IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 21, Issue 2 Ser. 1 (Mar. – Apr. 2024), PP 13-15
www.iosrjournals.org

Design And Stress Analysis Of An Iron Board

Mir Hayder[1], Jermiah Gray[2]
[1]*(Engineering Technology Department, Savannah State University, Savannah, Ga 31406, Usa)*
[2]*(Student, Engineering Technology Department, Savannah State University, Savannah, Ga 31406, Usa)*

Abstract:
In this study, a static analysis of an iron board was performed using SolidWorks simulation software. The study was carried out in two phases. In the first phase, a 3D model of an iron board was designed, and in the next step, a stress analysis was conducted with a 20 lbs load on the top plane of the iron board with its stand fixed to the floor. The leading focus was to understand where the maximum stress is developed when a specific load is applied. In addition to the stress data, a displacement data was also developed to gain a better understanding. Results show that the maximum developed stress does not exceed the yield strength of the material, i.e., no parts within the iron board failed/yielded with the application of the load. The maximum deformation takes place in the middle of the top plane.

Key Word*: 3D Model; Part and Assembly Files; SolidWorks Simulation; Stress Analysis; Yield Strength.*

Date of Submission: 13-03-2024 Date of Acceptance: 23-03-2024

I. Introduction

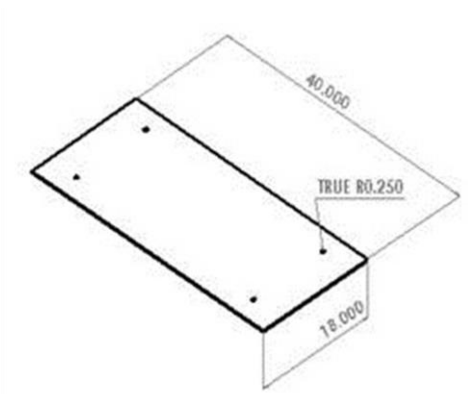
Many individuals sometimes face a time of confusion when something breaks or deform because of the amount of force applied. This is due to the misunderstanding of maximum load, points of maximum stress, and deformation.

The main objective of this study was to carry out a static analysis of an iron board using SolidWorks Simulation software. The main focus was to understand where the maximum stress is developed when the load is applied at the top of the iron board. In addition to stress data, displacement data is planned to be produced to develop a better understanding.

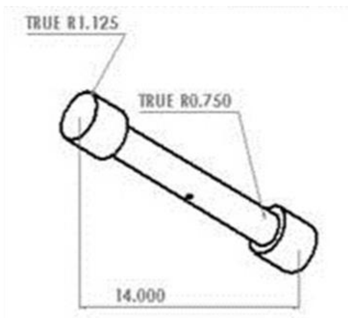
The analysis was carried-out into two stages:
1. To develop a suitable 3D model of an iron board, and
2. To perform a stress analysis keeping the stand of the iron board fixed to the floor.

II. Parts And Dimensions

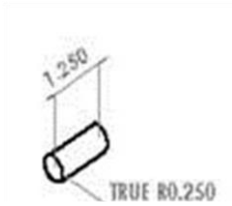
Five individual parts were created in SolidWorks: (1) Top board, (2) Balance, (3) Stand, (4) Pin 1, and (5) Pin 2 as part files. The dimensions of these parts are shown in Figure 1. The unit of dimension is in inch. It should be noted that the dimensions were arbitrary chosen by the authors; they were not replicated from any existing model of iron board.



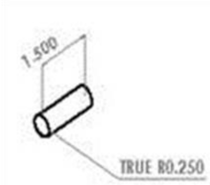
(a) Top Board



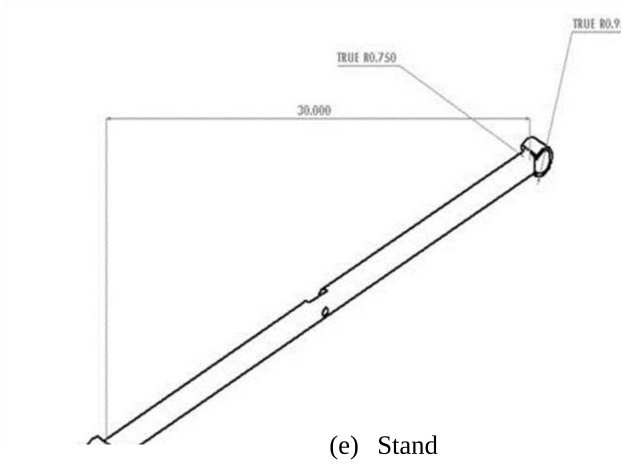
(b) Balance



(c) Pin 1



(d) Pin 2



(e) Stand

Figure 1: Part files created in SolidWorks for iron board.

III. Assembly Model

After creating the five separate Part files [1] according to the dimensions, an assembly file was prepared for SolidWorks simulation. Two balance parts were joined with the top board using four of pin 1. Two stands were joined in the shape of an 'X' by pin 2. The top of the stands was connected to the balances and the bottom was also joined to two balances that helped to support the entire assembly to stand on the ground. Figure 2 (a) and 2 (b) represent the exploded and assembly views of the iron board assembly, respectively.

Figure 2: Assembly model for the iron board: (a) exploded view and (b) assembly view.

IV. Simulation Model

Once the assembly model was complete, it is ready for simulation [2]. For simulation, the material for all parts was assumed to be 3003 Alloy Aluminum. The load applied from the top of the assembly was 20 lbs. This is shown by the downward purple arrows in Figure 3. The bottom of the assembly was made fixed to the floor for stability; the green marks on the bottom of the balances represents this situation.

V. Result

The analysis of results shows that, for 20 lbs of load applied on the top board, the Von Misess stress was found to be 16.603 MPa and the maximum displacement was 0.1762 mm.

Figure 4 represents the stress plots. Figure 4 (a) shows the stress distribution with a deformation scale of 577 and Figure 4 (b) shows the same with the exact scale. The displacement plots are shown in Figure 5. The plot with deformation scale of 577 is shown in Figure 5 (a) and that for exact scale is shown in Figure 5 (b). It should be noted that the deformation scale of 577 was arbitrarily chosen to provide a better understanding of the stress distribution and the deformation of the entire assembly.

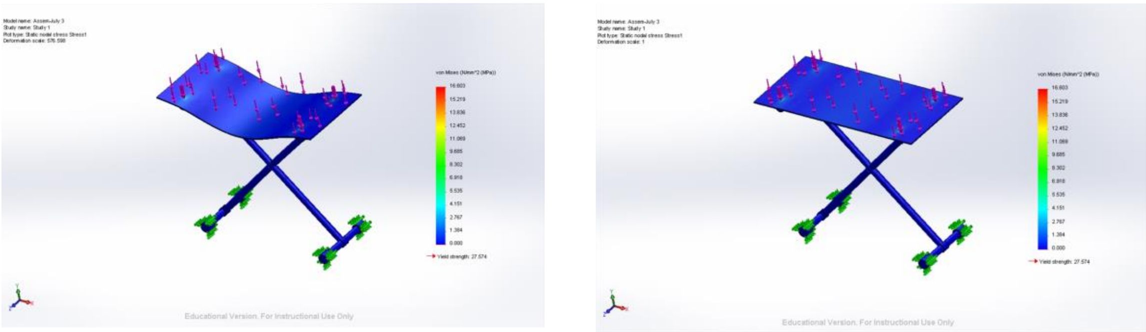


Figure 4: Stress distributions with (a) deformation scale of 577 and (b) exact scale.

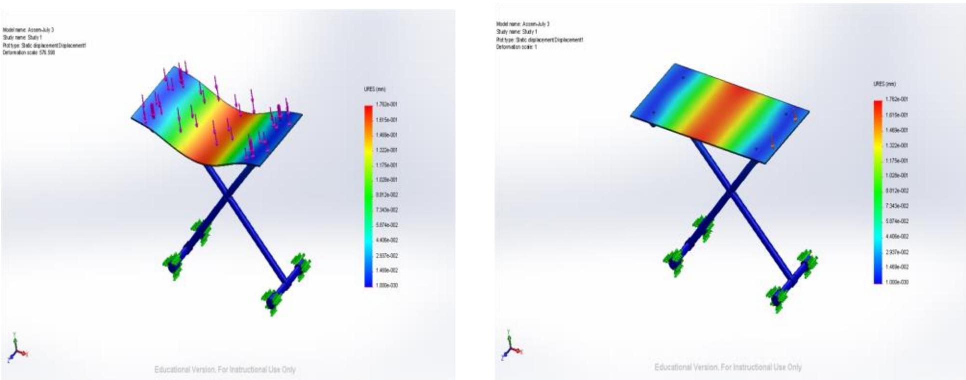


Figure 5: Displacement plots with (a) deformation scale of 577 and (b) exact scale.

The major conclusions from this study are summarized as below:

1. For 20 lbs load applied on the board of the iron board made of 3003 Alloy Aluminum, the maximum stress is developed on the top of the board in the near vicinity of the connection pins.
2. The maximum stress does not exceed the yield strength of the material, i.e., no parts within the iron board will fail/yield with the application of 20 lbs load.
3. The maximum displacement is developed in the center of the top of the ironing board.

References

[1]. SolidWorks 2023 Part I - Basic Tools: Parts, Assemblies and Drawings; By Paul Tran, SDC publications.
[2]. SolidWorks Simulation Essential Training Booklet, SolidWorks Corporation.