A study on structural health of bicycle frame using Finite Element Analysis
Bharati A. Tayade\(^a\), T.R. Deshmukh\(^b\)
\(^a\)PG student, Department of Mechanical Engineering
PRMIT&R Badnera, Amravati, India
tayade.bharati3@gmail.com
\(^b\)Professor, Department of Mechanical Engineering
PRMIT&R Badnera, Amravati, India.
trdeshmukh@mitra.ac.in

ABSTRACT:
Frame is very important part of bicycle as all the important accessories are mounted on the frame. The frame need to be very strong, stiff and light in weight, which is obtained by combining different materials and optimizing its shapes. The strength of frame construction is correct design of a frame because it is the most important part that ensures safe riding. This paper deals with the various design of bicycle frame. The modeling of bicycle frame is done in Computer Aided Design software CATIA and analysis of frame is done using the analysis software Ansys. This analysis is done by considering conditions like static start up, steady state paddling, vertical impact, horizontal impact, rear wheel braking etc. This paper gives us the stress, strain, factor of safety of particular bicycle frame.

Keywords: CATIA, FEA, Ansys, Bicycle frame, Impact.

I. INTRODUCTION
The innovation in the design of bicycle frame is still going on, the reason behind this is that the manufacturers and construction designers have innovative ideas related to minimize aerodynamic drag, to improve comfort, minimizing the mass of the frame, maximizing lateral stiffness in the load transfer from the hands and feet to the drive, maximizing the strength capabilities of the frame to allow for a higher load capacity or better load distribution, and adjusting the vertical compliance of the frame to tune the softness of the ride \([1,2]\) that are needed to provide rider comfort and safety ride. Most modern bicycle frames have the simple form. This shape emerged in about 1895 following several decades of vigorous development and evolution and it remained as basically unchanged since that time \([3]\). The trial and error method is used at that time but this method does not provide the relevant result and the intuition made is not necessarily all time correct so there is need of software that provide a way to get direct and appropriate result. This ultimately saves the cost and time of manufacturer. The solution is provided by Finite Element Analysis (FEA). FEA is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called field problems. The field is the domain of interest and most often represents a physical structure. The field variables are the dependent variables of interest governed by the differential equation. The boundary conditions are the specified values of the field variables on the boundaries of the field. Depending on the type of physical problem being analyzed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity to name only a few \([4]\).

II. LITERATURE REVIEW
As far back as 1986, Peterson and Londry (1986) used FEA to fine-tube the design of the Trek 2000 aluminium frame using two other designs (steel, aluminum) as performance benchmarks for mass, strength and stiffness characteristics. The model used beam elements to represent the tubular frame structure (excluding forks) with a variety of loading conditions to all frames to calculate their response characteristics \([5]\). In 1999, D. Arola et.al. explained the method used for an experimental evaluation of unique Prototype Bicycle motocross (BMX) frame \([6]\). In 2009, Thomas Jin-Chee Liu, Huang-Chieh Wu in their discussed the fiber direction and stacking sequence design for the bicycle frame of the carbon/epoxy composite laminates \([7]\). Alexandre Callens, André Bignonnet the methodology used for validation of bicycle frames and the fatigue strength prediction is excellent when compared to the standard tests \([8]\). The work presented by Derek Covill, et.al. outlined a FE model using beam elements to represent a standard road bicycle frame. The model simulates two standard loading conditions to quantify the vertical compliance and
lateral stiffness characteristics of 82 existing bicycle frames [9]. Recently in 2014, M. V. Pazare deals with the stress analysis of bicycle frame by using Finite Element Method. The analysis of frame is carried out in ANSYS software, and the F.E.A. results are compared with theoretical results. And it is found that there is good agreement between analytical and F.E.A results [3].

III. METHODOLOGY

The methodology used in this paper consists of modeling the bicycle frame in CATIA software and analyze the frame using the analysis software Ansys. In this paper the sample analysis of frame of Falcon Avon is presented here. CATIA is software which is used for creation and modifications of the objects. In CATIA, the design and modeling feature is available. Design means the process of creating a new object or modifying the existing one. Drafting means the representation or idea of the object. Modeling means creation 2D to 3D model. By using CATIA software, create the model of the bicycle frame. The modeling of various frames in CATIA is as follows.

3.1. Modeling of bicycle frame

![Figure 3.1: Modeling of Falcon Avon bicycle frame](image)

![Figure 3.2: Modeling of Sunami bicycle frame](image)

![Figure 3.3: Modeling of Foster bicycle frame](image)

![Figure 3.4: Modeling of Miss India Hero bicycle frame](image)

3.2. Analysis requirement

After modeling the analysis of frame is done in Ansys Software, for that purpose After preparing the model in CATIA it is improved to ANSYS, the file is imported from CATIA by file>import>IGES. To carry out the analysis various conditions are consider like, Static start up, Steady state pedaling, Vertical impact, Horizontal impact, Rear wheel braking. The input data for the analysis of bicycle frame Material IS2039 is as follows:

Young’s modulus: 2.e+005 MPa

Poisson’s ratio = 0.31

Density = 7.75e-006 kg/m³

Tensile Yield Strength =320 MPa

Tensile Ultimate Strength= 400 MPa

Physics type = Structural

Analysis type= Static structural, Solver target= Ansys mechanical
Force applied in various conditions:

Figure 3.5 (A) Static start up (B) Steady state pedaling (C) Vertical impact (D) Horizontal impact (E) Rear wheel braking

IV. RESULT ANALYSIS

The analysis of bicycle frame named Falcon (Avon) size 50.5cm (20") is done using following condition as follows:

4.1. Static start up
4.2. Steady state pedaling

Figure 4.2 (A) Deformation (B) Von Mises stress (C) Strain (D) Factor of safety

4.3 Vertical Impact

Figure 4.3 (A) Deformation (B) Von Mises stresses (C) Strain (D) Factor of safety

4.4. Horizontal impact
4.5. Rear wheel braking

Results in tabular form:

<table>
<thead>
<tr>
<th>Conditions</th>
<th>Total Deformation</th>
<th>Equivalent Elastic Strain</th>
<th>Equivalent (von-Mises) Stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>Static start up</td>
<td>1.9931e-002 mm</td>
<td>8.1359e-005 mm/mm</td>
<td>16.272 MPa</td>
</tr>
<tr>
<td>Steady state pedaling</td>
<td>1.554e-002 mm</td>
<td>1.0228e-004 mm/mm</td>
<td>20.456 MPa</td>
</tr>
<tr>
<td>Vertical impact</td>
<td>4.1937e-002 mm</td>
<td>2.7646e-004 mm/mm</td>
<td>55.292 MPa</td>
</tr>
<tr>
<td>Horizontal impact</td>
<td>0.10109 mm</td>
<td>3.1421e-004 mm/mm</td>
<td>62.842 MPa</td>
</tr>
<tr>
<td>Rear wheel braking</td>
<td>1.6719 mm</td>
<td>2.7345e-004 mm/mm</td>
<td>54.689 MPa</td>
</tr>
</tbody>
</table>

V. CONCLUSION

From the results of FEA, it is apparent that the stresses induced in the bicycle frame of Falcon Avon is least and the factor of safety is also well above the limit. Also the Von Mises stresses are less than ultimate strength for the material. Thus the design of bicycle frame is sturdy. The use of Ansys software makes the process of calculation fast and several iterations are permissible to arrive at the best possible results. The results are relevant provided the assumptions and boundary conditions are perfect.
References


