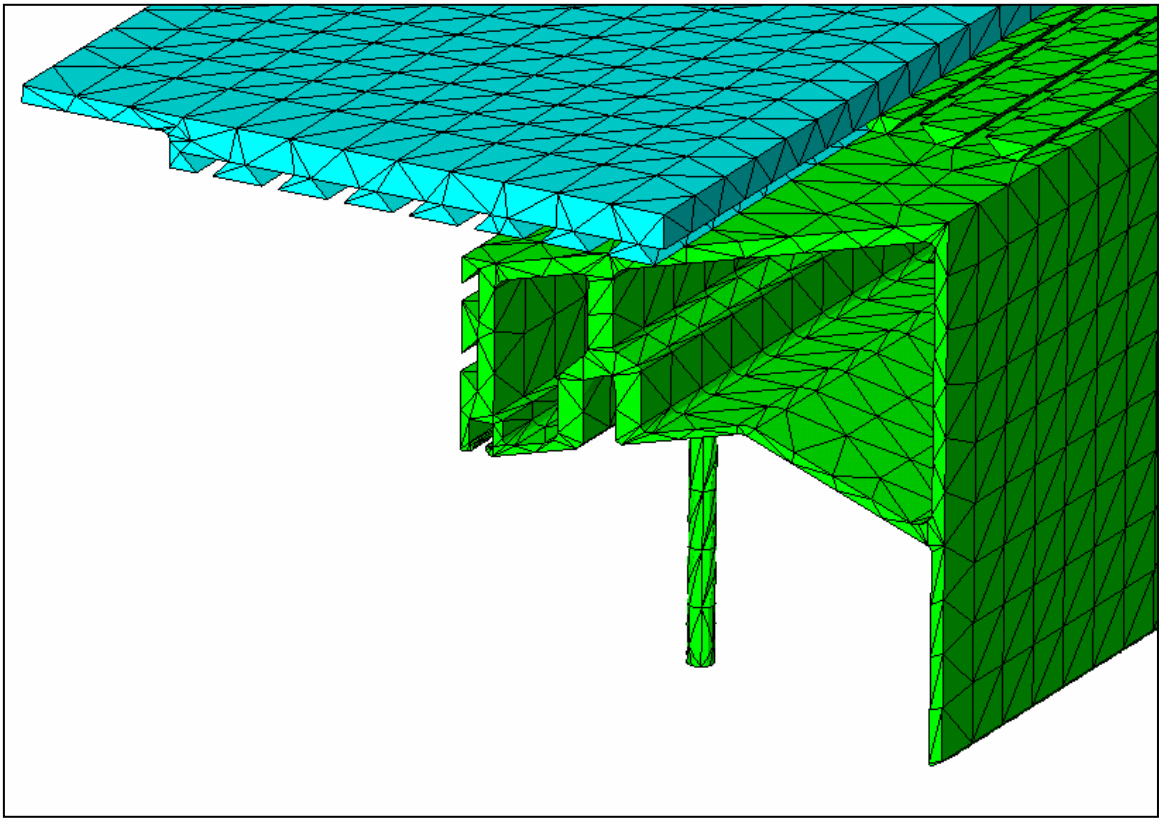

Figure 1 - Mesh shown for dovetail support, flange and bolts.

Boundary constraints were then added to the ledge that the storm basin rests on. The constraint location can be seen in Figure 2; the green arrows represent the whole bottom face of the grey 'ledge' as being fixed. A forty pound load was applied to the ends of each bolt to simulate a distributed load of 800lbs in the basin; shown as red arrows in Figure 3. The simulation was then run to determine the factor of safety of the flange, as well as the deflection and maximum stress of the flange. The material used for all parts in this simulation was T-6061 aluminum.

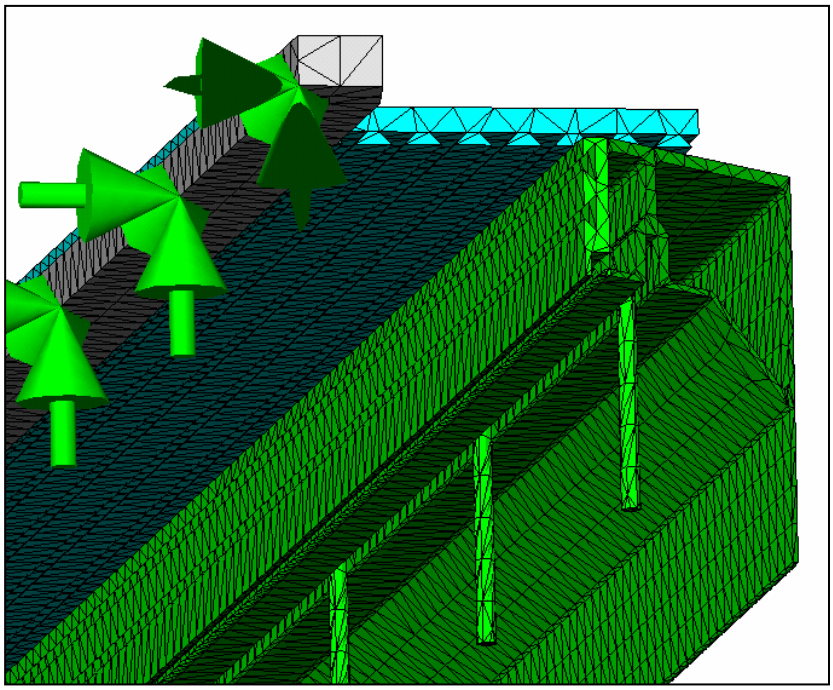

Figure 2 - Boundary constraint location

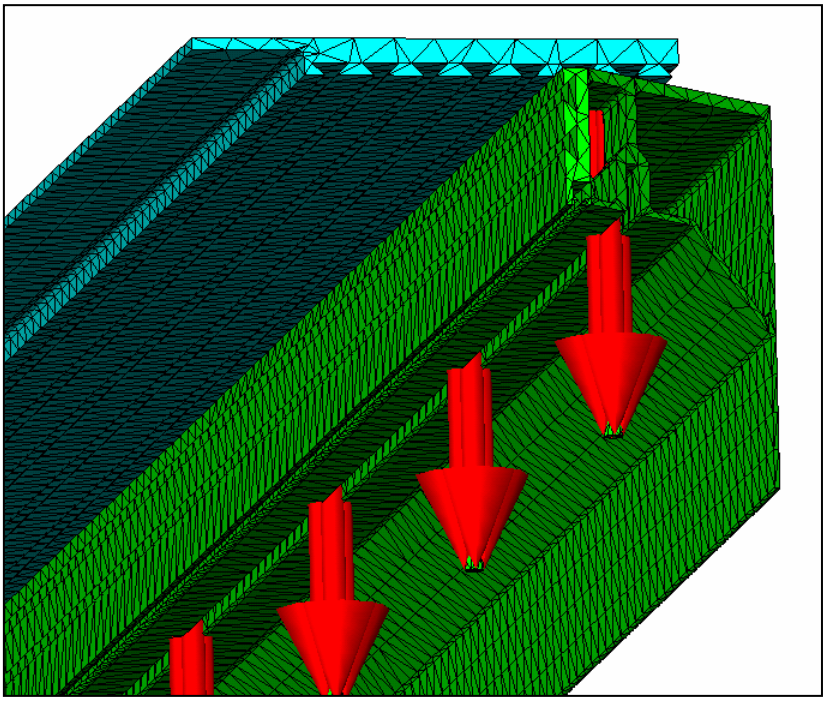

Figure 3 - Loads applied to bottom face of bolts

Results

The constraints and loading used for the simulation were conservative for this simulation. By constraining the support to the whole ledge a moment, and subsequent stress concentration, is added to the thinnest area of the part (see Figure 4.) If the fully assembled support and basin was modeled, instead of just one flange section, the deflection would most likely be reduced, due to the support of the basin on the lower flange section; as opposed to having no constraints in this simulation.

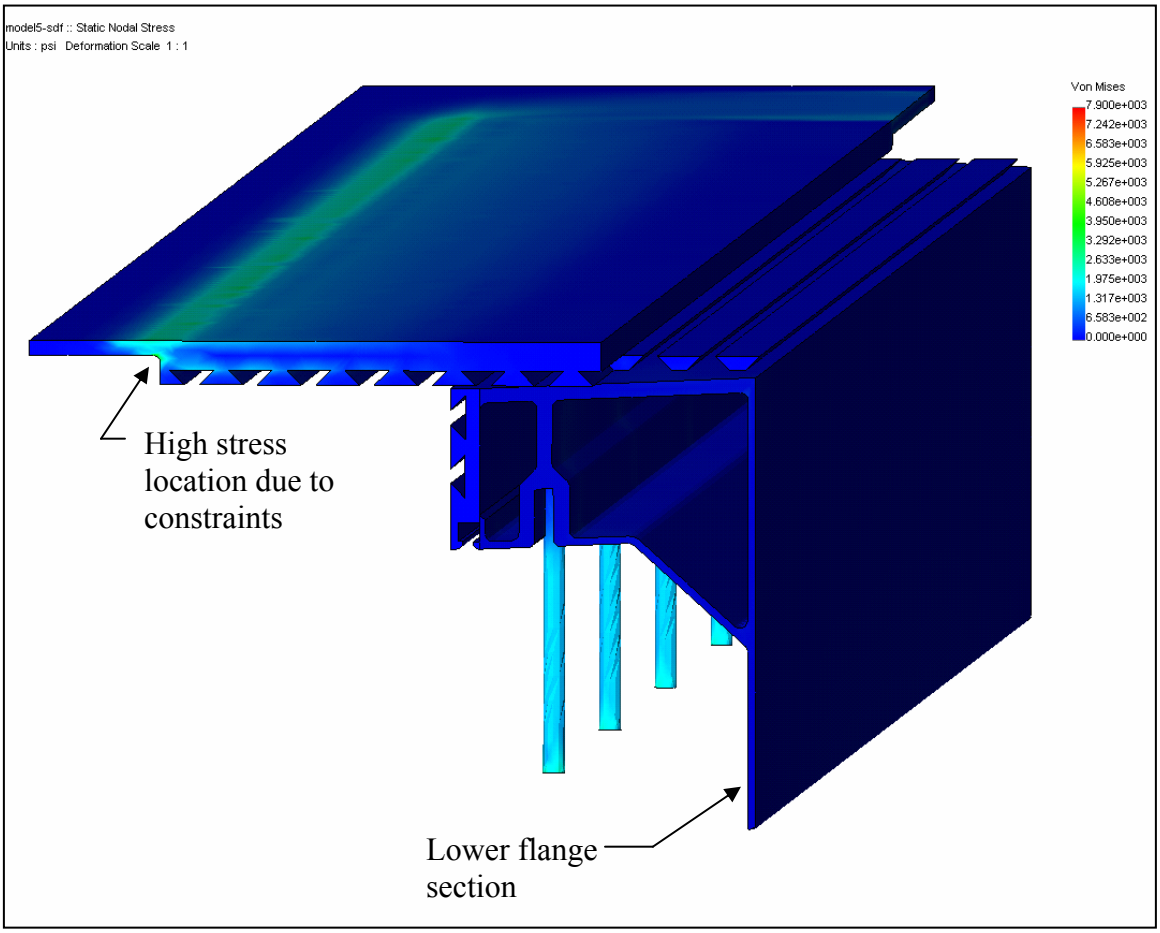

Figure 4 - Plot of Von Mises stress, showing stress concentration

The minimum factor of safety calculated in the simulation was 1.008, located at the end of the bolts. This was due to the fact the material for the bolts was specified as aluminum, as opposed to the actual stainless steel parts; as the bolts were 'bonded' to the flange, so the software viewed the flange and bolts as a single entity. Without the bolts, the factor of safety for the flange and support was acceptable. There is a small area where the FOS is less than two on the underside of the support, near the dovetails, as seen in Figure 5. This is due to the extra moment applied as discussed previously. When fully assembled, the sections of flange would support each other, as well as being constrained

by the storm basin itself; restraining the movement of the support and most likely yielding a higher FOS.

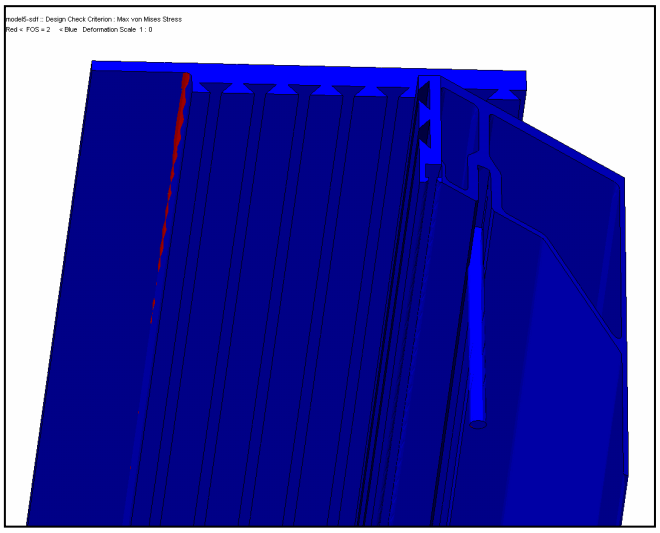

Figure 5 - Areas with FOS < 2 shown in red

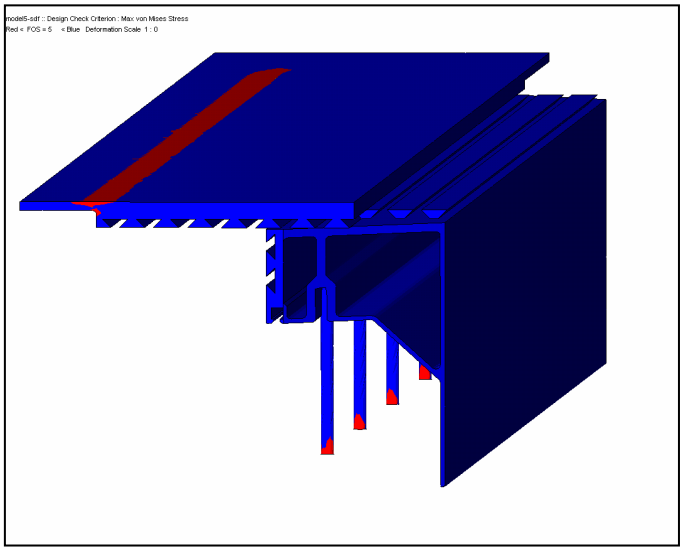

Figure 6 - Areas with FOS < 5 shown in red.

The maximum yield stress for the flange was 7940 psi; also concentrated at the end of the bolts. The yield stress at the thin section of the dovetail is a much more reasonable 1600 - 4500 psi; and there is very minimal stress for the majority of the flange. The maximum displacement was at the bottom of the flange lower section, and was calculated to be 0.0189 in. The deformation can be seen in Figure 7, shown in an exaggerated plot, with a ratio of 50:1.

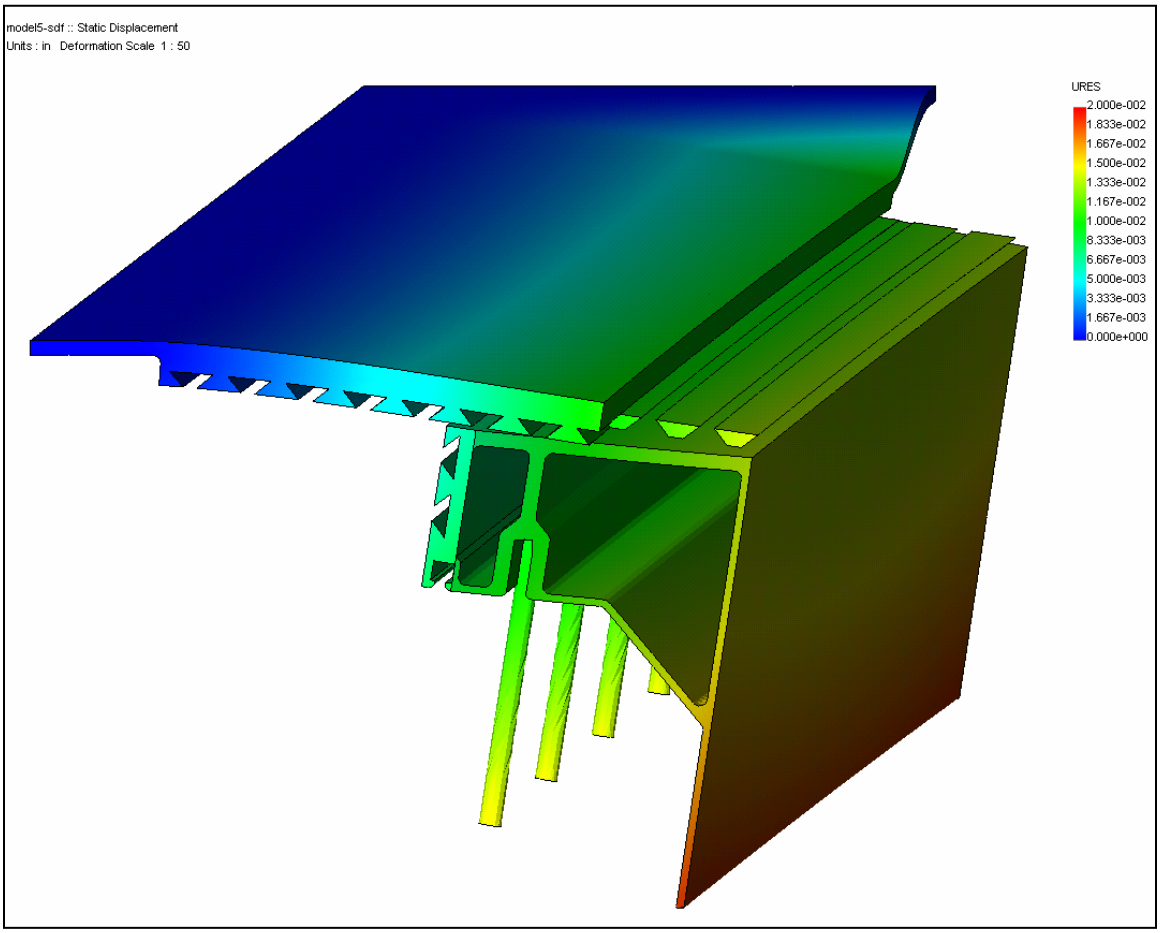

Figure 7 - Flange deformation; 50:1 scale

Conclusion

Based upon this simulation, it appears that this extruded flange will provide adequate support for Fabco's storm basin unit with the supplied operating conditions. This is due to the very small deformation shown, and the favorable FOS plots from the simulation. The area of the flange that does show a FOS less than 2 is due to the conservative simulation constraints; and will most likely be higher when in the fully assembled unit.