DESIGN AND ANALYSIS OF WHEEL RIM BY USING CATIA &ANSYS

G.Ashokkumar[1], M.UmaMahesh[2], S.Madhu Sudhan[3], T.CHOLAI RAJ[4]

1 Assistant Professor, Mechanical Department, Priyadarshini Institute of technology, ramachandrapuram, Tirupati Andhra Pradesh, India, gadhireddy.ashokkmar@gmail.com

2 Assistant Professor, Mechanical Department, Priyadarshini Institute of technology, ramachandrapuram, Tirupati Andhra Pradesh, India, umamahesh491@gmail.com

3 M tech(student), Mechanical Department, S.V.COLLEGE OF ENGINEERING, Karakbadi Road, Tirupati, Andhra Pradesh, India, sreeramamadhusudhan@gmail.com

4 Assistant Professor, Mechanical Department, Priyadarshini Institute of technology, ramachandrapuram, Tirupati Andhra Pradesh, India, solairajmech335@gmail.com

Abstract

The purpose of the car wheel rim is to provide a firm base on which to fit the tyre. Its dimensions, shape should be suitable to satisfactorily accommodate the particular tyre required for the vehicle.

In this study a tyre of car wheel rim belonging to the disc wheel category is considered. Design in an important industrial activity which influences the quality of the product. The wheel rim is designed by using modeling software CATIAv5R18. In modeling the time spent in producing the complex 3-D models and the risk involved in design and manufacturing process can be easily minimized. So the modeling of the wheel rim is made by using CATIA.

Later this CATIA model is imported to ANSYS for analysis work. ANSYS software is the latest software used for simulating the different forces, pressure acting on the component and also for calculating and viewing the results. A solver mode in ANSYS software calculates the stresses, deflections, bending moments and their relations without manual interventions, reduces the time compared with the method of mathematical calculations by a human.

ANSYS static analysis work is carried out by considered two different materials namely aluminum and forged steel and their relative performances have been observed respectively.

In addition to this rim is subjected to Vibration analysis (Modal analysis), a part of Dynamic Analysis is carried out and its performance is observed.

In this project by observing the results of both static and modal analysis obtained forged steel is suggested as best material.

Index Terms: *Wheel rim, Catiav5R18, Ansys, static analysis, Vibration analysis (Modal analysis).*

---***---

1. INTRODUCTION

Archaeologists and historians of today see the introduction of the wheel as the real genesis of any old civilization. The wheel is perhaps the most important discovery of old times. This discovery capitulated commerce to heights unknown before. The wheel has developed from nothing more than an over sized bearing to an fully integral part of any modern transportation vehicle. The modern vehicle is also seen today a fashion item to complement people's individual tastes.

Motor vehicles are produced according to very strict rules to ensure the safety of the passengers. Every component is therefore designed according to the criticality of the component. Wheels are classified as a safety critical component and international cods and criteria are used to design a wheel.

Materials to produce these wheels have become has sophisticated as a design and materials can range from steel to non ferrous alloys like magnesium and aluminum.

Automotive wheels have evolved over the decades from early spoke designs of wood and steel. Carry over's from wagon and bicycle technology, to flat steel discs and finally to the stamped metal configurations and modern cast and forged aluminum alloys rims of today's modern vehicles. Historically successful designs arrived after years of experience and extensive field testing. Since the 1970's several innovative methods of testing well aided with experimental stress measurements have been initiated.

In recent years, the procedures have been improved by a variety of experimental and analytical methods for structural analysis is (strain gauge and finite element methods). Within the past 10years, durability analysis (fatigue life predication) and reliability method for dealing with variations inherent in engineering structure have been applied to the automotive wheel.

1.2 FUNCTIONS OF A WHEEL RIM

In its basic form a wheel rim is a transfer element between the tyre and the vehicle. The following are the main functions of a wheel rim:

- Transfers torque (braking and acceleration).
- Support mass (support the mass of the motor vehicle).
- Adds mass (damped mass for driving comfort).
- Dissipates heat (from braking).
- Adds value.
- Absorbs impact (road hazards).
- Conserves energy (potential energy in momentum).

.

1.3 CLASSIFICATION OF CAR WHEELS

Car wheels are divided in to two main categories, steel wheels and alloy wheels. Alloy wheels are often fitted standard during the manufacturing of modern vehicles.

1.3.1 STEEL WHEELS

All steel wheels consist of two pressed components, the rim and the wheel disc, which are welded together. The rim is the part on which tyre is mounted. Its dimensions shape and condition must suitable to satisfactorily accommodate the particular tyre required for the vehicle. The wheel disc is the supporting member between the vehicles hub and the rim. Its dimensions shape and location in the rim must be suited to the design of the wheel hub and the suspension geometry of the vehicle to which it has to be mounted. The purpose of the rim is to provide a firm base on which to fit the tyre. Four vital dimensions are involved.

They are the wheel diameter (a precise fit between tyre and rim is of utmost importance), the rim width, the flange height (designed to give adequate support to the tyre beads without changing the flux area of the side wall) and the rim-well (to facilitate the easy mounting and demounting of the tyres.

Because the inside diameter of the tyre must fit precisely onto the rim, it would be impossible for the inside diameter of the tyre to pass over the large diameter of the tyre rim without causing damage to the beads. Forcing the tyre bead into the rim well opposite to the fitting head of the machine during the fitting or removal process, allows the tyre bead enough purchase to pass over the rim flange.

Steel Disc

1.3.2 ALLOY WHEELS

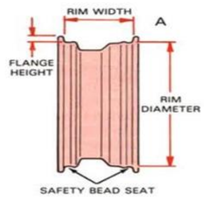
Alloy wheels are often incorrectly referred to as magnesium or "Mag" wheels. Magnesium is used in alloys. However, they are almost found only in racing rims meant for the track. Its brittle and highly flammable qualities make it unsuited as a road rim. Low pressure, die-casted aluminium alloy wheels are used and offer certain benefits over steel wheels. It is possible to design alloy wheels that alloy for the better air flow over the brakes and that are also slightly lighter and visually more appealing than steel wheels. Because alloy is lighter than steel, wider rims can be used without sacrificing unsprung weight.

Aluminium Alloy Wheel

1.4 STEPS INVOLVED IN PROJECT WORK

- Gathering all relative data for the design of wheel rim.
- Generation of model using CATIAV5.
- Importing the generated model to ANSYS for analysis work.

- Static analysis is carried out on the wheel rim to evaluate the performance.
- Modal analysis is carried out on the wheel rim.



Wheel Rim

1.4 MODELLING USING CATIA

1.4.1Modules of CATIA

1.4.1.1 Sketcher

A feature is able building block that describes the design, like a keyway on a shaft. Each feature indicates how to add material (like a rib) or remove a portion of material. Feature adjusts automatically to changes in the design there by allowing the capture of design intent. This also saves times when design changes are made. Because features have the ability to intelligently reference other features, the change made will navigate through design updating the 3D model in all affected areas.

In typical solid modelling software the designer can create a feature in two basic ways. One is to sketch of the shape to be added and then extrude, revolve or sweep it to create the shape. This are called sketched features.

Another type of the features is the pick and place feature. In this feature engineering operation such as placing a hole, chamfering, or rounding a set of edges, or shelling out the model is done.

1.4.1.2 Part

CATIA part enables you to design models as solids in progressive 3D solids modelling environment. Solid models are geometric models that offer mass properties such as volume, surface area and inertia. If you manipulate any model, a 3D model remains solid.

CATIA provides a progressive environment in which you create and change your models through direct graphical manipulation. You drive the design process for your project by selecting an object (geometry) and then choose a tool to invoke an action on that object. This object- action work flow provides greater control over the design of your models while allowing you to express your creativity. The user interface provides further support for this design process.

With this model, the context sensitive user interface guides you through the design process. After you choose an object an action, CATIA interprets the current modelling context and presents requirements and optional attempts to complete the task. This information is displayed in a non obstructive user interface called the dashboard that enhances your ability to directly work with your models by assessing your action and guiding you through the design process.

1.4.1.3 Assembly

The different parts developed in the PART module of CATIA are assembled in the ASSEMBLY module of CATIA. The components developed are assembled using the placement constraints available from the list in the component placement dialogue box.

1.4.2 Geometric Modelling

Computer representation of the geometry of component using software is called a geometrical model. There are three types of geometric modelling.
Those are
i. Wire-framing modelling
ii. Surface modelling
iii. Solid modelling

1.4.2.1 wire-frame modelling

2D Two dimensional representation used for a flat object.
2D.this goes somewhat beyond the two 2D capabilities by permitting a 3D dimensional object to be represented as long as, it has no side walls details.
3D, this allows for full three dimensional modelling of a more complex geometry.

Even three dimensional wire-frame representation of an object is sometimes in adequate for complicated shapes. Wire-frame models can be enhanced by several different methods. The figure shows the object with two possible improvements. The first uses dashed lines to portrait the rare edges of the object, those that would be invisible from the front. The second enhancement removes the hidden lines completely, those providing a less cluttered picture of the object for the viewer.

CATIA system has an automatic hidden lines removal feature while other systems require the uses to identify the line that are to be removed from view.

1.4.2.2Surface model

This is another enhancement of wire-frame model involves providing a surface representation which makes the object appear solid to the viewer. However the object is still stored in the computer as a wire frame model.

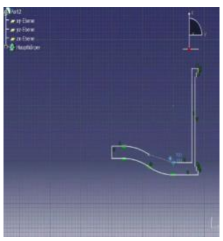
1.4.2.3Solid modelling

This is most advanced geometric modelling in three dimensions. This typically uses solid geometry shapes called picture to construct the object.

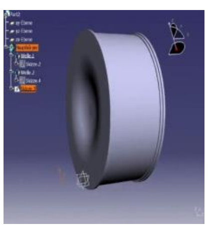
Another feature of the CATIA system is colour graphics capability. By means of colour, it is possible to display more information on the graphics screen coloured images help to clarify components is an assembly or highlight dimensions or host of other purposes.

1.4.3 STEPS INVOLVED IN DESIGN

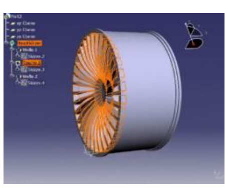
1. Draw the profile diagram of the wheel rim



2. Now revolve the profile body with respect to y-axis. Then we obtain the wheel rim body as



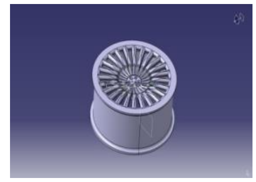
3. By selecting the face of wheel, the required design is drawn on the surface is removed by using POCKET operation



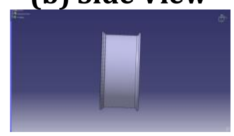
4. By using circular pattern the specific design is obtained all over the rim
5. Once again selecting the face draw the circle for and rotate them using
circular pattern.
6. Form holes using POCKET option.
7. And finally using the EDGE FILLET option the side edges are made filleted for final finishing.

1.4.3 VIEWS OF WHEEL RIM

(a) Top View



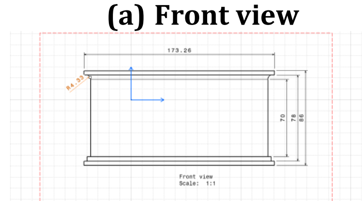
(b) Side View



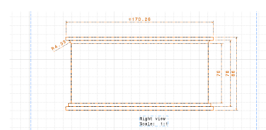
1.4.4 Drafting of Wheel Rim

Drafting is a process of making dimensions to a model in various views. Some of the drafted views of the wheel rim are as follows:

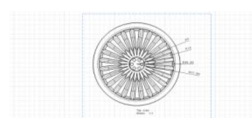
(a) Front view



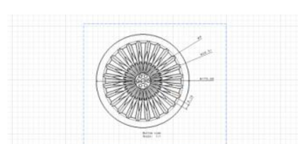
(b) Right view



(c) Top view

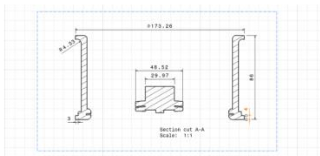


(d) Bottom view

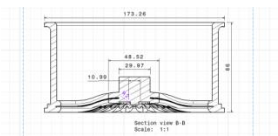


(e) Sectional views

i. Section cut A-A



ii. Section cut B-B



2.1 Finite Element Method (FEM):

FEM is the most popular numerical method.

Applications - Linear, Nonlinear, Buckling, Thermal, Dynamic & Fatigue analysis. FEM will be discussed in detail at later stage.

2.1.1 Software Based FEM

For using any commercial software there are 3 steps -

1) Preprocessing- Consumes most the out of the three steps.
2) Processing (or solution) - just click on "Solve"& it's the software's turn to do the job
3) Post processing- Result viewing & interpretation

2.1.2 Step 1 - Pre processing

a) CAD data
b) Meshing (or discretization to convert infinite dof to finite one)
C) Boundary conditions

In early stage of industrial applications of Finite Element Analysis, CAD, meshing & analysis al1 used to be carried out **by** a single engineer only. Soon it was realized that separation of the jobs &forming dedicated subgroups i.e. CAD group, Meshing group & Analysis or calculation group is necessary for optimum output and efficiency.

CAD & Meshing -There are specialized software's for CAD, Meshing & Analysis. CAD & meshing consumes most of the time For example - Typical time for a single person to mode1 (CAD) 4cylinder engine block is 6 weeks & for brick meshing 7 weeks (For tetra mesh about 2 weeks).

Boundary Conditions -Consumes least time but it is the most Important step (typically applying load cases is about 1 day job). 3 months hard work of meshing & CAD data preparation of engine block would be undone in just 1 day if boundary conditions are not applied properly.

After completion of preprocessing i.e., CAD, Meshing and Boundary conditions, software internally forms mathematical equations of the form [F] = [K] [δ].

2.1.3 Step 2 - Processing or Solution

During preprocessing user has to work hard while solution step is the turn of computer to do the job. User has to just click on solve icon & enjoy a cup of tea! Internally software carries out matrix formations, inversion, multiplication & solution for unknown e.g. displacement & then find strain stress for static analysis.

Today we are using FEA just because of availability of computers. FEM has been known to
Mathematicians & engineers right from late 50's but since solving so many equations manually was not possible, in true sense FEA got recognition only after emergence of high capacity Computers.

2.1.4 Step 3 - Post processing

Post processing is viewing results, verifications, conclusions & thinking about what steps could be taken to improve the design.

Consider a simple example which involves al1 the above Steps

Probably at the moment you are sitting on a chair or stool & reading this book. In this example
we will analyses the stool itself for stress & displacement for a load of 200 **kg** (assuming it could be used for sitting as well as supporting any object up to max. 200 kg wt.)

2.2 Types of analysis

1) Static analysis
2) Non linear analysis
3) Dynamic analysis
4) Buckling analysis
5) Thermal analysis
6) Fatigue analysis
7) Optimization
8) CFD analysis
9) Crash analysis
1 0) Vibration analysis (Modal analysis*).*

For this analysis I am using static analysis

2.2.1 static analysis

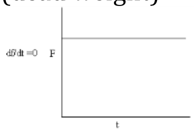
A static analysis calculates the effects of study loading conditions on a structure, which ignoring inertia and damping effects such as those caused by time varying loads. A static analysis can, however, includes study inertia loads(such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads(such as the static equivalent wind and seismic loads commonly defined in many building codes).

Static analysis is used to determine the displacements, stresses, strain, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Study loading and resonance conditions are assumed; that is, the loads and the structures response are assumed to vary slowly with respect to time.

Static:

There are two conditions for static analysis

1) Force is static i.e. no variation with respect to time (dead weight)



2) **E**quilibrium condition - **1** forces (F_x, $\mathbf{F_y}$, $\mathbf{F_z}$**)** and **1** Moments (M_x, M_y, M_z) = 0. FE model fulfils this condition at each and every node. For complete rnodel summation of external forces and moment is equal to reaction forces and moments.

2.2.2 Vibration analysis (Modal analysis).

Popular FEA codes provide for many types of vibration analyses. These allow the examination of the dynamic behaviour of structures. Some produce results that vary over time, and others give root-mean-square responses. Most dynamic analyses require a modal (natural frequency) analysis be performed on the model first. A few common types of vibration analysis are described below. Dynamic Time – time history. Involves real-time solution using a time stepping scheme. Damping is often important in this type of analysis. Often the type of damping model provided controls the methods effectiveness for low vs. high frequency vibrations. This type of simulation is good for examining the short-term effects of dynamic loads. Dynamic Frequency - involves a root-mean-square solution for repeating harmonic loads. The response generated by the results can be interrupted as the steady-state response of the structure. This can be used to see the long term effects of harmonic loads, such as machine vibrations. Dynamic Random – random vibration is used when the time-varying forces are of statistical nature. This type of simulation is most commonly used for transportation-induced vibrations. This type of simulation is most commonly used for transportation-induced vibrations. The input involves the power spectral density of the excitation. This input the mean square response of the system.

2.3 Commonly used Software's: Nastran, Ansys, Abaqus, i-deas NX, Radioss, Cosmos, UG, Pro-Mechanics, Catia etc.

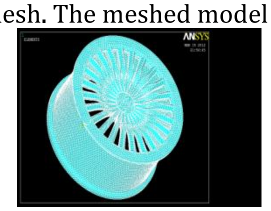
3.1 calculations

3.1.1 SPECIFICATIONS

a. Tyre diameter (approx) =560mm
b. Wheel size=14 inches
c. Length =86mm
d. Flange shape=J
e. Rim width=5 inches
f. Wheel type= disc wheel
g. Flange height= 0.68inches
h. Tyre type = radial
i. Aspect ratio=65
j. Off set=80.54

4.1 ANALYSIS USING ANSYS

1. After preparing the model in CATIA it is imported to ANSYS. The file is imported from CATIA by
   1. File>Import>IGES
2. The imported model is meshed by using TETRA mesh. The meshed model is as follows:



3. Later this meshed model is defined with two different materials namely ALUMINIUM and FORGED STEEL and subjected to static analysis.

4. **AT PRE-PROCESSOR STAGE**

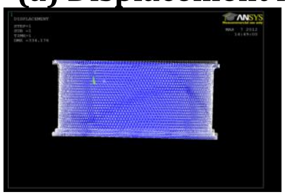
   Input data for ALUMINIUM:
   Young's modulus =0.71e5 N/mm^2
   Poisson ratio =0.33
   Density =2800 kg/m^3
   Circumferential pressure =200 kpa
   Input data for FORGED STEEL:
   Young's modulus =2.1e5 N/mm^2
   Poisson ratio =0.3
   Density =7600 kg/m^3
   Circumferential pressure =200 kpa

5. After this meshed model is constrained at holes by all DOF where the bolts has to be placed.
6. After constraining the meshed model, the model is subjected to a circumferential load of 200 Kpa.
7. Later the results were obtained in the SOLVER module.
8. In the similar way MODAL (VIBRATION) ANALYSIS is carried out. In modal analysis only all DOF is given, beyond that no external force is applied.
9. Later in the SOLVER module, analysis type is changed from static command to modal command and solution in done in solution window.
10. Next solution results such as Stress, Displacement, Von mises stress, ultimate strength etc.., can be observed in GENERAL POST-PROCESSOR.

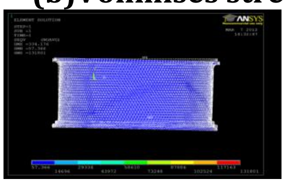
5.1 RESULTS AND DISCUSSION

5.1.1 RESULTS FOR ALUMINIUM WHEEL RIM

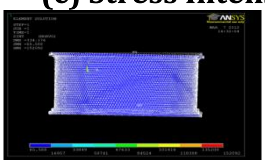
(a) Displacement result



(b)Vonmises stresses

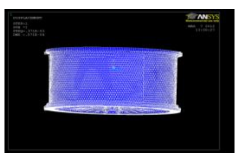


(c) Stress Intensity

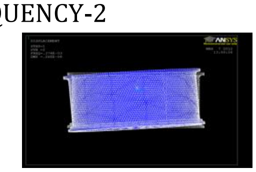


(d)Modal solution results:

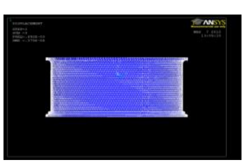
FREQUENCY-1



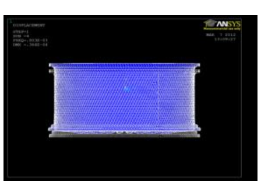
FREQUENCY-2



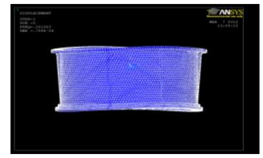
FREQUENCY-3



FREQUENCY-4

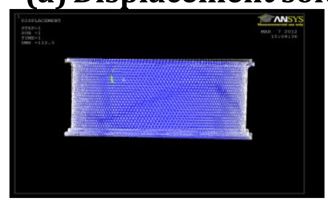


FREQUENCY-5

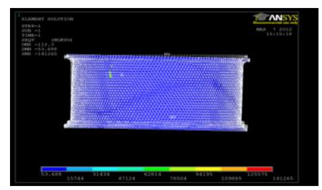


5.1.2 RESULTS FOR FORGED STEEL WHEEL RIM

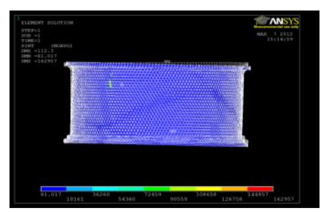
(a) Displacement solution



Von-mises Stresses

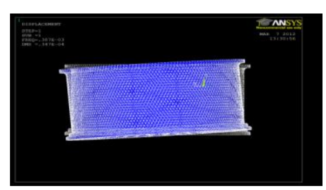


(b) Stress Intensity

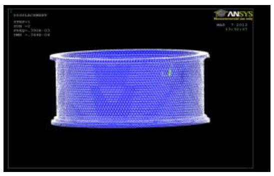


(c) **Modal Solution**

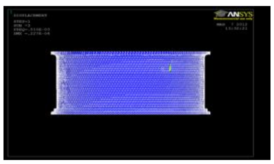
FREQUENCY-1



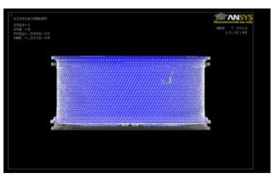
FREQUENCY-2



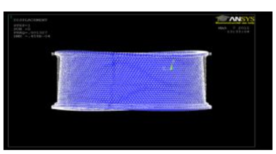
FREQUENCY-3



FREQUENCY-4



FREQUENCY-5



5.2 STRUCTURAL RESULTS

Type Of Result	For ALUMINIUM	For FORGED STEEL
Displacement (DMX value)	**334.176**	**112.3**
Von-mises stress	**131801**	**141265**
Ultimate stress Value	**152092**	**162957**

5.3 MODAL ANALYSIS RESULTS:

FREQUENCY SUB STEPS	DMX For Aluminium	DMX For Forged steel
1	**.571E-04**	**.347E-04**
2	**.568E-04**	**.344E-04**
3	**.375E-04**	**.227E-04**
4	**.364E-04**	**.221E-04**
5	**.758E-04**	**.495E-04**

6.1 CONCLUSION

CAD model of the wheel rim is generated in CATIA and this model is imported to ANSYS for processing work. An amount of pressure 200 Kpa is applied along the circumference of the wheel rims made of both ALUMINIUM & FORGED STEEL and bolt circle of wheel rims is fixed. Following are the conclusions from the results obtained:

1. Aluminum wheel rim is subjected to more stress compared to Forged Steel.
2. In both the cases Von-misses stresses are less than Ultimate strength.
3. Deflections in Aluminum are more when compared to Forged Steel.
4. Since in both the cases Von-misses stresses is less than the Ultimate strength, taking deflections into account , Forged steel is preferred as best material for designed wheel rim.
5. By observing modal analysis results frequencies 1 & 2 are considered to be safe in both cases.

6.1.1 FUTURE SCOPE OF THE PROJECT

1. In this thesis only pressure acting circumferentially on the wheel rim is only considered, this can be extended to other forces that act on the wheel rim.
2. In this thesis, only structural and modal analysis is carried out, this can be extended to transient analysis.

REFERENCES:

1. Mizoguchi, T,Nishimura, H., Nakata, K., and kawakami, J.,1982,"Stress analysis and Fatigue Strength Evaluation of Sheet Fabricated 2-Piece Aluminum alloy Wheels for Passenger cars" R&D,

Research & Development (Kobe Steel, LTD) ,Vol.32,n 2,pp,. 25-28.
2. Resiner , M., and DeVries, R., “Finite Element Analysis and Structural optimization of vehicle wheels, ” SAE Technical Paper Series, 1983.
3. Hoemson, R., “Structural Design of Agricultural Wheels,” American Society of Agricultural Engineers.
4. Lawerence.G and Miller.P., “Wheel Materials,” SAE technical publications 1988.