SIMULATION TEST OF AUTOMOTIVE ALLOY WHEEL USING COMPUTER AIDED ENGINEERING SOFTWARE

MOHD IZZAT FALIQFARHAN BIN BAHAROM

UNIVERSITI MALAYSIA PAHANG

I hereby declare that I have read this thesis and in my opinion this thesis is sufficient in term of scope and quality for the award of the degree of Bachelor of Mechanical Engineering.

Signature	: ……………………………………
Supervisor	: Mohd Rashidi Bin Maarof
Date	: …………………………………...

Signature	: ……………………………………
Panel	: ……………………………………
Date	: …………………………………...

SIMULATION TEST OF AUTOMOTIVE ALLOY WHEEL USING COMPUTER AIDED ENGINEERING SOFTWARE

MOHD IZZAT FALIQFARHAN BIN BAHAROM

A report submitted in partial fulfillment
of the requirements for the award
of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
University Malaysia Pahang

OCTOBER 2008

ABSTRACT

This paper will present and discuss the use of finite element technique for simulation test of functionality of automotive allow wheel nature during operation. Computer Aided Engineering Software is used to simulate the wheel trough loading method. Several common design of wheel that is been used in the automotive industry is used. The load for stress distribution and displacement data subjected to loading on alloy wheel is viable trough the finite element model. The result shows critical point takes places at certain point of the alloy wheel component

ABSTRAK

Projek ini akan menerangkan dan membincangkan penggunaan teknik finit elemen untuk ujian simulasi bagi penggunaan roda aluminum semasa operasi. Software komputer bantuan kejuruteraan digunalan untuk simulasi melalui kaedah beban. Rekaan spesifik bagi roda aluminum yang biasa digunakan dalam industri automotif digunakan untuk projek ini. Beban untuk terikan dan sesaran apabila dikenakan beban pada roda aluminum ditunjukkan melalui model teknik elemen finit. Keputusan menunjukkan lokasi kritikal yang berlaku di poin tertentu di komponen roda aluminum

STUDENT'S DECLARATION

I hereby declare that the work in this report is my own except for quotations and summaries which have been duly acknowledged. The report has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature :
Name : MOHD IZZAT FALIQFARHAN BIN BAHAROM
ID Number : MH05065
Date : 6 NOVEMBER 2008

ACKNOWLEDGEMENT

Alhamdulillah,

I would like to take this opportunity to express my sincere gratitude and appreciation especially to my supervisor, Mr Mohd Rashidi Maarof for his constant guidance, invaluable knowledge, and constructive idea in leading me to accomplish this project.

I also would like to express my sincere appreciation to all my friends, lecturers, faculty member and others whose comment and help most in preparing this project. Last but not least my parent for giving me full support and encouragement for me to do my best in this project. Thank you

To my beloved mother and father

Mr. Baharom Bin Abdul Ghani

Mrs. Norma Bt Taib

And my brother and sisters

ABSTRACT

This paper will present and discuss the use of finite element technique for simulation test of functionality of automotive allow wheel nature during operation. A specific design wheel is used. The load for stress distribution and displacement data subjected to loading on alloy wheel is viable trough the finite element model. The result and how wheel performed is discussed.

ABSTRAK

Projek ini akan menerangkan dan membincangkan penggunaan teknik finit elemen untuk ujian simulasi bagi penggunaan roda aluminum semasa operasi. Rekaan spesifik bagi roda aluminum digunakan untuk projek ini. Beban untuk terikan dan sesaran apabila dikenakan beban pada roda aluminum ditunjukkan melalui model teknik elemen finit. Keputusan dan bagaimana roda itu dijalankan di bincangkan dalam projek ini.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

CAD	Computational Added Design
3D	Three Dimensional
kPa	Kilo Pascal
mm	Millimeter
kN	Kilo Newton
MPa	Mega Pascal
W	Work
Rb	Bead radius

CHAPTER 1

INTRODUCTION

1.1 Introduction

The wheel is a device that enables efficient movement of an object across a surface where there is a force pressing the object to the surface. Early wheels were simple wooden disks with a hole for the axle. Because of the structure of wood a horizontal slice of a trunk is not suitable, as it does not have the structural strength to support weight without collapsing; rounded pieces of longitudinal boards are required. The spoke wheel was invented more recently, and allowed the construction of lighter and swifter vehicles. Alloy wheels are automobile wheels which are made from an alloy of aluminum or magnesium metals (or sometimes a mixture of both). Alloy wheels differ from normal steel wheels because of their lighter weight, which improves the steering and the speed of the car, however some alloy wheels are heavier than the equivalent size steel wheel. Alloy wheels are also better heat conductors than steel wheels, improving heat dissipation from the brakes, which reduces the chance of brake failure in more demanding driving conditions. Over the years, achieving success in mechanical design has been made possible only after years of experience coupled with rigorous field-testing. Recently the procedures have significantly improved with the emergence of innovative method on experimental and analytical analysis. Alloy wheels intended for normal use on passenger cars have to pass three tests before going into production: the dynamic cornering fatigue test, the dynamic radial fatigue test, and the impact test. Many alloy wheels manufacturing company had done numerous amount of testing of their product but their method on simulation test on alloy wheel information often kept limited.

Historically, successful designs was arrived after years of experience well aided worth extensive field -testing. Since the 1970's several innovative methods of testing and experimental stress measurements have been initiated. In more recent years, the procedures have significantly improved by the emergence of a variety of experimental and analytical methods for structural analysis. Durability analysis, that is: fatigue life prediction and reliability methods, for dealing with various inherent in engineering structures has been used for the study of automotive rims.

In its basic form a wheel is a transfer element between the tire and the vehicle. The function of a wheel can be further broken up into simple verb-noun combinations to describe its purpose. A wheel;

i. ♦ Transfers torque, (braking and acceleration)
ii. ♦ Support mass, (support the mass of the motor vehicle)
iii. ♦ Adds mass, (damped mass for driving comfort)
iv. ♦ Dissipates heat, (from braking)
v. ♦ Adds value, (aesthetically pleasing)
vi. ♦ Absorbs impact, (road hazards)
vii. ♦ Conserves energy, (potential energy in momentum)

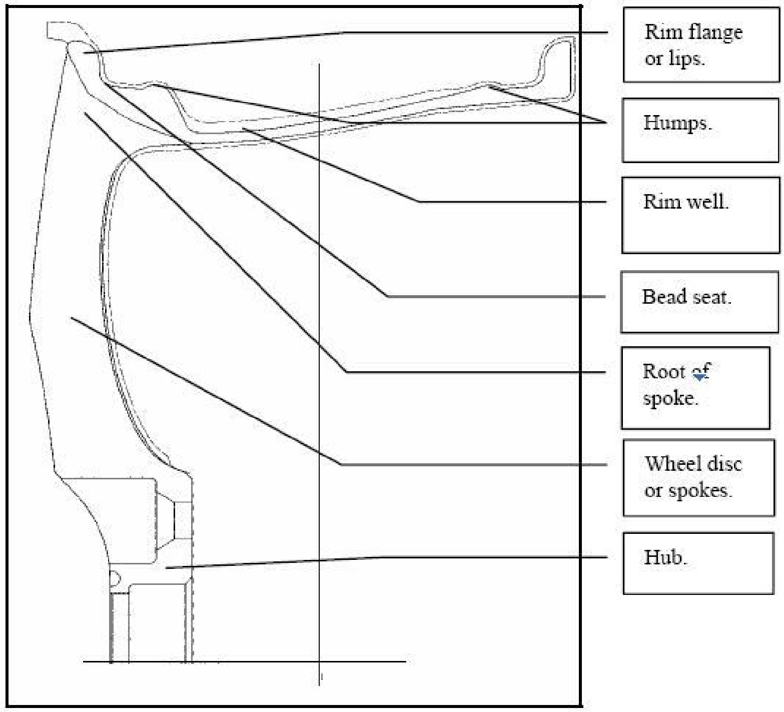


Figure 1: Wheel Assembly

i. **Hub**

This forms the interface between the hub of the vehicle's drive train and the wheel. This part will have the bolt holes to facilitate in the fastening of the wheel to the vehicle.

ii. **Wheel disc or spokes**

This is the part of the wheel that transfers all of the loads.

iii. **Rim flange or lips**

These lips retain the tire and also act as a strength-rib to protect the wheel against impact loads.

iv. **Bead seat**

The tire bead will rest against this area.

v. Humps

These humps will ensure that the tire stays against the rim flange in the deflated condition.

vi. Rim well

This area facilitates in the assembling and disassembling of the tire.

1.2 Problem Definition

This project will be focus on simulating the nature of alloy wheel during operation on road. A range of variables and parameter will be accounted in the simulation. These testing methods provide information on the stress analysis of the alloy wheel in different situation when loads are acted on different design of alloy wheels. Stress occur on the rim under radial load determine the performance characteristic of an alloy wheel for structural integrity. A wheel should maintain structural integrity without any cracks or plastic deformation. Under a radial load, the strength of the rim usually determines the fatigue life of a wheel, so the stress evaluation is mainly focused on the rim.

1.3 Objectives

Objectives of the project are:

i. To simulate the stress analysis of alloy wheel using computer aided engineering software
ii. To studies the stress analysis distribution occurs of an alloy wheel when static force is acted on.

1.4 Scopes

The scopes of the project are:-

i. Design of the automotive alloy wheel using CAD software product is generally used in automotive industry (14, 15, 16 inch alloy wheel, 6 inch wide, 5~8 spokes)
ii. Simulation data collection using FEA software
iii. Analysis of stress analysis of alloy wheel

CHAPTER 2

LITERATURE REVIEW

2.1 Loading Methods on Automotive Wheel.

Presented here are several methodologies for modeling the effect of the vehicle weight as it is transferred to the rim. Methods explained are referenced in the published literature and analogies taken from thick ring theory in stress analysis in the development of loadings on links and eyebars. As stated, the modeling of the tire on the rim is extremely complex and involves both non linear analysis (tire potion) coupled with linear static stress analysis (rim). The interface of the two components would be achieved by the use of gap or contact elements. This work has been done at The Goodyear Tire and Rubber Company. Results indicate that the loading shape is in the form of a cosine function about a central angle of about 30° from either side of the point of contact with the ground.

Others assumed this 30° angle is developed from the contact patch geometry of the tire. Meaning that when the tire is loaded there is flat at the point of contact with the ground, this is the so called contact patch, and the length of this patch is then converted to an angle swept by the bead seat area in contact with the rim. The other bead seat areas tend to be pulled away from the rim, so no loading is present in the areas or is considered negligible.

Finally, a method proposed assumes the area in contact with the rim spans half of the tire or 90° symmterical about the point of loading. The loading method is similar to a cylindrical bar in a clevis, assuming no gap exists.

The total weight of a car is balanced with a vertical reaction force from the road through the tire. This load constantly compresses the wheel radially. While the car is running, the radial load becomes a cyclic load with the rotation of the wheel. Hence, the evaluation of wheel fatigue strength under radial load is an important performance characteristic for structural integrity. According to the SAE, a wheel should maintain structural integrity without any cracks or plastic deformation for more than 4×10^9 rotations under a radial load, Q, expressed by the following equation.

$$Q\ S_r * W \ldots\ldots\ldots\ldots\ldots = (1)$$

where, S_r means acceleration test factor ($S_r = 2.2$) and W means maximum tire load. For this application W=4316 N, thus Q = 9496 N.

Under a radial load, the strength of the rim usually determines the fatigue life of a wheel, so the stress evaluation is mainly focused on the rim. In this analysis also, the contact condition between the discs spoke flange and the rim well is assumed to be tightly closed, and the contact area is modeled by one element with the summed thickness of the disc and the rim.

In an actual wheel, since a radial load is applied to the wheel on the bead seats with the tire, the distributed pressure is loaded directly on the bead seats of the model in this analysis. The pressure is assumed to have a cosine function distribution mode within a central angle of 40° in a circumferential direction as shown in Figure 3. The 40° was chosen based on previous literature, in which some determined this angle by strain gauge experiments or some researchers assuming the angle from the contact patch area. Using the idea of the contact patch width corresponds to the area of contact over the bead seats as shown in Figure 2.

This may not be valid for the run flat tire due to its stiff side walls, which would reduce the contact patch area. According to the manufacturer, a run flat tire deflects about 12 mm for every 4448 newtons of force; The P22560R16 tire under the load of 10.45 kN, the tire should deflect 29.71 mm. This is in good agreement with the experimental tests that predicted 8.9 mm at 3113 N load. For a corresponding height of 29.71 mm and based on a tire diameter of 663 mm, the angle ,〈, swept out is 48 degrees. The calculations were based on segments of circle geometry. This analysis however did not prove to be entirely accurate and was later found to be almost 90°. This can be rationalized, because of the additional strength needed in the bead seat area of the tire. By using the cosine function accordingly, the distributed pressure, W_r, is given by the following equation:

$$W_r = W_0 * \cos\left(\frac{\pi}{2} * \frac{\theta}{\theta_0}\right) \text{........}(2)$$

The total radial load W is calculated by using Eq. (2) as follows.

$$W = b \int_{-\theta_0}^{\theta_0} W_r * r_b \, d\theta \text{........}(3)$$

$$W = b\int_{-\theta 0}^{\theta 0} W_0 * r_b * \cos\left(\frac{\pi}{2\theta_0} * \theta\right) d\theta$$

$$W = b * W_0 * r_b * \frac{1}{\left[\frac{\pi}{2\theta_0}\right]} * \sin\left(\frac{\pi}{2} * \frac{\theta}{\theta_0}\right)\Bigg|_{-\theta 0}^{\theta 0}$$

$$W = 4 * b * r_b * \theta_0 * \frac{W_0}{\pi} \text{ ....(4)}$$

or

$$W_0 = \frac{W * \pi}{b * r_b * 4 * \theta_0}$$

b=19.8 mm, W=10.4 kN , r_b=7202 mm. Applying these yields W_0 = 2045 kPa. where, r_b is the radius of the bead seats and b means the total width of the bead seats. In this analysis, the total radial load W = 9496 N is applied to the model, and the magnitude of the load is the same as applied to the actual wheel in the stress measurement experiment.

In this stress measurement experiment, the wheel is assembled with a Goodyear Eagle aqua steel tire (P22560R16) which is inflated to a pressure of 241 kPa and pressed against a flat plate with a load of 3113 N. Strain gages are attached to the wheel in circumferential direction. The central angle 0° of the pressure distribution is 80 degree.

Pressure is applied to two bead seats on the inboard side and the outboard side. In detail, half of the pressure on the inboard side is applied to the inboard rim flange, and the other half is applied to the inboard bead seat. The reason for this is that the inboard rim flange deflects easily due to the long inboard rim leg, and so it is susceptible to 10 loading from the tire. The loading condition described above is determined from a comparison between the measured stress of the rim and the calculated stress under some tentative loading conditions.

The optimum dividing ratio of the applied load on the bead seat versus the applied load on the rim flange is thought to vary according to the contact condition between the tire and the rim. This, in turn, is affected by such factors as tire type (bias or radial), the tire air pressure, the reinforcement structure of the tire and the type of the rim [2]. In actual modeling the locations along the circumference were identified as element numbers (node numbers) and corresponding angle.

Previous experiment results on this subject have revealed that the actual load on the tire–rim unit takes the form of a cosine function having a central angle of about 40◦ measured from either side of the point of contact with the ground (Figure 2)[1-3]. Subsequent research efforts have confirmed the 40◦ angle and attributed it to contact patch geometry of the tire [4]. This essentially implied that when the tire is fully inflated there occurs a flat at the point of contact with the ground. This is referred to as the contact patch. The length of the contact patch is converted to an angle swept by the “bead seat” area in contact with the rim. Other bead seat areas tend to be pulled away from the rim, and no loading is assumed to be present in these areas. The method used in this study assumes that the area in contact with the rim spans half of the tire, i.e., 90◦ symmetrical about the point of loading. The loading method is quite similar to a cylindrical bar in a clevis and assuming the lack of a gap. This method is referred to as the eyebar loading method [5]. This stress and displacement of the experiment only limited to the bead seat of alloy wheel.

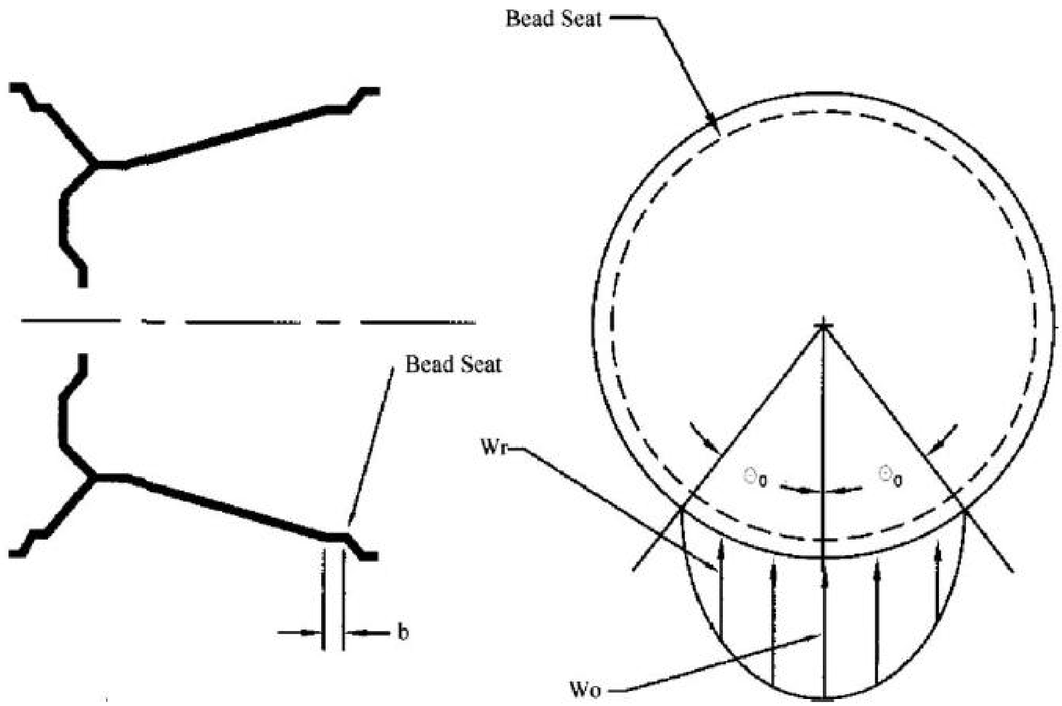


Figure 2.1: Radial load on Wheel

2.2 Material properties and manufacturing process for cast aluminum wheels

One of the leading aluminum alloys for wheels in use today is AlSi7Mg, with the chemical composition shown in Table 2.

Si	Fe	Cu	Mn	Mg	Zn	Ti
6.5–7.5	0.15	0.03	0.10	0.3–0.45	0.07	0.10–0.18

Table 2.1: Chemical composition of AlSi7Mg (in %)

The manufacturing of aluminum passenger car wheels is made up of low pressure die casting. The molten aluminum is kept in a gas tight heat insulated container from where it flows under mild pressure of approximately 70–100 kPa via a standpipe to escape through vent-holes, the molten aluminum enters the die without turbulence [8]. After solidification of the material in the die, the container is depressurized and the molten contents of the standpipe flow back into the container. The wheel is then sent for machining.

Following are the monotonic material data which were obtained from specimens taken from finished wheels. Table 2 Mechanical properties for Aluminum A356-T6:

Ultimate tensile strength (S_u)	250 MPa.
Yield strength (S_y)	230 MPa.
Elongation (e)	5%.
Hardness (HB)	90.

Table 2.2: Mechanical properties for Aluminum A356-T6

CHAPTER 3

METHODOLOGY

3.1 Introduction

Methodology is underlying principles and rules that govern a system while method can be defined as systematic procedure for a set of activities. Methodology start with concept design is several of idea and concept to produce the best results. Several concepts will be evaluated for their design and fabricate, then comparing them to get the best design. This chapter also discussed about software involved in producing the design.

3.2 Project Methodology

i. Literature review
ii. Measuring alloy wheel parameter
iii. Drawing using CAD software
iv. Simulation using CAE software
v. Analysis

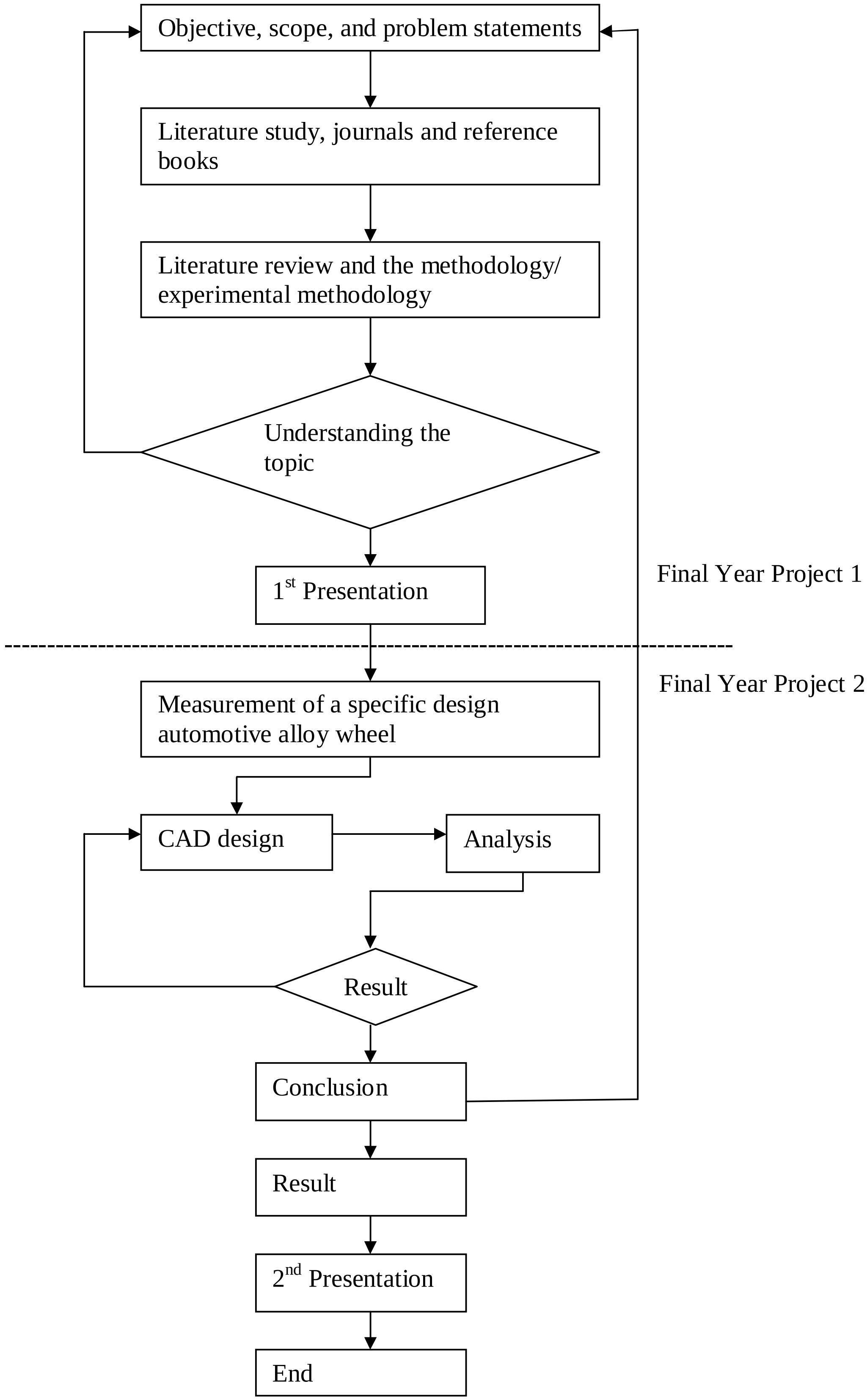


Figure 3.1: Project development activities phase

3.4 Determining the Alloy Wheel Design and Specification

Firstly, determination for the wheel specification is done to ensure the physical parameter and dimension.. Commonly used design in the automotive industry is used for this simulation test. The design and specifications of the alloy wheel is stated in table 3

Figure 3.2: Actual 7 Spokes Alloy Wheel Design **Figure 3.3**: Actual 6 Spokes Alloy Wheel Design

Table 3.1: Specifications of Alloy Wheel for Simulation Test

Parameter	Size/ Feature
Diameter	16 inch
Width	6 inch
Offset	38mm
Bead Seat	20mm
Pitch Center Diameter	100mm
Spoke size	40mm/50mm/60mm
Number of spoke	0/5/6/7/8.

3.5 Measuring and 3 dimensional design of alloy wheel

The following steps below are the phases that need to be followed in order to create 3 dimensional models for the Alloy wheel

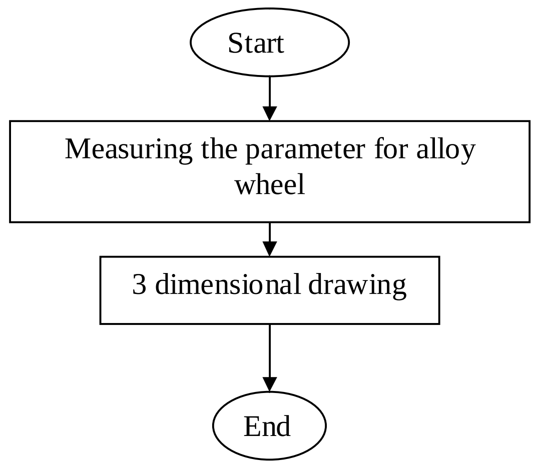


Figure 3.4: Steps to create 3D model for an alloy wheel

3.6 Stage 1: Measuring

A measurement process is needed to get an actual dimension on the alloy wheel parameter. Dimensions then is transferred to CAD software to make 3 dimensional design of the alloy wheel. Vanier caliper, protector, masking tape and ruler are used when taking the measurement of the alloy wheel.

Measuring is one of the important aspects in order to get precise and less tolerance results of an analysis operation. In measuring the alloy wheel there are several things that are important that is the diameter, bead seat size, spoke size, offset and the pitch center diameter. All these measurement need to be accurate and precise as possible to reduce tolerance. After finishing with the measurements, the next step is to draw in 3 dimensional using the suitable CAD software. The 3 dimensional design of the alloy wheel is done using the Solid Works

Figure 3.5: Alloy wheel bead seat design

Figure 3.6: Masking tape is used to determine the width of the bead seat of the alloy wheel

3.7 Stage 2: 3 dimensional drawing in Solid Works

The dimension and parameter of the alloy wheel is then transferred to Solid Works. Solid Works is a 3 dimensional computer aided design software (CAD). Solid works approach is to modeling and assembling part. The creation of the solid for alloy wheel design in 3 dimensional environments is typically begins with the definition of topology in 2 dimensional sketches. The dimension is then added to the topology to determine the length and sizes for the curves and locations for the vertices in conjunction with topological constraints

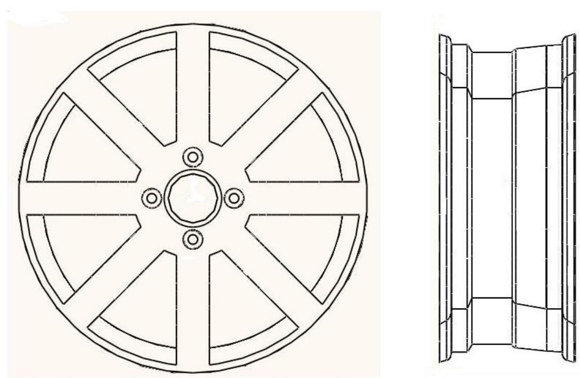


Figure 3.7: Front and side of 2 Dimensional designs for the 8 spokes alloy wheel

From the 2 dimensional designs the 3 dimensional designs can be made. The drawing of the design is made. 6 different number of spoke with 3 different spoke sizes is shown on figures below

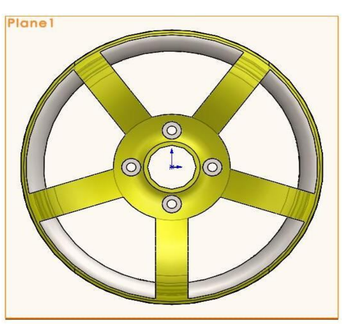


Figure 3.7.1 40 mm spokes

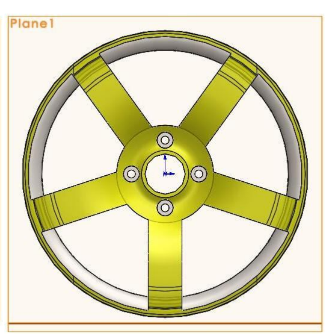


Figure 3.7.2 50 mm spokes

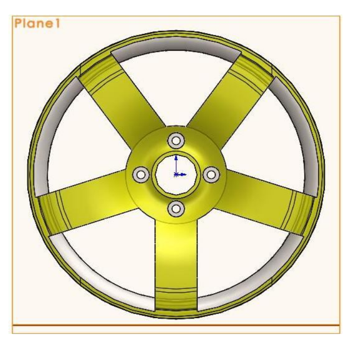


Figure 3.7.3 60 mm spokes

Figure 3.8: 5 spokes wheel

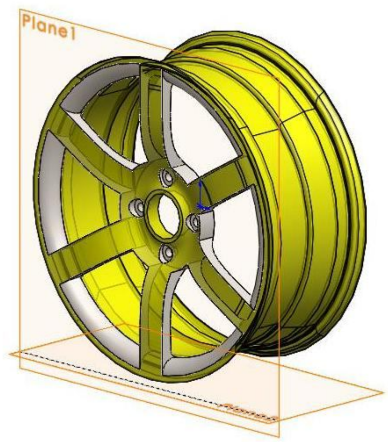


Figure 3.9: 6 spokes wheel

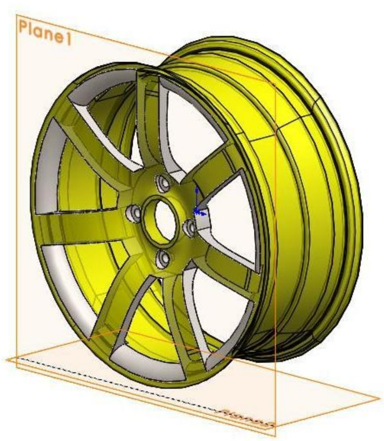


Figure 3.10: 7 spokes wheel

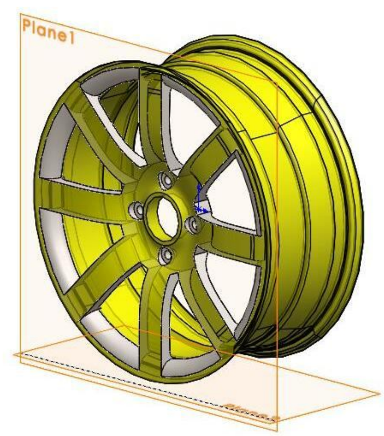


Figure 3.11: 8 spokes wheel

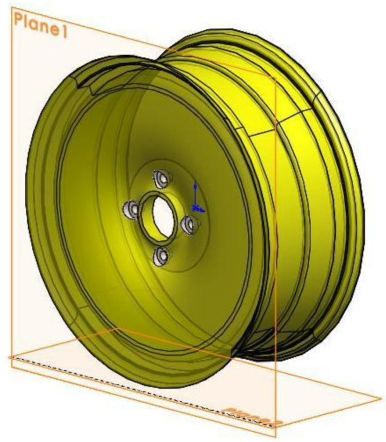


Figure 3.12.: No spoke wheel

3.8 Finite Element Analysis

The process of generation of finite element model to simulate the static force is described in the following sequel. Finite element modeling and analysis is carried out using a general purpose finite element software Algor Fempro V16

Algor Fempro V16 software is used for finite element analysis of the alloy wheel. Algor Fempro V16 is a tool for 2 dimensional and 3 dimensional finite element analyses (FEA) model creation. From Algor Fempro V16 software 2 dimensional and 3 dimensional structured meshes can be generate faster and more easily. The generated mesh can be modified and their meshing capability is an advantage to get more accurate result in the analysis.

The assembly files from Solid Works in IGES format is imported into the Algor Fempro V16 software. For the mechanical simulation model, the contact diagnostic probes identify areas are automatically display

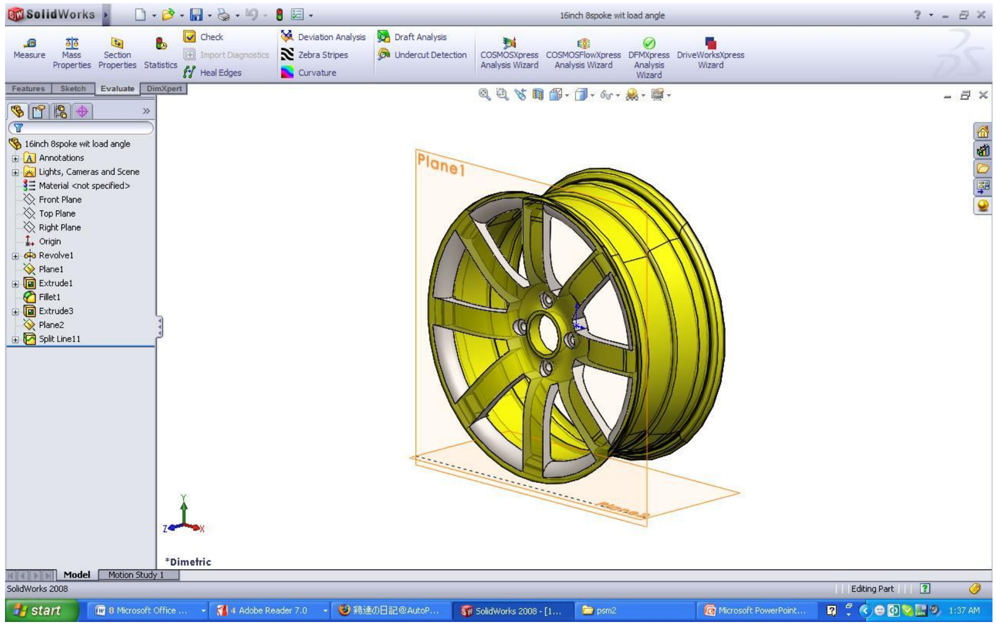


Figure 3.13: Alloy wheel design in Solid Works

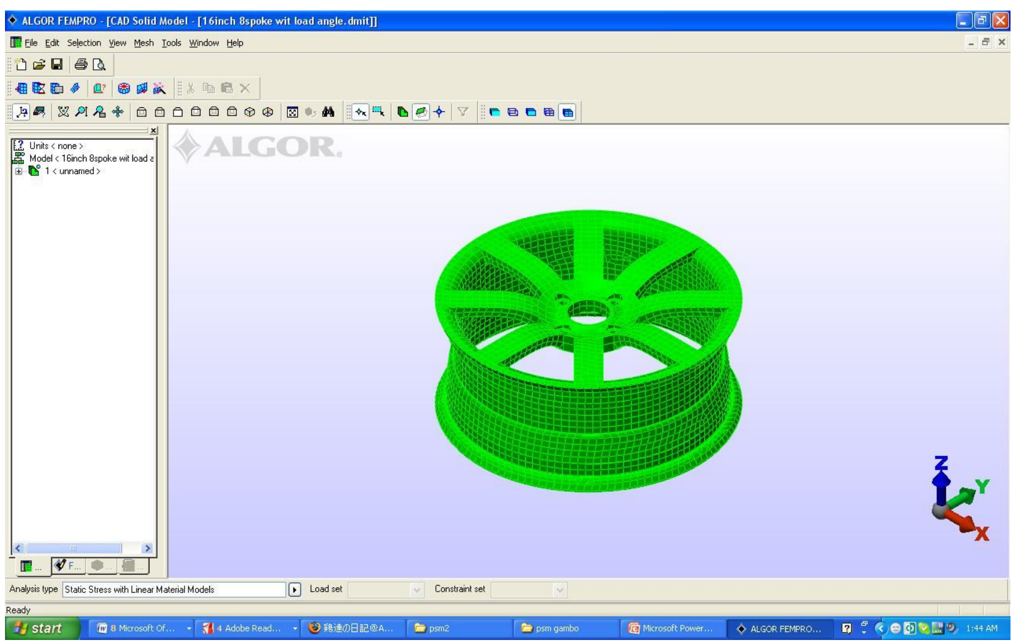


Figure 3.14: Alloy wheel in Algor Fempro V16

Linear static stress is used for the analysis. The alloy wheel is fixed on the hub hole in X axis, Y axis and Z axis. This is to simulate the bolt and nut effect on the hole of the wheel. The entire node inside the hole is fixed. The load is acted below the bead, while considering the contact patch load on the alloy wheel. The load is acted at Y axis to simulate the ground force acted on the body of the alloy wheel. Load ranging from 2500N to 5500N is applied 40 degree load angle [9]. The application of the free body diagram of the rim is shown in figure 3.14

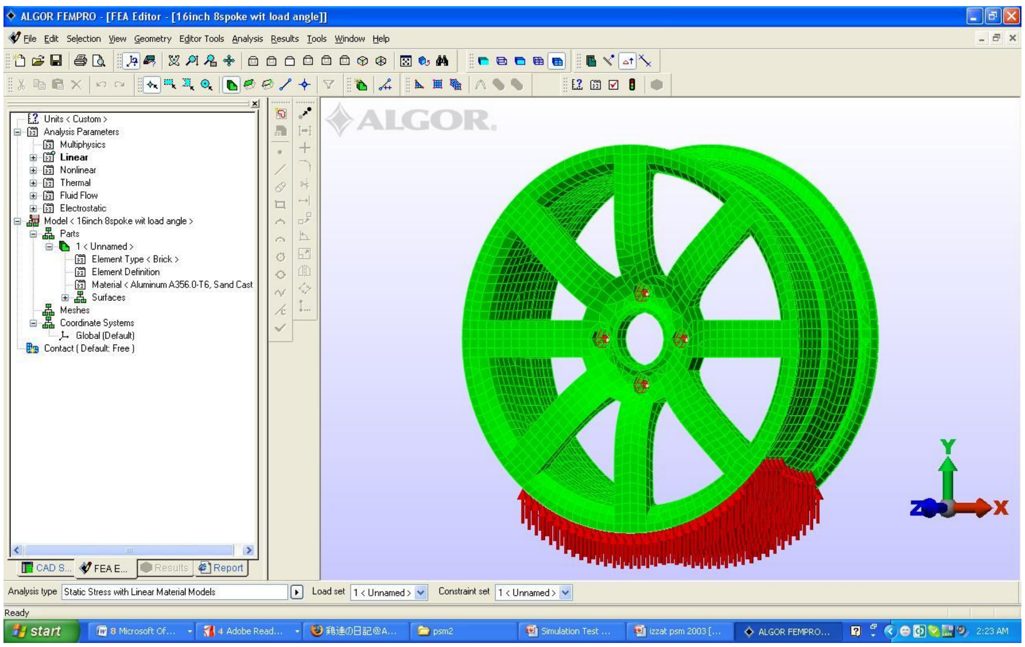


Figure 3.15: Free body diagram of alloy wheel in Algor Fempro V16

Aluminum A356-T6 is used for the material selection for the alloy wheel based on the literature study. Finally the simulation and analysis can be done.

CHAPTER 4

RESULT AND DICUSSION

4.1 Mesh sensitivity

Force and displacement is use to find the mesh sensitivity of the CAE software for the alloy wheel. Mesh percentage is range from 30,40,50,60,70,80,90 and 100 is used to find the best mesh percentage. Higher mesh percentage lead to coarser mesh per area but lower mesh percentage lead to finer mesh per area. One alloy wheel design is used to find the best mesh percentage while using 2500N of force. The force is constant.

Table 4.1: Mesh percentage vs stress and displacement

Mesh Percentage	100	90	80	70	60	50	40	30
Stress N/mm^2	123.1618	148.6226	211.466	226.8941	211.466	224.3051	232.1965	464.5736
Displacement mm	1.271845	1.523641	3.28985	3.98422	3.68985	4.057667	4.068958	5.904094

From the data the force displacement value is decreasing respected to mesh percentage. The data tend to be nearly constant at 40% to 80% mesh. 12.8391N/mm^2 different occur for highest stress at 40% mesh (232.1965N/mm^2) and lowest stress at 60% (211.466N/mm^2) in the range of 40% to 80% mesh. 0.770918mm different occur for highest displacement at 40% mesh (4.057667mm) and lowest displacement at 80% (3.28985mm) in the range of 40% to 80% mesh. The best mesh percentage is chosen to be 50%. This is because the stress value and displacement value is nearly constant at 50%. The graph for Mesh percentage vs. Stress and Mesh percentage vs. displacement is shown.

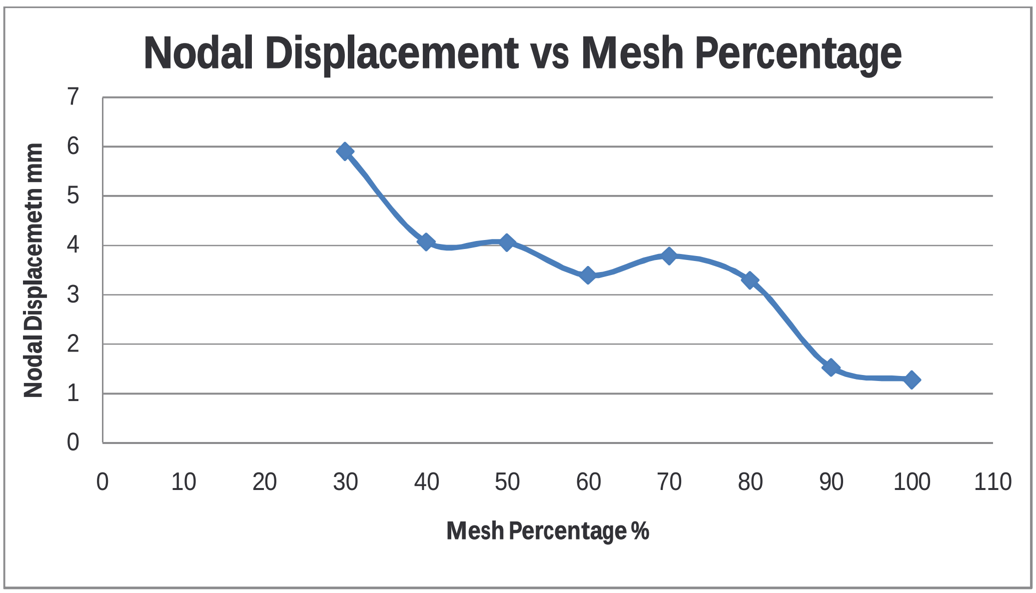


Figure 4.1: Nodal displacement vs mesh percentage

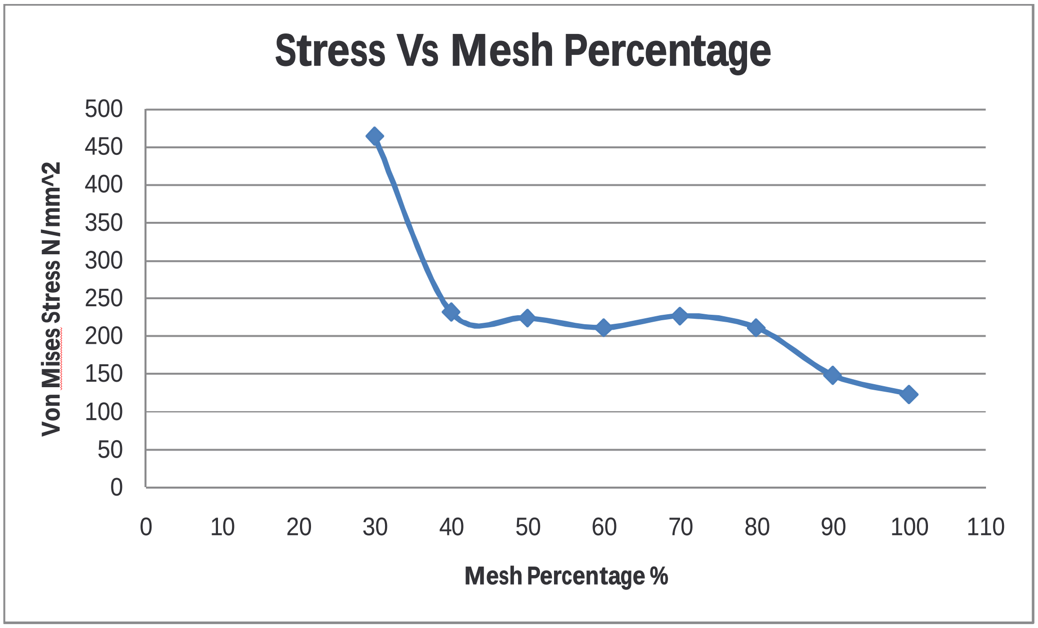


Figure 4.2: Stress vs Mesh percentage

4.2 Stress and Displacement result

The stress and displacement results obtained for different design of alloy wheel are shown below. The variation of von misses stress of the alloy wheel with respect to different loading for different design is shown

The purpose of performing stress analysis on the wheel is to predict the stress region on the alloy wheel. In order to get a good sense of how the wheel's material performs under 2500 N , 3500 N, 4500N and 5500N of force[9], we will need to compare the maximum effective stress values (also called Von Mises criterion) observed in the results to a known value that represents the limits of performance for the

Table 4.2: Result of Von misses stress for 40mm spoke wheel

Force (N)	2500	3500	4500
5spokes	288.033 N/mm^2	403.2462 N/mm^2	518.4956 N/mm^2
6 spokes	278.9337 N/mm^2	390.5071 N/mm^2	502.0808 N/mm^2
7 spokes	266.3567 N/mm^2	372.8995 N/mm^2	479.4423 N/mm^2
8 Spokes	224.3051 N/mm^2	314.027 N/mm^2	403.7492 N/mm^2
No spoke	149.1843 N/mm^2	208.8581 N/mm^2	268.5318 N/mm^2

Table 4.3: Nodal displacement for 40 mm spoke wheel

Force (N)	2500	3500	4500
5spokes	4.912502mm	6.98123mm	8.842505mm
6 spokes	4.739102mm	6.634744mm	8.530387mm
7 spokes	4.281721mm	5.994411mm	7.707101mm
8 Spokes	4.057667mm	5.680733mm	7.303802mm
No spoke	2.310542mm	3.234759mm	4.158977mm

Table 4.4: Result of Von misses stress for 50mm spoke wheel

Force (N)	2500	3500	4500
5spokes	246.6723 N/mm^2	345.3413 N/mm^2	444.0104 N/mm^2
6 spokes	212.3918 N/mm^2	297.3484 N/mm^2	382.3053 N/mm^2
7 spokes	193.1777 N/mm^2	270.4487 N/mm^2	347.7198 N/mm^2
8 Spokes	166.678 N/mm^2	240.8933 N/mm^2	310.8923 N/mm^2
No spoke	149.1843 N/mm^2	208.8581 N/mm^2	268.5318 N/mm^2

Table 4.5: Result of displacement for 50mm spoke wheel

Force (N)	2500	3500	4500
5spokes	4.429807mm	6.20173mm	7.973655mm
6 spokes	3.953245mm	5.534542mm	7.115842mm
7 spokes	3.572516mm	5.001522mm	6.43053mm
8 Spokes	2.970109mm	4.158148mm	5.534236mm
No spoke	2.310542mm	3.234759mm	4.158977mm

Table 4.6: Result of Von misses stress for 60mm spoke wheel

Force (N)	2500	3500	4500
5spokes	211.4752 N/mm^2	296.0652 N/mm^2	380.6555 N/mm^2
6 spokes	200.9299 N/mm^2	281.3019 N/mm^2	361.674 N/mm^2
7 spokes	185.7253 N/mm^2	253.7295 N/mm^2	313.3661 N/mm^2
8 Spokes	166.2496 N/mm^2	234.7494 N/mm^2	293.2493 N/mm^2
No spoke	149.1843 N/mm^2	208.8581 N/mm^2	268.5318 N/mm^2

Table 4.7: Result of displacement for 60mm spoke wheel

Force (N)	2500	3500	4500
5spokes	3.943466mm	5.520852mm	7.09824mm
6 spokes	3.695366mm	5.173513mm	6.65166mm
7 spokes	3.314084mm	4.639719mm	5.965353mm
8 Spokes	3.073485mm	4.302878mm	5.532274mm
No spoke	2.78676mm	4.04656mm	4.987987mm

4.3 Interpretation of Stress values

The purpose of performing stress analysis on the wheel is to predict the stress distribution on the different alloy wheel design. In order to get a good sense of how the wheel's performs under static load of 2500N, 3500N, and 4500N [9] , the maximum effective stress values (also called Von Mises criterion) observed in the results to a known value that represents the limits of performance for the wheel to find the highest stress concentration area

In the case of the material of the wheel used in this simulation test, the maximum predicted stress of the A356-T6 alloy value is 250 MPa. Depending on the units set in the beginning of the analysis, this value may appear in N/mm^2, or KPa (1000 Pascals). The numeric value can be read from the vertical color bar, where dark blue is the minimum stress and red is the maximum stress.

Comparisons of the predicted maximum stress are made to commonly known and experimentally derived values of yield strength for the appropriate material. The yield strength tends to refer to a value at which the material exhibits substantial deformation or breaks. Different design of alloy wheels has different yield strengths:

Comparing the maximum stress value, the no spokes design wheel has the No spokes design alloy wheel has the lowest stress occur that is 149.1843N/mm^2 when subjected to the lowest force (2500N) and 268.5318 N/mm^2 when subjected to the highest force (4500N).

For multispokes design alloy wheel with 40mm spoke, the highest stress occurs on the 5 spokes design alloy wheel. That is 288.033N/mm^2 when subjected to the lowest force (2500N) and 518.4956 N/mm^2 when subjected to the highest force (4500N). For the 40mm spokes design, the wheel is exceeded the ultimate tensile strength of the material (302MPa) at the 3500N and 4500N of force for the 5,6,7 and 8 spokes wheel design

For the 50mm spokes size wheel, the design with less spokes have the higher stress values that is on the 5 spokes design. That is 246.6723N/mm^2 when subjected to the lowest force (2500N) and 444.0104 N/mm^2 when subjected to the highest force (4500N) For the 50mm spokes design, the wheel is exceeded the ultimate tensile strength of the material (302MPa) at the 3500N for 5 spokes wheel, 4500N of force for the 5,6,7 and 8 spokes wheel design

For the 60mm spoke size wheel, alloy wheel with 5 spokes wheel shows the highest stress values that is 211.4752N/mm^2 when subjected to the lowest force (2500N) and 380.6555 N/mm^2when subjected to the highest force (4500N). For the 60mm spokes design, the wheel is exceeded the ultimate tensile strength of the material (302MPa) only at the 4500N for 5, 6, and 7spokes wheel.

From the stress values data the stress is decreasing when the spoke size is increasing. This resulting with lower stress concentration on the alloy wheel. While if number of spokes of the wheel increase the stress value is decreasing.

4.4 Interpretation of Displacement values

Displacement (deformation) values are interpreted in a fashion similar to that of stress values. The reported values of displacement are in Meters, if the units were set to Newton/Meter in the beginning of the analysis. Depending on the units, the minimum and maximum reported values must be interpreted accordingly

In the case of the all the wheels design, the maximum displacement takes place in the bottom regions that is again the region in which the force was applied. Higher force on that particular area will lead to higher deformation. No spoke alloy wheel design have the lowest deformation on the wheel that is 2.310542mm when subjected to the lowest force(2500N) and 4.158977mm when subjected to the highest force (4500N)

For multispokes design alloy wheel with 40mm spoke, the lowest deformation occurs on the 8 spokes design alloy wheel. That is 4.057667mm when subjected to the lowest force (2500N) and 8.842505mm when subjected to the highest force (4500N)

For the 50mm spokes size wheel, the design with more spokes have the lowest deformation that is on the 8 spokes design. That is 2.970109mm when subjected to the lowest force (2500N) and 7.973655mm when subjected to the highest force (4500N)

For the 60mm spoke size wheel, alloy wheel with 8 spokes wheel shows the lowest deformation that is 2.78676mm when subjected to the lowest force (2500N) and 7.09824mm when subjected to the highest force (4500N)

From the multispokes alloy wheel design, alloy wheel with the highest number of spokes (8 spokes) have the lowest deformation, 2.78676mm. While comparing with the no spokes design wheel with 2.310542mm.

From the deformation data, the values of deformation is decreasing when the number of spokes increasing. The deformation value data is proportional with the stress value data.

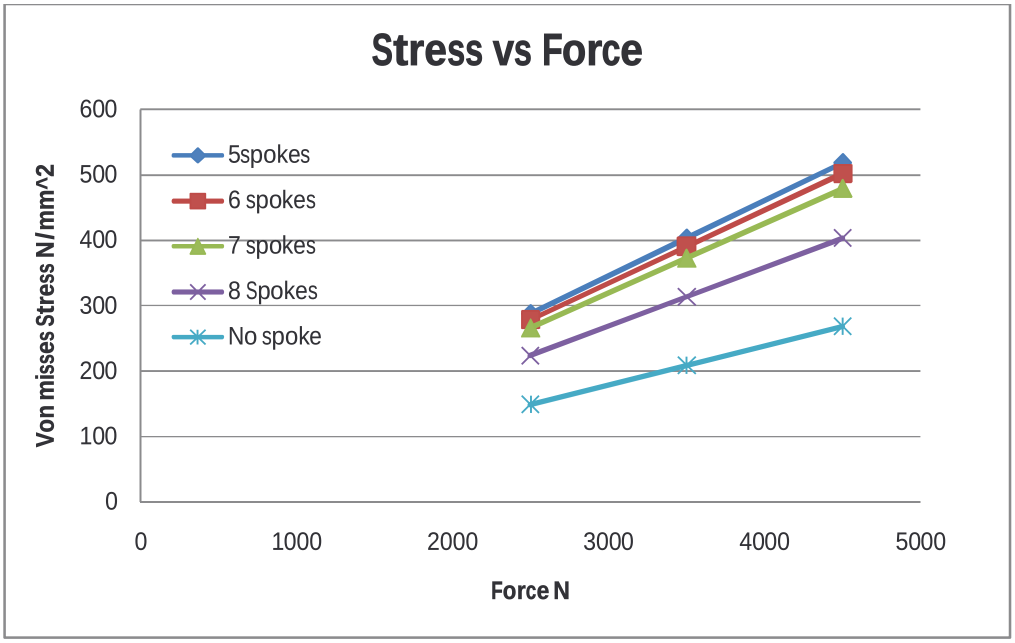


Figure 4.3: Stress vs force for 40mm spokes wheel

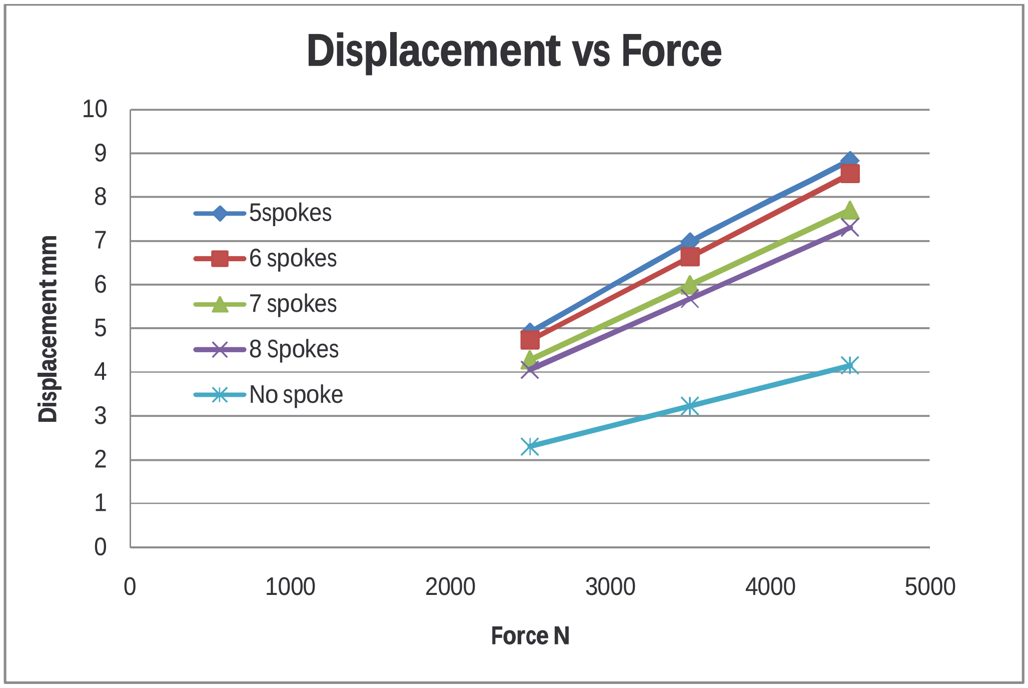


Figure 4.4: Displacement vs force for 40mm spokes wheel

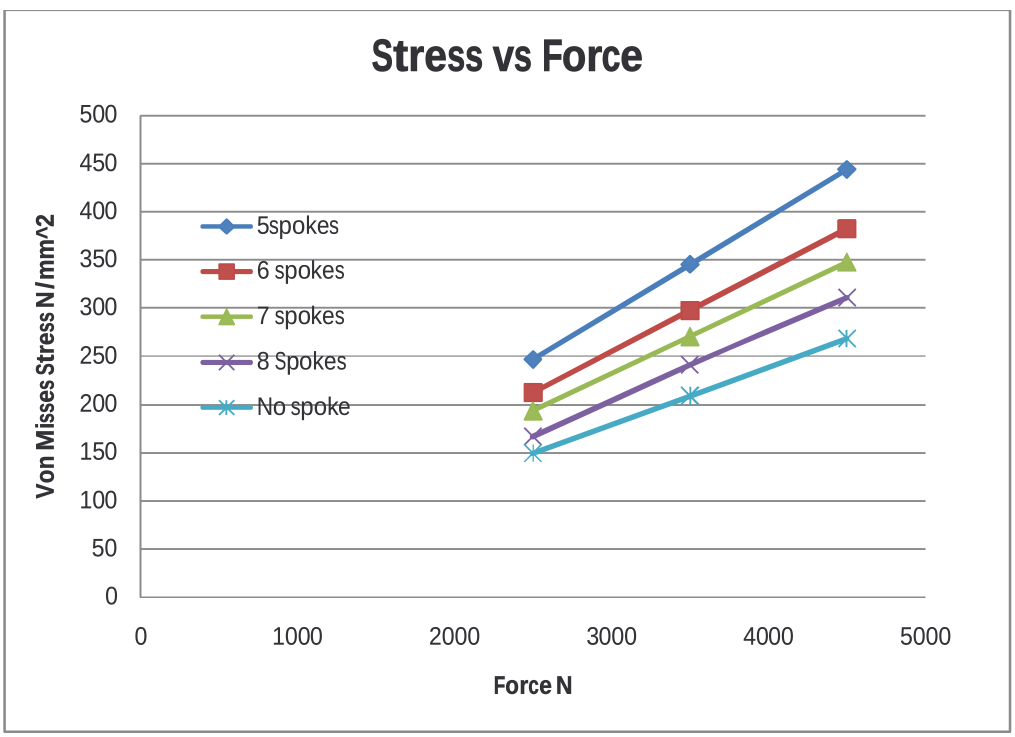


Figure 4.5: Stress vs force for 50mm spokes wheel

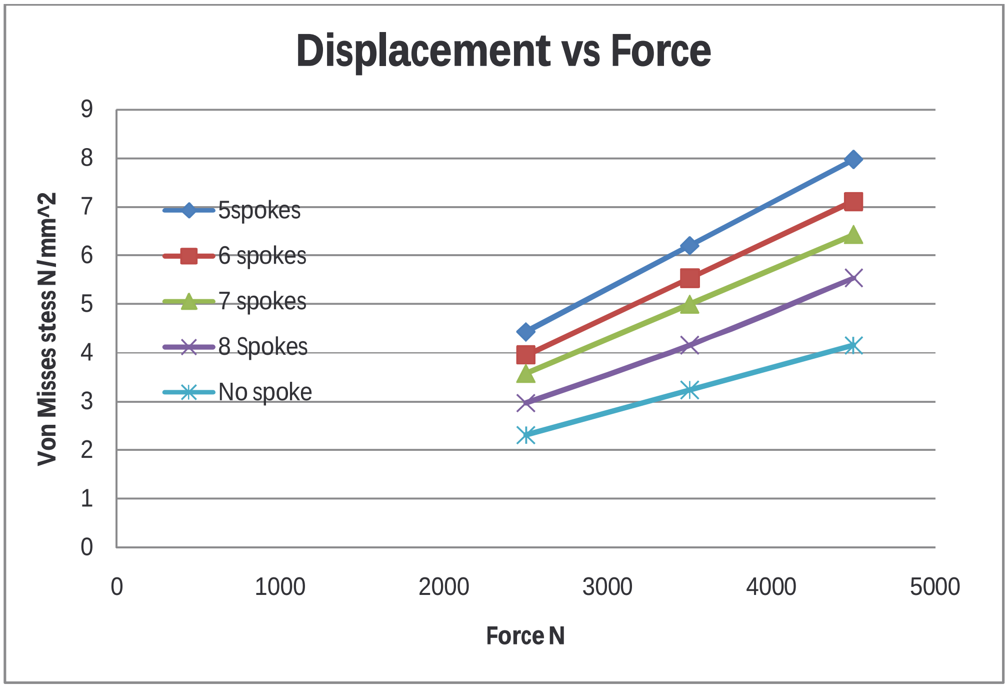


Figure 4.6: Displacement vs force for 50mm spokes wheel

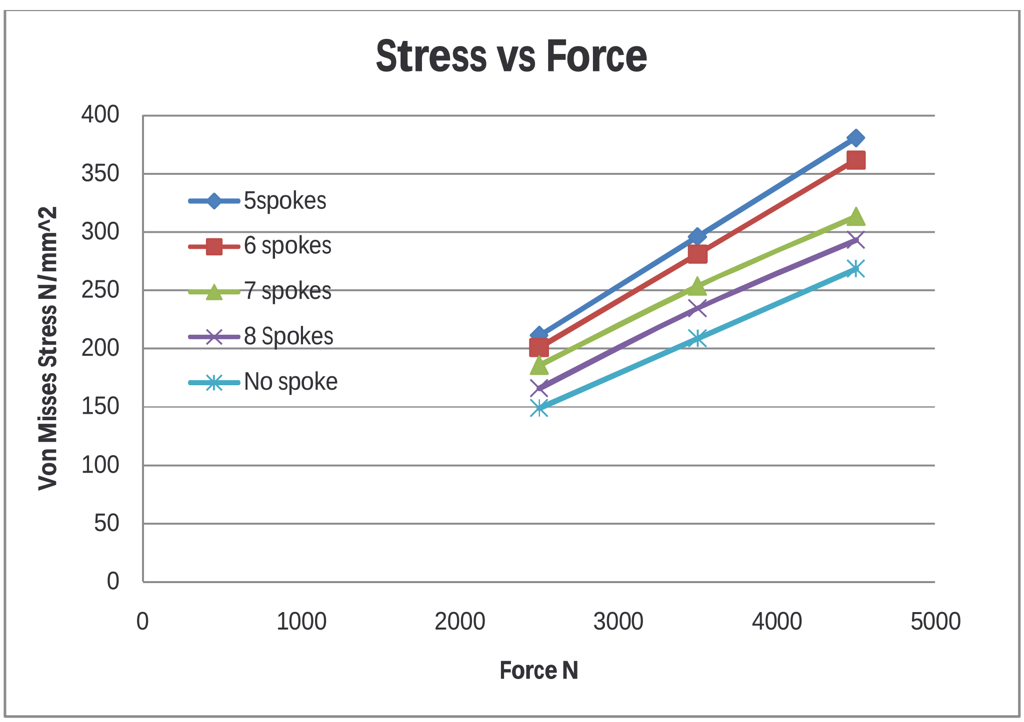


Figure 4.7: Stress vs force for 60mm spokes wheel

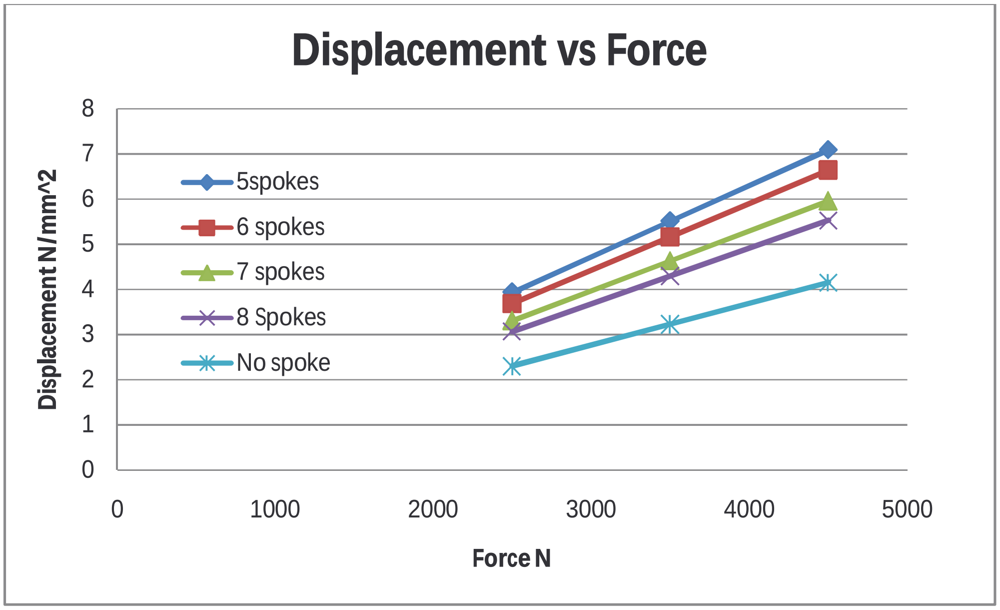


Figure 4.8: Displacement vs force for 60mm spokes wheel

4.5 Critical location for the stress analysis result

While the numeric values give the minimum and maximum predicted stress values, there are need to identify the specific areas, in which these values are observed. To this end it will take a closer look at the graphical, rainbow-colored representation of stress on the wheel. Most of the wheel, in particular the top and side region experiences little stress (blue regions), whereas the higher values of stress concentrate near the bottom region, between street level and hub. This corresponds to the region in which all of the pressure has been applied.

While the spokes are still in a blue/cyan/green range of stress, clearly note the extreme stress (orange/red) in the corners, where spokes and rim meet. It is generally the goal of such analysis to identify regions of high stress, which can then be re-modeled to minimize and spread out the stress. In the case of this wheel, the corners between spokes and rim have been rounded off, but obviously not enough to spread out the stress. In most structural designs, there are many rounded corners, as this tends to minimize the stress. The stress region of the alloy wheel design is shown

4.6 Critical location on no spokes wheel design

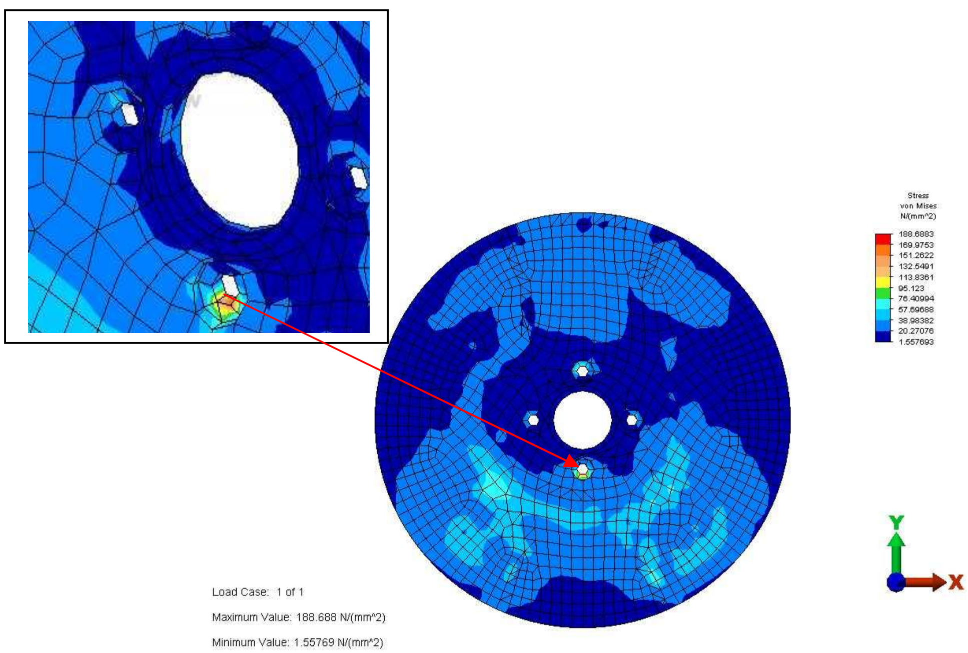


Figure 4.9: Front view of No spoke wheel stress distribution

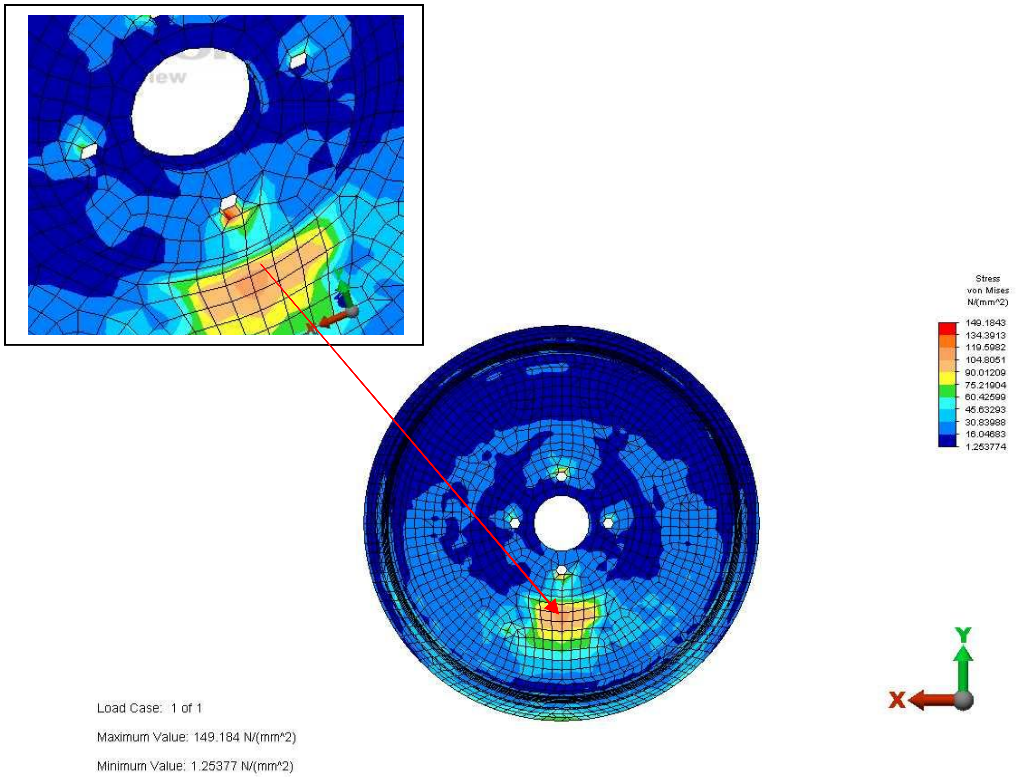


Figure 4.10: Back view of No spoke wheel stress distribution

The no spokes wheel design has the least stress and displacement, the stress is concentrated on the area below the hole. Highest stress is on that area because the force acted on the ground is pushing the wheel while the hole is fixed thus stress is concentrated on that particular area. The Stress is well distributed on the no spokes wheel because it has more material to hold the stress acted on the wheel. Higher stress also occurs inside the hole of the wheel when it is fixed. The stress is occurring on the area below the hole. From actual condition, the bolt from automotive knuckle will acted as the force that contact from below to the hole of the alloy wheel.

4.7 Critical location Multispokes design

\

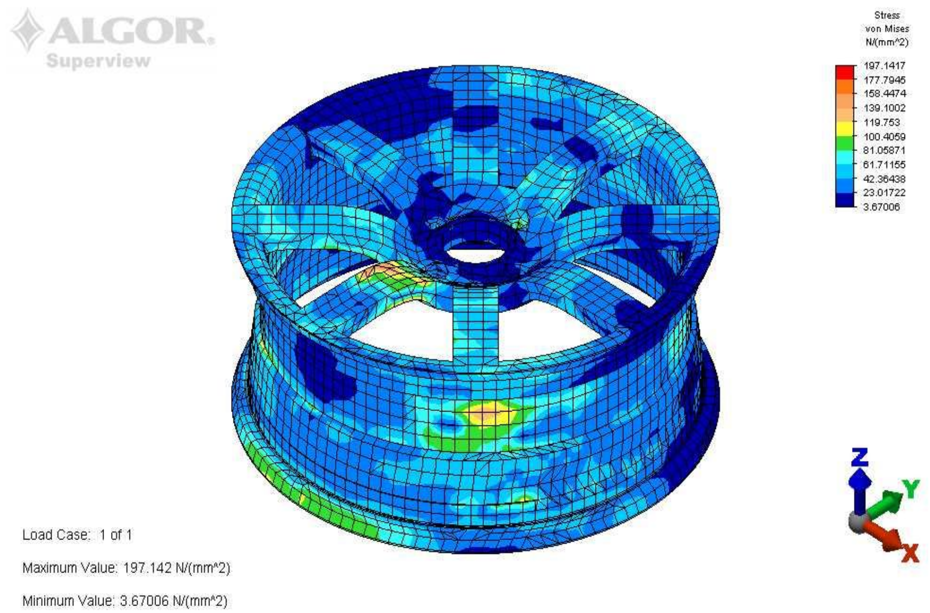


Figure 4.11: Overall view of 8 spokes wheel stress distribution

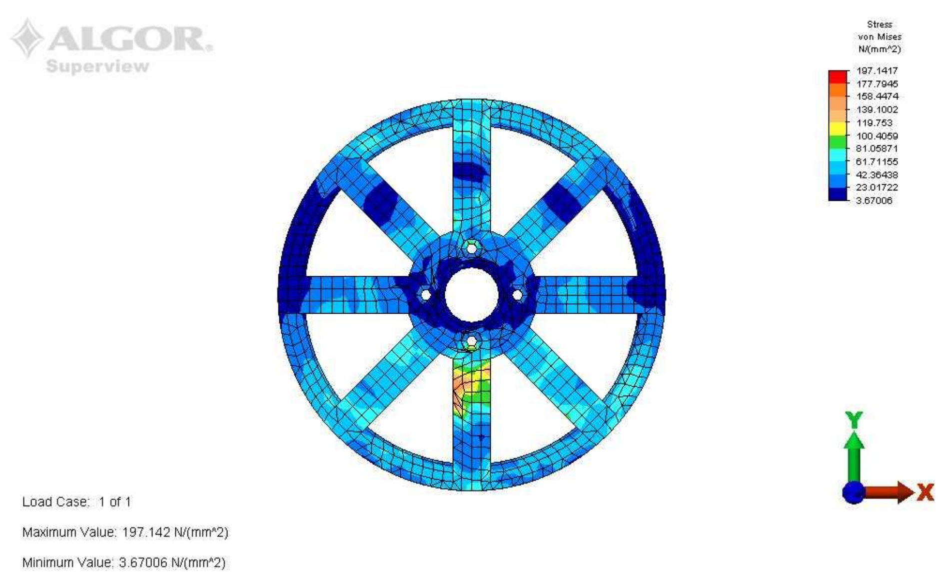


Figure 4.12: Front view of 8 spokes wheel stress distribution

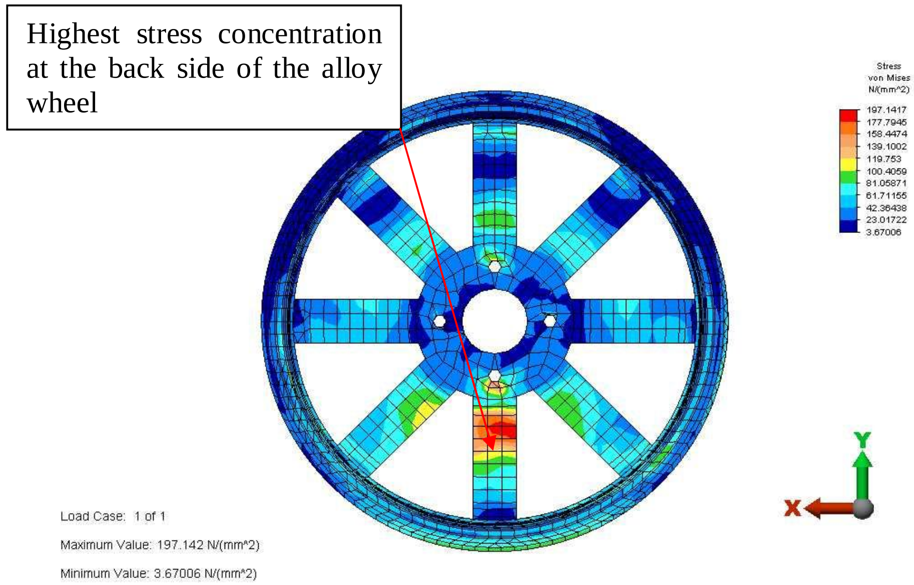


Figure 4.13: Back view of 8 spokes wheel stress distribution

For multispokes alloy wheel, it has the highest stress and displacement. The spoke below the hub and hole suffer the most stress. Force is higher at the area because only 1 spoke holding all the stress of the wheel and most of the wheel stress is not distributed well among the spokes. The hub hole and the backside of spoke reveals the greatest stress and it is because stress is concentrated on that area. The inside bend surface of the spoke gather all the stress on that area thus making it more vulnerable to crack or fail.

Higher stress also occurs inside the hole of the wheel when it is fixed. The stress is occur on the below area of the inside hole. From the analysis also, there are stress occur on the bead seat at which the force is acted.

CHAPTER 5

CONCLUSION

5.1 Conclusion

The location of stress concentration compare with the literature review is found to be the same. These are drawn based on the comparison of the result obtained from the test and finite element analysis

Without doubt, the no spokes alloy wheel experiences the least amount of stress and the least amount of displacement; this is because it is structurally most sound. However, no material was saved in the no spokes wheel design because of the solid feature of the wheel. This may cause higher production costs, and they may not warrant the marginal improvement in strength.

The next best design is the wheel with 8 spokes design wheel. This wheel experiences more stress and more displacement than the no spokes wheel, design but the difference is negligible, even though the wheel remains structurally sound in half the range of the no spoke wheel design ,yet the wheel can still remain under ultimate tensile strength even though higher force acted on it. The 8 spokes wheel can remain under ultimate tensile strength of the material for the highest force that is 4500N.

The wheel with 5 solid spokes or less would work, but a design of this kind is would impact the structural quality over time, as most displacement occurs in the area without spokes. This could expect the wheel to become more elliptical over time. Thus making it more prone to fail or crack overtime when use. 5 spokes wheel is generally lighter, but the disadvantage is it could not remain under the ultimate tensile strength for the lower force because of the less material used on the wheel design

5.2 Recommendation

Further to the analysis that has been conducted, there are several action plans or further work to be done. The following are some recommendations that should be consider achieving a good analysis result:

Using different Computer aided engineering software to compare the stress result. This will provide more variety of result to compare thus, provide the validity of the result from this project result

Comparing stress result from Computer aided software with the actual experimental result. The actual experiment can provide the accurate result for the result for the computer aided engineering software. Actual experimental result should be done exactly the same with the same parameter and variables to provide accurate result.

5.3 Contribution

The goal is to identify regions of high stress on different alloy wheel design, the stress region that affected the wheel to failure and crack from which can then be re-modeled to minimize and spread out the stress of the future design of alloy wheel. Better design and high performance alloy wheel can be made without affecting the safety of the wheel. This will provide additional information and knowledge to make better and safer wheel. With this optimum use of material and less time needed to design a new wheel thus can speed up the production to make new wheel for automotive manufacturer today.

REFFERENCE

[1] R. Hoemsen, Structural Design of Agricultural Wheels, American Society of Agricultural Engineers, Paper no. 84-1558, 1984, p. 10.

[2] Y. Morita, Sumimoto Metals Publication 39 (3) (1987) 245–263.

[3] K. Ishihara, Nippon Kikai Gakkai Ronbunshu Chen 55 (513) (1989) 1254–1258.

[4] H. Konishi, A. Fujiwara, T. Katsura, K. Takeuchi, M. Nakata, Nippon Kikai Gakkai Ronbunshu Chen 62 (599) (1996) 2884–2890.

[5] A. Blake, Practical Stress Analysis in Engineering Design, McGraw-Hill, New York, 1990, pp. 363–367.

[6] JIS D 4103. Japanese Industrial Standard. Disc Wheel for Automobiles. 1989.

[7] Grubisic V, Fischer G. Automotive wheels, methods and procedures for optimal design and testing. SAE Technical Paper Series 830135; 1984:1.508–1.525.

[8] Fischer G, Grubisic V. Cast aluminum wheels and buses – testing and evaluation. SAE Technical Paper Series 841705; 1985;6.1051–6.1062.

[9] John Stearns C. An investigation of stress and displacement distribution in a aluminum alloy automobile

APPENDIX

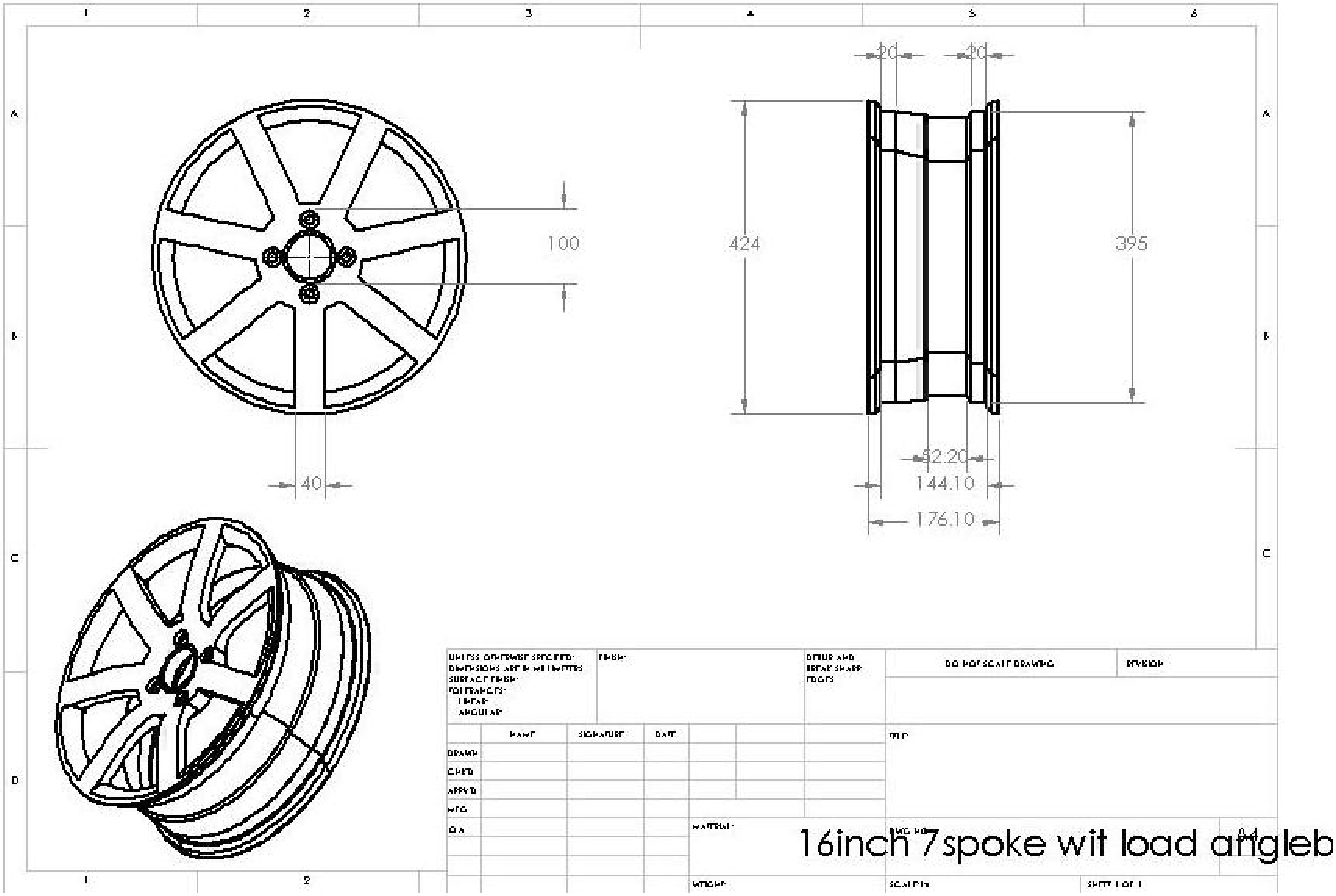
GANT CHART FOR FINAL YEAR PROJECT 1

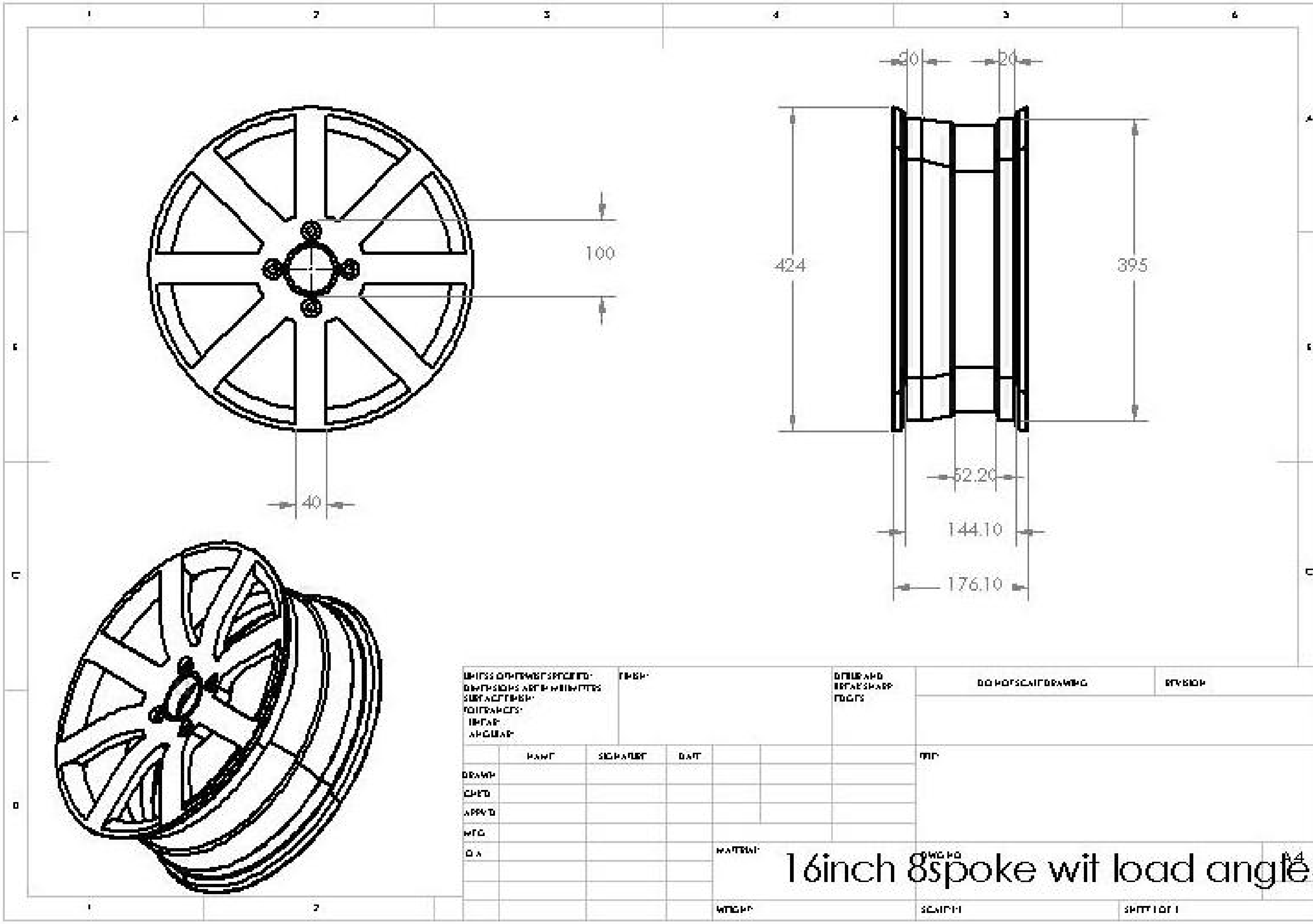
PROJECT ACTIVITIES		WEEKS													
		1	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14**
1	Discuss title and objectives														
2	Discuss scope and problem statement														
3	Chapter 1														
4	Discuss the format of project														
5	Literature - journals and ref. books														
6	Discuss literature on the Alloy wheel														
7	Discuss literature on loading method														
8	Discuss literature on types of material														
9	Chapter 2 - literature review														
10	Discuss the analysis and methodology														
11	Chapter 3 methodology														
12	Preparation for presentation 1														

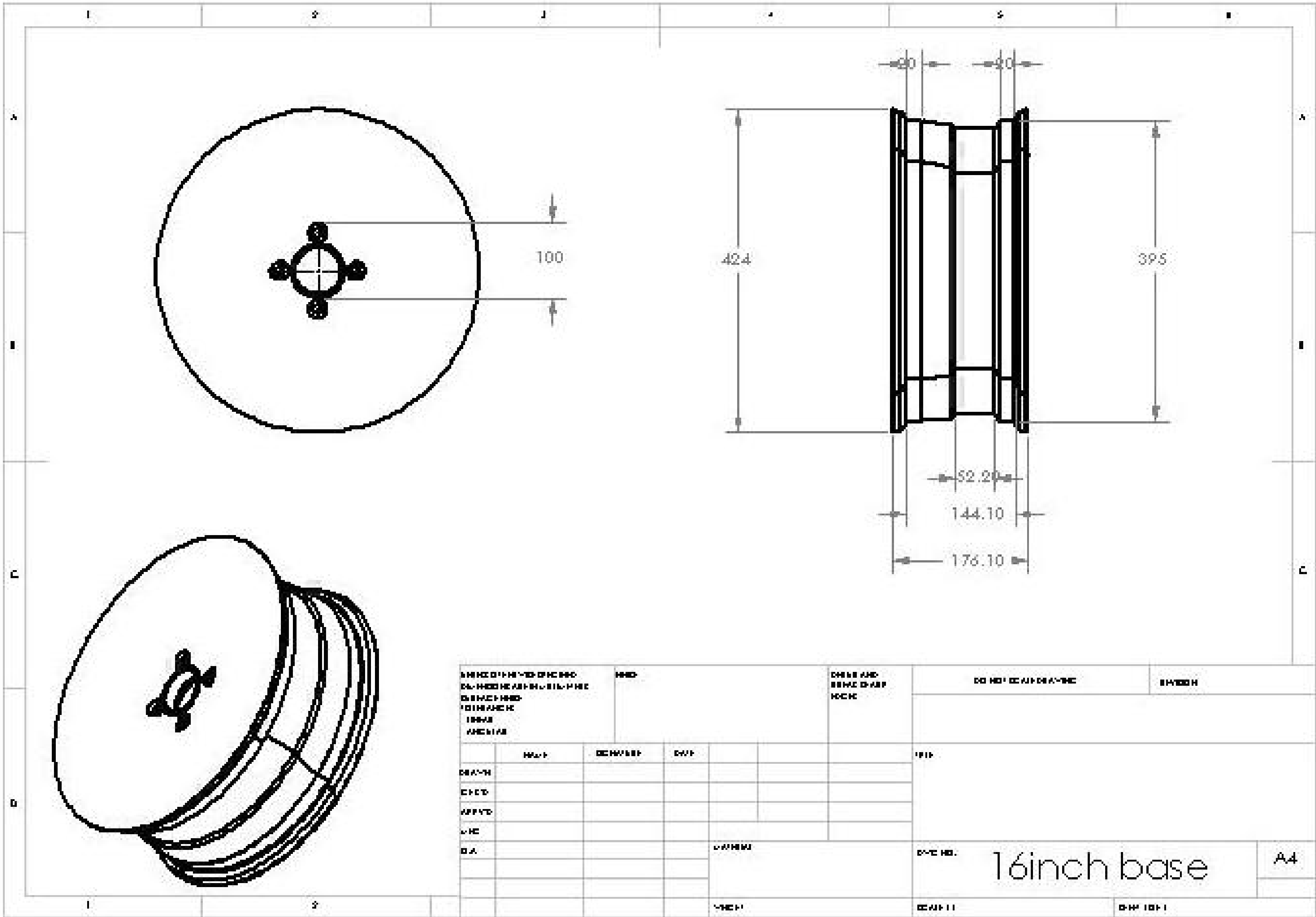
GANT CHART FOR FINAL YEAR PROJECT 2

PROJECT ACTIVITIES		WEEKS												
		1	2	3	4	5	6	7	8	9	10	11	12	13
1	Literature review													
2	Measure the dimension of alloy wheel													
3	Design using SOLIDWORKS													
5	Discuss on analysis, constraints and loads													
6	Analyze intake valve design using ALGOR													
7	Analyze and discuss the simulation results													
8	Conclude the project													
9	Complete chapter 4 and 5													
10	Final report preparation													
11	Preparation for presentation 2													

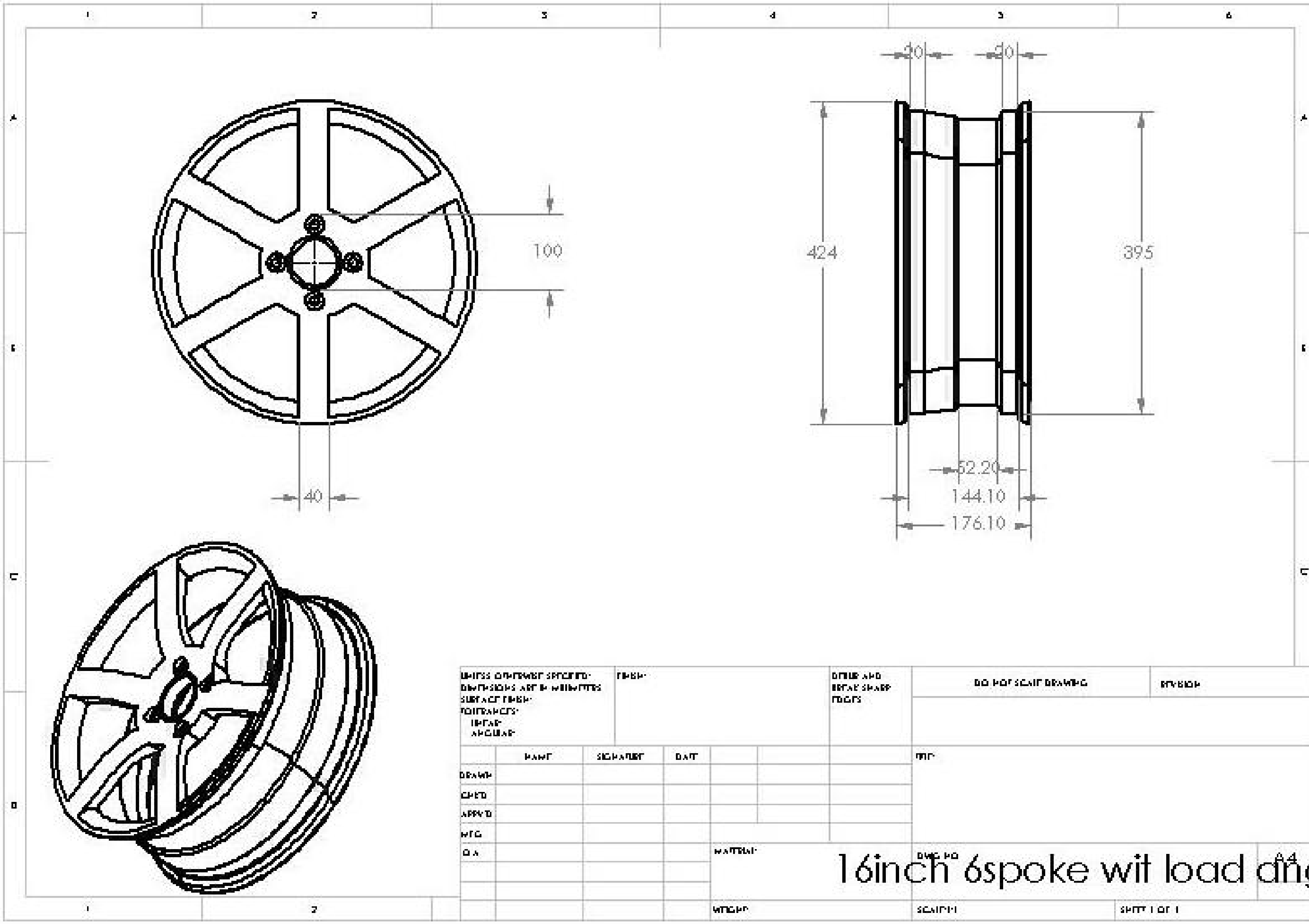
TECHNICAL DRAWING




20
20
100
424
395
40
52.20
144.10
176.10
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:
FINISH:
DEBUR AND
BREAK SHARP
EDGES
DO NOT SCALE DRAWING
REVISION
NAME
SIGNATURE
DATE
DRAWN
CHK'D
APPV'D
MFG
Q.A
MATERIAL:
TITLE:
DWG NO.
16inch 8spoke wit load angle4
A4
WEIGHT:
SCALE:1:1
SHEET 1 OF 1

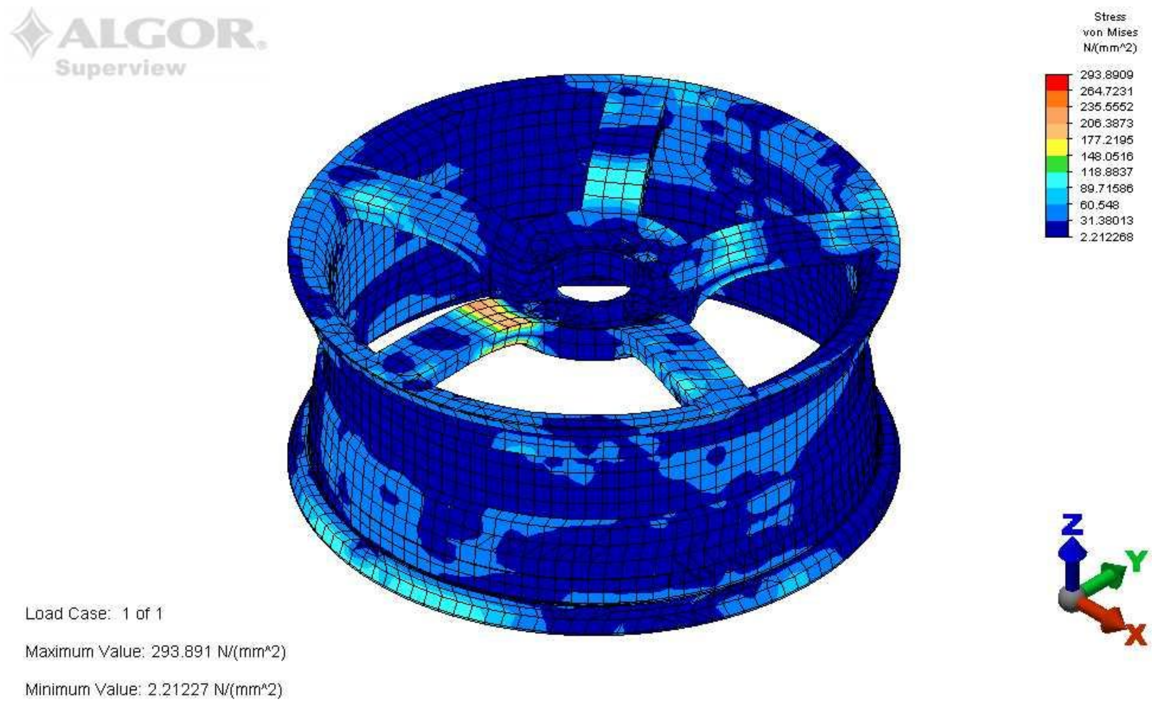


20
20
100
424
395
52.20
144.10
176.10
16inch base
A4

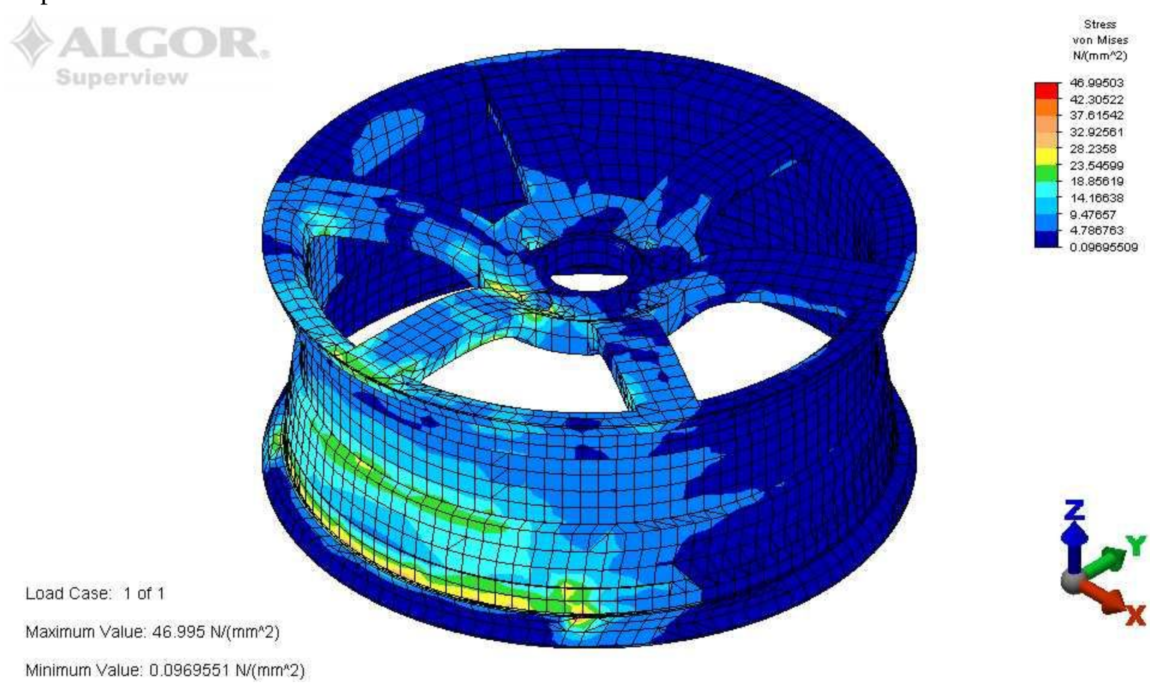

100
40
424
395
20
20
52.20
144.10
176.10
DO NOT SCALE DRAWING
REVISION
NAME
SIGNATURE
DATE
DRAWN
CHK'D
APPV'D
MFG
Q.A
MATERIAL:
DWG NO.
16inch 6spoke wit load drg
A4
WEIGHT:
SCALE:1:1
SHEET 1 OF 1

Stress Contour

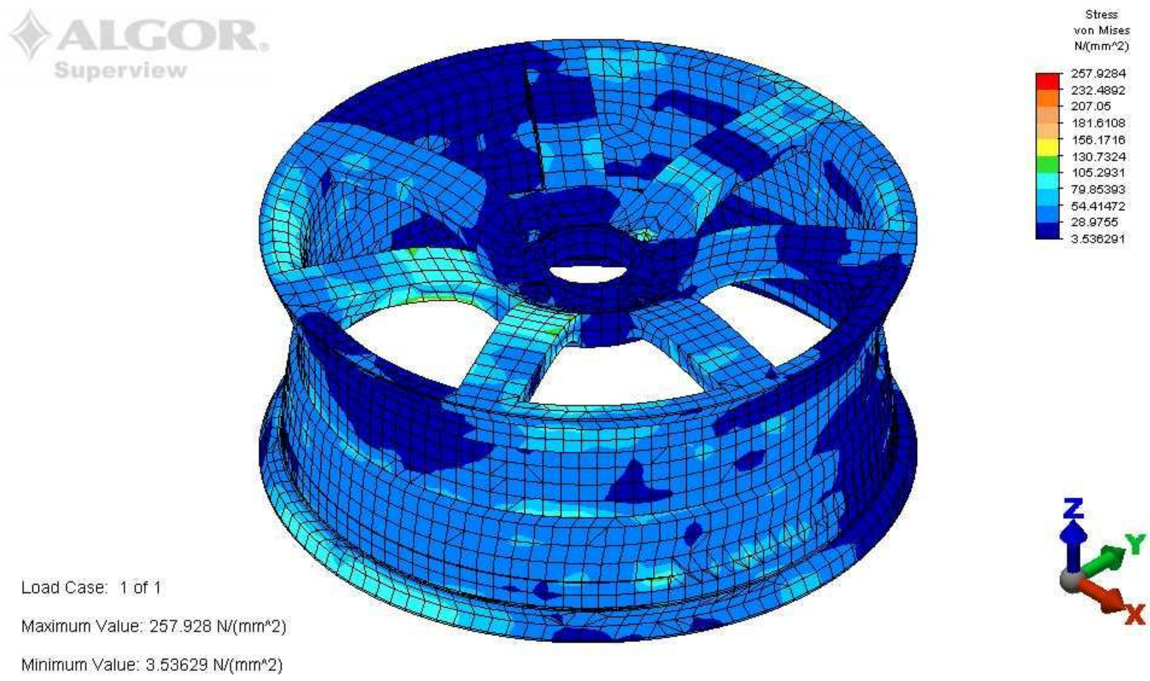
5 spokes



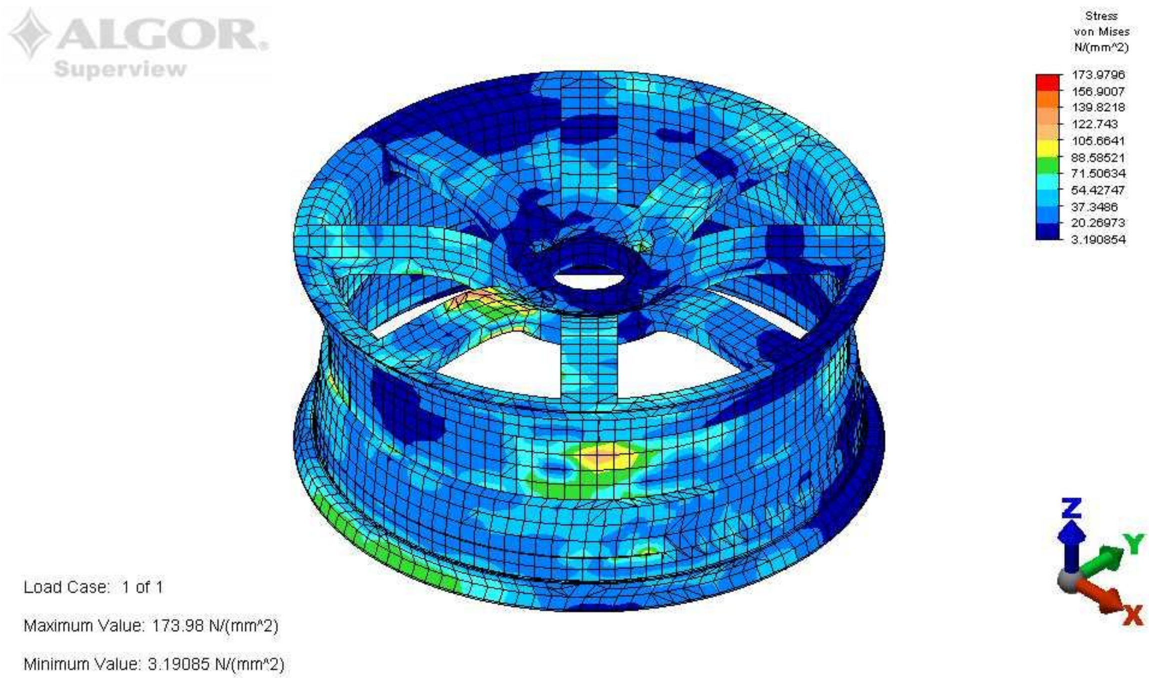
6 Spokes



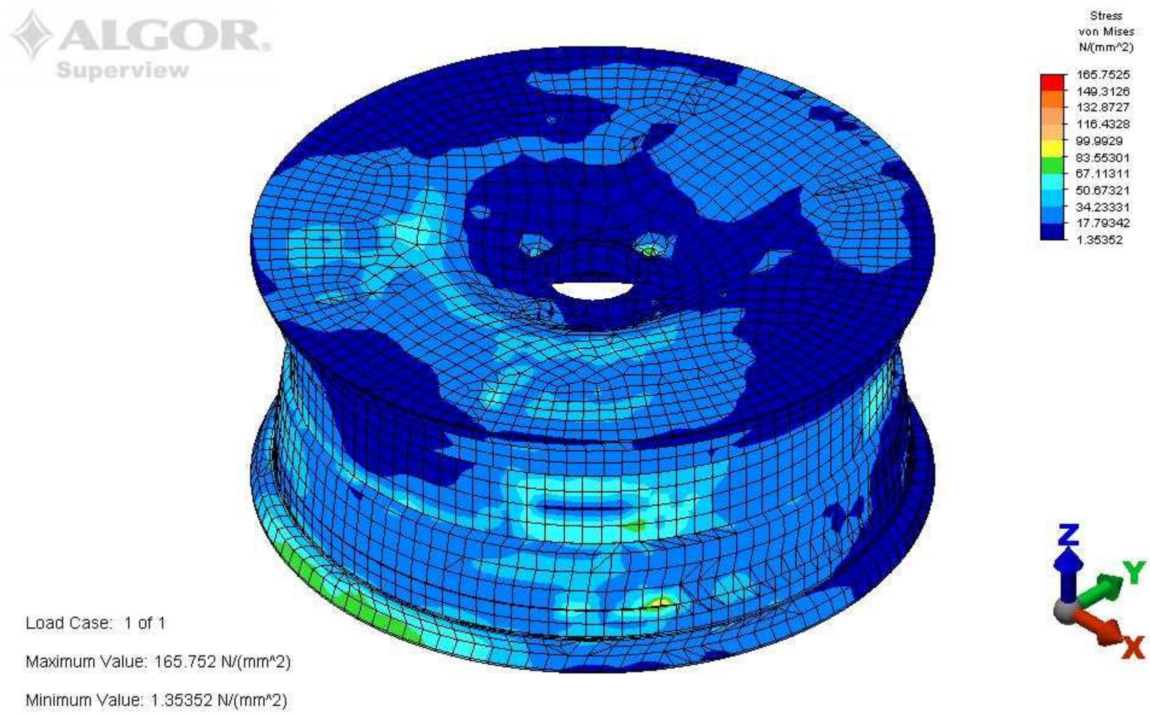
7 Spokes



8 Spokes

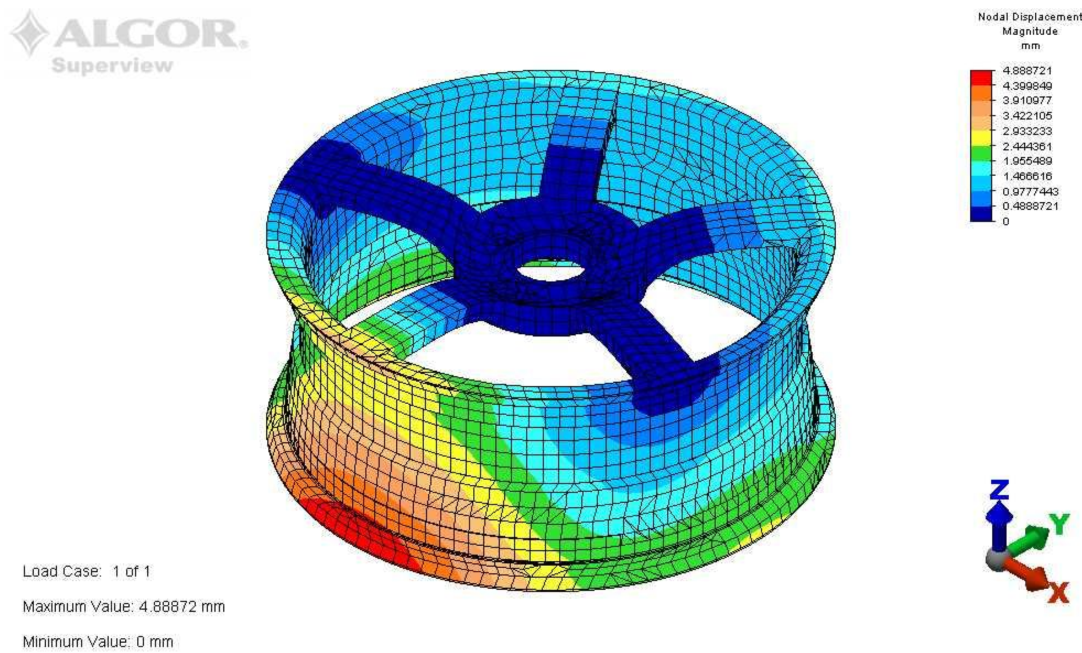


No Spokes

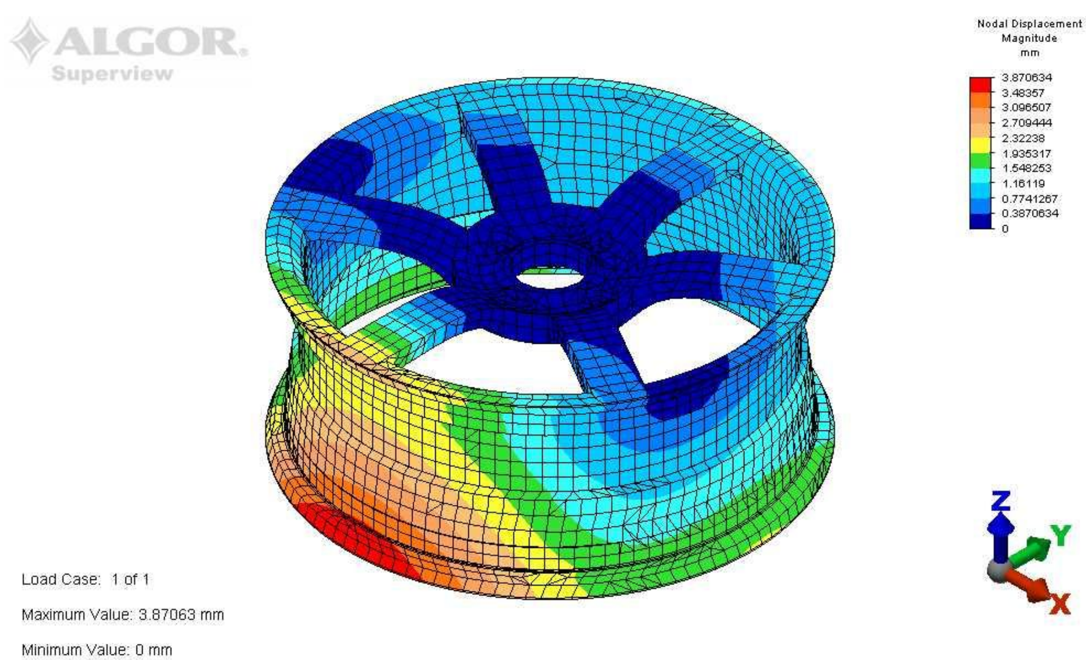

ALGOR
Superview
Stress
von Mises
N/(mm^2)
165.7525
149.3126
132.8727
116.4328
99.9929
83.55301
67.11311
50.67321
34.23331
17.79342
1.35352
Z
Y
X
Load Case: 1 of 1
Maximum Value: 165.752 N/(mm^2)
Minimum Value: 1.35352 N/(mm^2)

Displacement Contour

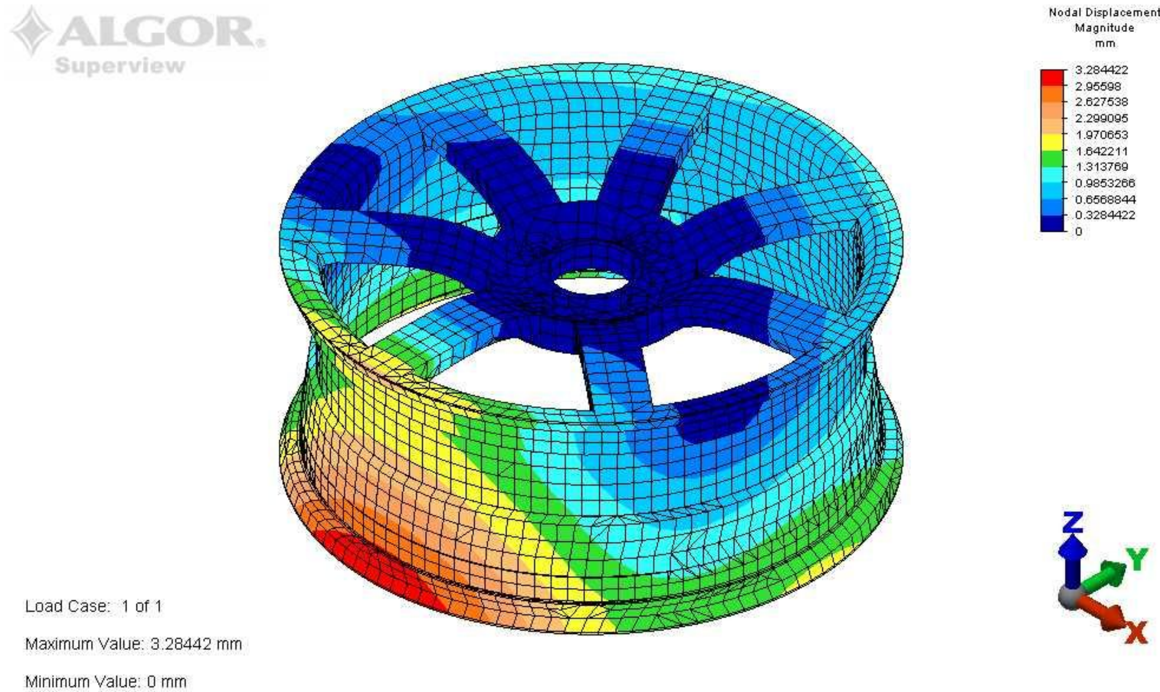
5 Spokes



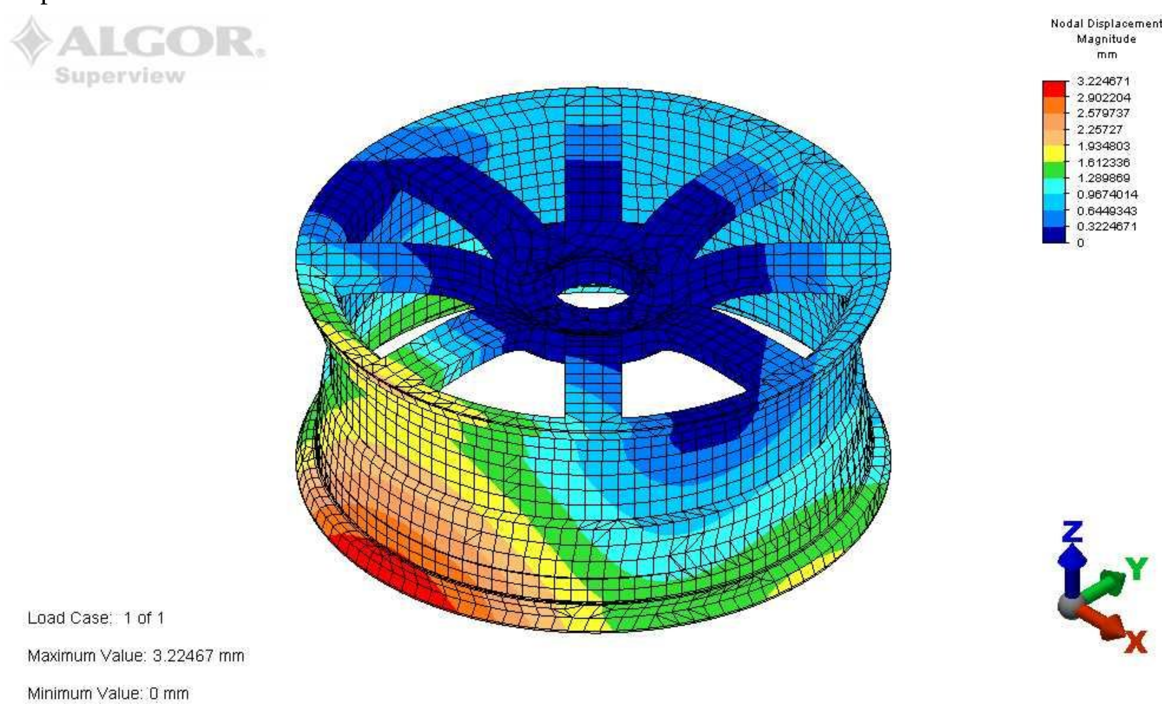
6 Spokes



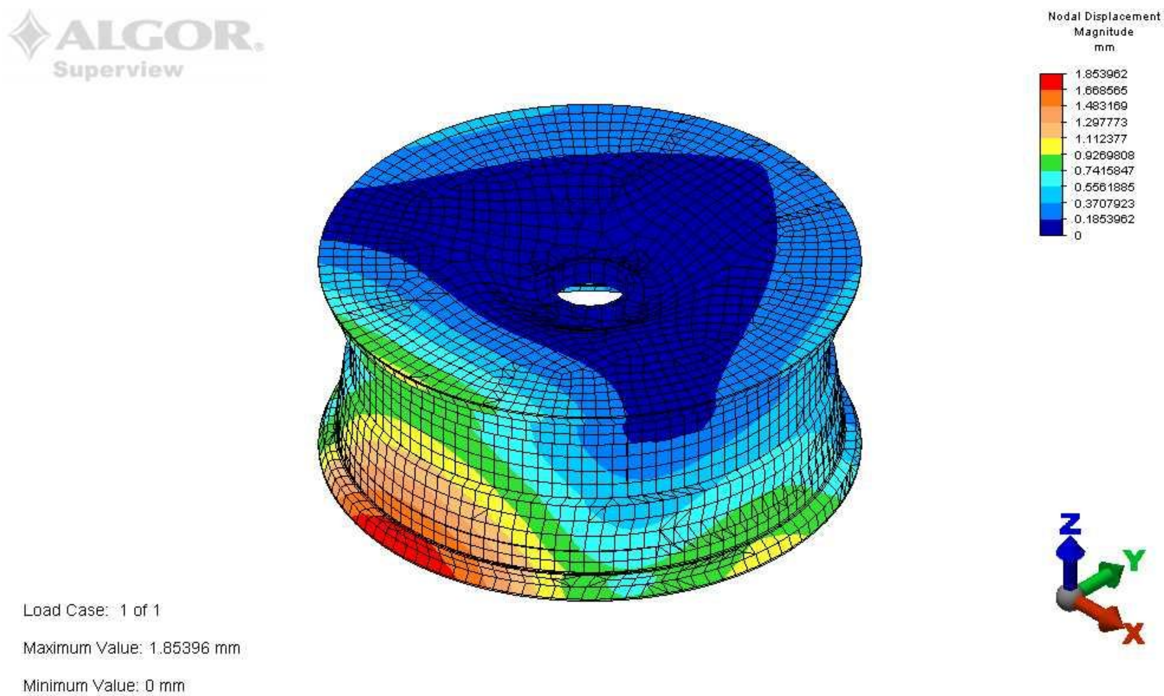
7 Spokes



8 Spokes



No spokes


ALGOR
Superview
Nodal Displacement
Magnitude
mm
1.853962
1.668565
1.483169
1.297773
1.112377
0.9269808
0.7415847
0.5561885
0.3707923
0.1853962
0
Z
Y
X
Load Case: 1 of 1
Maximum Value: 1.85396 mm
Minimum Value: 0 mm