Explicit dynamic analysis of gear tooth of a synchromesh manual transmission gear box

SHREEKANTH R D [1], H S MANJUNATH[2], BYRE GOWDA K C[3], AMITH KUMAR S N[4]

[1]M.Tech Scholar, Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bengaluru
[2]Assistant Professor, Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bengaluru
[3]Assistant Professor, Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bengaluru
[4]Assistant Professor, Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bengaluru

---

Abstract - *New ideas and developed solution are the best compliments for the desired solution in order to avoid failure in gears and improve the service life using new technologies makes accomplishment of objectives. Material properties plays a very important role during the process we need to compare it with different other material properties .using simulation software all the desired functions can be checked analytically and mathematical ways .since the basis of experiments were on trial and error methods upon that we need to design some model using CAD software. Later we import through ABAQUS software, so simulation is done further. Since by using the software it makes work faster and accurate results can be achieved without any loss of data. It helps in realizing the stress levels in the desired manner.*

Key Words: *PET bottle; Structural Optimization; Light weight Design.*

1. INTRODUCTION

Gears are one of the primitive sector of human inventions surprisingly machines as that uses gearing as one of the kind of gear design considered for a specific importance place a very important role for the industrial point of view gear is considered as a important aspect of vibration analysis in order to realize the behavior of the gears when they are meshed quiet importantly it would perform through system monitoring and control of the gear transmission system basic understanding of gear mesh needs to be confirmed.

Stress are produced when gears are meshed together some kind of activities that takes place such as tooth bending and shearing this characteristic was finely noted at a nonlinear problem by applying codes and different elements and algorithms. Due to complicated condition solution converge results those are obtained are challenging some solution angular displacement gear pitch circle rolls on each other without slipping at constant torque load contact region similarly change can accomplish from single tooth contact to double contact region due to mesh cycle tool at line of action normal contact force of tooth mesh stiffness tend to act and gear transmission can be easily found out. Coupling among transverse motion and torsion of the system where linear mesh stiffness gives the feasible approach to learn coupling. Gear modelling and analysis was not only the important objective linear tooth mesh stiffness and finite element analysis we used to compare among those experiments conducted a aim is to validate gear box diagnostic methodologies we need to identify the vibration frequency associated with gear and bearing.

Fig -1: A typical multi-mesh gear system

2. METHODOLOGY

The main basic steps is create a geometric model of gear structure will be modeled using solid edge modeling tool. Once the model is created as per specification then it is imported into ABAQUS for meshing. Extract mid surface and mesh the model using shell elements. Then the performance linear analysis for bending loads using the ABAQUS solver and finally comparing the strength required for standards.

2.1 Dynamic analysis

Dynamic analysis carried out at different loading condition to get the stress level of the component

2.2 Material Properties

Material property for AISI 8630 steel

Density	7805 kg/m^3
Young's Modulus	210 GPa
Poisson's Ratio	0.3
Tensile yield Strength	550 MPa
Ultimate Tensile yield Strength	620 MPa

Table -1: Material Property

2.3 Geometric modeling

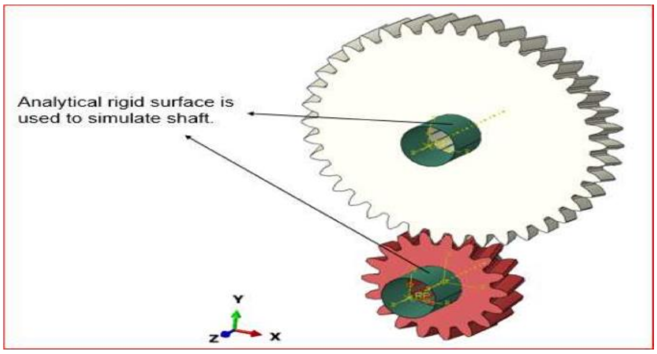
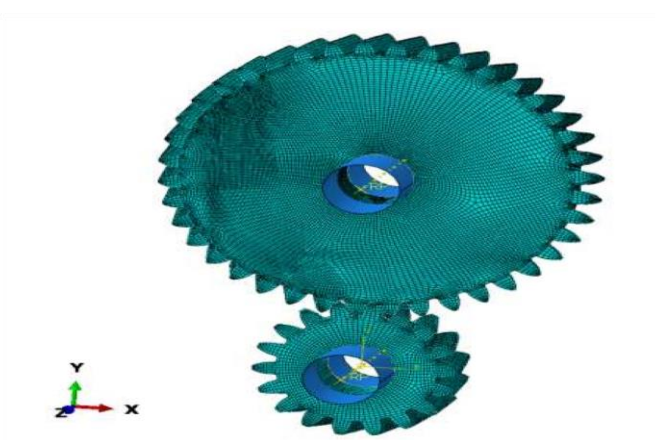
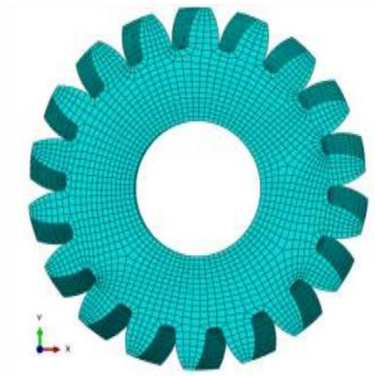


Fig -2: A typical multi-mesh gear system

The above figure 2 gives the representation of geometric modeling structure and The modeling has been done using the modeling software, SOLID EDGE. Solid works is a modeler which utilizes a parametric feature-based approach to create model and assemblies. Constraints are referred as parameters whose values determine the shape or geometry of the model or assembly. Parameters can be either numeric or geometric, numerical such as circle diameter or line lengths and geometric, such as parallel, concentric, tangent etc.
Model is built in solid works by 2D sketch. Then to define the size and location, dimensions are added to the sketch. Relations are used to define attributes such as perpendicularity, parallelism and tangency. Later drawings are created from parts.
Based on this drawing views are generated automatically and to this tolerance and dimensions are added.

After creation of model then this model will be imported to abaqus where meshing is done. During meshing elements are selected based on the requirements and the meshing of the model is done as shown below following the procedure.





```
Total number of nodes: 239278
Total number of elements: 209485
      209485 linear hexahedral elements of type C3D8R
```

Fig -3: FE Meshed model

The meshed model shown n the above figure is the finite element model of gear structure which is discredited using hexahedral element of type C3D8R. These elements are selected because of their ability to capture the geometry of any complex model. Since the model is of various size and shapes, hence hexahedral element is used. The meshed gear total number of node is 239278 and total number of the element is 209485.

2.4 Loads and Boundary conditions

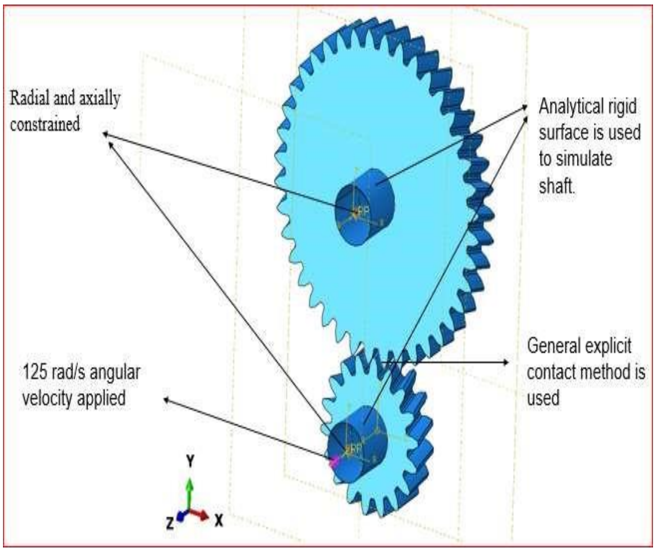


Fig -4: loads and boundary condition

The boundary condition is the one of the major important factors to govern the output result of FEA and loading condition or loads are forces, acceleration or deformations applied to a structure or its components. Load cause deformations, displacement and stress in structures. Applying the loads and boundary condition plays a significant role boundary applied whenever needed or required since that closely matched motion as loads acts at the system depending on the mannerism of the abaqus boundary condition are applied condition varies if explicit procedure had been chosen degree of freedom restricts for both pinion and gear except the rotational motion therefore it is free rotate the gear is axially and radially constrained then analytically rigid surface used to simulate shaft method is used and applied at 125 rad per sec of angular velocity .

3. RESULTS

3.1 Explicit Dynamic Analysis, Displacement

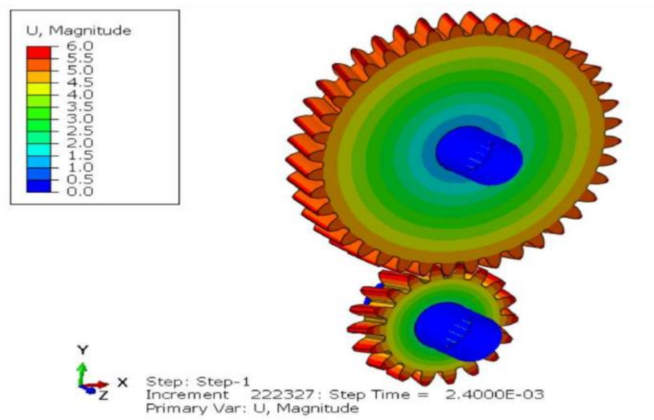


Fig -5: Explicit Displacement of Gears

The two gears are contact with each other and one gear rotation in clock wise direction and other is rotated in the antilock wise direction. The load is increases and corresponding to the stress is also increases. The displacement of the above model is 6mm with respect to the step time 2.400e-3

3.2 Stages of displacement

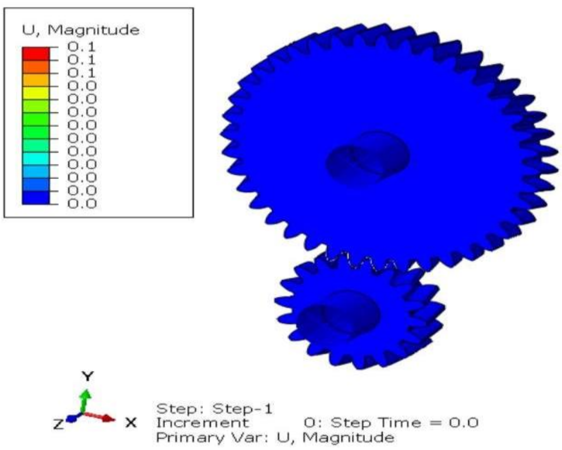


Fig -6: First stages of displacement of the gears

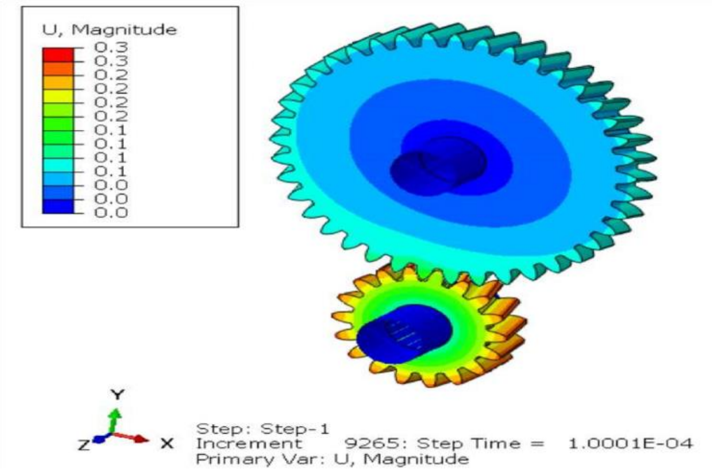


Fig -7: Second stages of displacement of the gears

The stages displaced are find out by using Abqus solver in Fig 6 shows the magnitude of the model is 0.1mm with respect to the step time 0 and Fig 7 shows second stage of the displacement is 0.3mm with respect to the step time 1.0001e-04 in the figure 3.2 you can see the gear and the pinion at the rest position the displaced activity is absolute zero and there is no variation in the movement of the gears hence there is no transient vibration .since the explicit methods is not applied here in the case of any vibration that is caused during the vibration in the figure 3.3 it undergoes a displacement so explicit methods are imbibed here since there should be some amount of accuracy is high and fast that can be easily realized. It is significant to understand the displacement where the exact location of gear approached and what position the gears are revoked the abaqus plays a important role in realizing the movements of the gear due to actual load condition.

3.2 Explicit dynamic stress analysis

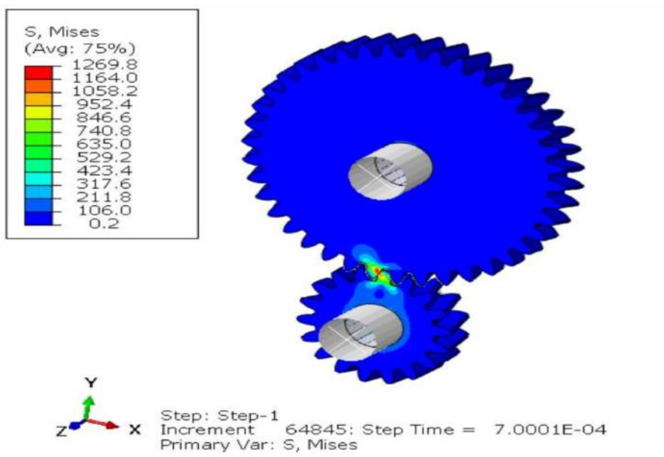


Fig -8: Explicit Dynamic Stress Analysis

Abaqus in the case of linear and nonlinear dynamic analysis work will be carried out at abaqus addition to it such as inertial affect and dynamic nonlinear system response to the torque input of the driving gear was successfully examined. In the case of nonlinear analysis there can be a choice between modes and direct integration methods. So as the performance of the program changes we can roll on to slightly nonlinear system where modal method can be performed using Eigen modes as a platform for calculating the response. Then commuting the problem becomes less expensive and simple to use the direct integration applied to all the degree of freedom. Maximum stress goes up to 1269.8Mpa.

3.3 Stages of Stress Development

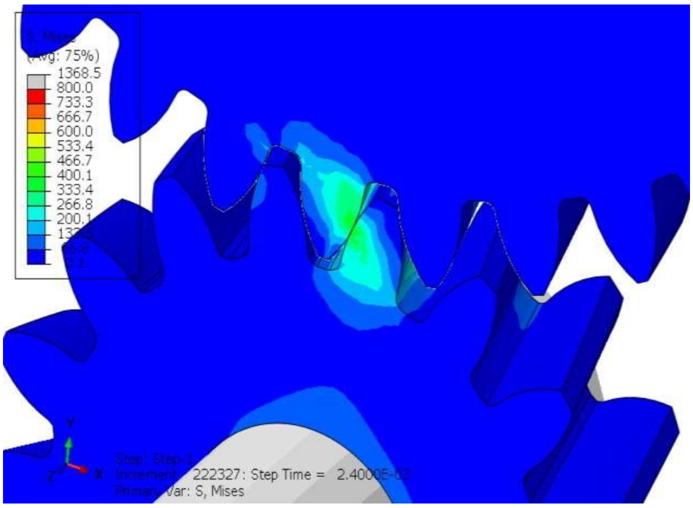


Fig -9: First Stages of Stress Development

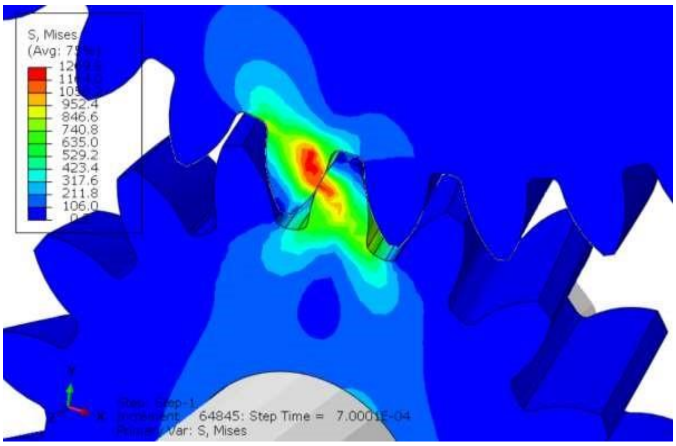


Fig -10: Second Stages of stress development

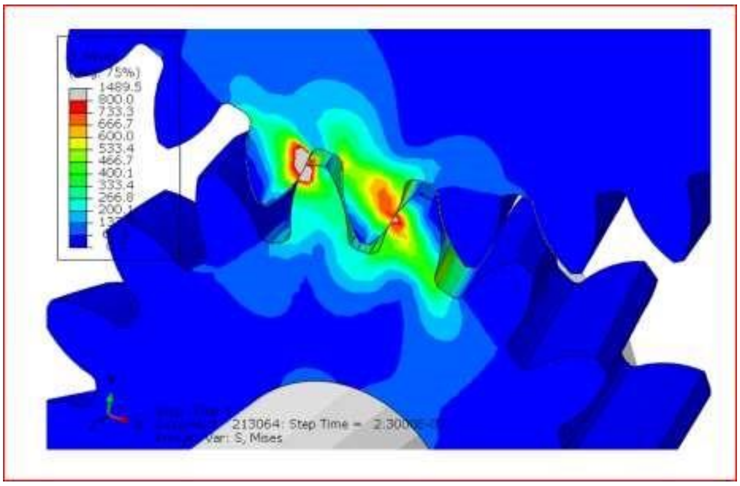


Fig -11: Third Stages of stress development

3.4 Contact pressure

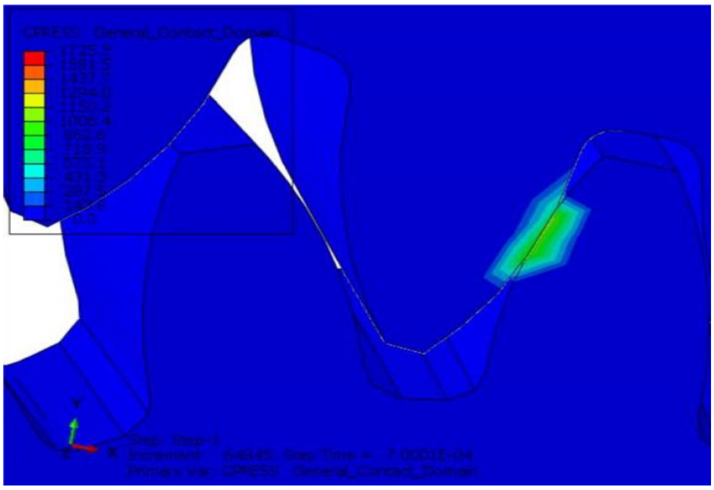


Fig -10: Contact Pressure

The contact stress created when surface of two bodies are bodies are pressed together by external loads are significant stress. The FEA results for the contact pressure distribution at different position along the contact path of gear teeth as shown above the figure. The variation of contact pressure on the gear tooth. Which shows similar trend to its load sharing pattern. The contact pressure developed two surface of gear contact with each other the maximum pressure developed is 1725.3Mpa using ABAQUAS software. Along the z direction the stress value is found to be zero, mean stress is found to be about 1006.4 Mpa .The increment values 64845 at the step 1 and inhibited step time would be 7.0001E0.4. Pressure depends upon the path of engagement and dis engagement of the gears and the line of contact of the stress is concentrated to one values. Since the pressure angle between the common tangent and common normal goes on decreasing due to the variation of the loads one of the stress components becomes zero in the z direction

4. CONCLUSION

The synchromesh gear system is analyzed for the strength and dynamic simulation of the gear movements is captured. Explicit dynamic analysis proves to be a great tool to study the time dependent behavior of the gears where the sequence of events is calculated for a very small time period of impact conditions. The high speed gear contact is found to be in accordance with the expected behavior. The contact stress developed is slightly higher at 1270 Mpa, owing to the nature of dynamic impact stress of up to two times the normal static stress which can be acceptable. The rotation sequence shows constant engagement and disengagement of the gears and subsequent stress transfer can be seen from the sequence of event. The contact pressure plot shows the normal contact which is desired is obtained. The stress values those are calculated during the engagement and dis engagement is said to be convinced with the desired values. Displacement during the action of the loads is said to be limited hence leads to the smooth operation of gears. The main objective is to obtain the stress value below 2000 MPA hence it is achieved

REFERENCES

1. S.Sai Anusha, P.Satish Reddy, P.Bhaskar, M Manoj "Contact Stress Analysis of Helical Gear by Using AGMA and ANSYS" Vol 2, Issue 12, 2014.
2. Mrs.Shinde S.P, Mr. Nikam A.A , Mr. Mulla T.S. "Static Analysis of Spur Gear Using Finite Element Analysis" ISSN: 2278-1684, PP: 26-31.
3. Ivana Atanasovska, Vera Nikolic-Stanojlovic, Dejan Dimitrijevic, Dejan
4. Momcilovic "Finite Element Model for Stress Analysis and Nonlinear Contact Analysis of Helical Gears" Vol.LVIX,No.1,2009.
5. Mohammad Asaduzzaman Chowdhury, Dewan Muhammad Nuruzzaman, Biplov
6. Kumar Roy "Experimental Investigation of Friction Coefficient and Wear Rate of
7. Stainless Steel 304 Sliding against Smooth and Rough Mild Steel Counter faces" 2013.