Analysis of Loader Arm of Pneumatic High speed Loader

Mohammed Naser Farooqui
Suresh Arunachalam

Department of Mechanical Engineering
Blekinge Institute of Technology
Karlskrona, Sweden
2006

Analysis of Loader Arm of Pneumatic High speed Loader

Mohammed Naser Farooqui
Suresh Arunachalam

Department of Mechanical Engineering
Blekinge Institute of Technology
Karlskrona, Sweden
2006

Thesis submitted for completion of Master of Science in Mechanical Engineering with emphasis on Structural Mechanics at the Department of Mechanical Engineering, Blekinge Institute of Technology, Karlskrona, Sweden.

Abstract:
The function of the loader arm of the high-speed pneumatic loader is to load the work piece from the conveyer belt into the die machine and vice-versa. The purpose of the work was to develop a Finite Element Model of the loader arm by using ANSYS Software. And with the help of the FEA model, the loader arm is optimized from stress and minimum deflection point of view. The knowledge of FEA and optimization is required to achieve the desired optimization in the existing structure.

Keywords:
Finite Element Modeling, Finite Element Analysis (FEA), ANSYS.
Acknowledgements

The thesis work was carried out in Mechanical Design Department at Cybernetik Technologies Pvt. Ltd, Pune, India. Under the supervision of Dr. Ansel J Berghuvud from the university and Mr. Mahesh Wagle from the company.

We wish to express our sincere appreciation to Mr. Mahesh Wagle, Technical Director, Cybernetik Technologies Pvt. Ltd for his guidance, support and professional engagement in the thesis work in spite of his busy schedule. At the Department of Mechanical Engineering, Blekinge Institute of Technology we wish to thank Dr. Ansel J Berghuvud for his valuable support and guidance.

Our thanks would have been incomplete without mentioning the name of Dr. Nirav Desai, C.E.O, Cybernetik Technologies Pvt. Ltd for granting us the permission to carry out the thesis work in his firm.

We would also like to thank Mr. M. Kaduskar for his support throughout the thesis work, lastly we would thank all the staff at Cybernetik, for their support and coordination during the thesis work.

Pune, January 2006

Mohammed Naser Farooqui

Suresh Arunachalam
# Contents

1 Notation 5
2 Introduction 6
   2.1 Objectives 6
3 Analysis Approach 9
   3.1 Description of loader arm 9
   3.2 Description of work piece 10
4 Finite Element Analysis 11
   4.1 Steps of Finite Element Analysis 11
5 Finite Element Modeling 13
   5.1 Stiffness Matrix 13
   5.2 Computation of nodal load vector 14
   5.3 Computation of Final Stresses 14
6 Modeling in ANSYS 16
   6.1 Stages of modeling in ANSYS 17
      6.1.1 Pre-processing 17
      6.1.2 Solution stage 20
      6.1.3 Post-processing 20
   6.2 Results for Original Design 21
   6.3 Criteria for Accuracy 25
7 Optimization 27
   7.1 Steps Incorporated before Optimization 27
   7.1 First Stage of Optimization 28
   7.2 Second Stage of Optimization 33
   7.3 Comparison of Results 38
8 Discussion and Further Work  39
  8.1 Future Work  39
9 Conclusions  40
10 References  41
1 Notation

\( E \) \hspace{1cm} \text{Young’s modulus}

\( M \) \hspace{1cm} \text{Bending Moment}

\([B], [C]\) \hspace{1cm} \text{Matrices for the Beam Element}

\( I_z \) \hspace{1cm} \text{Moment of inertia}

\([k_m]\) \hspace{1cm} \text{Stiffness Matrix for 3-D Beam}

\( x_m, y_m, z_m \) \hspace{1cm} \text{Degrees of Freedom}

\( \sigma \) \hspace{1cm} \text{Stress}

\( \{Q\}, \{Q_m\}\) \hspace{1cm} \text{Nodal Load Vector}

\( \{d_m\}\) \hspace{1cm} \text{Nodal Displacement}

\( \{d_m', d'\}\) \hspace{1cm} \text{Displacements at One Node with Reference to the Global Axes}

\( \{S_0\}\) \hspace{1cm} \text{Stress Resultants}

\( \mu \) \hspace{1cm} \text{Poisson’s Ratio}
2 Introduction

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. 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 .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 a loading cycle in complete 3 seconds. Hence the task was to optimize the loader arm from stress and deflection point of view. And to obtain an optimum design that could show the least deflection in the loader arm. Images of the loader arm are shown in Figures 2.1-2.4.

2.1. Objective

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. 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. In this way the structural behaviour of the arm is studied.
Figure 2.1 Position of the loader arm during still position.

Figure 2.2 View of the loader arm during loading the work piece condition.
Figure 2.3 View of the loader arm during unloading the work piece.

Figure 2.4 Normal view of the loader arm.
3 Analysis Approach

The static analysis approach will highlight the various considerations and steps that were taken during the analysis of the loader arm. For the analysis of the loader arm finite element method was used to study the deflection and the stresses induced in the structure as mentioned by C.S.Krishnamorthy [1], Klaus Jurgan Bathe [2]. Once the loader arm is analyzed using FEA the loader arm is then modelled as studied in chapter no.5 in this thesis report and then the arm is modelled using ANSYS software.

3.1 Description of the loader Arm

As seen from Figure 2.1, 2.2, 2.3, and 2.4 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. 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 a loading cycle in complete 3 seconds. 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.

3.2 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 6.3 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. The various shapes of the work piece are seen in Figure 2.1 to Figure 2.4 as shown in chapter 2 of this report.

The strategy adopted here is first the structure is analyzed using FEA and later it is modelled 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 in chapter no.6.

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 within the safe design limit. 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.
4 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 [1], Klaus Jurgan Bathe [2] and Tirupathi R. Chandrpatla and Ashok D. Belegundu [3] 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 [1], [3] 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.

4.1 Steps involved in Finite Element Analysis

The following are the steps that are followed in the analysis of the structure by finite element analysis (FEA).

**Discretization and Pre-Processing of Finite Element Model:**

The structure is first distributed into smaller elements of finite dimensions called as finite elements. The structure is then considered as an assemblage of these elements connected at a finite number of joints called as nodes or nodal points. The properties of the element are formulated and combined to obtain the solution for the entire structure.

**Computation of Element Properties:**

Using the above step, the strain displacement matrix, element stiffness matrix and nodal vector are computed for each element.

**Assemblage of Elements:**

The direct stiffness method is used to constitute the global stiffness matrix $[k]$ and nodal load vector $\{P\}$. 
The connectivity relation between element and global degree of freedom is used to compute the contribution from an element to the global stiffness matrix $[k]$.

**Solution of Equation of Equilibrium:**
The linear simultaneous equations for equilibrium $[k][r] = \{P\}$ are solved for the nodal displacement of the structure.

**Computation of Stresses:**
The stresses at various points in the element are computed.

Graphics based software ANSYS is used for this stage, since it provides with all the necessary outputs in the graphical form which highlights the various stresses induced in the structure and its quiet compactable for taking necessary iterations in the design.
5 Finite Element Modeling

The finite Element 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 Jurgan Bathe [2], 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 [1], [2] and the beam element selected for this framed structure is a three dimensional beam element.[2],[3].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.

5.1 Stiffness Matrix for a Three-Dimensional Beam

The following is the element stiffness matrix for the three dimensional beam element which has been described by Klaus Jurgan Bathe [2].

$$[k_m] =$$
5.2 Computation of element nodal load vector

The nodal loads due to loads acting on the element can be transformed to the global system using the given equation as

\[ \{Q\} = [T]^T \{Q_m\} \]  

(5.1)

5.3 Computation for Final Stress Resultants

In finite element analysis the stress is computed as

\[ \{\sigma\} = [C][B]\{d\} \]  

(5.2)

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 can be expressed as

\[
\{S\} = [k_m] \{d_m\} \quad (5.3)
\]

To the above stress resultants we should add the stress resultants due to loads on the member under fully restrained condition.

\[
\{\sigma\} = [k_m] \{d_m\} + \{S_0\} \quad (5.4)
\]

Where \{S_0\} represents 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.
6 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 [4].

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 [4]. The properties of this element are mentioned as follows:

PIPE16 is a uniaxial element with tension-compression, torsion, and bending capabilities. 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 (BEAM4), 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 [4] of the loader arm are obtained as shown in Figure 6.1 and the Figure 6.2 shows the yellow portion on the meshed structure that is the application of the degrees of freedom on the meshed structure.

Figure 6.3 shows the application of the static loading on the meshed structure, in the Figure 6.3 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 output of the stresses present in the structure, stresses like bending, axial and Von Mises stresses which are nothing but the generalized stress distribution on the surface of the structure. The graphical output of the deflection in the structure is also seen and it can be simulated to see the motion of the deflection in the structure.
6.1 Stages of modeling in ANSYS

The following are the three main processes involved during the modeling of ANSYS. ANSYS, Analysis Guide [4].

6.1.1 Pre-processing

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 [4]. 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 Figure 6.1, where the entire arm is show in the meshed view by making use of PIPE 16 element in ANSYS.

![Figure 6.1 Meshed model of the structure using PIPE16 element in ANSYS](image)

*Figure 6.1 Meshed model of the structure using PIPE16 element in ANSYS*
Figure 6.2 shows 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. In the Figure 6.2 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.

The Figure 6.3 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 Figure 6.3 indicate the load acting in the downward direction, along with the complete meshed model of the loader arm.
Figure 6.3 View 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 6.4 where the yellow portion shows the degrees of freedom and the red arrows shows the application of the static load in the downward direction.
6.1.2 Solution stage
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.

6.1.3 Post-processing
This is the stage the results of the performed analysis are obtained as both numerical and graphical output.

Figure 6.4 View showing the Application of Load and Degrees of Freedom on the Meshed Model.
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.

### 6.2 Results for Original Design

The following where the graphical output obtained when the ANSYS program was run. This shows the various stresses acting on the loader arm.

![Static Analysis of Loader Arm](image)

**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 Figure 6.5.
**Figure 6.6 Bending stresses in the arm.**

A bending stress of 98.016 N/mm$^2$ is seen in the Loader arm during the ANSYS analysis as seen in the Figure 6.6.
Figure 6.7 Deflection observed in the loader arm.

A deflection of 13.276mm is seen in the loader arm in the above Figure 6.7.
Figure 6.8 Deflection as seen in the ANSYS model of the arm.

The Figure 6.8 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 Figure 6.9.
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.

### 6.3 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 in the safe design limit or not.

Calculations where 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 within in the safe design limits.
7 Optimization

Optimization plays a crucial role in enhancing the performance of any product and it is also important from an economic point of view. Optimization has a major role to play in the present work.

It helps to get the 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.

7.1 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 seen in Figures 7.1 and 7.6. The simulations were performed on the altered structures and the results were obtained.
7.1 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 7.1 shows the solid model of the loader arm when its geometry is altered.

Figure 7.1 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.
Figure 7.2 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 were observed when the geometry of the structure was changed as seen in Figure 7.3. Bending stress of 71.237 N/mm$^2$ was seen.
A deflection of 7.698 mm was noticed when the geometry of the structure was altered for the process of optimization. As seen in the Figure 7.4.
The following where the results that were obtained when the first stage of optimization is performed, as seen in figures 7.1 to 7.5 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.

- 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.
7.2 Second Stage of Optimization

A second stage of optimisation was performed to reduce the deflection even more than in the first optimisation 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 7.6.

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 7.6 View showing the changes performed on the geometry of the arm for the second stage of optimization.
Figure 7.7 Axial stresses for second stage of optimization.

The Figure 7.7 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.
Figure 7.8 Bending stresses for second stage of optimization.

A Bending Stress of 48.399 N/mm^2 is noticed from the Figure 7.8 when the second stage of optimization is performed.
A deflection of 2.971mm is seen in the loader arm when the simulation is performed for the second stage of optimisation, see Figure 7.9.

Figure 7.9 Deflection for the second stage of optimization.
The Max. 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 were obtained during the second stage of optimization:

- Max. Deflection = 2.971 mm.
- Max. Bending stress = 48.399 N/mm².
- Max. Axial stress = 30.329 N/mm².
- Max. Von Mises Stresses = 48.212 N/mm².

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 within the safe limit.
7.3 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 with in the safe design limit.

Table 7.1 Results after Optimization.

<table>
<thead>
<tr>
<th></th>
<th>Original Design</th>
<th>First Optimization</th>
<th>Second Optimization</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Max. Deflection</strong></td>
<td>13.276 mm</td>
<td>7.698 mm</td>
<td>2.971 mm</td>
</tr>
<tr>
<td><strong>Max. Bending stress</strong></td>
<td>98.016 N/mm²</td>
<td>71.237 N/mm²</td>
<td>48.399 N/mm²</td>
</tr>
<tr>
<td><strong>Max. Axial Stress</strong></td>
<td>96.721 N/mm²</td>
<td>71.220 N/mm²</td>
<td>30.329 N/mm²</td>
</tr>
<tr>
<td><strong>Max. Von Mises Stress</strong></td>
<td>96.967 N/mm²</td>
<td>71.223 N/mm²</td>
<td>48.212 N/mm²</td>
</tr>
</tbody>
</table>

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 Cybernetik Technologies Pvt. Ltd, Pune, India.
8 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 in the Figure 11.1 in chapter 11 cannot be altered further; it needs to be restricted to the position as seen in Figure 11.1 in chapter 11 of this report. 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.

8.1 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.
9 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.
10 References
