IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

# REVERSE ENGINEERING OF IC ENGINE CONNECTING ROD OF HERO- HONDA MOTORCYCLE ENGINE

**Suresh Batni*, Dr. M.L. Jain, Dr. Prashant Sharma**
*Sanghavi Inst. Of Management and Science, Indore
SGSITS, Indore
Pacific Inst. Of Technology, Udaipur

**DOI**: 10.5281/zenodo.49814

## ABSTRACT

The connecting rod is modeled from cloud points obtained by laser beam scanning. The points are joined by lines and then creating bounding areas and filling the volume.The laser beam scanning method for inspecting a reflective surface has been studied with analytical derivations and numerical modeling. Specular reflection off the surface was assumed and mathematical relations consistent with analytic geometry were used to trace the laser beam and predict its screen projection. It was found that deviation of the screen projection is proportional to the surface slope along the projection direction at the reflecting point and the distance from the point to the screen. The 3-D modeling was validated with 2-D analytical relationship as a special case and was tested with a profiled SMC (sheet molding compound) panel.

**KEYWORDS**: Connecting Rod, Laser beam, Modeling, The laser beam scanning method.

## INTRODUCTION

Surface quality is crucial for many industrial products. The surface flatness of silicon wafers for electronic components, of glass plates for optical instruments, of magnetic discs for disc storage systems, and of rolled metal sheets or strips for downstream processing, and the surface shape and smoothness of optical lenses or mirrors, of precision machined work pieces, and of automotive exterior body panels are just a few examples.

The connecting rod is modeled from cloud points obtained by laser beam scanning. The points are joined by lines and then creating bounding areas and filling the volume. Later the geometric solid model is imported to solid work package. Specification of Hero-Honda motorcycle:

150 cc -4 Stroke single cylinders, Air cooled
Max. B.P.=10.6 kW At 8500 rpm,
Displacement=149.2 cc
Bore Stroke=57.3x57.8 mm
Max. Torque = 12.8 Nm @ 6500 rpm

## LASER BEAM SCANNING

The following procedure is followed to obtain the cloud point data.

1. Clamp the connecting rod with three jaw chuck to restrict the degrees of freedom.
2. All surfaces are scanned separately and patches are created for each surface.
3. The surfaces and patches created in step ii, in IGES format, directly imported to ‘solid work’ to create the solid model.
4. Using the patches full extended surfaces are obtained foe each face of the connecting rod by extrapolating, extending, blending, intersecting and trimming operations of wire frame and surface design module of solid work to create a closed surface model.
5. The 3D solid model of the connecting rod is then created from closed surface model, and is important to ANSYS.

## SURFACE RECONSTRUCTION

Two main categories in surface reconstruction are free from fittings and quadric fitting. The surfaces meshes generated in these approaches are inadequate for current CAD/Cam software which typically require surface to be represented in the Non Uniform Rational B-Splines (NURBS) form. CAD model has the flexibility of incorporating changes and interfacing with CAM.[4] Thus through a CAD, one can achieve faster production rates. Surface Modeling and Solid Modeling of the part is done from the cloud point data using modeling packages like Solid Edge.

**What is the need to conduct FEA analysis of connecting rod?**

These are the possible reasons that necessitate conducting FEA analysis of connecting rod.

1. To check whether the connecting rod sustains the compressive gas force that is transferred through it at the point of combustion.
2. To find out the maximum stress induced due to compressive gas force
3. To find out the best configuration and suitable material for connecting rod.

Engine designer will be in the position to answer these questions only when he conducts FEA analysis of connecting rod.

## METHODOLOGY

**I. Design and analysis Connecting rod assembly** For any meaningful analysis of connecting rod, it is imperative that an appropriate piston and piston pin assembly is incorporated into the study since the dynamic performance of the connecting rod is a function of the contact pressure between the piston and cylinder, and the flexing and oval distortion of the pin which are all important criteria for assessment of whether the forces occurring can be transferred from the connecting rod to the pin safely. Due to the complex geometries and the interactions involved the finite element analysis approach in an attractive and easy means to understand such interactions. Consequently, in this study, an appropriate assembly was designed to suit the reversed engineered connecting rod. This connecting rod is shown in below. The model was meshed with Solid186 elements using standard mechanical shape checking in ANSYS workbench using 8144 nodes and 4471 elements. A preliminary mesh convergence analysis showed that this level of mesh refinement was adequate to perform the linear static analysis of the model.

***Fig. 1 Use of 3D White Light Scanner***

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

*Fig 2 Connecting Rod*

## II. Structural Static analysis Connecting rod assembly

Static Structural analysis is one in which the load/field conditions does not vary with time and the assumptions here is that the load or field conditions are gradually applied (Not suddenly applied). The most common application of FEA is the solution of stress related design problems. As a result, all commercial packages have an extensive range of stress analysis capabilities.

The behavior of the system could be either linear or non–linear as we would discuss in future.

Typically in a structural analysis the kind of matrices solved is: [K]*[X] = [F]

Where K is called the stiffness matrix, X is called the displacement matrix and F is the load matrix. This is a force balance equation. At times, the elements of matrix [K] are a function of [X]. Such a system is called non-linear system.

With the help of information about nodes, physical properties and material propert ies the stiffness matrix for individual elements is calculated and assembled to form the global stiffness matrix.

Then the known displacements are substituted in the displacement matrix. Also the force matrix is updated as per the loading conditions. This forms a complete set of simultaneous equations which upon solving give out the displacements at each node as results. The displacement values are then differentiated to evaluate the strain values. Then the strain values are multiplied by the Young's modulus to get the stress values. The same is explained using bar elements as follows:

The bar element has only one degree of freedom say, Ux shown in Fig.3.2. That is it can take either tension or compression.

If the length of bar element is "L" and if "A" is its area of cross section, the deflection under unit load is given by EA/L where "E" is the Young's modulus of the material. Then the stiffness matrix of the element "1" is given by: EA/L

From a formal point of view, three conditions have to met in any stress analysis, equilibrium of forces (or stresses), compatibility of displacements and satisfaction of the state of stress at continuum boundaries. The kind of loads that a system can experience here could be:

- Force load applied at one or several points.
- Pressure loads that can be distributed over one or multiple regions.
- Inertia loads due to motion as a result of velocity, acceleration or deceleration.
- Specified displacements applied at one or more locations. The outputs that can be expected from software are:
- Displacement at one or more points.
- Strains at one or more points.
- Stresses at one or more points.
- Reaction forces.

It all starts off with the formulation of the components 'stiffness' matrix. The square matrix is formed from details of material properties, the model geometry any assumptions of the stress-strain field (plane stress or strain).

Once the stiffness matrix is created, it may be used with the knowledge of the forces to evaluate the displacements of the structure (hence the term displacement analysis). On evaluation of the displacement, they are differentiated to give six strain distributions, 3 mutually perpendicular direct strains and 3 corresponding shear strains.

## LINEAR STEADY STATE SOLUTIONS

Despite the fact that all physical phenomena are non-linear and time dependent to some degree, linear static analysis remain the most useful and prolific form of FE analysis carried out today. The reason for its widespread use is that linear analyses are fast, oftentimes sufficiently representative of physical phenomena and the ease with which the analysis can be performed.

Linear analysis deals with problems in which the structural response is linear. Therefore, if the applied forces are doubled, then the displacements and internal stresses also double. Problems that fall outside this domain are usually classified as non-linear.

Static or steady state analyses are those where the solution is independent of time. Inertial forces are either ignored or neglected and so there is no requirement to calculate actual time derivatives. Problems that require inertial terms to be evaluated are usually classified as dynamic and/or transient analyses.

Linear static analyses are usually sufficient for situations where loads are known and the instance at which peak stress occurs is obvious. When performing a linear static stress analysis, the analyst applies static loads (forces, pressures or prescribed displacements) to the model.

As with all types of analyses, linear static ones are based on a set of assumptions. The main assumptions are listed here:

- All deformation and strain are small.
- Structural deformations are proportional to the loads applied. This infers that the loading pattern does not change due to the deformed shape and no geometric stiffening occurs due to the application of the load,
- All materials behave in a linear elastic fashion. Hence, the material deforms along the straight line portion of the stress-strain curve (no plasticity or failures occur). Highly localized stress concentrations are usually permitted as long as gross yielding does not take place.
- No boundary condition varies with time or application of load.

## RESULT

| S.No. | Analysis Perameter | Reverse engineering | | Forword engineering | |
|---|---|---|---|---|---|
| | | Max | Min | Max | Min |
| 1 | Total Deformation | 4.1341e-010 m | 0. m | 3.9164e-010 | 0. m |
| 2 | Directional Deformation | 9.3317e-011 m | -9.3175e-011 m | 9.2112e-011 m | -9.1962e-011 m |
| 3 | Equivalent Elastic Strain | 8.3255e-009 m/m | 2.0453e-014 m/m | 8.2641e-009 m/m | 2.597e-014 m/m |
| 4 | Equivalent Stress | 1653.8 Pa | 2.8669e-003 Pa | 1640.1 Pa | 1.7083e-003 Pa |
| 5 | Normal Stress | 1304.9 Pa | -1118.8 Pa | 1373.7 Pa | -1078.1 Pa |

1.Max total deformation in reverse engineering gives more value comparative FE because of the dimensional difference. The dimension of the connecting rod comes less than forword engineering design.
2.Normal stress , equivalent stress of reverse engineering are under permissible limit by considering same factor of safety.
3. Reverse engineering take lesser time than conventional method.
4. RE connecting rod weight get reduce, by which we can save the quantity of material under mass production.

## CONCLUSION

In this paper, we have focused on predicting the screenprojections of reflected laser scans off a surface whose topographyis known or prescribed. The reversed calculation, i.e.,constructing the surface topography from a given screen projection ,is more involved; a procedure similar to that in asubsequent paper (Lee, 1999b) could be used. It is recommended for future research..

\

[1] Brown, E. B,, 1965, *Modern Optics,* Reinhold Publishing Corporation, New York,p. 6.
[2] Clarke, D. A., Reynolds, R. L., and Pryor, T. R., 1986, "Panel Surface Flaw Inspection," U.S. Patent 4,629,319.
[3] Georg, G., 1984, "PALAPREG SMC, A Case of New Resin Systems Being Used for Automotive Parts," *Proceedings, 14th Reinforced Plastics Conference, British Plastics,* pp. 129-132.
[4] Gunter, J., 1983, "Method for Determining the "Long Term Waviness" of Large SMC Panels," SAE International Congress and Exposition, Detroit, Michigan, Paper No. 830142.
[5] Hupp, S. S., 1988, "A Quantitative Method for Analysis of Surfaces of Molded SMC Parts," 43rd Annual Conference, RP/C, SPI, Session 10-A.
[6] Hupp, S. S., and Hackett, T. B., 1988, "Quantitative Analysis of Surface Quality for Exterior Body Panels," SAE International Congress and Exposition, Paper No. 880358.Hupp, S. S., and Hackett, T. B., 1989, "The Characterization of Painted Surfaces for Exterior Body Panels," 44th Annual Conference, Reinforced Plastics/Composites