International Academy of Science, Engineering and Technology
Connecting Researchers; Nurturing Innovations
IASET

APPLICATION OF MODELING AND SIMULATION
TOOLS FOR DESIGN OF PLASTIC PRODUCTS

D. L. Gunapala, O. Gunapala & S. Egodage

Research Scholar, Department Chemical and Process Engineering, University of Moratuwa, Sri Lanka

ABSTRACT

This paper is related to the application of finite element analysis to designing a plastic product particular a rim for unitary wheel used with Ground Service Support Equipment such as trolleys, container dollies, luggage trailers. The workwas carried out paying special attention to the optimization of the wheel structural architecture for load bearing capability.

Modeling and simulation work was carried out by considering thermoplastic material namely polyamide 6 filled with 30% short glass fibers. The selection of plastic material was based on several factors, including mechanical strength under static and dynamic load, resistance to prolonged action of elevated temperatures, cost and ability to be molded with conventional techniques.

A plastic model of the steel rim was developed based on general plastic product design standards. The model was simulated in order to identify areas of possible potential failure. After that model was optimized by changing and re-arranging its structural elements for further reduction of stresses in the identified potential failure areas. Final modification of the proposed model was carried out for product mold ability to ensure its manufacture with conventional molding technique. The finite element analysis showed that the stresses generated in the optimized model components were well below the actual yield stress of the plastic. The destructive load ofthe plastic wheel model estimated under static radial load condition with finite element analysis agreed well with the results obtained by physical testing the molded prototype samples under the same loading conditions. This fact affirmed suitability of the Solidwork design and Simulation packages for plastic part design. The application of this techniques lead to the reduction of prolonged product implementation period and significant cost saving due to reduced reproduction number of prototypes to bemade for evaluation of the product suitability, thereby making design successful and efficient.

KEYWORDS: *Rim for Unitary Wheel, Design, Simulation, Modeling, Validation*

INTRODUCTION

Plastics have been widely used in automotive industry to innovate safety, performance and fuel efficiency. Replacing metals with plastics is a viable option as light weighted vehicle requires less energy to maintain a constant speed and run more mileage out of a volume of fuel with the minimum emissions to the environment.

Nowadays plastic wheels made out of high performance thermoplastic based composites are available for highest level trim vehicles and advantages they offer are equivalent to the price being quoted. The high cost is associated with mainly with design, materials, crash analysis, and prolonged implementation procedure.

In recent years, the design and implementation procedures have been improved and simplified by a variety of experimental and analytical methods developed for structural analysis as a strain gauge, finite element methods. Some of them like durability analysis or fatigue life prediction and reliability method for dealing with variations inherent in engineering structure has been applied to the automotive wheel. Several studies have been carried out in these directions [1-4]. Observations indicated that deformation induced in wheels of typical vehicles and mobile machineries was clearly three-dimensional and simulation of the 3D model made it possible to determine dynamic cornering fatigue[1], predict vehicle-induced destruction on the land [2], optimize wheel mass, reduce rolling resistance, predict fatigue life[3,4].

Deformational behavior of plastic wheel is complex, as failure could occur in micro modes, such as matrix cracking, de-lamination, uneven fiber breakage, and others, that are not typical for traditional steel wheels. Most previous theoretical works were based on two dimensional formulations consisting of deformation only within the plane of wheel motion and only some of them were devoted to simulation of the 3 D models of the automobile wheels considering mainly some material related aspects [5],while this paper made an attempt to provide a reference for complete study of the design process starting from creating a 3D model, including structural design, material selection, simulation, optimization and completing with validation of the created model by means of mechanical tests physically performed on molded prototype samples.

The Major Objectives of this Research Were Related to:

- designing and arranging the structural elements of the wheel assembly,

- analyzing the stress and deflection levels generated in the designed model to predict its load bearing capability ,

- molding the prototype samples and testing

- validation of the created model by comparing load bearing capability of the designed model predicted with physical tests performed on molded prototype samples.

Experimental Part

- Design procedure included:

- Creation of the rim geometry as the CAD model

- Defining material properties

- Defining boundary limits for loading

- Meshing

The designing procedure was started with a simulation of the standard steel rim structure of 3.00-8 size. For this purpose, a 3D model of the steel rim was created and radial force of 6000 N was applied to the rim in the radial direction. The most loaded areas of high stress and deflection levels were identified. Then the rim structure was redesigned for plastic one by introducing structural reinforcing elements to the identified week areas and the 3D plastic model was simulated.

Based on simulation results, the model was re-designed again by changing structural elements shapes and their arrangements in order to reinforce week areas of the potential failure. This procedure was repeated several times until stresses and deflections were brought down well below the yield stress of the plastic with acceptable safety factor.

Bolt holes on the navering around axel hole were the points in where the rim was affixed to axle hub face. So they were selected as zero displacement fixtures in simulation studies (Figure 1).

Figure 1

The Mesh Refinement Techniques was used in a simulation study. To have a good result and accuracy the fine meshing was used. The mesh size selection was started from 15mm and was reduced to 5mm gradually to cover all small elements of the model (table 1). Meshed model is shown in figure 2.
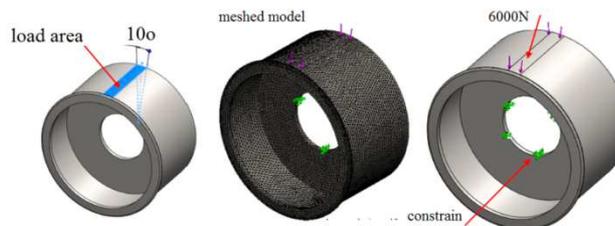

Figure 2

Table 1

Study No	Description	Applied Static Force (N)	Mesh Size (mm)	Stress (MPa)	Displacement (mm)
1	Steel Rim of Original Design	6,000	10	32	0.018
2		6,000	8	31.6	0.016
3		6,000	6	32.6	0.017
4		6,000	5	33.2	0.019

As stresses generated in the model were depended on the rim surface area to which radial force was applied, simulation was done for four cases in where radial force was applied on the rim surface within the sector angle of 5°, 50°, 100°, and 150° (Figure 3).

When the load was concentrated within a small area, stresses and deflection in the structure were at the high level and there was a significant drop in the stress level when contact angle increased from 10° to 150°. Based on this, the load on the wheels was applied within 10o contact angle area for all simulation experiments (Figure 4).
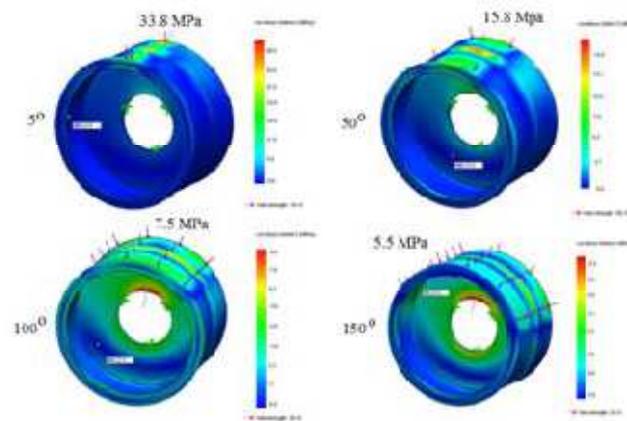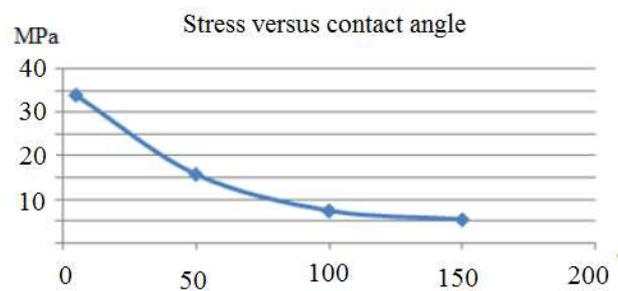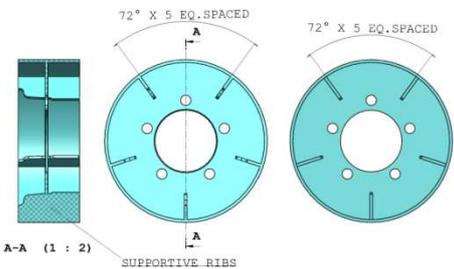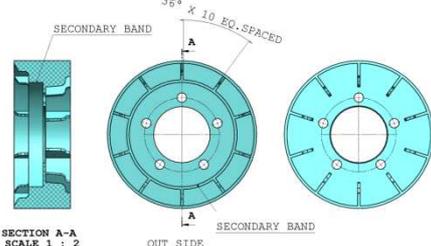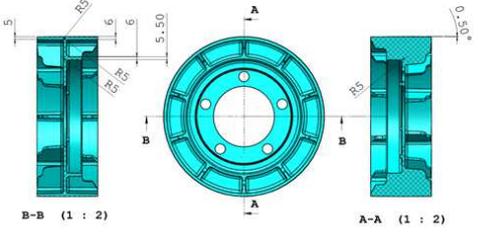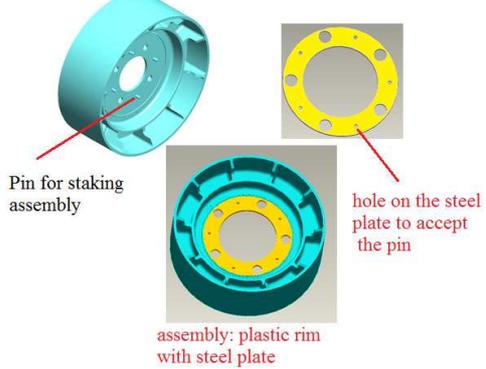
Figure 3

Figure 4

Prototype samples of the designed rim model were injection molded out of polyamide 6 filled 30% short glass fibers. Bench test was used to evaluate breaking force and deflection of molded samples. For this purpose, a plastic rim was fixed to the axel that was fitted to test bench. The initial force of 6000N was applied in the radial direction and then it was gradually increased until break occurred. Force value versus deflection was recorded.

RESULTS AND DISCUSSIONS

In order to convert the steel rim into plastic one the following changes were made to standard steel rim design (Figure 5):

- Most outer rim diameters was increased

- Hard base rubber layer performing a smooth transition function between soft rubber tread and hard steel rim was eliminated as its function was done by plastic.

- Rim width remained unchanged

- Rim band thicknessof 5 mm was maintained the samefor all parts of the plastic rim in order to keep uniformity for efficient and trouble-free injection molding.

Table 2: Design Comparison

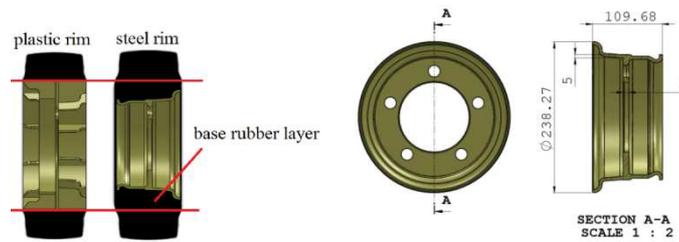| No	Model	Description
1		Five equally spaced ribs were arranged on both sides of the central plate in order to improve the stiffness of the structure and reduce deflection under load
2		The number of reinforcing ribs was increased from 5 to 10 keeping equally spaced distance corresponding to 36 ° angle. A secondary band was introduced to the center plate on the side opposite to the side facing vehicle axel.
3		Drafts of 0.5 ° were introduced to the outer and inner band rim faces,to each rib on the outer band and secondary band. Fillets were made between ribs andouter band, secondary band and central plate to avoid sharp edges.
4		A steel plate of 4 mm thick was placed to the central part of the rim on the bolt fixing side opposite the vehicle axel hub. To attached steel plate to rim by staking assembly, five pins were made on the rim surface and five holes of the same diameter were made on the steel plate for locking.

Figure 5

In order to improve the structural rigidity of the plastic model, five ribs were placed on the nave ring perpendicular to the inner surface of the band (model No1, table2).Simulation showed, that stress of 51.3MPa was found in the band area at edges and mid part, this value was half of the yield strength of the plastic (Fig.6, table 2). To increase the strength of the rim at the edges and mid part, the proposed model No1 was modified as described in table No2, model No2. As per simulation data, the stresses in the rim of newly created model No2 were dropped down and the maximum stress of 24.4 MPa was observed at the mid part of the outer band, while the edges were almost free of stresses(Fig.7, table 3).
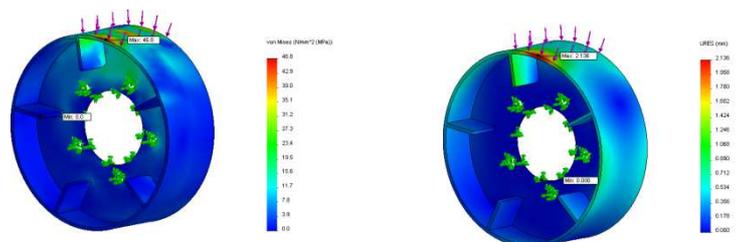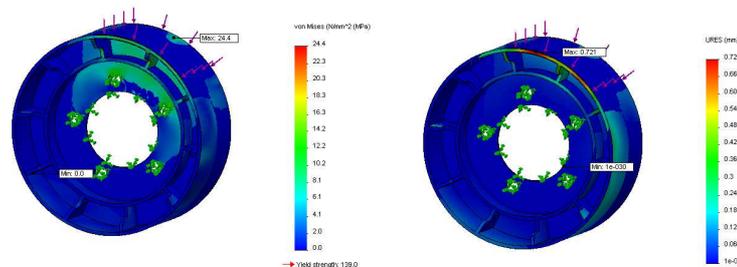

Figure 6

Figure 7

Further changes made to the model were related to model mold-ability (model No3, table 1)Smoothing sharp edges and corners would make it possible to use conventional injection molding to produce plastic rim. The simulation study showed that outer and inner bands of the rim, including all ribs, experienced low-stress level, while the major load was shifted to the area around bolt holes /fixtures on the nave ring. This area was stressed and simulation showed 54.6 MPa (Fig. 8, table3). This could initiate cracks in case of excessive torque applied to the rim when bolting to the axel hub. It was suggested to re in force bolt holes area with steel plate as described in model No4 table2. Addition of steel plate to nave ring to reinforce the bolt hole area resulted in stresses dropped down to 26.7MPa, that was sufficient for the required application (Fig.9, table 3).

Analyzing simulation data the breaking force was estimated asa product of safety factor and applied static force in the final model:

$$\text{Breaking force} = 7.03 \times 6000 = 42180N$$

Breaking force obtained by physical testing of prototype samples which were injection molded was determined and given in Figure 10. As it could be seen from the graph increasing in radial force applied to rim caused growth in deflection. Break occurred when force applied was about 42000N. Obtained value agreed well with results derived from the analysis of simulation data.
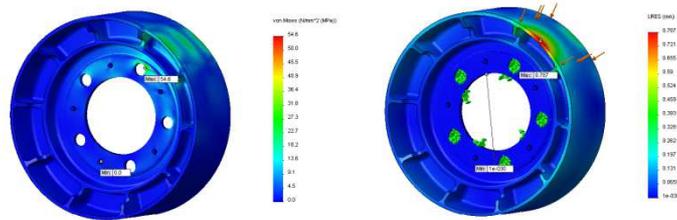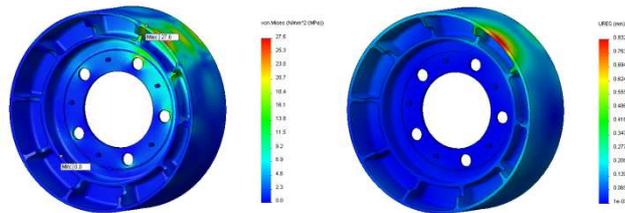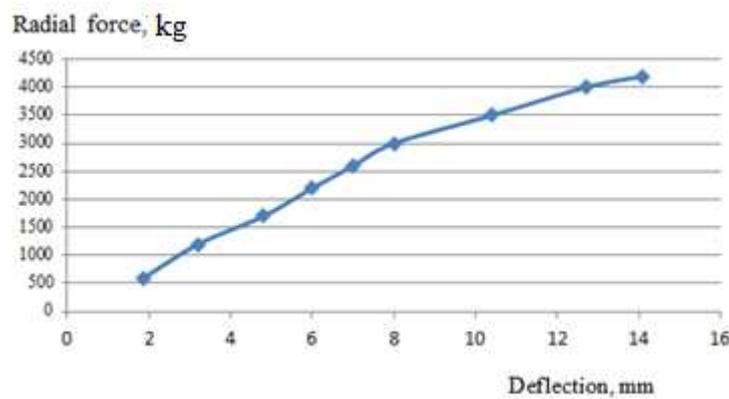

Figure 8

Figure 9

Figure 10

Table 3

Study No	Applied Static Force (N)	Mesh Size (mm)	Stress (MPa)	Displacement (mm)	Safety factor
1	6,000	5	46.8	2.136	3.44
2	6,000	5	24.4	0.721	5.5
3	6,000	5	54.6*	0.767	6.6
4	6,000	5	27.6	0.832	7.03

CONCLUSIONS

After completion of the study, the following conclusion can be made:

- Based on the staticloading analysisofthe 3D model,the plastic rim structure for the unitary wheel was designed.

- It comprised a subtracted solid body of the rim portion with a center bore configured to receive axle hub, an inner band, a circumferentially extended outer band that margined the rim portion, nave ring that extended outwardly and radially of the center bore with a plurality of holes circularly positioned and configured to receive bolts, a plurality of ribs extending outwardly and radially at right angles from the nave ring up to the outer band positioned on both sides of the rim and configured to improve flexural rigidity of the structure.

- Excessive stresses in the bolt hole area on the nave ringwere eliminated with steel plateplaced around bolt hole area and fixed to the rim by means of stacking assembly

- The destructive radially applied load on the plastic rim model estimated understatic radial load condition with finite element analysis agreed well with the results obtained by physical testing the molded prototype samples under the same loading conditions.

- Validation of the designed model affirmed suitability of the Solidwork design and Simulation packages for plastic part design. The application of this techniques lead to the reduction of prolonged product implementation period and significant cost saving due toreduced reproduction number of prototypes to bemade for evaluation of product suitability, thereby making design successful and efficient.

REFERENCES

1. Hambleton J.P. "On modeling a rolling wheel in the presence of plastic deformation as a three- or two-dimensional process". Department of Civil Engineering, University of Minnesota, USA,International Journal of Mechanical science, September 2009.

2. Dresher A. "Modeling wheel-induced rutting in soils: Rolling"Journal of Terra-mechanics 46(2):35-47, April2009.

3. Sourav Das," Design and Weight Optimization of Aluminium Alloy Wheel" International Journal of Scientific and Research Publications, Volume 4, Issue 6, June 2014.

4. M. Rajeshkumar, Finite Element Analysis of Composite Material for Automotive Wheel Rim, International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), Volume 8, Issue 2, March-April 2018, pp. 1349-1356

5. Taek-Young Kim, Ho-Kyung Kim, "Three-dimensional elastic-plastic finite element analysis for wheel-rail rolling contact fatigue" International Journal of Engineering and Technology, June 2014.'

6. Harish Panjagala, "Design & Weight Optimization of a Wheel Rim for Sport Utility Vehicle".MATEC Web of Conferences 172, 03006,2018.