Analysis of Flywheel

Akshay P. Punde, [1] G.K.Gattani[2]

[1]Student, M.E. [CAD/CAM] Department of Mechanical Engineering Babasaheb Naik college of Engineering, Pusad,
[2]Associate Professor Department of Mechanical Engineering BNCOE, Pusad (445215), S.G.B.A.U. Amravati (MS)

Abstract: In present investigation, to counter the requirement of smoothing out the large oscillations in velocity during a cycle of a I.C. Engine, a flywheel is designed, and analyzed. By using Finite Element Analysis are used to calculate the stresses inside the flywheel, we can compare the Design and analysis result with existing flywheel

Keywords: *flywheel, stress analysis, FE analysis*

I. Introduction

A flywheel is an inertial energy-storage device. It absorbs mechanical energy and serves as a reservoir, storing energy during the period when the supply of energy is more than the requirement and releases it during the period when the requirement of energy is more than the supply A flywheel used in machines serves as a reservoir which stores energy during the period when the supply of energy is more than the requirement and releases it during the period when the requirement of energy is more than supply.

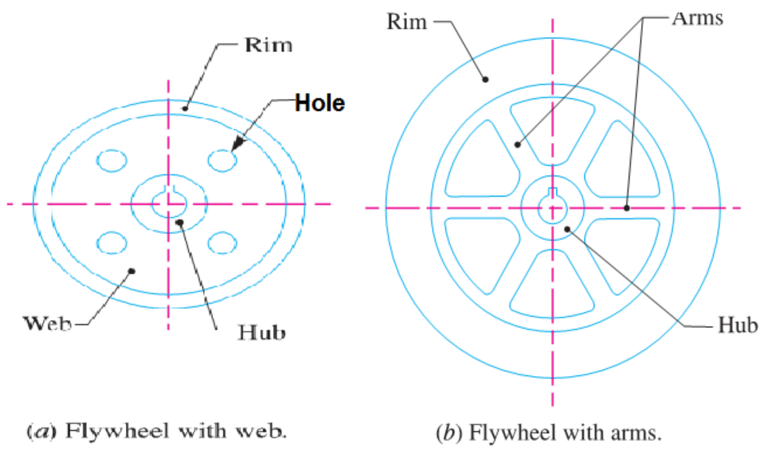


Fig-1 Types of flywheel

II. Literature Review

Literature review is an assignment of previous task done by some authors and collection of information or data from research papers published in journals to progress our task. It is a way through which we can find new ideas, concept. There are lot of literatures published before on the same task; some papers are taken into consideration from which idea of the project is taken.

In 2005 JohnA.Akpobi & ImafidonA.Lawani [1] have proposed, a computer-aided-designs of software for flywheels using object-oriented programming approach of Visual Basic. The various configurations of flywheels (rimmed or solid) formed the basis for the development of the software. The software's graphical features were used to give avisual interpretation of the solutions. The software's effectiveness was tested on a number of numerical examples, some of which are outlined in this work.

In 2012 Sushama G Bawane, A P Ninawe and S K Choudhary had proposed [2] flywheel design, and analysis the material selection process. The FEA model is described to achieve a better understanding of the mesh type, mesh size and boundary conditions applied to complete an effective FEA model.

Saeed Shojaei , Seyyed Mostafa Hossein Ali Pour Mehdi Tajdari Hamid Reza Chamani [3] have proposed algorithms based on dynamic analysis of crank shaft for designing flywheel for I.C.engine , torsional vibrasion analysis result by AVL\EXCITE is compared with the angular displacement of a desire free haed of crank shaft ,also consideration of fatigue for fatigue analysis of flywheel are given.

Sudipta Saha, Abhik Bose, G. SaiTejesh, S.P. Srikanth have propose [4] the importance of the flywheel geometry design selection and its contribution in the energy storage performance. This contribution is demonstrated on example cross-sections using computer aided analysis and optimization procedure. Proposed Computer aided analysis and optimization procedure results show that smart design of flywheel geometry could both have a significant effect on the Specific Energy performance and reduce the operational loads exerted on the shaft/bearings due to reduced mass at high rotational speeds.

Bedier B. EL-Naggar and Ismail A. Kholeif [5] had is suggested the disk-rim flywheel for light weight. The mass of the flywheel is minimized subject to constraints of required moment of inertia and admissible stresses. The theory of the rotating disks of uniform thickness and density is applied to each the disk and the rim independently with suitable matching condition at the junction. Suitable boundary conditions on the centrifugal stresses are applied and the dimensional ratios are obtained for minimum weight. It is proved that the required design is very close to the disk with uniform thickness.

III. Development Tools

3.1CATIA (Computer Aided Three-dimensional Interactive Application):

It is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Syste mes. Written in the C++ programming language, CATIA is the cornerstone of the Dassault Systemes Commonly referred to as a 3D Product Lifecycle Management software suite, CATIA supports multiple stages of product development from conce ptualization, design (CAD), manufacturing (CAM), and engineering (CAE). CATIA facilitates collaborative engineering acr oss disciplines, including surfacing & shape design, mechanical engineering, equipment and systems engineering.

CATIA can be applied to a wide variety of industries, from aerospace and defense, automotive, and industrial equipment, to high tech, shipbuilding, consumer goods, plant design, consumer packaged goods, life sciences, architecture and construction, process power and petroleum, and services.

3.2ANSYS

The ANSYS Workbench environment is an intuitive up-front finite element analysis tool that is used in conjunction with CAD systems and/or Design Modeler. ANSYS Workbench is a software environment for performing structural, thermal, and electromagnetic analyses. The class focuses on attaching existing geometry, setting up the finite element model, solving, and reviewing results. The class will describe how to use the code as well as basic finite element simulation concepts and results interpretation. The finite element method (FEM) is a method for dividing up a very complicated problem into small elements that can be solved in relation to each other. Its practical application is often known as finite element analysis (FEA)

IV. Modeling of Flywheel

There are generally two types of model used for analysis that are used in industry: 2-D modeling, and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively .So the flywheel needs to be modeled into a 3-D solid.
Maruti 800 Flywheel was selected for Modeling

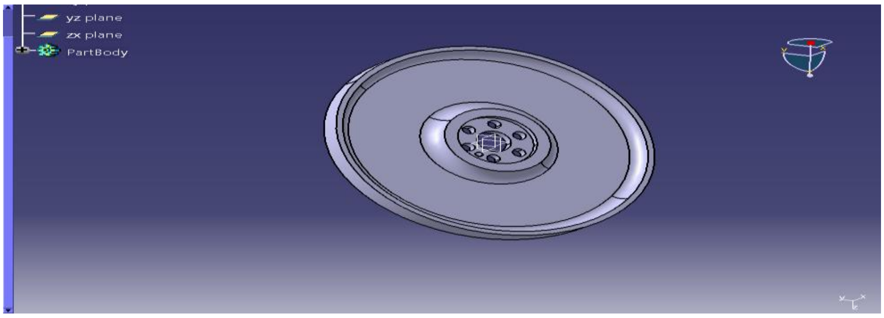


Figure 2: Model Of flywheel In CATIA

V. Analysis of flywheel with FEA

FEA (Finite Element Analysis) consists of a computer model of a material or design that is stressed and analyzed for specific results it is utilized to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

To build the physical system into a finite element model easily, some assumptions are needed:

Assumption 1: A rigid installation is used to connect the flywheel and its drive-shaft, no key-ways are needed to fix/drive the flywheel and there exist no slide, built-in stress and deformations on the connection surface; therefore, displacement constraints can be simply applied on the shaft hole;
Assumption 2: The flywheel only works in the vertical plane (X-Y plane) so that the gravity can be simply applied;
Assumption 3: The material used are isotropic;
Assumption 4: The aerodynamically resistance can be neglected;
Assumption 5: There exists no vibration;
Assumption 6: The fillets/chamfers can be neglected unless dimensioned

5.1 Material Properties

Two materials that are used for Design of Flywheel & their properties are given below table.

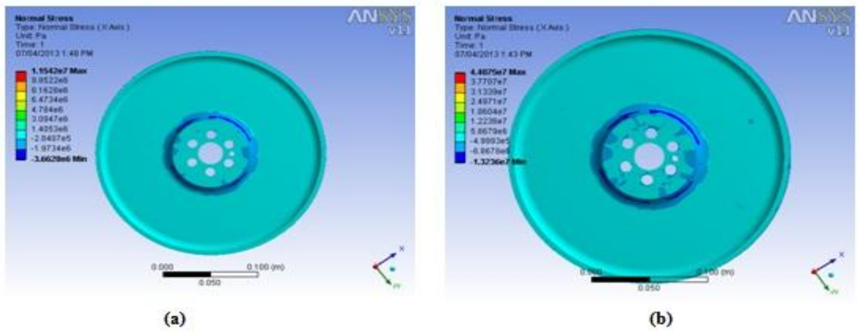
Table 1: Material properties

Material	Ultimate stress(MPa)	Density	Poissons ratio
Gray Cast Iron	214	7510	0.23
S-glass-Epoxy	4800	2000	0.25

5.2 Element Type

Based on the consideration of rotational deformations in the flywheel, the element Soilid72, a 3D 4-node tetrahedral structural solid with rotations, is used to model meshes. The element is defined by 4

Figure 3: Normal Stress in (a) S-Glass-Epoxy Flywheel (b) cast Iron Flywheel



Nodes with 6 DOFs at each node and well suitable to create irregular meshes It also has stress stiffening capability. But in ANSYS workbench Program control element are used.

5.3 Meshing method

Free mesh with smart element sizing is adopted to automatically and flexibly mesh the model. Compared to mapped mesh, which is restricted to only quadrilateral (area) or only hexahedron (volume) elements, free mesh has no restrictions in terms of element shapes. Smart sizing gives the user a greater opportunity to create reasonably shaped element during automatic element generation.

5.4 Boundary conditions and loads

Moment: MZ, applied on the nodes on the shaft-hole surface ANSYS 11 will be utilized for finite element analysis The ANSYS Workbench, together with the Workbench project and tabs, provides a unified working environment for developing and managing a variety of CAE information and makes it easier for you to set up and work with data at a high level.

VI. Results and Discussions

A structural analysis of was performed and normal stress are shown in fig 3. Flywheel was discredited into 19735 elements and 36539 nodes.
Table 2 shows that normal stress obtained for Gray cast iron is 44.07 Mpa and similarly for S glass Epoxy is 11.54 Mpa.

Table 2: Comparison result by ANSYS

Material	Normal stresses (Mpa)	Total Deformation (m)
Gray Cast Iron	44.07	$1.0484*10^{-3}$
S Glass Epoxy	11.54	$5.3399*10^{-4}$

VII. Conclusion

Based on the above work of flywheel and its optimization methods the following conclusion can be drawn.

It is clear that, cast iron flywheels are having higher Stress and deformation. S Glass Epoxy can be used in flywheels to store energy with less mass. It can be also used in high speed applications.