Analysis of Loader Arm of Pneumatic High Speed Loader

Mr. Ayub Ashwak[1], Mr G Balaji[2] Mr. M Leela Ramesh[3,]
[1, 2, 3] Dept of Mech. Engineering
[1, 2, 3] Holy Mary Institute of Technology & Science, Hyderabad, INDIA

Abstract- *The pneumatic high – speed loader is employed to load and unload the auto sheet components in high speed metal forming press machine. Cycle time of 3 second for loading is required. Loader consists of tubular bridge frames; the tubular frames are mounted on the columns of the press machine. The loader consists of cross travel saddle, and vertical travel saddles, which are provided to have x-y axis, adjusted for the loading arm. The high – speed pneumatic arm is mounted on the vertical saddle Balancing of the loading arm is achieved by means of a rigid pantograph structure linking the counterweight to the outboard arm. The counterweight is designed to rotate, thereby enabling the counterweight to cater for two different movements. During operations, the arm will be fully balanced irrespective whether the product pipes are full or empty. The counterweight is adjustable to enable the balance of the arm to be fine tuned. Special care has been given to the rotating pivots of the pantograph bar, both in terms of design and material selection, ensuring smooth operation with a minimum of required maintenance.*

The loader arm also has up and down motion to pick and place the metal sheet in and out of the press machine; a travel time of 1500mm is completed in less than 1.2 sec. the pneumatic arm then places the metallic sheet on the press tool rapidly completing a loading cycle in complete 3 sec.

Keywords- pneumatic high – speed loader; Modeling; Finite elements; Hollow bodies,vertical travel saddles

I. INTRODUCTION

The present work involves analysis of the loader arm and optimization of the arm from stress and minimum deflection point of view. While doing the analysis it is important to have a sound knowledge of the finite element analysis (FEA).In the process of analysis, a finite element model of the loader arm is created and it is analysis using ANSYS software.

The static force loading condition is analyzed. The structure is then optimized from stress and minimum deflection point of view by

1. Changing the Limb length
2. Static analysis,
3. Transient analysis.

During optimization the structural geometry is varied and its effect on the stresses and deflection induced are seen, two different iterations are taken for changing the structural geometry and the respective deflection induced in the loader arm are studied with the help of ANSYS software, the software provides with a graphical output of the stress analysis of the structure.

Figure-1.1: Loader-arm in still position.

In this way the structural behavior of the arm is studied. Pneumatic systems employ gas that is compressed under extremely high pressure. The practical use of pneumatics comes in putting that compressed gas to use, or should i say the use of the rapid expansion of compressed gas.

1.2 Pneumatics Theory:

A fluid power system is one that transmits and control energy through the use of pressurized liquid or gas. In Pneumatics, this power is air. This of course from the atmosphere and is reduced in volume by compression, thus increasing its pressure. Compressed air mainly used to do work by acting on a piston or vane. While this energy can be used in many facets of industry, the field of industrial pneumatics is considered here. The correct use of pneumatic control requires an adequate knowledge of pneumatic components and their function to ensure their integration into an efficient working system. Although electronic control using a programmable sequencer or other logic controller is currently specified, it is still necessary to know the function of the pneumatic components in this type of system.

Pneumatic cylinders, rotary actuators and air motors provide the force and movement of most pneumatic control systems to hold, move, form and process material. To operate and control these actuators, other pneumatic components are required i.e. air service units to prepare the compressed air and valves to control the pressure, flow and direction of movement of the actuators.The drive can be bidirectional controlled and is capable of unlimited movement (provided that the "umbilical cord" is sufficiently long). It can be constructed with a very small diameter and achieve very fast step sequences. The two clamping heads are joined to the Fluidic Muscle via a flexible connection. The strategy adopted here is first the structure is analyzed using FEA and later it is modeled by the steps used in FEM using the ANSYS software, and then the results are obtained from this analysis.

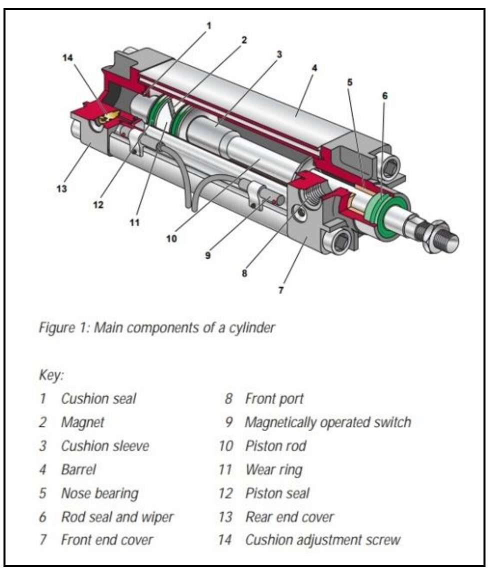

Figure-1.2: Typical Pneumatic-actuator

1.3 Description of the loader Arm

The loader arm is made up of hollow tubular pipes of Mild Steel (MS) having the internal diameter of 20mm and external diameter of 22mm. The high-speed pneumatic arm is mounted on the vertical saddle as shown in Figure-1.5 It has a rapid loading movement in and out of the press machine. The loader arm also has up and down motion to pick and place the metal sheet in and out of the press machine; a travel time of 1500mm is completed in less than 1.2 seconds hence the name of high speed loader is given to it. The pneumatic arm then places the metallic sheet on the press tool rapidly. Completing the cycle of loading in 3 seconds.

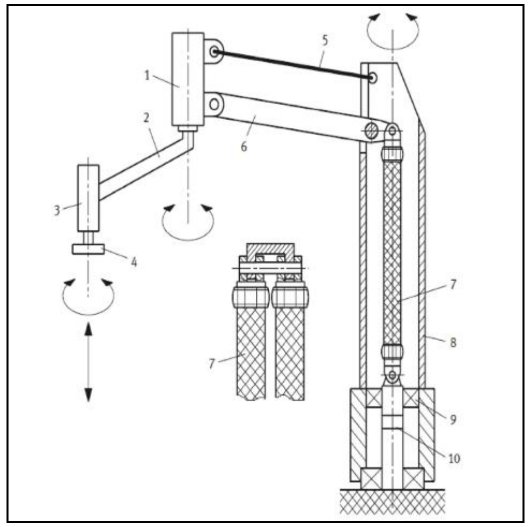

Figure-1.3: schematic diagram of loader arm

The material properties of the loader arm are as follows. Material Properties: Young´s Modulus = 200000 N/mm^2. Poison´s Ratio = 0.3. Two suction pads are mounted on each of the two limbs of the arm at a distance of 975.7mm and 1341.5 mm each for the extreme right end of the loader arm that is considered as the reference point for the measurement. So in all there are four suction pads that are mounted on the limbs of the arm.

The function of the suction pads is to get in contact with the work piece, due to the suction created in the pads the work piece get stuck to the pads and when the suction is released the work piece is let free.

1.4 Speed control

For many applications, cylinders can be allowed to run at their own maximum natural speed. This results in rapid mechanism movement and quick overall cycle times of machine. However, there will be applications where uncontrolled cylinder speed can give rise to shock fatigue, noise and extra wear and tear to the machine components.
The factors governing natural piston speed and the techniques for controlling it are covered in this section.

The maximum natural speed of a cylinder is determined by:

- Cylinder size
- Port size
- Inlet and exhaust valve flow

• Air pressure
• Bore and length of the hoses
• Load against which the cylinder is working.

From this natural speed it is possible to either increase speed or as is more often the requirement, reduce it. First we will look at how the natural speed for any given load can be changed by valve selection. Generally, the smaller the selected valve the slower the cylinder movement. When selecting for a higher speed however, the limiting factor will be the aperture in the cylinder ports. Valves with flow in excess of this limitation will give little or no improvement in cylinder speed. The aperture in the cylinder ports is determined by the design. Robustly constructed cylinders will often be designed with full bore ports. This means that the most restrictive part of the flow path will be the pipe fitting. These cylinders are the type to specify for fast speed applications and would be used with a valve having at least the same size ports as the cylinder. Lighter duty designs, particularly small bore sizes, will have the port aperture much smaller than the port's nominal thread size. This has the desired effect of limiting the speed of the cylinder to prevent it from self destructing through repeated high velocity stroking. The maximum natural speed of these cylinders can often be achieved with a valve that is one or two sizes down from the cylinder port size.

1.5 Description of the work piece

The work piece weighs 10 kg in weight and the work piece is loaded and unloaded by the arm with the help of the suction pads that are mounted on the limbs of the arm. So in all a load of 10 kg acts combing on all the four suction pads in the downward direction. As seen in Figure 6.3 in chapter 6 of this report the four arrows pointing towards the downward direction shows the load acting on the limbs of the arm the Figure 1.7 is generated in ANSYS highlighting the static loading condition of the arm. That means that a load of 2.5 kg each is acting on each of the suction pads which are at a distance of 975.7mm and 1341.5 mm from the reference measurement.

	STEEL AISI 1022	ALUMINUM 7075
DENSITY (Kg/m^3)	7858	2810
YOUNG'S MODULUS (MPa)	20500	71700
POISSON'S RATIO	0.21	0.33
STRENGTH (MPa)	380	572

Table-1.1: Properties of work piece material

The strategy adopted here is first the structure is analyzed using FEA and later it is modeled by the steps used in FEM using the ANSYS software, and then the results are obtained from this analysis. The various steps followed during the ANSYS analysis are studied. Depending on the results obtained the deflection is studied in the structure and then the structure is optimized to reduce the deflection. During the optimization procedure it is of importance to maintain the stresses induced

In the structure with in the safe design limits the two simulations are obtained called as first stage of optimization and second stage of optimization by making the necessary changes made to the geometry of the loader arm which is studied in chapters called as first stage of optimization and second stage of optimization. This is then preceded by the results and conclusion chapters.

1.6 Finite Element Analysis

The method of analysis adopted in this current work is the finite element method of analysis. In the FEA solutions are based on the displacement method of analysis which works on the principle of virtual work. Depending on the theories by C.S.Krishnamorthy, Klaus Jorgen Batheand Tirupathi R. Chandrpatla

Following steps of the FEA where adopted and incorporated to achieve the design of the loader arm. The loader arm is a framed MS structure which behaves as a beam when statically loaded and the conventional way of design for a three dimensional beam is a tedious work which can be effectively overcome by FEA. And the three dimensional beam analyses by FEA gives the exact amount of displacement induced in each of the beam element as compared with the conventional way of design.

II. DESIGN ANALYSIS

This method is well known method of analysis in structural design and is a universally accepted method. The method constructs a discrete system of matrix equations describing the mass and stiffness effect of a continuous structure. The geometric complexity of the structure puts no restrictions when the mass and stiffness matrices are assembled from each simple element with simple shape and sizes, each element is formulated mathematically in association with a simple geometric description with no respect to the overall geometry of the structure. In this work we make use of the beam element for modeling the structure as the loader arm behaves as a beam when it is subjected to static loading conditions as stated by Klaus Jorgen Bathe since the structure that is the loader arm is a framed structure of hollow MS. pipes of outer diameter 22mm and inner diameter 20mm, it behaves as a beam when it is statically loaded and the beam element selected for this framed structure is a three dimensional beam element. so the following formulations for a three dimensional beam elements are considered in this thesis work. The steps of the modeling in the finite element analysis are discussed below.

The element stiffness matrix for the three dimensional beam element which has been described by Klaus Jurgan Bathe m*k the Computation of element nodal load vector is same due to The nodal loads due to loads acting on the element can be transformed to the global system Computation for Final Stress Resultants In finite element analysis the stress is computed as

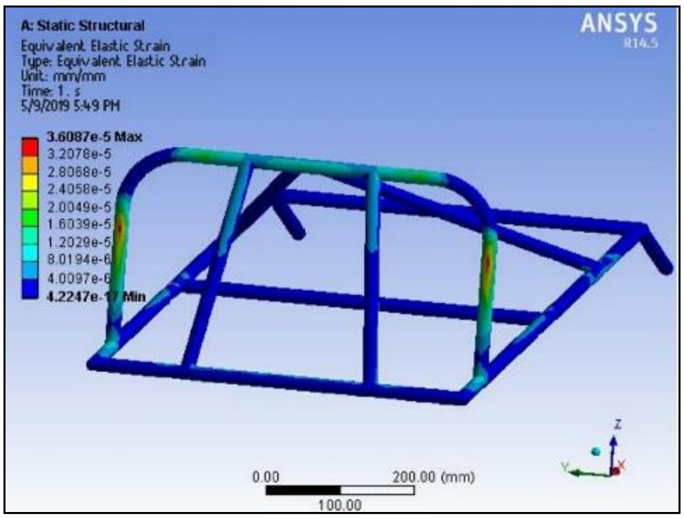

Figure-2.1: Bending stresses in the arm

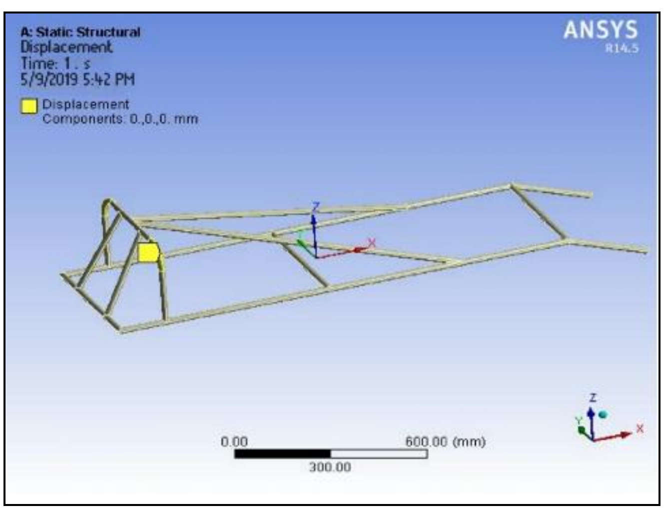

Figure-2.2: Meshed model of the structure using PIPE16 Element in ANSYS

These stress resultants correspond to the degrees of freedom as axial and shear forces and bending moments. Hence the stiffness coefficients give the value of these actions due to unit displacements. Then after assembling the stiffness matrices and solving them and retrieving the displacements at the ends of the members by the finite element analysis. The end actions due to end displacements for member in the local axes system To the above stress resultants we should add the stress resultants due to loads on the member under fully restrained condition.

$$\{\sigma\} = [\,Mk\,\{dm\}+\{S0\}\,]$$

Where the stress resultants corresponding to the nodal degrees of freedom due to loads on the member under fully restrained end conditions. It may be noted that the above explanation holds good for a three-dimensional beam element and gives the final stress resultants at the ends of the member. Modeling in ANSYS The software used for the Finite element analysis calculations is ANSYS. Which is widely used commercial simulation software. The three basic features of this software are pre-processing, solution and post-processing stages. ANSYS, Analysis Guide For the analysis of the loader arm 3D-beam element is used. The loader arm is a made up of hollow MS pipes of outer diameter 22mm and inner diameter 20mMSo PIPE 16 is used to model the loader arm The properties of this element are mentioned as follows: PIPE16 is a un-axial element with tension-compression, torsion, and bending capabilities.

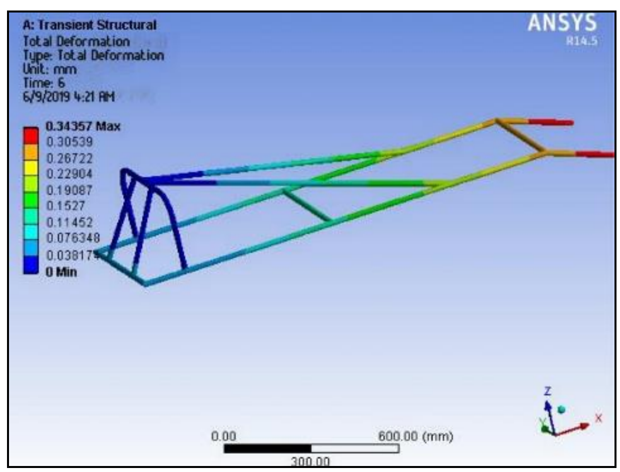

Figure-3.3: Meshed model of the structure using PIPE16

The element has six degrees of freedom at two nodes: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. This element is based on the 3-D beam element, and includes simplifications due to its symmetry and standard pipe geometry. Now in ANSYS the structure is modeled and its meshed diagram is generated then the loading conditions are applied. After application of the loading conditions the meshed model of the structure is applied with the degrees of freedom and the various diagrams are generated. During the ANSYS analysis the meshed figures of the loader arm are obtained as shown in Figure3.3

The Figure 3.2 shows the yellow portion on the meshed structure that is the application of the degrees of freedom on the meshed structure. the application of the static loading on the meshed structure, in the red arrows pointing towards the downward direction are indicating the load acting on the red arrows present on each limb of the loader arm. Apart from this the analysis gives graphical out put of the stresses present in the structure, stresses like bending ,axial and Von Misers stresses which are nothing but the generalized stress distribution on the surface of the structure.

The graphical out put of the deflection in the structure is also seen and it can be simulated to see the motion of the deflection in the structure. Due to the specificity of operation objects, the end-effector for robot must have adequate active flexibility, adaptability and security. Based on Flexible Pneumatic Actuator (FPA), a flexible pneumatic bending joint was designed which had both contact rigidity and grasping flexibility. The mathematic model of bending joint was built based on force/torque balance principle.

The bending angle and output force relationships with the pressure of cavity were analyzed. Based on flexible pneumatic bending joint, the agricultural fruit picking end-effector was proposed, and its grasping mathematic model and control system were built. Experimental setup was designed to carry out the testing and comparing investigation with simulation. The results indicate that the theory models match the experiment result. This end-effector can produce enough force for grasping fruit like apple, and has enough active compliance and adaption.

Totally different from conventional rigid robot, bioinspired soft robot has characteristics of deformable body, flexibility, continuous changeability and high active adaptability. It has become a new evolution in robotics. In this context, a flexible pneumatic robotic actuator FPA was proposed to act as both driver and executor for soft robot configuration

The structure, principle and mathematic model of FPA are described. Based on FPA, a series of soft joints, such as soft bending joint, link-embedded soft bending joint, hybrid bending joint, torsion joint, spherical joint and side-sway joint, are developed. Then soft robots based FPA are reported, including climbing robot, cucumber gripper, end-effector for cone, multi-fingered dexterous hand and hand rehabilitator. All the structures, working principles and experiments of the soft joints and robots are elaborated.

III. FINITE ELEMENT MODELING

Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variation calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a paper published in 1956 by M. J. Turner, R. W. Clough, H. C. Martin, and L. J. Top established a broader definition of numerical analysis. The paper centred on the "stiffness and deflection of complex structures".

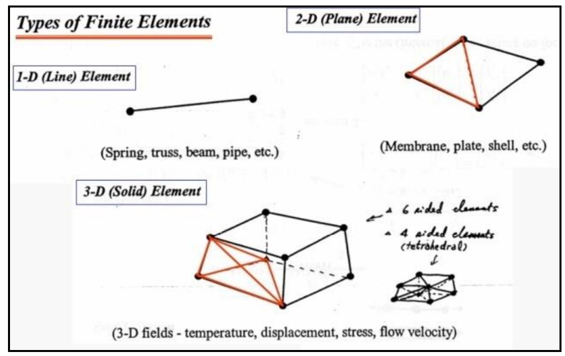

Figure-4.1: Elements and nodes (Nodal quantity)

The finite element method (FEM), or finite element analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Boundary value problems are also called field problems. The field is the domain of interest and most often represents a physical structure. The field variables are the dependent variables of interest governed by the differential equation. The boundary conditions are the specified values of the field variables or related variables such as derivatives on the boundaries of the field.

IV. MODELING IN ANSYS

4.1 Stages of modeling in ANSYS

The following are the three main processes involved during the modeling of ANSYS. ANSYS, Analysis Guide The element used to model the arm in ANSYS is PIPE16. PIPE16 is well suited for the meshing of the structure that´s way it is being used, It has both bending and membrane capabilities. Both in-plane and normal loads are permitted. ANSYS, Analysis Guide The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. Stress stiffening and large deflection capabilities are included. The loader arm is made up of MS. The material of the structure is having

Young´s modulus of 200000 N/mm^2, having Poisson ratio of 0.3. The meshed model of the loader arm can be seen in the Figures, where the entire arm is show in the meshed view by making use of PIPE 16 element in ANSYS. Meshed model of the structure using PIPE16 element in ANSYS, the application of various degrees of freedom on the structure of the loader arm when the analysis of the arm is performed by using ANSYS. We can see the red arrows pointed towards the downward position that highlights the load acting on the arm during the static loading condition. The red arrows also show the position of the suction pads the arrows are located at the location of the suction pads where the load acts. View showing the application of degrees of freedom on the meshed model The highlights the application of the force load on the limbs of the loader arm. The static load acts at a distance of 975.7 mm and 1341.5 mm from the reference point. The red arrows in the Figures indicate the load acting in the downward direction, along with the complete meshed model of the loader arm. The Views Showing the Application of Load on the Meshed Model. The complete overview of the meshed model in ANSYS of the loader arm along with the application of degrees of freedom and the static load of 10 kg acting is shown in the Figure 3.2 where the yellow portion shows the degrees of freedom and the red arrows shows the application of the static load in the downward direction. View showing the Application of Load and Degrees of Freedom on the Meshed Model. This is the next step of analysis in ANSYS. Once the pre-processing stage is finished the results are obtained in this. In the solution phase of the analysis, the computer takes over and solves the simultaneous set of equations that the finite element method generates. The element solution is usually calculated at the elements' integration points.

4.2 Post-processing

This is the stage the results of the performed analysis are obtained as both numerical and graphical output. Once the results are obtained the software ANSYS generates graphical outputs of the analyzed structure which are extremely useful to have an idea about where the stresses and deflections are occurring and in which portion of the structure needs to be stronger to withstand the various stresses induced in them. Results for Original Design

The following graphical output obtained when the ANSYS program was run. This shows the various stresses acting on the loader arm. Figure 6.5 Axial stresses in the loader arm. An axial stress of 96.721 N/mm^2 is seen in the loader arm when the ANSYS analysis is performed on the loader arm as seen in Bending stresses in the arm. A bending stress of 98.016 N/mm^2 is seen in the Loader arm during the ANSYS analysis 2

4.3 Deflection observed in the loader arm.

A deflection of 13.276mm is seen in the loader arm in the above Figure 6.8 Deflection as seen in the ANSYS model of the arm. The Figure 6.1 shows the position of the arm when the deflection is seen in it while performing the ANSYS analysis. The maximum von Mises stresses are shown in Figures Hence the results obtained by the above analysis are as follows:

- Max. Deflection = 13.276 mm.
- Max. Bending stress = 98.016 N/mm^2.
- Max. Axial stress = 96.721 N/mm^2.
- Max. Von Mises stresses = 96.967 N/mm^2.

The above obtained values are used as the reference values in an optimization of the structure. Criteria for Accuracy

Since the finite element method is an approximation therefore it is important to consider the accuracy of the calculations therefore during the process of analysis the accuracy was considered by the following parameters described below: Now for testing the accuracy of the obtained results two criteria´s where considered they include, comparing of the results with the results obtained by the conventional and analytical way of calculations that where performed at the design department at Cybernetic Technologies Pvt Ltd. And secondly the calculated stresses where seen that they are within the safe design limit or not. Calculations were seen to be in the safe design limit. But during the current task the accuracy of the calculations where consulted with the design engineers at the design department at Cybernetic Technologies Pvt. Ltd.

Thus the results obtained where found to be in the safe design limit by the respective design authorities, and where considered matching closely with in the safe design limits.

V. OPTIMIZATION

5.1 The Optimization

Optimization plays crucial role in enhancing the performance of any product and it is also important from economic point of view. Optimization has a major role to play in the present work. It helps to get best possible design from the original design. Optimization helps to come up with a new design, which is complete and desirable. In the case of the loader arm the initial calculation results show a level of deflection considered as not safe for the loader arm. Therefore it was decided to reduce the deflection to the least possible, while keeping all the stresses within the safe elastic limit. Steps Incorporated before Optimization

Optimization was the most important aspect of this work that is being carried out in this thesis. The initial results obtained after performing the Finite Element Analysis on the loader arm using ANSYS software, were used as a basis for an optimization of the studied structure. The deflection was found to be the parameter that was to be optimized so the process of optimization was performed for minimum deflection and stresses. The optimization criterion was to have a least possible deflection by keeping the stresses within the safe design limit. It was decided to reduce the deflection induced in the arm by making changes in the geometry of the loader arm. Now the most crucial task was to decide which link in the loader arm to alter so as to reduce the deflection induced in the loader arm. For deciding this lots of trial and errors were taken on the entire structure of the loader arm. Then finally out of trial and error it was decided that the inclined links supporting the long horizontal limbs should be changed. The altered geometry of the structure can be solved by von mises simulations were performed on altered structures and the results were obtained.

5.2 First Stage of Optimization

Now changing the position of the link in the arm from its original position that is when the position of the link is at 465.5 mm from the reference point of measurement, to the position of 577mm from the reference end of measurement. The Figure 6.1 shows the solid model of the loader arm when its geometry is altered. View showing the change in geometry for the loader arm for the first stage of optimization. After changing the geometry of the structure the simulations where performed on the modified structure of the arm and the following results where obtained graphically. Axial stresses for first Stage of optimization. shows the axial stresses induced in the loader arm when the structural geometry of the arm is changed. And it was found to be 71.22 N/mm^2. Bending stresses where observed when the geometry of the structure was changed as seen in Figure 6.1. Bending stress of 71.237 N/mm^2 was seen. Deflection for first Stage of optimization. A deflection of 7.698 mm was noticed when the geometry of the structure was altered for the process of optimization. Von Mises Stresses for first Stage of optimization. The following where the results that were obtained when the first stage of optimization is performed, as when the position of the link is changed from 465.5mm from the reference point of measurement, to the position of 577mm from the reference end of measurement.

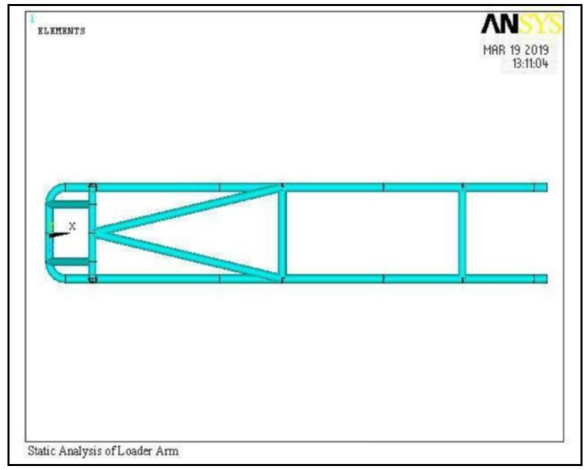

Figure-5.1: View showing the change in geometry for the loader arm for the first stage of optimization

- Max. Deflection =7.698 mm.
- Max. Bending stress = 71.237 N/mm^2.
- Max. Axial stress = 71.22 N/mm^2.
- Max. Von Mises Stresses =71.223 N/mm^2.

Now it is seen that the deflection has been drastically reduced along with the stresses which are within the safe design limit.

5.3 Second Stage of Optimization

A second stage of optimization was performed to reduce the deflection even more than in the first optimization step. Now for the second stage of optimization we will change the position of the link from 577mm, which was taken for performing the first stage of optimization, to a position 711.65mm from the reference point, shown in Figure This position of the link is just above the position where the suction pads are mounted on the loader arm. Positions further away from the reference point are restricted. Simulations where performed on the altered structure to obtain the various stresses along with the deflection. Figure 6.2 View showing the changes performed on the geometry of the arm for the second stage of optimization. Figure 6.2 Axial stresses for second stage of optimization.

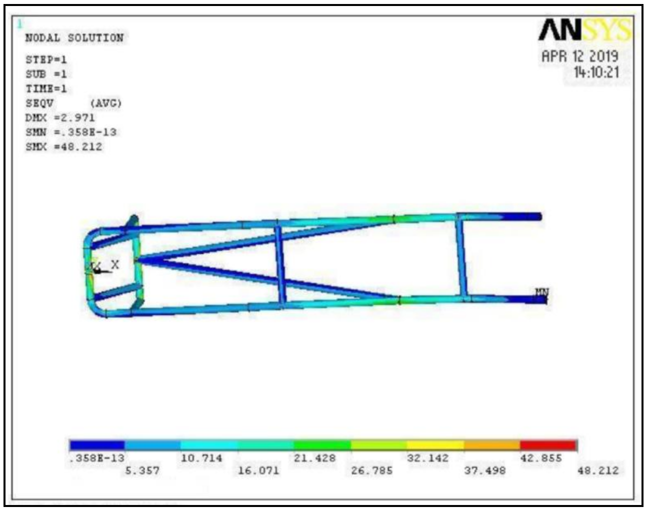

Figure-5.2: Von Mises Stresses for second stage of optimization

- Max. Deflection = 2.971 mm.
- Max. Bending stress = 48.399 N/mm^2.
- Max. Axial stress = 30.329 N/mm^2.
- Max. Von Mises Stresses = 48.212 N/mm^2.

As inferred from the obtained results in the second stage of optimization it is seen that the deflection is reduced to the least possible by keeping the stresses with in the safe limit.

The Figure 6.3 shows the various axial stresses found in the loader arm during the simulation. An axial stress of 30.329 N/mm^2 is seen in the above figure. Bending stresses for second stage of optimization. A Bending Stress of 48.399 N/mm^2 is noticed from the Figure 6.1 when the second stage of optimization is performed.

A deflection of 2.971mm is seen in the loader arm when the simulation if performed for the second stage of optimization. Figure 6.2 Von Mises Stresses for second stage of optimization. The Maximum Von Mises Stresses are seen when the second stage of optimization is performed on the structure of the loader arm. The following are the results that where obtained during the second stage of optimization:

5.4 Comparison of Results

The table 7.1 shows the comparison of the results obtained during the analysis of the loader arm using the ANSYS software. As inferred from the table above it can be seen that the deflection is being reduced as compared to the original design in the second stage of optimization by keeping the stresses within the safe design limit.

	Original Design	First Optimization	Second Optimization
Max. Deflection	13.276 mm	7.698 mm	2.971 mm
Max. Bending stress	98.016 N/mm^2	71.237 N/mm^2	48.399 N/mm^2
Max. Axial Stress	96.721 N/mm^2	71.220 N/mm^2	30.329 N/mm^2
Max. Von Mises Stress	96.967 N/mm^2	71.223 N/mm^2	48.212 N/mm^2

Figure-5.3: Optimization results comparison

5.5 Results after Optimization.

Therefore the second stage of optimization is considered the most appropriate one because we get the least of all the possible deflections as seen in the above results. The deflection is almost reduced to 2.971 mm. where as it was seen as 7.698 mm in the first stage of optimization and therefore this stage of optimization is considered the most suitable and the best possible which is obtained by making the necessary changes in the geometry of the loader arm. The results obtained were incorporated into the production and manufacturing of the loader arm at Cybernetic Technologies Pvt. Ltd, Pune, India.

5.6 Discussion and Future Work

From the results obtained in the second stage of optimization it can be seen that the deflection is reduced to 2.971mm. This is a considerable reduction in deflection as compared to the original design as well as the first stage of optimization. Then a question may arise that if the position of the link can be altered further.

The position of the link as seen and cannot be altered further; it needs to be restricted to the position as seen. The reason for this is that, if the length of the link is altered further till the tip of the loader arm, then it may restrict the entry of the arm inside the press machine. As the link may strike to the edge of the press machine and since this is a high speed loader then the altered changes may cause damage to the loader arm. The process of optimization is restricted till the second stage of optimization only and it cannot be extended further. Therefore the second stage of optimization is considered the most viable and convenient way out, and the deflection is also reduced to the least of the values as seen in the original design and the first stage of optimization.

5.7 Future Work

The current work only deals with the static analysis of the loader arm of the pneumatic high speed loader. The pneumatic high-speed loader is employed to load and unload the auto sheet components in high-speed metal forming press machine. Cyclic time of 3 second for loading is required. The loader arm also has up and down motion to pick and place the metal sheet in and out of the press machine; a travel time of 1500mm is completed in less than 1.2 seconds hence the name of high speed loader is given to it. The further scope of the work can include the dynamic analysis of the loader arm.

The use of aluminium alloys for loader arm is to be investigated practically since the working conditions will be different from that of conditions taken for analysis. The loads considered in the present work are only loads to be picked and placed, but the use of aluminium alloys for other applications like to loading the work piece from the conveyer belt into the die machine and vice-versa has to be analyzed. Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

VI. CONCLUSIONS

The main intention of the work was to carry out the optimization of the loader arm. For this purpose finite element analysis was used. The FEA was solved by using ANSYS software. ANSYS was the most suitable software for this purpose as it gives the stresses induced in the parts that are of interest to be modified along with a nice graphical output highlighting the various stresses induced in the structure that is being studied. The intention was to get the minimum possible deflection of the loader arm by keeping the stresses within the safe design limit.

The overall model of the structure was created and the required deflection and stress analysis was performed, it was found that the least deflection was obtained when the geometry of the structure was altered from the existing geometry. As shown in the figures in optimization section of this report. It can be inferred that by changing the geometry of the structure a considerable change can be obtained which can enhance the product function ability, as this was seen during the changes that were made to the structure during the process of analysis and optimization. As the changes made in the geometry of the structure were carried out, it was observed that the deflection was reduced considerably and was within the safe design limit.

The simulations carried out were of great importance as it eliminated the process of manufacturing the physical prototype of the loader arm and then checking its deflection, with the help of this work a lot of time involved in physical manufacturing and testing is saved and the alteration in the geometry of the loader arm can be performed with great ease. Therefore time consumption in the conventional way of design and optimization is easily overcome by this work.

Finally the changes made during the optimization were found to increase the performance of the loader arm, since the deflection induced in the loader arm was considerably reduced by making the necessary changes in its geometry. A further scope of the work can include the dynamic analysis of the loader arm.

In the present thesis, analysis is performed on the loader arm to optimize the arm from minimum stress and deflection. The model of the loader arm is changed by changing the limb lengths. The limb lengths are decreased from 900mm to 700mm and 500mm. The present used material is steel; it is replaced with Aluminium alloy 7075.The 3D models of the loader arm are done in Pro/Engineer. Analysis is done in ANSYS. By observing the static analysis results.

Stresses and deformations are increasing by decreasing the limb lengths. But the stress values are slightly deferring which is safe. By comparing the results between materials, the deformations are more for Aluminium alloys than steel but stresses are less for aluminium alloys than steel.

Transient analysis is performed by increasing the pressures by varying time, at 6secs and 12secs. By observing the analysis results, the stresses are within the range for all materials even at increased pressures. By comparing the results between materials, the deformations are more for Aluminium alloy than steel but stresses are less for aluminium alloys than steel.

It can be concluded that by decreasing the limb lengths increases the stresses on the arm but the variations in stresses are minimal which is safe under working conditions. And also by decreasing the limb lengths, the weight of the arm decreases. By changing the material from steel to aluminium alloys, the weight decreases and also the stresses are within range. So using aluminium alloys is better.

REFERENCES

[1] 1.C.S.Krishnamorthy, (2005), “Finite Element Analysis”. Tata McGraw Hill Publications. India.

[2] 2.Analysis of Loader Arm of Pneumatic High speed Loader by Mohammed Nasser Farooqui, Suresh

[3] 3.Klaus Jurgan Bathe, (2001), “Finite Element Procedure”. Prentice Hall. India.

[4] 4.Rocker arm: A Review by Syed Mujahid Husain, Prof. Siraj Sheikh, G.H.Raisoni

[5] 5.Tirupathi R. Chandrpatla and Ashok D. Belegundu, (2001), Introduction of Finite Elements in Engineering, Prentice Hall, India.

[6] 6.ANSYS, Analysis Guide, ANSYS release 6.5, SAS IP Inc, Houston.

[7] 7.Design and Analysis of Axle Arm for ARJUN MBT Track Tensioned by G.Maheedhara Reddy N. Amanda Kumar E.Balaji

[8] 8.Broman.G, (2004), Computational Engineering, (2004), Department of Mechanical Engineering, University of Karlskrona/Ronneby, Sweden..

[9] 9.Embedded optimum controller for process loop using arm controller by Sonal J Rane

[10] 10.Ottosen N.S. and Petersson H, (1992), Introduction to the Finite Element Method, Prentice Hall, Sweden

[11] M. Kleiner, A. Gartzke, R. Kolleck, J. Rauer, T. Weidner, Toolconcept for the high-pressure sheet metal forming (HBU Process),in: H.J.J. Kals, et al., Sheet Metal 1996: Proceedings of the FourthConference, vol. 2, University of Twente, NL, 1996, pp. 123–130.

[12] H.M. Shang, S. Qin, C.J. Tay, Hydroforming sheet metal withintermittent changes in the draw-in condition of the flange, J.Mater. Process. Technol. 63 (1997) 72–76.

[13] M. Ahmed, M.S.J. Hashmi, Finite element analysis of bulgeforming applying pressure and in plane compressive load, Proceedingsof Advances in Materials and Processing Technologies’95, Dublin City University, Dublin, 1995, pp. 932–942.

[14] D.M. Woo, Analysis of the cup-drawing process, J. Mech. Eng.Sci. 6 (2) (1964) 116–131.

[15] Z. Marciniak, J Duncan, Mechanics of Sheet Metal Forming,Edward Arnold, London, 1992.