Application of ANSYS in Optimal Structural Design of Clara Table

Jessica Song
JSJ(China) Machinery & Electronics Co., Ltd.
Tianjin, China
Bruce E. Olmsted
JSJ Corporation
Grand Haven, Michigan, USA
Bob Benefiel
Counter Point
Spring Lake, Michigan, USA

Abstract
This paper outlines design optimization of Clara Table leg wall thickness using ANSYS software. A detailed finite element analysis of table leg assembly structure was undertaken. The applied load is according to BIFMA(Business & Institutional Furniture Manufacturer’s Association) standards. In the finite element analyses, contacts are defined between adjacent parts to represent real force transitions. The complexes of bracket geometries necessitated simplification for reducing finite element model size. The nonlinear finite element analysis can reasonably predict the structural response to the loading. The steps of modifying structure in ANSYS were highlighted to reduce deflections and stresses within permissible limits. The leg deflection was reduced by about 20%.

Introduction
Concurrent engineering practices provide clear benefits, increasing design flexibility, increasing productivity, and reducing costs. FEA tools should be effectively moved to earlier in the product development process, and non-analyst designers must use these tools. Without integrating the usage of FEA, many components are not validated until prototype testing, yet problem discovered this late in the process still have to return to the designer’s desk for rework. A common practice for non-analyzed parts is simply to over-design them in an attempt to account for their functional unknowns, resulting in sub-optimal products. FEA technologies are not limited in aerospace and automotive industries. Furniture manufacturers have adopted FEA. Clara Table is a new product in our company.

In view of the above situations, it was decided that it is vital to carry out FEA of the leg assembly and optimize the thickness of the leg wall, identify each of the components having large stress and strain and modify the design such that the stresses and deflections are kept within the safe limits.

The Model
For carrying out the FEA, ANSYS software, a Leading Finite Element Package was selected. The 3D geometries of the leg, mounting base and bracket were created in ProE. ProE translator was employed to translate the assembly into ANSYS and create volumes in order to be meshed using solid element. Since the 3D geometries were created in CAD system, there were a lot of small features such as small rounds, small surfaces. All these small features were removed from the model before the translation. This can significantly reduces the model size and speeds up the analysis time. The model was meshed using tetrahedral element, solid 92. Tetrahedral element has some of the unique advantages such as, during auto mesh most of the algorithms support this element; Complicated structures where we can not fit in the regular brick elements this element can be used. Due to intricacy of structure, it was decided to use tetrahedral element.
**The Boundary Condition**

3D surface to surface contacts between the leg and mounting base; leg and bracket were defined with elements 170 and 174. Although the employment of contacts makes computation time longer, it gives more accurate results. The bracket and mounting base were rigidly connected by common nodes, see Figure 1 and Figure 2. The bottom area of the bracket was fixed in 6 dof, see Figure 3.

![Figure 1. FEA Model of Clara Table Bracket and Mounting Base](image)

---

*Figure 1. FEA Model of Clara Table Bracket and Mounting Base*
Figure 2. Common Nodes Between Bracket and Mounting Base
Material

The materials of the bracket and mounting base were considered Aluminum 380.0_F Die Casting Alloy and have the following material properties:

- Elastic Modulus: 10,298,000 psi
- Poisson’s Ratio: 0.33
- Tensile Yield Stress: 23,061 psi
- Tensile Ultimate Stress: 45,977 psi
- Elongation @ break: 2.5%

The leg was considered Aluminum 6063-T5 and have the following material properties:

- Elastic Modulus: 10,000,000 psi
- Poisson’s Ratio: 0.33
- Tensile Yield Stress: 21,000 psi
- Tensile ultimate Stress: 26,800 psi
- Elongation @ break: 12%

Linear elastic materials were assumed in this analysis.
**Loading**

130lbf were applied on the top surface of leg according to ANSI/BIFMA standard in two directions, see Figure 4. The load shown in Figure 4 represents Down direction. The Up direction is opposite with the loading direction in Figure 4.

![Figure 4. Clara Table Leg Assembly FEA Model](image)

**Analysis Results & Discussion**

The objective of the analysis was to obtain the performances of the leg and find the best thickness of the leg wall.

The analysis started with a baseline model: a model, which the leg wall thickness is the same as line data provided, 0.60in. The baseline model was used to get an understanding of what is actually happening to the leg during the test. Maximum deflection of the leg is the critical criteria to evaluate the performance of the leg under the loading. Two scenarios were analyzed for the leg wall thickness of 0.60in with 130 lbf in two directions.

The next step in the analysis was to vary the leg wall thickness. This approach included a series of simulation. The total eight scenarios which leg wall thickness are 0.65in, 0.70in, and 0.75in, 0.80in, respectively, and in two loading directions were performed. The leg wall thickness geometries changes were carried out in ANSYS using extruding and operating commands. The results for these analyses are summarized in Table 1. Considering the bracket geometries, leg wall thickness was decided to be 0.70in. The deflection of leg wall thickness 0.70in was 0.417in when loading is along Down direction, see Figure 5. The corresponding deflection was 0.496in with leg wall thickness 0.60in. About 20% deflection was reduced.
Figure 5. Clara Table Leg Displacement

Table 1. Leg deflection vs leg wall thickness.

<table>
<thead>
<tr>
<th>Force direction</th>
<th>Leg wall thickness</th>
<th>Up</th>
<th>Down</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.060 in</td>
<td>0.495 in</td>
<td>0.496 in</td>
</tr>
<tr>
<td></td>
<td>0.065 in</td>
<td>0.452 in</td>
<td>0.454 in</td>
</tr>
<tr>
<td></td>
<td>0.070 in</td>
<td>0.416 in</td>
<td>0.417 in</td>
</tr>
<tr>
<td></td>
<td>0.075 in</td>
<td>0.385 in</td>
<td>0.386 in</td>
</tr>
<tr>
<td></td>
<td>0.080 in</td>
<td>0.358 in</td>
<td>0.359 in</td>
</tr>
</tbody>
</table>
Conclusion

Finite element analysis is a very efficient tool for the analysis of office furniture, and can help designers to evaluate the designs and provide modification guidelines. The product quality was greatly improved and the cost was reduced.

References