TECHNOLOGY STATION: REINFORCED & MOULDED PLASTICS

Design using FEA techniques of a HDPE planter

Project: **Planter box design**

Client: **River Rock Designs**

Order Number: **N/A**

Design method: **Patran/Nastran/3D modelling**

Consulting Company: **Technology Station, Durban University of Technology**

Consultant: **Ryan Hamilton (M.Tech: mechanical engineering)**

Supervisor: **Prof Mark Walker (PhD: engineering)**

1. Scope

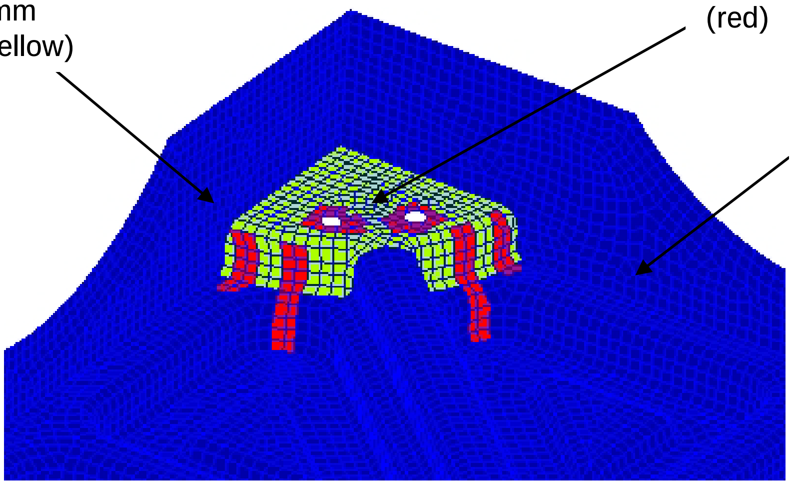
The following report details a design using 3D modelling techniques and finite element methods undertaken by the Technology Station (TS) to determine the shape (including ribs) that will allow the planter to undertake certain loads yet remain at a reasonable weight with a overall thickness of 2mm. Certain limitations were imposed on the design, yet a fair amount of leeway was given regarding the internal structure. The goal weight was between 850-900g.

2. Materials of Construction

For the purpose of this design analysis, the planter was modelled using solid elements of a homogenous material High Density Poly Ethylene (HDPE). The individual material and mechanical properties are typical values and are shown in Table 1.

Material	Density	Elastic Modulus E	Shear Modulus G	Yield Strength	Ultimate Tensile Strength
HDPE	955 kg/m^3	860 GPa	928 GPa	15 - 30 MPa	24 - 45 MPa

The thickness of the model was a constant 2mm with the exception of the mounting pads where the clip holes are, where an additional 1mm was added around the edge of the hole for added strength. Ribs were also added along the verticals to offer additional strength to reduce bending.



3. Boundary conditions

The following boundary conditions were used in the global analysis:

1. The planter was clamped at a total of 8 points, with 2 in each corner to represent the attachment clips as shown in Figure 3.1 below

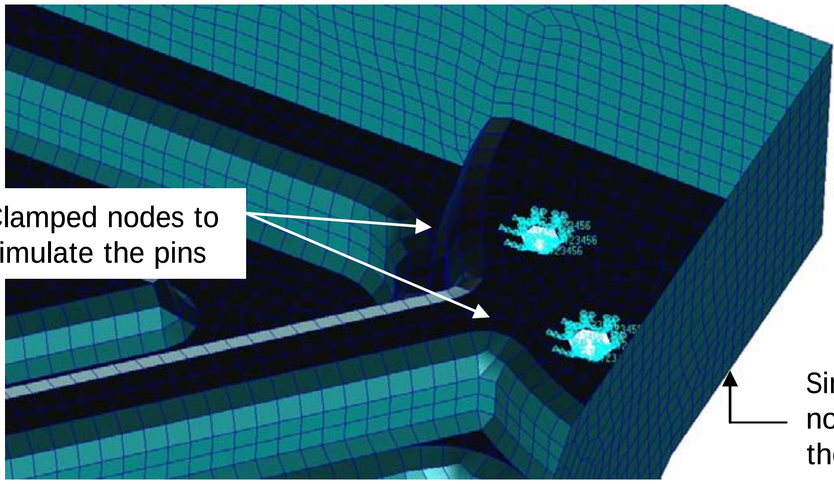


Figure 3.1: Boundary conditions of blade

4. Loads

A linear static analysis was performed on the structure with the following loads applied:

- Total load: an evenly distributed force was applied to the finite element model in the center as a worst case scenario. This load was to represent a human stepping on the planter. The final value was specified as 85kg (833.85N).

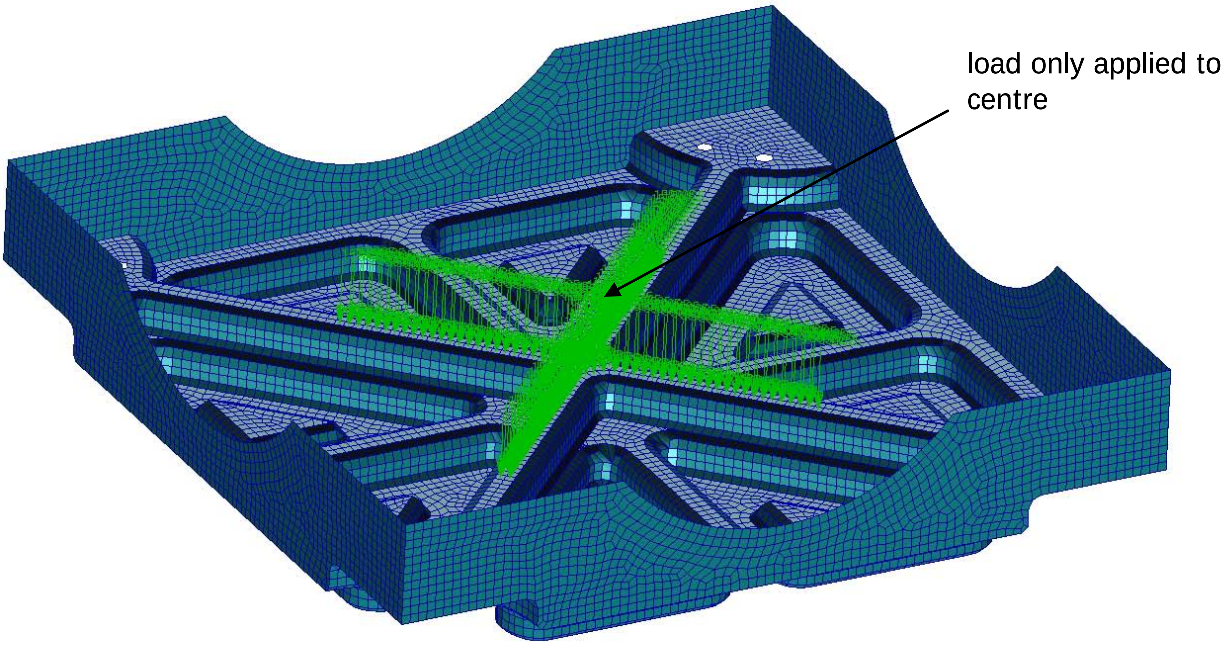


Figure 4.1: Human load (foot print)

- Total load: an evenly distributed force was applied to represent the planter filled with moist soil and was applied along all the horizontal faces in the planter. The final value was specified as 35kg (343.35N).

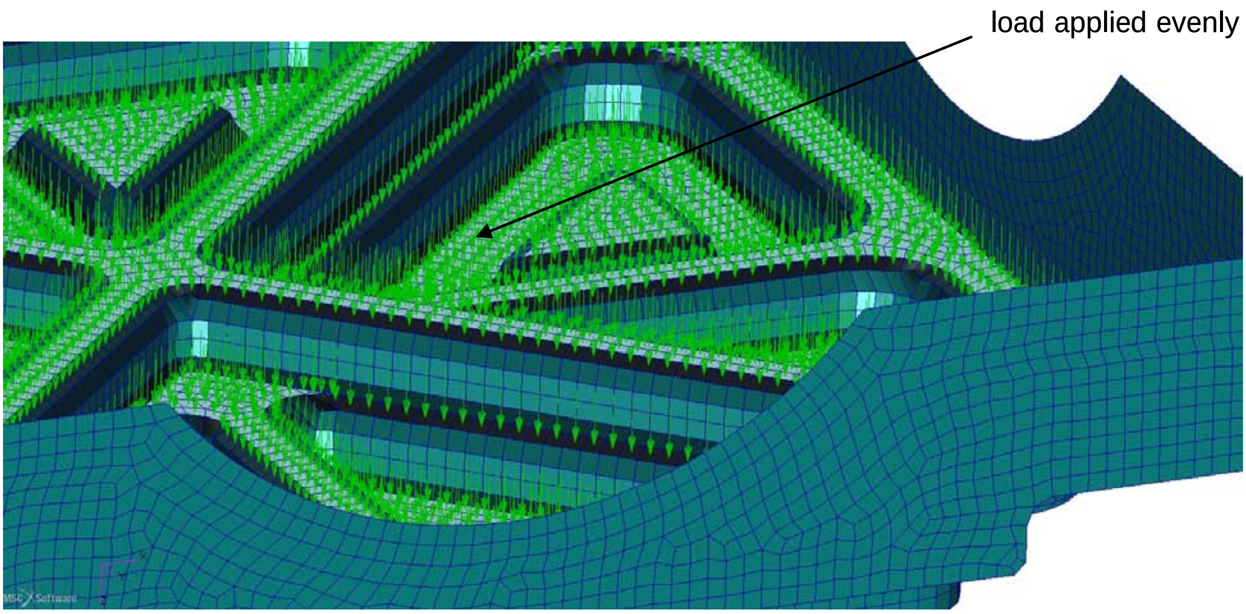


Figure 4.2: Sand load

5. Analysis/Design Philosophy

The full HDPE planter structure was analyzed using a linear static analysis. The design criterion was a stress based analysis with the yield strength as the limiting factor.

Assumptions

(1) The resulting strains in the finite element models are small enough to ensure an accurate representation by linear static analysis
(2) The effects of seismic loading have been excluded.
(3) There are no physical defects in the moulding process.
(4) The use of von Mises equivalent stress is an appropriate indication of the stress states

6. Results

A single load case was examined including both the sand and the human loading. The planter was supported on the flat pads on each corner and clamped around the clip holes. The maximum von Mises stress was situated in the region of the pads as this is where most of the bending takes place as it is near the supports. Figure 6.1 below illustrates the stress plot with the green areas the areas of probable high stress at 27.8MPa.

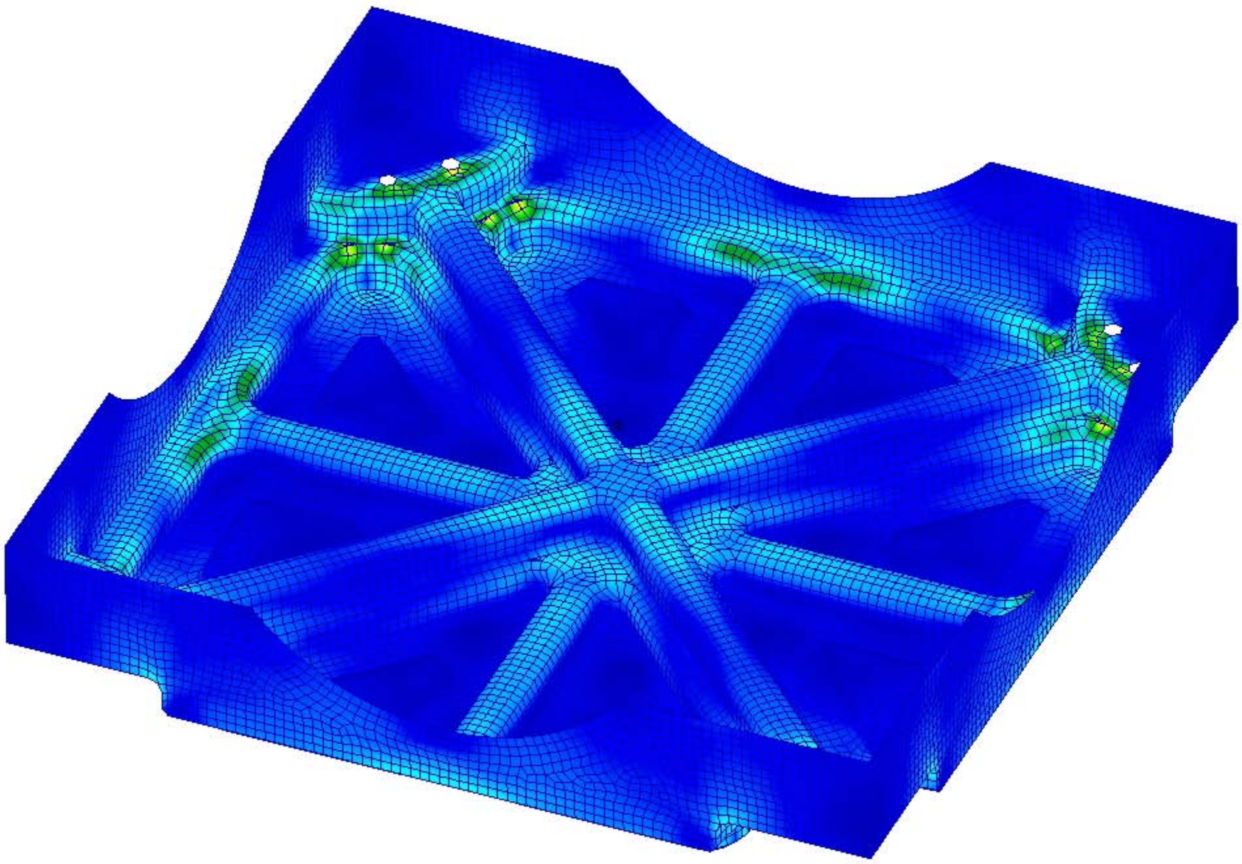


Figure 6.1: Maximum resultant von Mises plot

It should be noted that the model is a simple representation of the actual product and is merely used as a design aid. The distribution of the human load will be greater than used here as the soil will give way and buffer the load.

Figure 6.2 illustrates the maximum deflection as shown below. The deformation is negligible at only 0.027mm at the centre

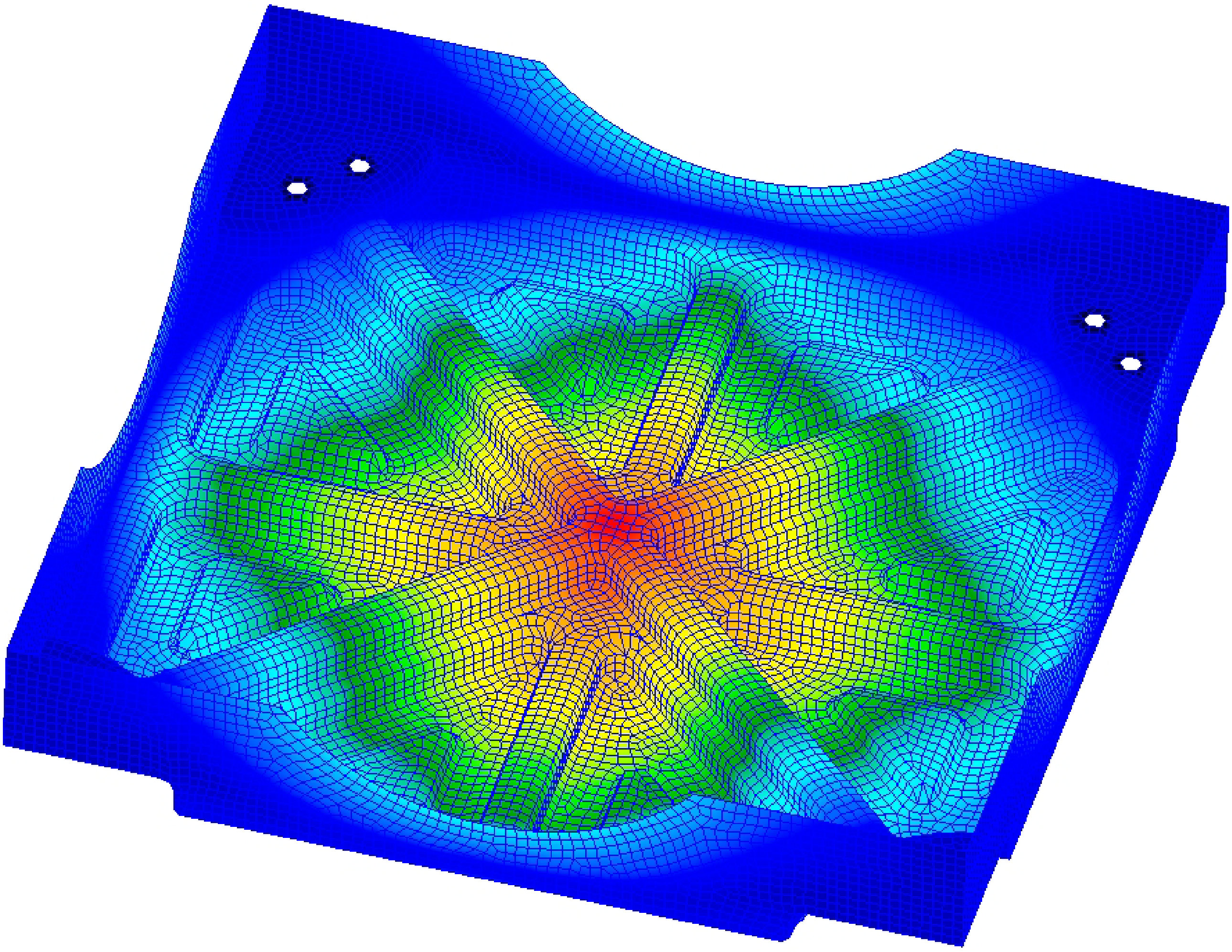


Figure 6.2: Maximum resultant deflection plot

7. Closure

A report detailing the design, assisted with FEA, of a planter box has been presented. After many attempts of the using the original design had failed, a new star rib arrangement was used. With an attempt to keep the weight to below 900g strategically placed channels were used as well as thickening appropriate areas of high stress. Ribs were used on the four corners to increase stiffness as shown below.

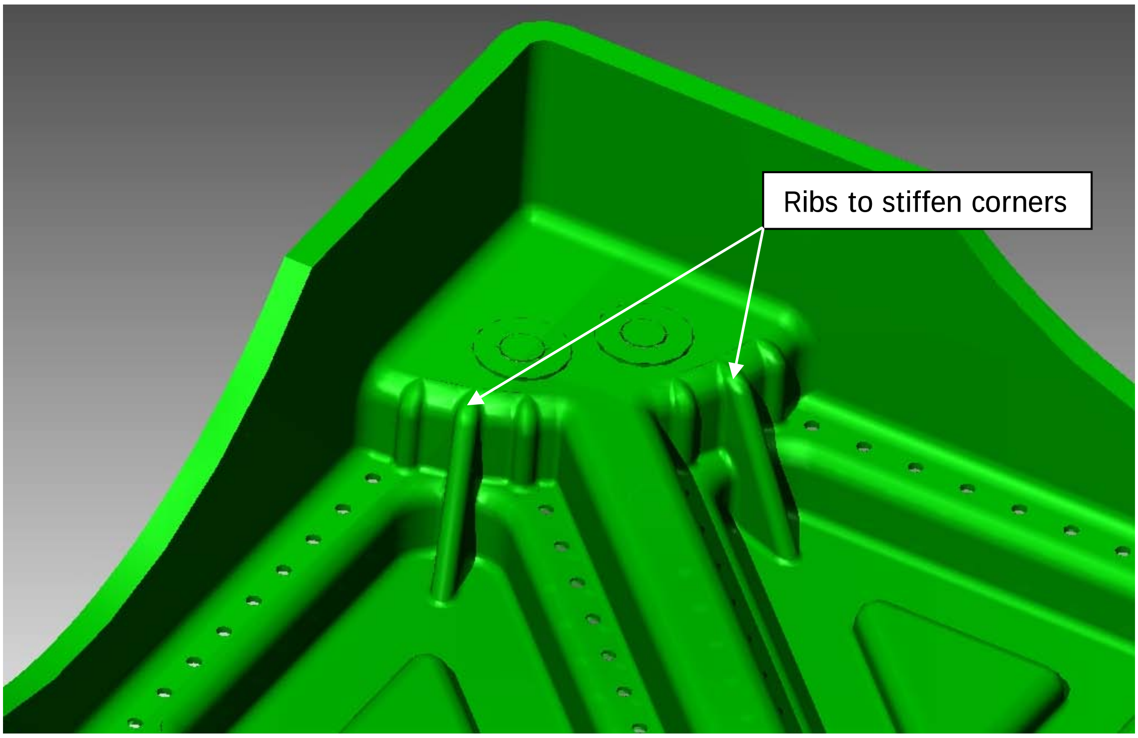


Figure 7.1: ribs in place to strengthen the corners

The eventual design meets the weight requirement as the current design weighs in at 870g and the resultant stresses are within the limits of the material with the addition of the ribs and thickened corner platforms.

Clients Signature of acceptance

____

Date: