Dynamic Load Analysis Of A Connecting Rod

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Abstract: ***The main objective of this study was to explore weight and cost reductionOpportunities for a production forged steel connecting rod. This has entailed performing a detailed load analysis.The conclusion of this study that the connecting rod can be designed andOptimized under a load range comprising tensile load corresponding to 360Crank angle at the maximum engine speed as one extreme load, and compressive load correspondingto the peak gas pressure as the other extreme load.Furthermore, the existing connecting rod can be replaced with a new connecting rod made of C-70 steel that is 10% lighter and 25% less expensive due to the steel's fracture crackability.***

C-70= (C-0.7, Si-0.2,Mn-0.25)

Keywords: connecting rod, load

1. Introduction

The connecting rod is the intermediate member between the piston and the Crankshaft. Its primary function is to transmit the push and pull from the piston pin to the crank pin, thus converting the reciprocating motion of the piston into rotary motion of the crank. Existing connecting rod is manufactured by using Carbon steel. Aluminum alloys are preferred engineering material for automobile, aerospace and mineral processing industries for various high performing components that are being used for varieties of applications owing to their lower weight, excellent Thermal conductivity properties. Among several series of aluminum alloys, heat Treatable Al6061 and Al7075 are much explored, among them Al6061 alloy areHighly corrosion resistant and are of excellent extricable in nature and exhibits Moderate strength and finds much applications in the fields of construction (Building and high way), automotive and marine applications.

1 MODEL GENERATION

Creation of a 3-D model in CatiaV5R20 can be performed using three workbenches i.e., sketcher, MODELLING and assembly.

1.1 SKETCHER

Sketcher is used to create two-dimensional representations of profiles associated within the part. We can create a rough outline of curves, and then specify conditions called constraints to define the shapes more precisely and capture our design intent. Each curve is referred to as a sketch object.

1.2 CREATING A NEW SKETCH

To create a new sketch, chose Start→Mechanical Design→Sketcherthen select the reference plane or sketch plane in which the sketch is to be created.

1.3 SKETCH PLANE

The sketch plane is the plane that the sketch is located on. The sketch plane menu has the following options:

Face/Plane:With this option, we can use the attachment face/plane icon to select a planar face or existing datum plane. If we select a datum plane, we can use the reverse direction button to reverse the direction of the normal to the plane.

XC-YC, YC-ZC,and ZC-XC**:** With these options, we can create a sketch on one of the WCS planes. If we use this method, a datum plane and two datum axes are created.

2MODELLING

2.1 FEATURE CREATION

"Feature" is an all-encompassing term that refers to all solids, bodies and primitives used in CatiaV5R20 Form Features are used to supply detail to the model in the form of standard feature types. These include hole, slot, groove, pocket, rib and pad. We can also create our own custom features using the UserDefinedoption. All of these features are associative.

Reference Featuresallow creating reference planes, reference lines and reference points. These references can assist in creating features on cylinders, cones, spheres and revolved solid bodies. Reference planes can also aid in creating features at angles other than normal to the faces of a target solid. Dress up Featureoptions let's us modify existing solid bodies and features. These include a wide assortment of options such as edge fillet, variable fillet, chamfers, draft, offset face, shell and tapers.

Wire frame and Surface designlets us create surface and solid bodies. A surface body with zero thickness, and consists of a collection of faces and edges that do not close up to enclose a volume. Most Free Form Feature options create surface bodies.

2.2 CREATION OF SOLID/SURFACE BODIES

We can create solid bodies by padding the sketch geometry to create associative features or Creating primitives for the basic building blocks, then adding more specific features (for example, holes and slots etc.).

Shafting the sketch and non-sketch geometry lets us to create a solid body with complex geometry. This method also gives us total control over the editing of the body. Editing is done by changing the swept creation parameters or by changing the sketch. Editing the sketch causes the swept feature to update to match the sketch.

Dress-up features are used to modify the part bodies according to given specifications these are the most important features to modify the objects.

3MODELLING PROCEDURE

3.1 CONNECTING ROD & SHANK

To create the above part following features are used.

a) Pad
b) Pocket
c) Edge Fillet

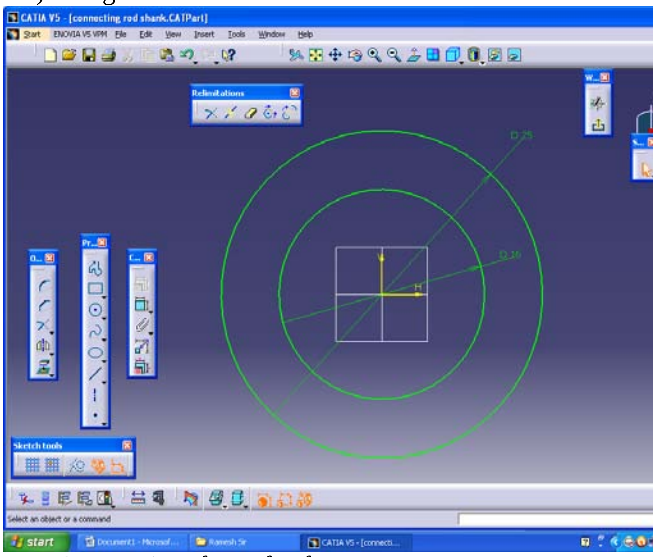


Figure 1 – user interface of software

Create a plane parallel to right plane with required dimension as shown below.

On the newly created plane draw a sketch as shown below and exit work bench. Use the pad tool to add the material normal to sketch with required dimension we get the part as shown in below fig On the top plane draw a sketch for shank as shown below and exit work bench. Use the pad tool to add the material

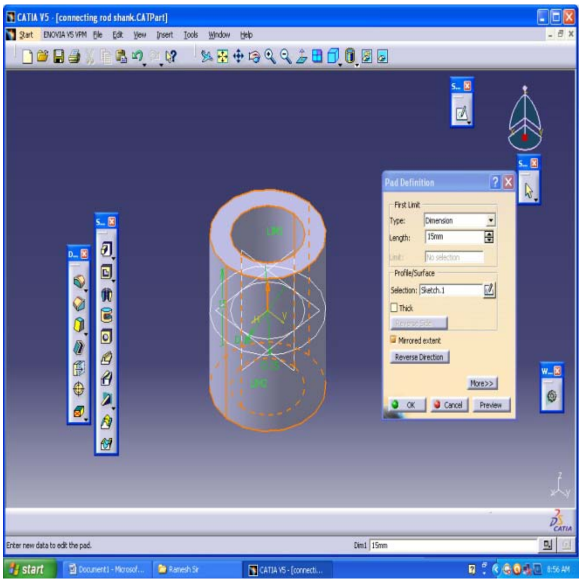


Figure 2 part modelling in CATIA

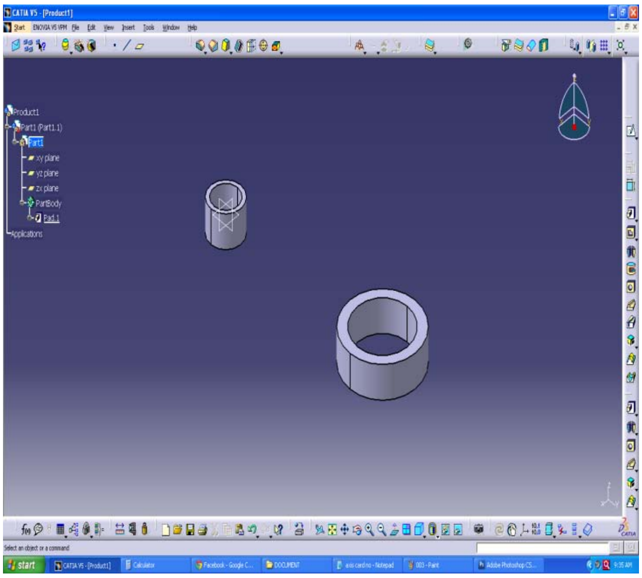
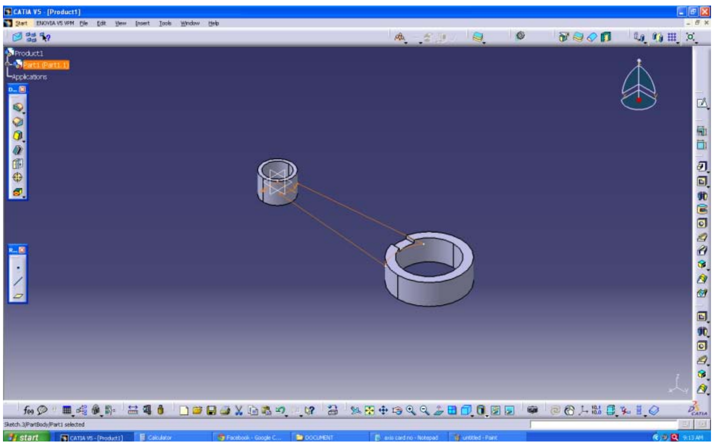


Figure 3- Making of shank

Next step is to create a shank between those to cylinder this can be done by taking normal plane which is perpendicular to those cylinder.



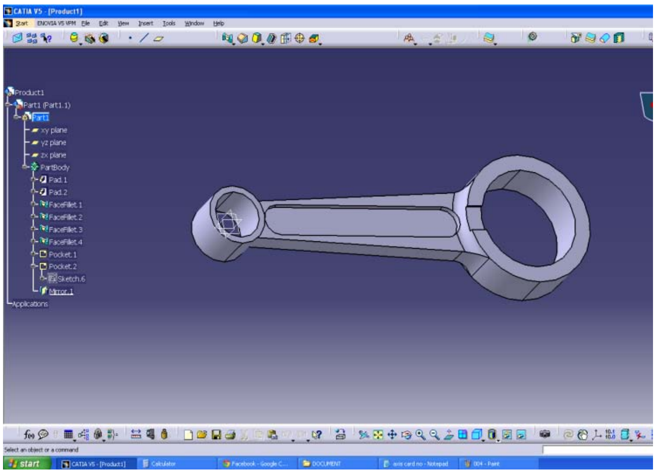


Figure 4- part creation

4CONNECTING ROD ASSEMBLY

After completing all the parts we will enter into assembly module In CATIA Assembly is also one of module to assemble the components whichever we designed in part MODELLING, in this we have two assembly methods one is bottom-up assembly and the other is top down assembly, in bottom-up assembly existing parts only can be assembled where as in top down assembly we can create parts in assembly module also, here we have already designed components so we follow bottom-up assembly.

To enter into catia assembly, open catia as usual then the default window is the assembly module and it shows the product window.

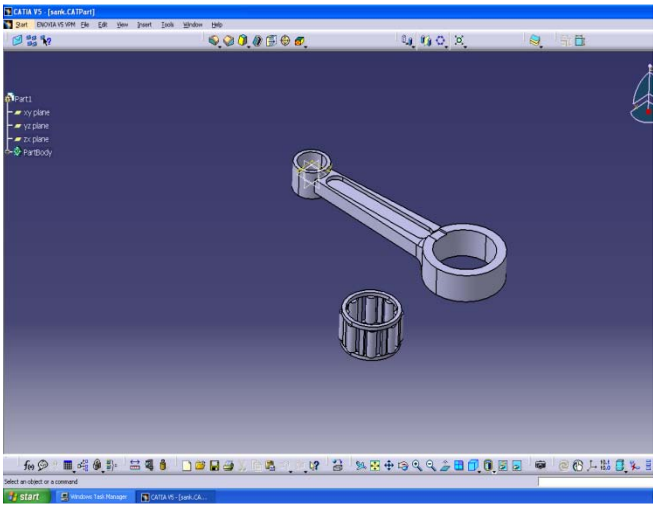


Figure 5- parts modelling

Now to insert the components go to insert menu – existing component – select product tree label it opens file selection window then browse the location of file and select – open it opens the part in assembly module. Now it is not constrained so to constrain the part use fix constraint tool from the constraints tool bar and select the component the component will be fixed at current location as shown below. (Used constraint is fixed constraint only).

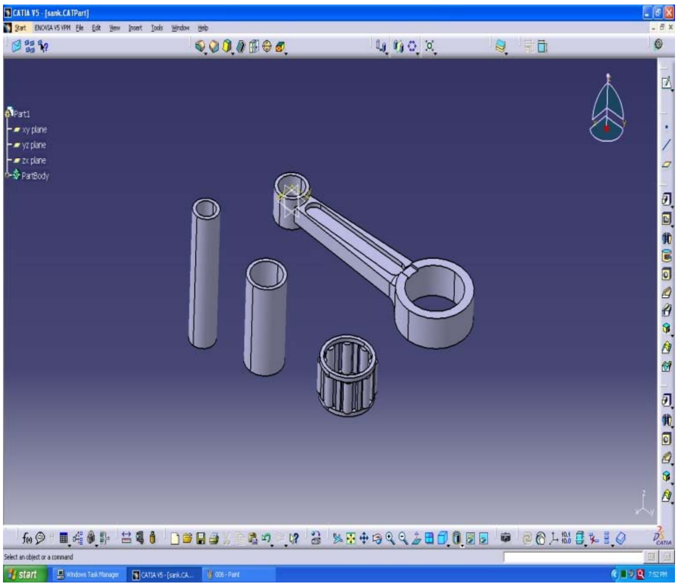


Figure 6- all features created

6FINAL ASSEMBLING

In this process all the components of connecting rod are assembled together.

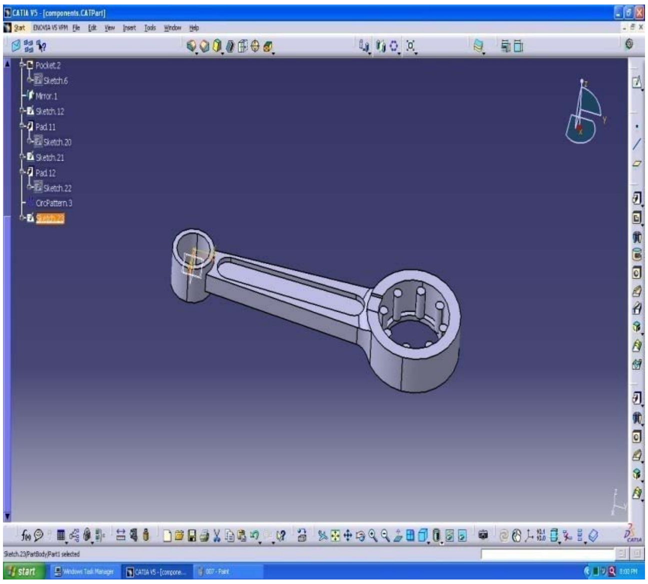


Figure 7- connecting rod

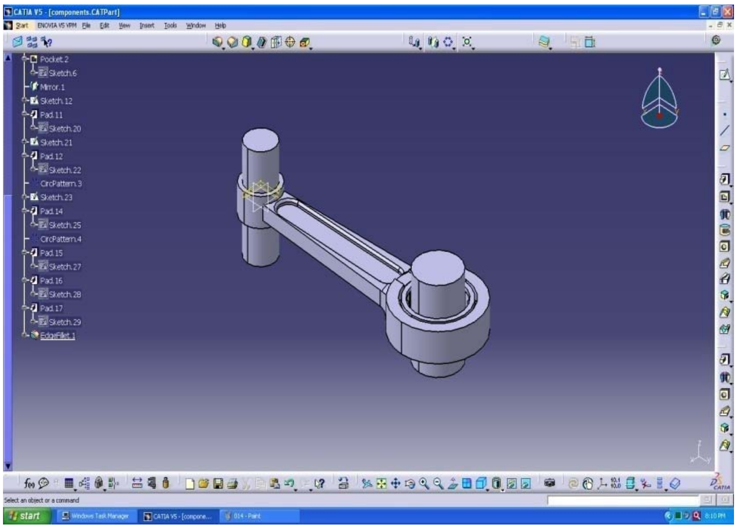


Figure 8- final assembling

7 ANALYSIS & DETAILS OF CONECTING ROD

Crankshaft radius 48.5 mm
Connecting rod length 141.014 mm
Piston diameter 86 mm
Mass of the piston assembly 0.434 kg
Mass of the connecting rod 0.439 kg
Is about the center of gravity 0.00144 kg m
Distance of C.G. from crank end center 36.44 mm
Maximum gas pressure 37.29 Bar
SLIDER CRANK MECHANISM-

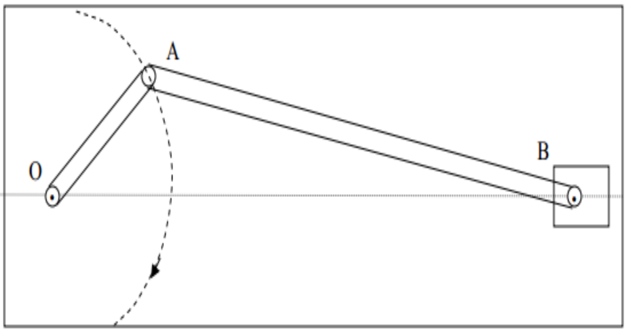


Figure 9- SLIDER CRANK ANALYSIS
Effect of speed on pv diagram at constant delivery ratio

8 FEA MODELING OF CONNECTING ROD

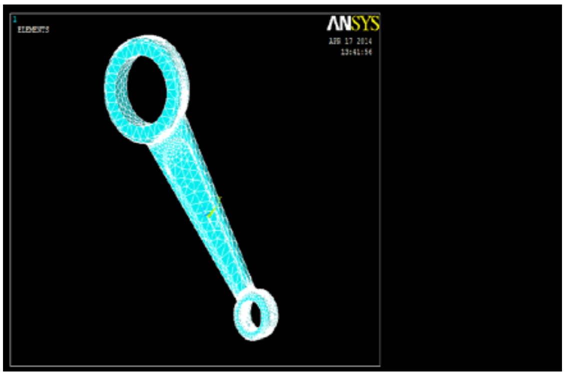


Figure 10- mesh generation of finite element model of connecting rod

Finite element mesh was generated using parabolic tetrahedral elements with various element lengths of 2.5 mm (20719 elements), 2 mm (37373 elements), 1.5 mm (77316 elements), and 1 mm (226409 elements).

8.1 Boundary condition-

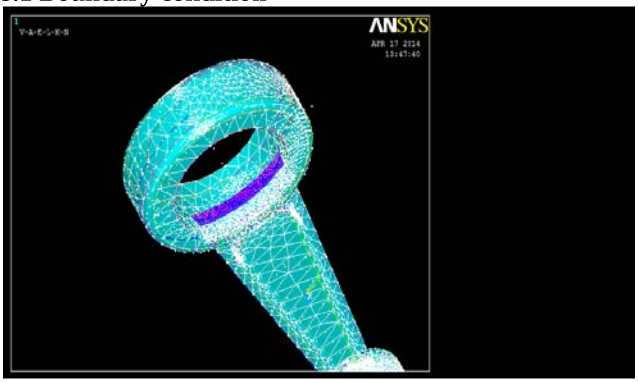


Figure 11- Force applied on partial surface

BOUNDARY CONDITIONS

Loading

Static FEA

The crank and piston pin ends are assumed to have a sinusoidal distributedloading over the contact surface area

This is based on experimental results . The normal pressure on the contact surface is given by:

p = po cos Θ

The load is distributed over an angle of 180o. The total resultant load is given by:

$$Pt = \int_{-\pi/2}^{\pi/2} po\,(\cos 2\Theta)\, r\, t\, d\Theta = po\, r\, t\, \pi/2$$

r, -radius of head
t –thickness of head
Θ.-total angle at which the load is applied

The normal pressure constant po is, therefore, given by:

po = Pt / (r t π/2)

The tensile load acting on expression from the force analysisthe connecting rod, Pt, can be obtained using the of the slider crank mechanism.compressive loading of the connecting rod, the crank and the piston pin ends are assumed to have a uniformly distributed loading through 120o contact surface,The normal pressure is given by:p = po

The total resultant load is given by:

$$Pc = \int_{-\pi/3}^{\pi/3} po\,(\cos\Theta)\, r\, t\, d\Theta = po\, r\, t\, \sqrt{3}$$

The normal pressure constant is then given by: po= Pc / (r t √3)

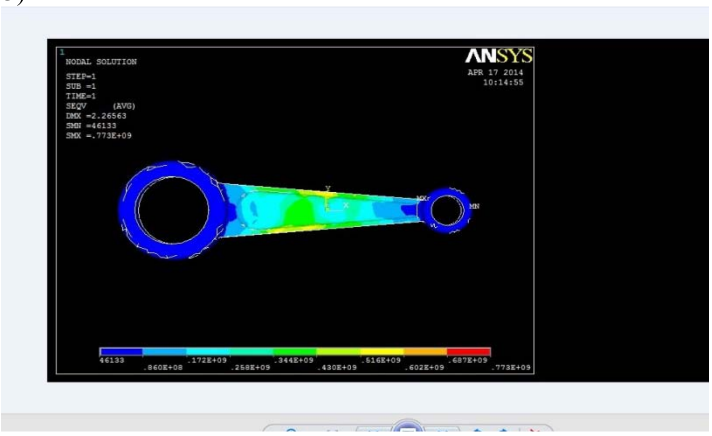


Figure 12- structural analysis

In this study four finite element models were analyzed. FEA for both tensile and compressive loads were conducted. Two cases were analyzed for each case, one withload applied at the crank end and restrained at the piston pin end, and the other with load applied at the piston pin end and restrained at the crank end. In the analysis carried out, the axial load was 26.7 kN (6 kips) in both tension and compression.

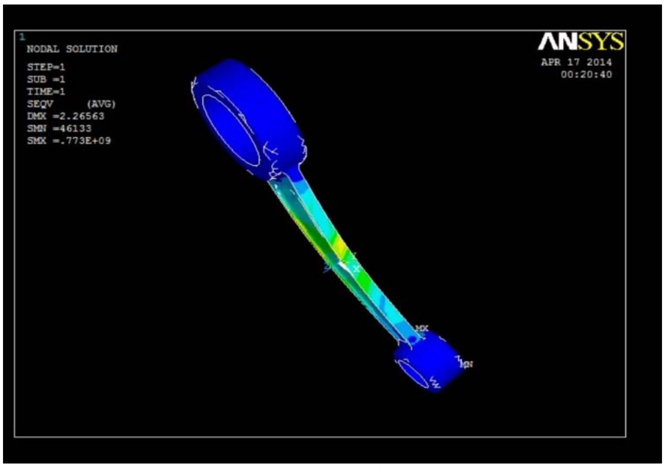


Figure 13 – nodal solution

9 RESULT

The pressure constants for 26.7 kN are as follows:

Compressive Loading:

Crank End:

po = 26700/ (24 x 17.056 x √3) = 37.66 MPa

Piston pin End:

po = 26700/ (11.97 x 18.402 x √3) = 69.98 MPa

Tensile Loading:

Crank End: po = 26700/ [24 x 17.056 x (π/2)] = 41.5 MPa

Piston pin End: po = 26700/ [11.97 x 18.402 x (π/2)] = 77.17 MPa

References

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