ANALYSIS AND OPTIMIZATION OF CONNECTING ROD BY FEA

[1]Mr.Ajit Lonkar
[1]M-Tech Student dept CAD/CAM
[1]Narayana Technical Campus, Telangana, India[Affiliated to JNTU, Approved by AICTE]

ABSTRACT: The automobile engine connecting rod is a high volume production, critical component. The main objective of this study was to optimize an alloy steel connecting rod for its weight. The optimized alloy steel connecting rod is intended to be more attractive option for auto manufacturers. Opportunities in reduction of eight are explored here. The idea behind optimizing is to retain just as much strength as is needed. Commercial software such as ANSYS was used to obtain the variation of quantities such as dimensional properties and loads. Objective of the optimization task was to minimize the mass of the connecting rod under the effect of load range. The stress analysis of the connecting is done using the prepared Finite Element Model. The connecting rod with the boundary conditions of compressive loading is optimized with the given design variables satisfying the maximum value of compressive stress. The optimized connecting rod is lighter than the original one by the amount of 12.33 %. Therefore there is weight reduction and cost reduction of the connecting rod. Also because of the reduced weight there is reduction in the inertia forces and so the operating cost of the engine will also be reduced.

1. BACKGROUND

The automobile engine connecting rod is a high volume production, critical component. It connects reciprocating piston to rotating crankshaft, transmitting the thrust of the piston to the crankshaft. Every vehicle that uses an internal combustion engine requires at least one connecting rod depending upon the number of cylinders in the engine. Connecting rods for automotive applications are typically manufactured by forging from either wrought steel or powdered metal. They could also be cast. However, castings could have blow-holes which are detrimental from durability and fatigue points of view. The fact that forgings produce blow-hole-free and better rods gives them an advantage over cast rods. Between the forging processes, powder forged or drop forged, each process has its own pros and cons. Powder metal manufactured blanks have the advantage of being near net shape, reducing material waste. However, the cost of the blank is high due to the high material cost and sophisticated manufacturing techniques with steel forging, the material is inexpensive and the rough part manufacturing process is cost effective. Bringing the part to final dimensions under tight tolerance results in high expenditure for machining, as the blank usually contains more excess material so in order to reduce the material cost and thus production cost it is better to optimize the weight or volume. And thus due to its large volume production, it is only logical that optimization of the connecting rod for its weight or volume will result in large-scale savings. It can also achieve the objective of reducing the weight of the engine component, thus reducing inertia loads, reducing engine weight and improving engine performance and fuel economy.

2. LITERATURE REVIEW

The connecting rod is subjected to a complex state of loading. It undergoes high cyclic loads of order of 10^8 to 10^9 cycles, which range from high compressive loads due to combustion, to high tensile loads due to inertia. Therefore, durability of these components is of critical importance. Due to these factors, the connecting rod has been the topic of research for different aspects such as production technology, materials, performance simulation, fatigue etc. For the current study, it was necessary to investigate finite element modeling techniques, optimization techniques, developments in production technology, new materials, fatigue modeling and manufacturing cost analysis. This brief literature survey reviews some of these aspects.

We modeled the connecting rod of 35 c.c. two stroke single cylinder engine with ANSYS software by parametric modeling (command line interfacing) and thus we performed three-dimensional finite element analysis of connecting rod under static compressive load and static tensile load by giving standard boundary conditions and thus we got the structural behavior of connecting rod under the static load conditions. The approach used finite element routine to first calculate the stresses and displacement in the rod. After the static finite element analysis of connecting rod we proceed to optimize the connecting rod. On the basis of study of static finite element analysis of connecting rod we concluded that there is much scope of weight reduction in existing connecting rod. Therefore after static finite element analysis, optimization of connecting rod of weight was carried out under specified limits. Objective of the optimization task was to minimize the mass of the connecting rod under the effect of load range. Comprising the two extreme loads, the peak compressive gas load and then tensile load at crank end. The weight of the new connecting rod or the "optimized connecting rod" was found to lower than the existing connecting rod. But this may not be minimum possible weight under the set of constraints defined. What has been attempted here is an effort to reduce the weight of connecting rod under specified inputs. The following factors have been addressed during optimization: Buckling load factor, the stress under loads dimensional parameters, all of these have been checked to be within permissible limits. So optimization section discusses the constraints under which weight was reduced and how the optimized connecting rod compares with existing one. And the study results in optimized connecting rod is 12.33% is lighter as compared to the existing connecting rod.

3. OBJECTIVE AND OUTLINE

The main objective of this study was to optimize an alloy steel connecting rod for it's weight. The optimized alloy steel connecting rod is intended to be more attractive option for auto manufacturers. Opportunities in reduction of eight are explored here. The idea behind optimizing is to retain just as much strength as is needed. Commercial software such as ANSYS was used to obtain the variation of quantities such as dimensional properties and loads.

The following objectives were carried out with ANSYS software:

- Modeling of connecting rod of 35c.c single cylinder two stroke engine. Complete model of the connecting rod done using ANSYS software. As the model is symmetric about two planes, it is further reduced to one forth. Then this quarter model is used for the analysis.

- Static finite element analyses of connecting rod under static compressive load and static tensile load. There is compressive force acing on the piston and so on connecting rod due to gas pressure, at some phase of the cycle and the tensile load acting due to tensile load acting on it due to inertia of the moving components. As the load is varying from compressive load to tensile load, Maximum compressive load and maximum tensile load conditions are two cases taken for the analysis. The details of boundary conditions (how loads and restraints are applied to the model) are discussed in the next chapters. It also discuses the various stresses coming in two cases at various locations.
- Optimization of connecting rod. Optimization is done to reduce the weight and the cost of the connecting rod, which is subject to constraints provided by the OEM. The first order method is used in ANSYS Optimization subroutine.

4. FE MODELING OF THE CONNECTING ROD

4.1 GEOMETRY OF CONNECTING ROD:

This chapter discusses geometry of connecting rod used for finite element analysis, its generation, simplification and accuracy. A solid model of the connecting rod of 35 C.C. two stroke single cylinder SI engine is shown in figure is generated using ANSYS using PARAMETRIC MODELING. In the case of connecting rod the degree of non-symmetry in the shank region, when comparing the areas on the either side of axis of symmetry perpendicular to the connecting rod length and along the web, was about 5%. This non-symmetry is not the design intent and is produced as a manufacturing variation. Therefore, the connecting rod has been modeled as symmetric component. For the simplicity point of view fillet and chamfers have been ignored at the transition end of pin end and transition end of crank end. Figure shows crank end, pin end, shank with slot, oil hole, slots on crank end and pin end. The theoretical weight of connecting rod calculated is 43.92 grams.

Various dimensional parameters are shown in the table as below.

Sr. No.	Design variable	Variable Description	Default value for this model
1	Lng	Connecting rod length	65
2	Bid	Crank end inner dia.	17
3	Bodb	Crank end bottom outer dia	24
4	Bodt	Crank end top outer dia	23
5	Sid	Pin end inner dia	13
6	Sodb	Pin end bottom outer dia.	19.5
7	Sodt	Pin end top side outer diameter	19
8	Ethk	End thickness	10
9	Shthk	Shank Thickness	6
10	Shsbd	Shank slot big dia	8
11	Shssd	Shank slot small dia	5
12	Shsthk	Shank slot thickness	1.5
13	Bsthk	Crank end slot thickness.	2
14	Bswdth	Crank end slot width	4.5
15	Shssd	Pin end slot thickness	2
16	Sswdth	Pin end slot width	5.75
17	Ohd	Oil hole dia	2.5
18	Ohtheta	Oil hole angular position from symmetry axis	135
19	Ced	Pin end circular extrusion dia	3.6

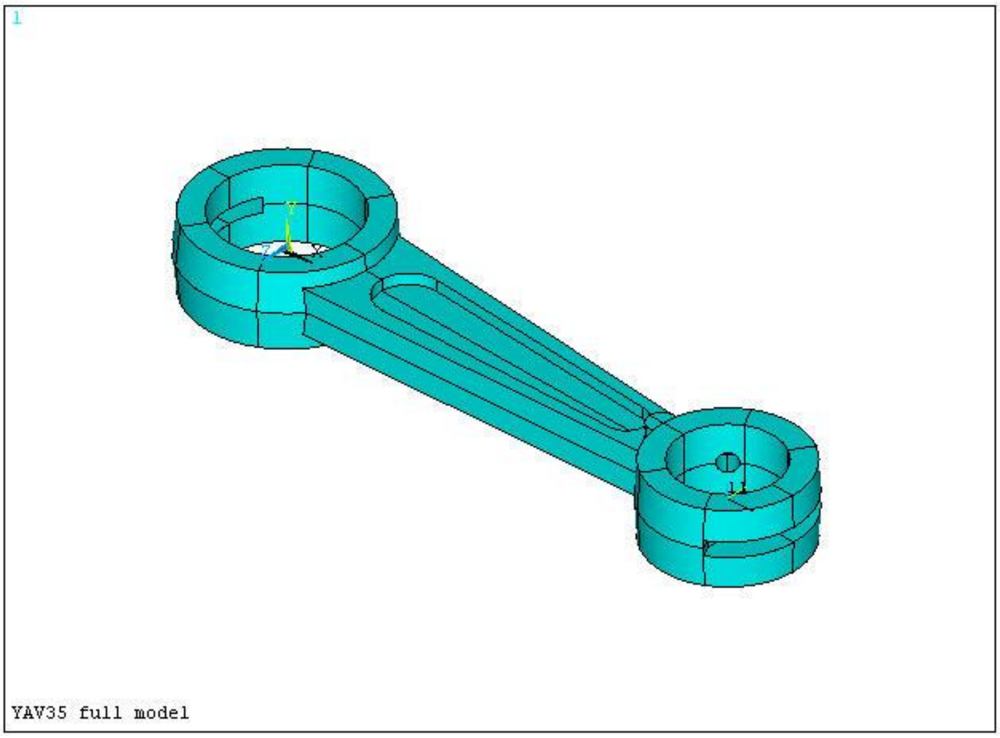

Figure 4.1: Full model of the connecting rod

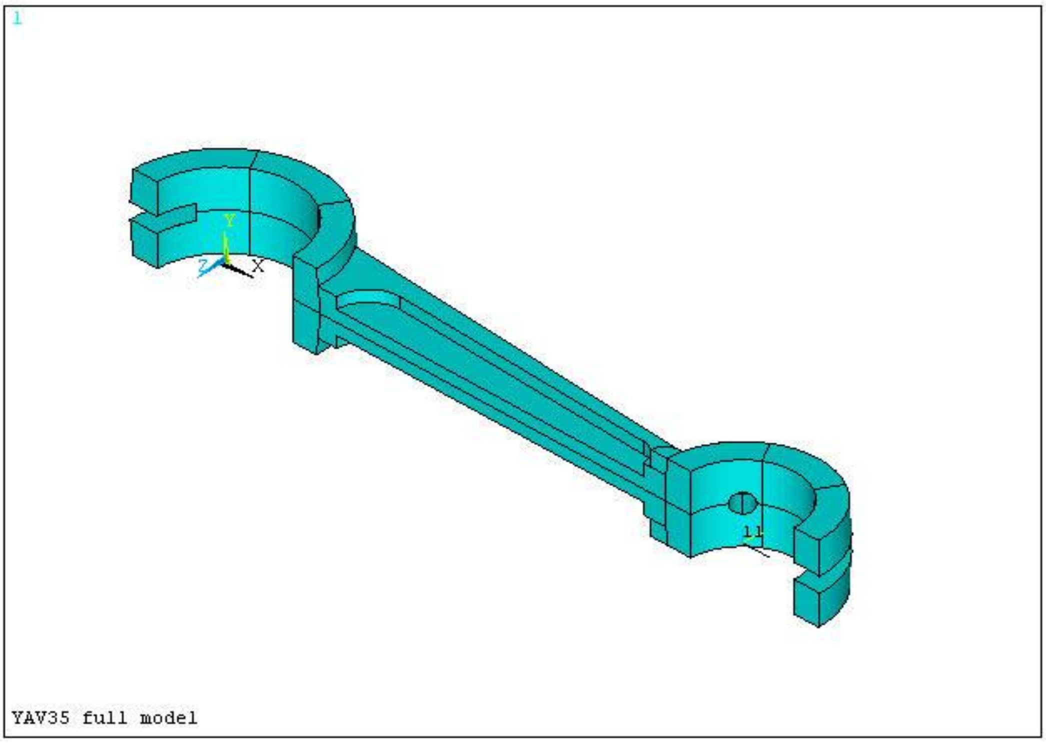

Figure 4.2: Half model of the connecting rod

4.2 PARAMETRIC MODELING OF CONNECTING ROD

Generally modeling in the ANSYS is performed by using two approaches

1. GRAPHICAL USER INTERFACE (GUI)

2. COMMAND LINE INTERFACE (CLI)

From the above two approaches command line interface is more accurate for the modeling of connecting rod in ANSYS, command line interface is more accurate. Therefore, for the current study we have performed command line interface to model the connecting rod of 35 C.C two stroke single cylinder S.I. engine. Various dimensional parameters as shown in the table are fed into the program, thus this makes the parametric approach under command line interface, and thus programming is called PARAMETRIC MODELING. One of the main advantages of parametric program modeling is that once the basic program for the modeling is generated we can model the connecting rod of various sizes by just changing the values of the parameters in program. Thus, modeling has been achieved using geometric programming technique (i.e. parametric modeling).

5. STATIC FINITE ANALYSIS OF CONNECTING ROD UNDER COMPRESSIVE LOADS

Figure shows quarter model in which compressive load is applied at the pin end and crank end is restrained. When the connecting rod is under axial compressive load, 120 degree of contact surface area is totally restrained (i.e. X,Y,Z, translation of all the nodes on the surface are set to zero, if connecting rod is in compression) Figure shows the compressive force is assumed to be uniformly distributed, which is also called cosine distribution. The uniformly distributed load has been applied 60 degree on either side of the direction of the resultant load, whereas for the full model area will be for 120 degree.

5.1 BOUNDARY CONDITIONS

A loaded body or structure is free to experience unlimited rigid body motion unless some supports or constraints are imposed that will ensure the equilibrium of the loads. These loads and constraints are the boundary conditions.

Since in structural static loads analysis the compressive loads cause maximum effects on the connecting rod then tensile loads.. Figure shows quarter model in which compressive loads are applied at the pin end and crank end is restrained. The compressive force is assumed to be uniformly distributed which is also called cosine distribution. As shown in figure, for the quarter model, uniformly distributed load and constraints are applied 60 degree either side of the direction of the resultant load, but for the full model connecting rod compressive load at the pin end and constraints at the crank end are applied 120 degree.

LOADING:

Figure show quarter model of connecting rod. The crank end and piston end are assumed to have a cosine distribution loading over the contact surface area of angle 60 degree, for quarter model, under static compressive loading (for full model, contact surface area will be for angle 120 degree). But there are very small amount of variation in results is observed when the constant pressure is applied instead of cosine loading. So the constant pressure is applied using surface load in ANSYS The pin end of the connecting rod has been applied standard gas force for 60 degree (as it is quarter model)

Gas force at pin end F= 951 N

Area of compression A=31.568

Therefore, the Normal pressure on the contact surface is given by

Normal pressure P= F/A

=951/31.568

CONSTRAINTS:

Since connecting rod is under static compressive loading, thus the crank end of the connecting rod has been restrained for 120 degrees. The FE model here is the quarter model so the all nodes of the area of 60 degrees from the axis of symmetry are selected and all three degrees of freedom (translations in x, y and z directions) are fixed (constrained) as shown in figure.

PROPERTIES OF CONNECTING ROD MATERIAL

Parameter	Unit	Scalar value
Modulus of elasticity	N/m^2	2.1e5
Poisson's Ratio	Unitless	0.30
Mass Density	Kg/m^3	8030
Yield Strength	MPa (N/mm^2)	537
Ultimate strength	MPa (N/mm^2)	966

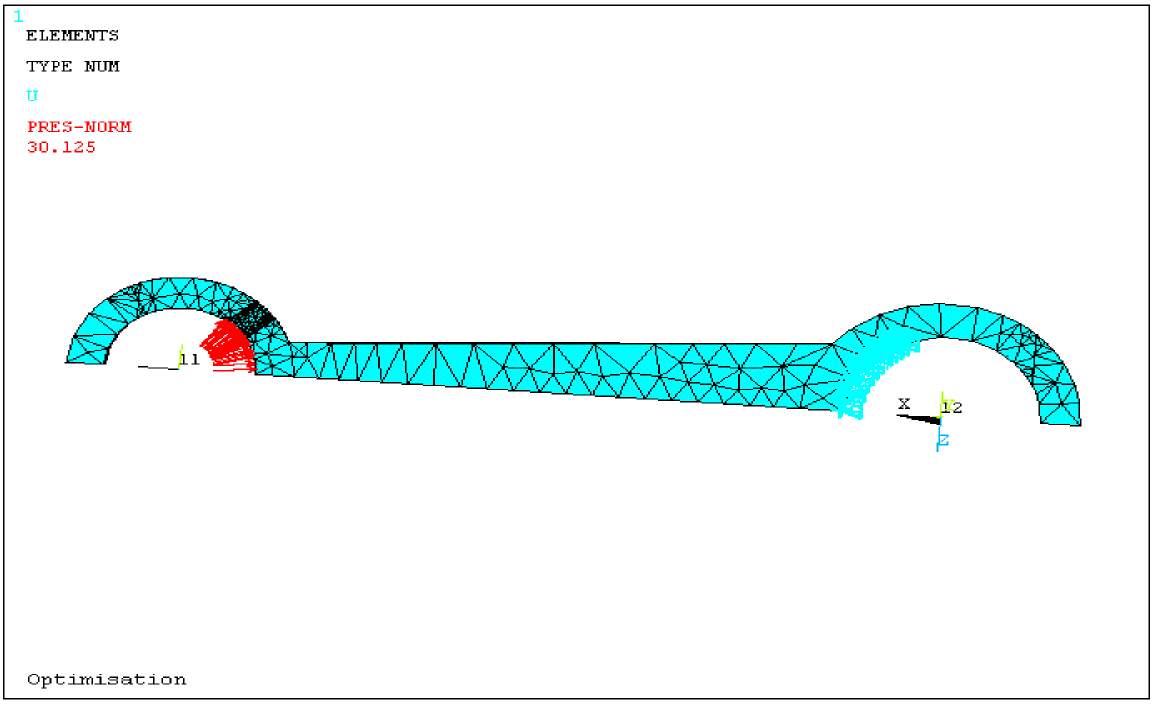

Figure5.1 : Boundary conditions for compressive loading

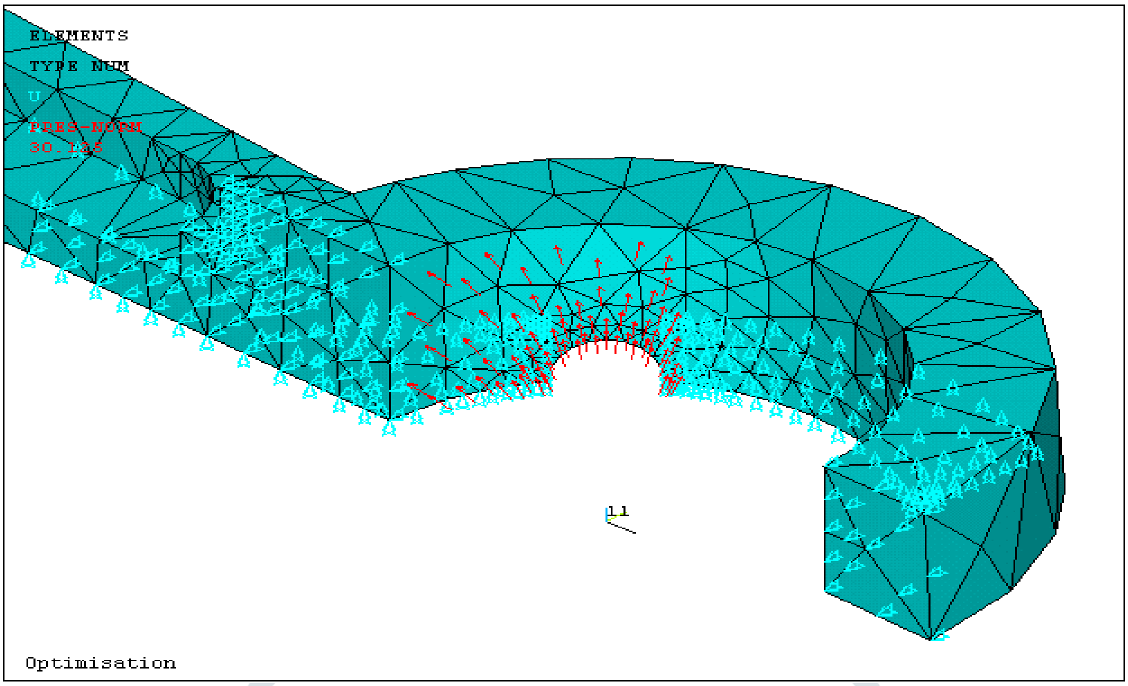

Figure 5.2: Pressure acting as a compressive load

5.2 RESULTS OF FINITE ELEMENT STRESS ANALYSIS OF CONNECTING ROD UNDER COMPRESSIVE LOAD

The structural analysis has been carried out by applying boundary conditions and constraints to get stress distribution as shown in figure. Figure shows the principle stress distribution in connecting rod under compression.

Under compressive load, the critical regions are the crank end transition and pin end transition, also the web at the crank end has a high stress region. Thus plot gives us the general idea of the stress variation along the length of connecting rod. The static loads for which these stresses are plotted is a compressive load of 951 N.

Figure shows the maximum stress produced under axial compressive load comes out to be 232.077 MPa which is at the pin end transition and crank end transition and the minimum stress is 0 MPa. Since analysis has been carried out under static compressive load therefore stresses induced have been assigned –ve sign. The minimum value (MN) with –ve sign shown in figure is the maximum stress developed in connecting rod at pin end transition and the maximum value (MX) with –ve sign is the minimum stress developed in connecting rod.

Mainly more stresses are coming in the shank part because it is the weaker part under compression. There is increase in the stresses in shank with reduction in the cross section of the shank. It goes to highest value at the joint with the end.

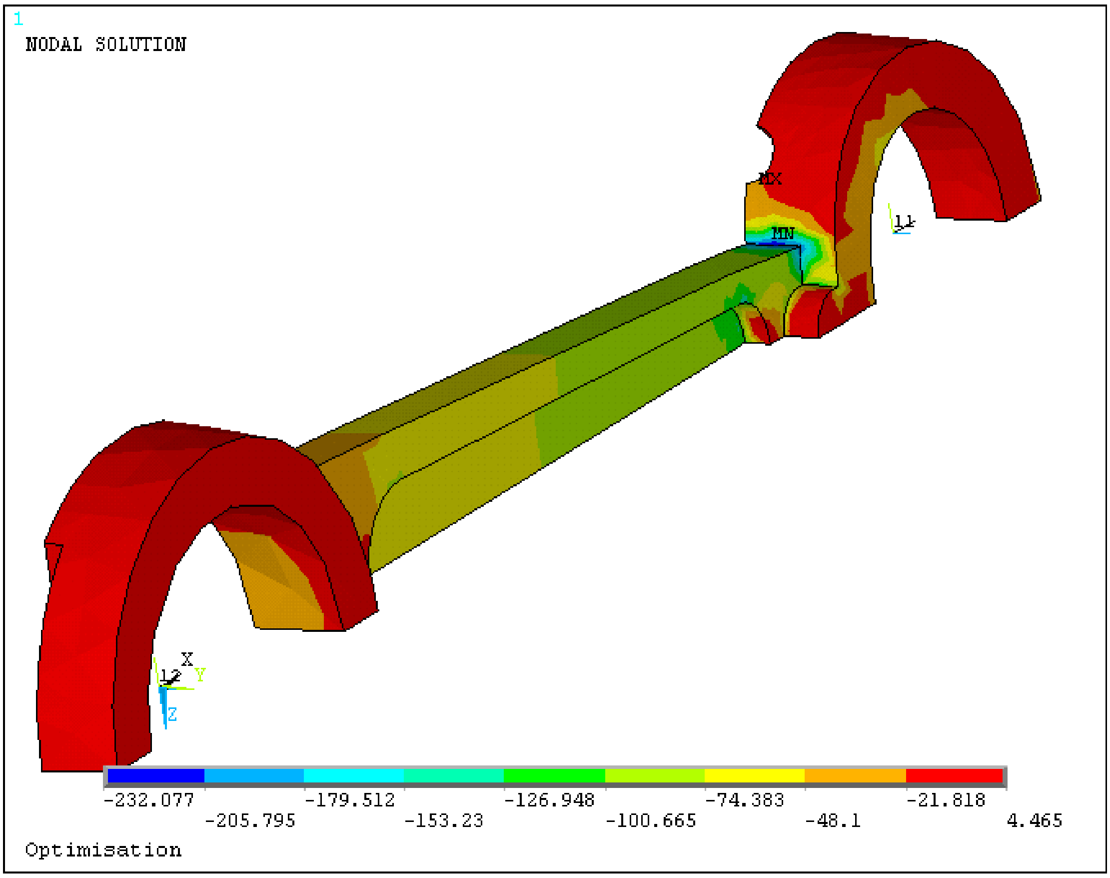

Figure 5.3: Principle stress distribution over the connecting rod under compressive loading

5.3 DEFLECTION IN CONNECTING ROD UNDER COMPRESSIVE LOAD

The previous section showed how boundary conditions and constraints are applied and also we analyzed the connecting rod with stress distribution under static compressive load .Since the connecting rod is applied under compressive load, thus, maximum deflection will occur at pin end of the connecting rod and minimum deflection will occur at crank end of the connecting rod.

Figure shows the deflection at various sections of connecting rod.

Maximum deflection produced is 0.23759mm at the pin end and minimum deflection produced is 0.00264mm.

Maximum deflection = 0.23759mm

Minimum deflection = 0.00264 mm

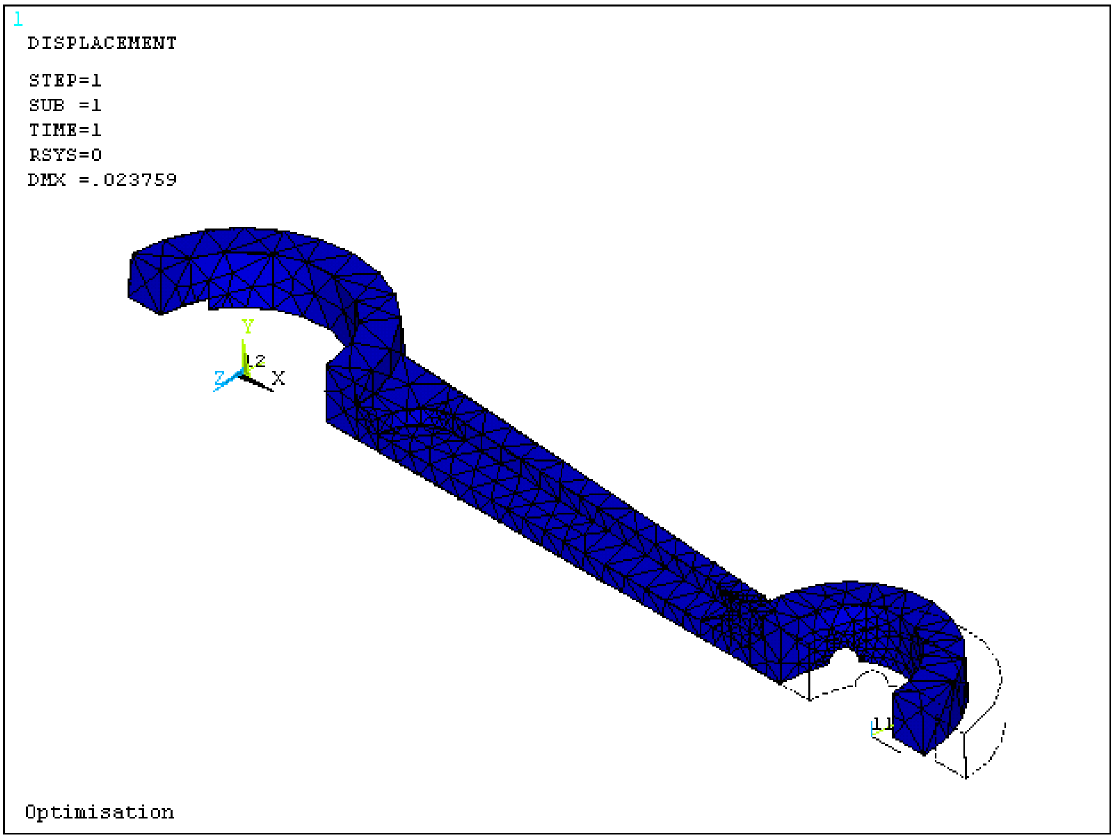

Figure 5.4 : Deflection of the connecting rod

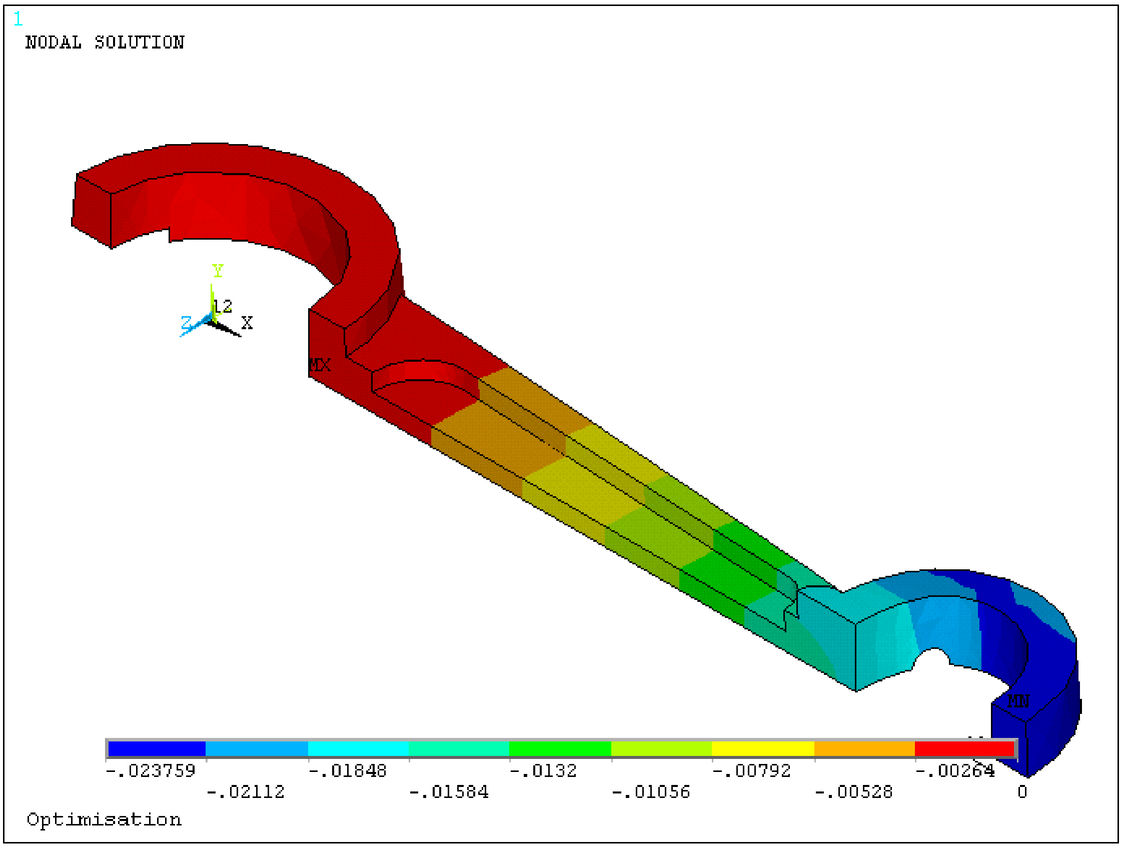

Figure 5.5: Deflection in axial direction

BUCKLING OF CONNECTING ROD

Buckling of connecting rod is like a column which experiences the axial compressive loading, so it is necessary to check it for buckling. As moment of inertia and cross sectional area of connecting rod is continuously decreasing from crank end to pin end, it is recommended to use average cross sectional area for buckling calculation.

Average cross sectional area calculated to be,

A=42.214 mm^2 for the connecting rod

The least moment of inertia of cross section is = 558.23 mm^4

So the slenderness ratio is as follows

$$k = \sqrt{\frac{I}{A}}$$

$$\therefore \sqrt{\frac{558.23}{42.214}}$$

$\therefore$ k = 3.636

$$\frac{l}{k} = \frac{65}{3.636}$$

=17.876

Therefore slenderness ratio being less than 30, it is concluded that buckling does not occur in this connecting rod and it is to be designed for compressive stress only.

6 OPTIMIZATION OF CONNECTING ROD UNDER COMPRESSIVE LOAD

Study identifies the potential for weight reduction in the existing connecting rod, as the maximum stresses in the case of compressive load are lower than the allowable stresses, and the difference is sufficiently higher. It also highlights the fact that if the component is designed on the basis of axial static load or a load range based on the load variation at the crank end, it will be over designed. In actual operation, few regions of the connecting rod are stressed to much lower stress levels than under static load corresponding to the load at crank end. The objective is to optimize the connecting rod for its weight. The weight of the new connecting rod or the “optimized connecting rod” is definitely lower than the existing connecting rod. But this may not be the minimum possible weight under the set of constraints defined.

To find the optimal weight of connecting rod within specified limits of various dimensions of connecting rod. Various dimensional parameters of connecting rod are given in table as shown with maximum and minimum limits.

The software used ANSYS has an optimization module called ‘Design Opt’. The optimization module in ANSYS has the capability to perform optimization under a set of loads. In the case of this problem of connecting rod, these loads consist of peak compressive gas loads (static) acting on the pin end as described earlier.

The optimization of quarter model connecting rod has been carried out with the given limits (maximum and minimum) of dimensional parameters of connecting rod. These values of the limits of the optimization parameters are given set according to the thumb rules of the design and the previous practical experience. These dimensional parameters with limits are shown in the table.

DIMENSIONAL PARMETERS WITH LIMITS

Sr. No.	Design variable	Variable Description	Limits	
			Minimum	Maximum
1	Shthk	Shank thickness	5.5	6.5
2	Shsthk	Slant slot thickness	1.5	2
3	Ssthk	Pin end slot thickness	2	3
4	Bsthk	Crank end slot thickness	2	3
5	Bswdth	Big slot width	4.5	5.5
6	Sswdth	Small slot width	5.75	6.75
7	Ced	Small end circular extrusion diam.	3	3.6
8	Shsbd	Shank slot big diam.	8	8.5
9	Shssd	Shank slot small dia.	5	5.5

Optimization has been performed by preparing parametric program under command line interface technique with ANSYS software.

6.1 PROCEDURE OF OPTIMIZAION

Optimization process has been performed with ANSYS software which follows the parametric program in which various instructions are given:

- Various dimensional parameters with maximum and minimum limits.
- Maximum allowable stress (i.e.300MPa) and other instructions as shown in optimization program.

Optimization Variables: There are some parameters which are to be declared as variables in the process of optimisation. Some are independent variables and some are independent. They are classified into three types as follows

1. Design Variables : Design variables are the independent quantities that are varied to achieve the optimized design. The design variables for the optimization of the connecting rod are shown in the table above. There are total nine dimensional parameters declared as design variables over here.
2. State Variables: State variables are the variables that constrain the design. They are generally stresses temperature, deflection, heat flow rate, frequency etc. Here principal stress 3 is the state variable having maximum limit of 300 MPa.
3. Objective Function: Objective function is the dependent variable to be optimized. The Objective function here is the weight of the connecting rod.

Optimisation Method: Optimization methods are traditional techniques that strive for minimization of a single function (the objective function) subject to constraints. Two methods are available: the subproblem approximation method and the first order method. In addition, you can supply an external optimization algorithm, in which case the ANSYS algorithm will be bypassed. To use one of these methods, you must have an objective function defined.

- Subproblem Approximation Method: This is an advanced zero-order method which uses approximations (curve fitting) to all dependent variables (SVs and the objective function). It is a general method that can be applied efficiently to a wide range of engineering problems.
- First Order Method: This method uses derivative information, that is, gradients of the dependent variables with respect to the design variables. It is highly accurate and works well for problems having dependent variables that vary widely over a large range of design space. However, this method can be computationally intense.
-

6.2 RESULTS OF OPTIMIZATION OF CONNECTING ROD UNDER COMPRESSIVE LOAD

As per the limits of dimensional parameters and maximum allowable stress (300 MPa) given in parametric program for optimization, ten loops are specified as the maximum limit of iteration as per instruction given in program, but we got optimized solution at sixth iteration (set) itself. As per each iteration carried out, ANSYS software gives out sets of various dimensional parameters and weight. All the sets showing dimensional parameter are within the specified limits, thus all sets are feasible. The result shows the sixth set as an optimized solution, which gives an optimum weight. The weight of the new connecting rod or the "optimized connecting rod" is definitely lower than the existing connecting rod. But this may not be the minimum possible weight under the set of constraints defined.

Parameters	Set 1 (Feasible)	Set 2 (Feasible)	Set 3 (Feasible)	Set 4 (Feasible)	Set 5 (Feasible)	*Set 6* (Feasible)
STR3	244.82	261.46	262.29	265.17	264.52	265.52
SHTHK	6.0000	5.5000	5.5000	5.5000	5.5000	5.5000
SHSTHK	1.5000	1.6279	1.7367	2.0000	2.0000	2.0000
SSTHK	2.0000	2.0584	2.1076	2.1674	3.0000	3.0000
BSTHK	2.0000	2.0735	2.1216	2.2377	2.9927	2.9927
BSWDTH	4.5000	4.5225	4.5440	4.6120	4.9762	5.2259
SSWDTH	5.7500	5.7643	5.7500	5.8042	6.0632	6.1978
CED	3.6000	3.5931	3.5904	3.5812	3.5432	3.4981
SHSBD	8.0000	8.0335	8.0474	8.0876	8.3432	8.5000
SHSSD	5.0000	5.0126	5.0248	5.0585	5.3330	5.4700
WT	0.10983E-01	0.10379E-01	0.10259E-01	0.99679E-02	0.96887E-02	0.96255E-02

Table shows the results of optimization which shows six sets. Each set shows various dimensional parameters. Values of dimensional parameters get changed with each set under the specified limits. As shown in the table SIXTH SET is the OPTIMIZED SOLUTION which gives OPTIMIZED WEIGHT.

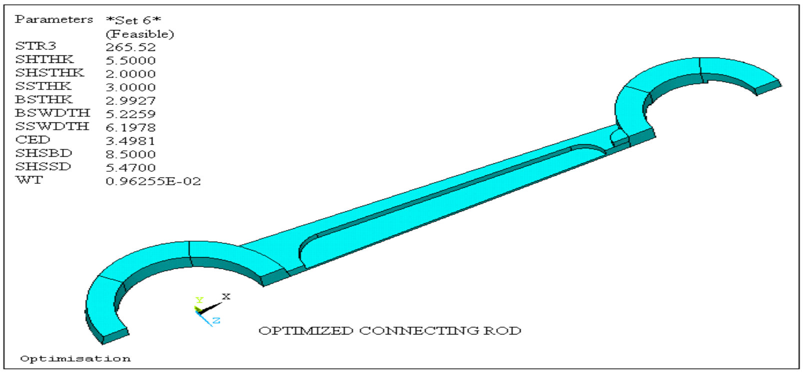

Figure 6.1 : Optimized Connecting rod

6.3 CALCULATION

Total volume of quarter model connecting rod,

$V_{tot} = 1198.69059\, m^3$

Mass density of quarter model connecting rod,

(rho) $= 8.03 \times 10^{-6}$ kg/m^3

Old weight of quarter model

$= V_{tot} \times$ rho

$= 1198.69 \times 8.03 \times 10\text{-}6$

$= 1.098 \times 10^{-2}$ kg

Total weight of full model connecting rod

$(W_{old}) = 4 \times 1.098 \times 10^{-2}$

=0.04392 kg

=43.92 gms

Optimized weight (New) of quarter model

$W_{new} = 9.625 \times 10^{-3}$ kg

Total weight of optimized (new) full model

$= 4 \times 9.625 \times 10^{-3}$

=0.0385 kg

= 38.5 gms

Therefore, Total weight reduction in existing connecting rod,

Percent reduction = $(W_{old}-W_{new}) \times 100/W_{old}$

=12.33%

Therefore total weight reduction in weight is 12.33%

6.4 GRAPH BETWEEN DESIGN VARIABLES AND NUMBER OF ITERATIONS

The graph between design variables and number of iterations shows variation of dimensional parameter with subsequent iteration until the optimized solution is reached. Here as shown in graph, various parameters get varied until the optimized solution at sixth iteration is obtained.

The values of the design variables (parameters) are changed in such a direction that the material of the connecting rod will get reduced and the weight will be minimum. The values of these parameters at each set are shown here.

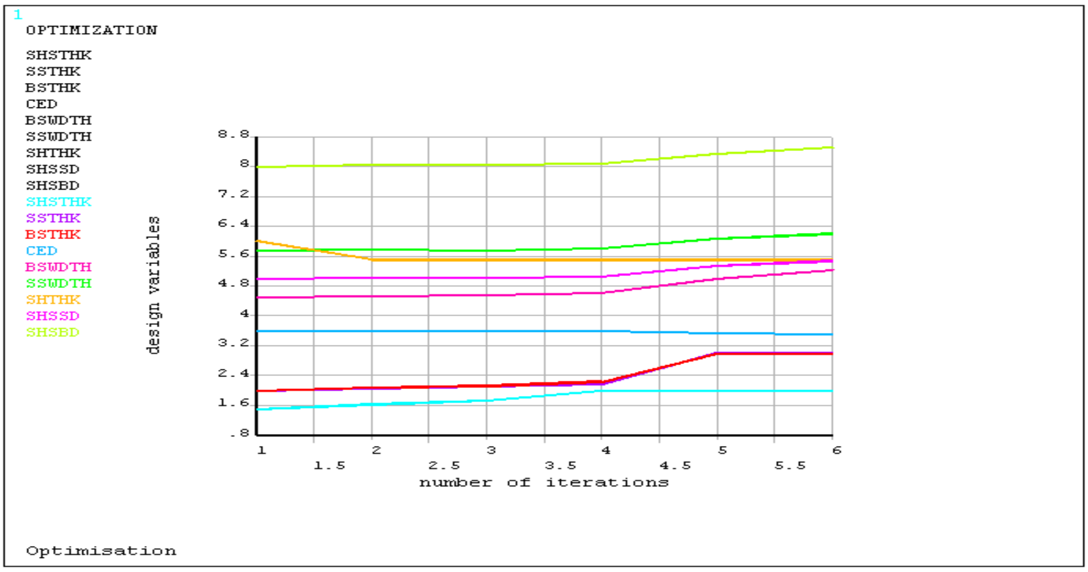

Figure 6.2: Variation of Design Variables (different parameters) with iteration

6.5 GRAPH BETWEEN MAXIMUM STRESS AND NUMBER OF ITERATIONS

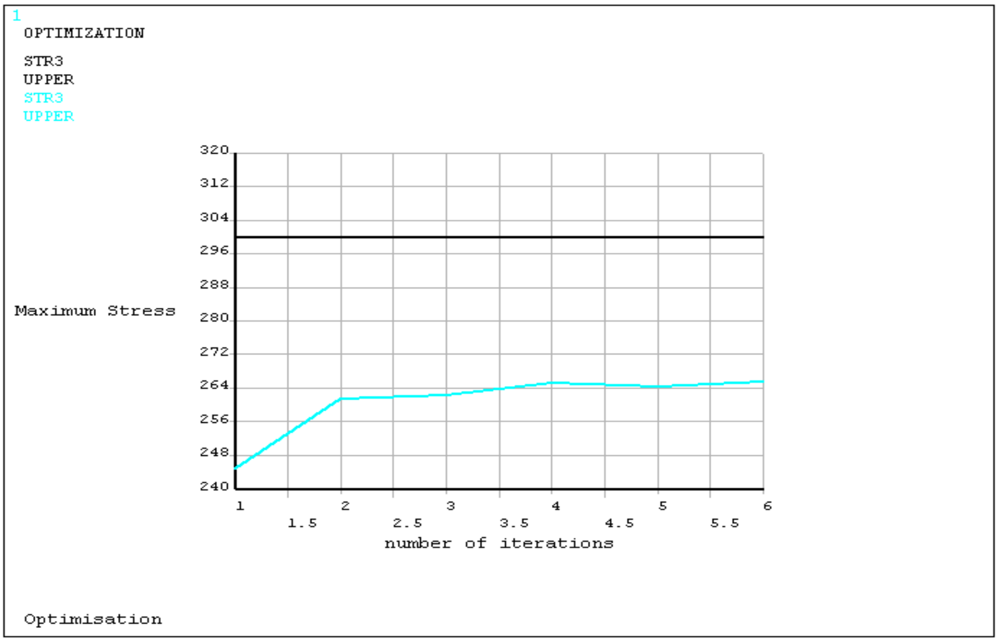

Figure 6.3: Variation of state variable (Stress) with iteration.

At end of each set or iteration there is change in the value of state variable (maximum stress), but it always less than the allowable stress and it maximum at the end of last iteration

6.6 GRAPH BETWEEN WEIGHT AND NUMBER OF ITERATION

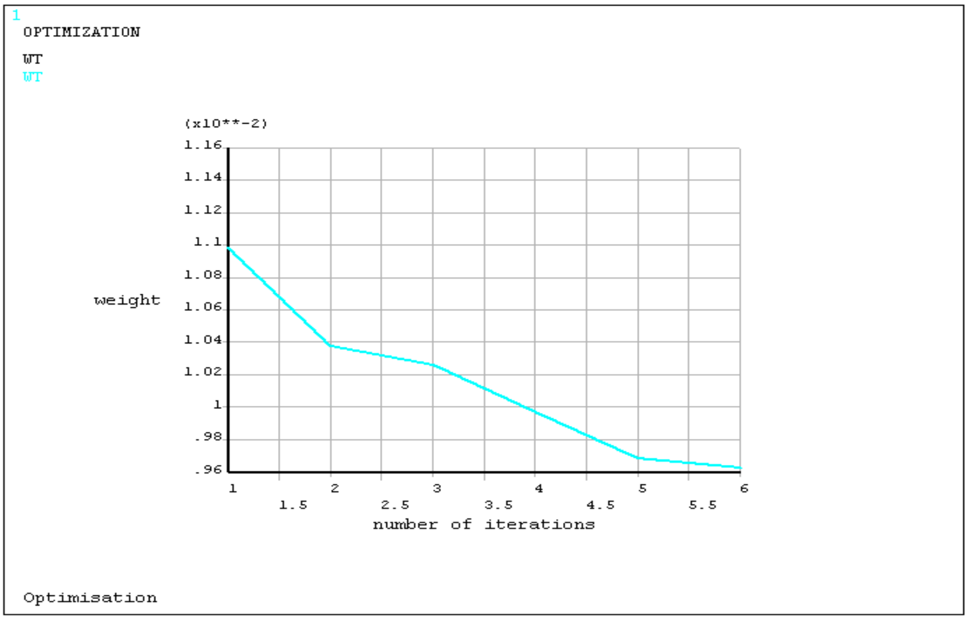

Figure 6.4: Variation of the objective function (weight) with iteration.

As shown in the graph vertical coordinate shows the weight of **quarter model** and horizontal coordinate shows the number of iterations upto optimized iteration (set 6). The weight is continuously decreasing at each iteration. It is minimum at last (sixth set).

CONCLUSION:

The given connecting rod is first modeled using ANSYS. Then the modeling is made Parametric; which is not possible in case of procedure of modeling using any CAD software and analyzing it using any CAE software. As the complete part and the boundary conditions of the part are symmetric about two perpendicular planes, quarter model of the connecting rod is prepared and the symmetry is applied. The stress analysis of the connecting is done using the prepared Finite Element Model. Two cases of tensile and compressive loadings are studied. The stresses coming under compressive loading are far more than that of under tensile loading. So the Optimization is performed under the compressive load. The connecting rod with the boundary conditions of compressive loading is optimized with the given design variables satisfying the maximum value of compressive stress. The optimized connecting rod is lighter than the original one by the amount of 12.33 %. Therefore there is weight reduction and cost reduction of the connecting rod. Also because of the reduced weight there is reduction in the inertia forces and so the operating cost of the engine will also be reduced.