International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 8 Aug. 2022, pp: 1603-1619 www.ijaem.net ISSN: 2395-5252

Analyzing the Vibration Failure in Pick Head of the Pick and Place Robot Arm

S.Sivakumar

Assistant Professor, Ganesh College of Engineer Salem, Tamilnadu

Submitted: 15-08-2022 Revised: 27-08-2022 Accepted: 30-08-2022

ABSTRACT: this project presents the Analysis of stress distribution and deformations occur in the transfer arm of the Pick and Place Transfer Module. Initially, the transfer arm is designed in CAD tool Pro-E and analyzed by using the FEA tool – ANSYS 10. The result shown that the maximum Von Mises stress occur at the mounting portion of linear bearing blocks and ball screw mechanism, whose wear is hazardous to the transferring accuracy of the module. Similarly the maximum deformations occur at the far end portion of the Transfer Arm, which impair the Unit per hour (UPH) productivity. In this project, an attempt will be made in order to reduce the stress distribution at the critical area as well the deformation of the pick head by selecting suitable materials. From this analysis, the obtained result is the alloy 7075 is till rigid enough even the transfer arm running at 10g acceleration under the mass of 10kg.

Keywords— Transfer arm, stress distribution, FEA-Finite element analysis, deformation, Duralumin, Alloy 7075.

I. INTRODUCTION

In electronics manufacturing, integrated circuit packaging is the final stage of semiconductor device fabrication, in which the tiny block of semiconducting material is encased in a supporting case that prevents physical damage and corrosion. The case, known as a "package", supports the electrical contacts which connect the device to a circuit board.

Nearly all the IC packages handling systems operated in the back-end process of semiconductor industry there have pick & place transfer module. The pick head of the module is driven by servomotor through time belt or ball screw mechanism. It picks up the IC package from its vehicle and transfers it to a designated location with high accuracy, where other devices such as lead conditioning combs or laser marker will perform their function onto the IC package (Fig 1.1).

This transferring is required to be as fast as possible in order to reduce the process cycle time of the overall handling system. However, based on Newton's law, (Force = Mass x derivative of speed), higher UPH (unit per hour) requirement usually leads to great inertia force induced from huge acceleration at the starting of each transferring and huge deceleration at the end of it. If the mass of moving part is heavy, the induced force not only intensifies the vibration of the handling system calling for more time to settle down, but also demands bigger sized motor and thus induces more cost, and the stress distribution inside the
Related mechanical part becomes worse.

Figure 1.1

Lead Conditioning Machine with Pick & Place Transfer Module (X-Y-Z)

Due to the considerable periodic impact force generated from the sharp accelerating and decelerating, the life span of the pick & place transfer module is shortened. To reduce the impact force and consequently reduce the critical stress and system settle down time, the mass of the moving part has to be minimized, as derivatives of speed of the moving part is expected to increase.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 8 Aug. 2022, pp: 1603-1619 www.ijaem.net ISSN: 2395-5252

1.2 EXISTING PROBLEMS

Following the merciless competition in the market, manufacturers rack their brains on how to enhance their competency. One of the methods they are applying is increase their machine's Units per Hour (UPH). According to Newton's law, (Force = Mass x derivative of speed), this will generate considerable impact force inside the machine and greaten its vibration. The force stress distributions of the involved parts are worsen. Normal design method by simply increasing the part's sizes is too risky under the high UPH condition. Finite Element Method reveals its advantages.

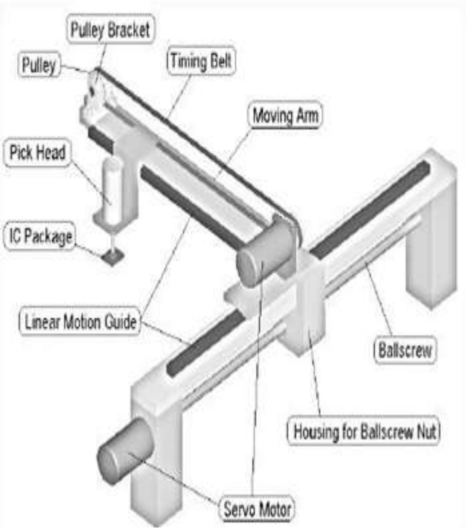


Figure 1.2
Pick & Place Transfer Module (X-Y)

To meet the UPH requirement from the customer, the machine is speeded up and the Transfer Module is running with high acceleration and deceleration. The resultant considerable machine vibration and the significant noise come from the transferring ball screw mechanism bring into being the doubts that can the Transfer Arm (figure 1.2) withstand the additional periodic impact force? When does this module fail? How fast can it run?
Normal design experience cannot be based to make any convincing assurance when dealing with such reliability issues on complex mechanical parts in a high precision, high speed machine.

1.3 A PROFESSIONAL PICK & PLACE AUTO PROGRAMMER FOR ICS

Figure 1.3
AT3-300A - A Professional Pick & Place Auto Programmer for ICs

1.3.1 Excellent performance
The system can operate a variety of input and output options with tray, tube and tape thus provides programming, marking and packing conversion for most of IC products on / in tray, tube and tape.

1.3.2 Intelligent operation
Automatic pick & place operation in sequence of loading / positioning / marking (optional for tray / tube ICs) / programming /sorting / marking (optional for tape-out ICs) / unloading with powerful software control.

1.3.3 Efficient programming
The new generation programmers resided in System, ALL-100G, are designed with high speed CPU, built-in FPGA, high capacity pin drivers and USB interface, thus providing a high speed, low noise, stable and reliable programming platform.

1.3.4 Optional marking
System provides dot / alphanumeric marking for tray, tube and tape ICs.

1.3.5 Flexible packing conversion
System offers IC packing conversion between tray, tube and tape through fast change-over tray / tube / tape I/O devices installed to the working platform.

1.3.6 Accurate positioning
Being equipped with two precise CCD. One fixed, upward CCD for IC positioning while the other

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 8 Aug. 2022, pp: 1603-1619 www.ijaem.net ISSN: 2395-5252

carried, downward CCD for sockets / pick & place spots positioning.

1.3.7 Multi-site programming
Up to 4 sets / 32 sites for AT3-300A and 8 sets / 64 sites for AT3-300AL can be programmed simultaneously, thus minimizing handler idle time when programming high density memory devices.

1.3.8 Fast change-over and easy maintenance
Power-on self diagnostics available. Module design allows easy maintenance, repair and replacement as well as fast change-over between I/O devices.

1.3.9 High throughput
900 UPH, 4.0 sec/unit handler index time for AT3-300A and 800 UPH, 4.5 sec/unit for AT3-300AL. System offers 900~800 units/hour consistent throughput for ICs with programming time less than 96 sec. for AT3-300A and 252 sec. for AT3-300AL.

1.3.10 Minimizing setup change
Universal pin driver design makes socket module universal for various product types with same package, thus minimizing setup change when switching product type for programming. If it comes with the same package, no any hardware change is needed. If package is different, it only needs a simple pull-plug for socket module replacement.

1.3.11 Powerful operation software
Auto socket positioning makes system easy to setup. Both setup data and test results are automatically saved for next power-on operation as well as for quality / yield traceability. Graphical user interface makes it easy to access.

II. LITERATURE SURVEY

Tiefu Shao-Analysis of Potential Failure in Pick & Place Transfer Module: In this project, the stress distribution in the critical part -Transfer Arm of a Pick and Place Transfer Module is primarily studied by applying Finite Element Method and using the advanced FEA tool – ANSIS5.7. He identified that the maximum Von Mises always occur at the mounting area of linear bearing blocks whose wear and tear is hazardous to the transferring accuracy of the module; and the maximum deformation always occur at the far end portion of the Transfer Arm at where the Pick Head requires more time to settle down which impair the UPH - the index of the machine handling capability. Finally he gave some Corresponding suggestions are made for the design of the Pick & Place Transfer Module and for the future advanced investigation to the module.

Takeshi Takaki and Toru Omata - 100g-100N Finger Joint with Load-Sensitive Continuously Variable Transmission: in this project instant of DC motor a five bar linkage CVT-Controlled variable transmission finger joint was taken. It is analyzed by finite element method. The strength of the machine can be improved by selecting the suitable material, and also the machine exerts greater fingertip force. They conducted experiment and identified the machine exerts the fingertip force more than 100n even running at angular velocity more than 550 deg/sec. the speed of CVT can be controlled by shape memory alloy brake of 0.56 g, which consumed less power compared to DC motor**.**

S. S. ohol, s. r. kajale - Simulation of Multi finger Robotic Gripper for Dynamic Analysis of Dexterous Grasping: in this project a multi fingered grippers was designed to improve the grasping contact. For better grasping the sensor used at the fingers. Finally the force exerted by multi finger has been analyzed and simulated.

Damanbir Singh Virk, V. K. Banga - Overloading Failures in Robot Manipulators :This paper presents the different types of robots used in industries and also the mechanical hazards associated with robot manipulator with special emphasize on overloading. The key parameter in providing the motion to a robot i.e. DOF has also been discussed. This research paper has reviewed the hazards correlation with overloading; different categories of loads have also been listed to give a brief preface about overloading.

R. Marappan **-** An experiment is conducted with three different hyper elastic materials and the deformations and contact width for various loads are determined. The material properties of finger materials are computed using measured deformation. The suitable soft material for developing robot soft fingers is then selected based on the deformation of the finger and frictional coefficients between finger material and the selected target materials. The numerical simulation is also done for the developed model, and it well matches with the experimental results. The developed soft finger model and its force function can be usefully applied to soft-fingered object manipulations in future.

I.J. Polmear – in this project three series of cast and wrought alloys emerged that are based on the Al-Cu- (Mg), Al-Mg-Si and Al-Zn-Mg-(Cu) systems are used for the material of transfer arm. But recently, a major effort has been made to exploit the age hardening potential of alloys containing the light element lithium is used as material for transfer arm.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 8 Aug. 2022, pp: 1603-1619 www.ijaem.net ISSN: 2395-5252

2.1 PROBLEM IDENTIFIED

In existing project, for the same design of the mechanism used in the pick & place transfer module. Aluminum alloy 6061 used as the material for transfer arm and maximum stress distribution and maximum displacement were calculated under various mass and acceleration. Alloy 6061 reduces the stress distribution in considerable value but the stress distribution increases due to load increases and displacement will reach maximum value under increasing of mass.

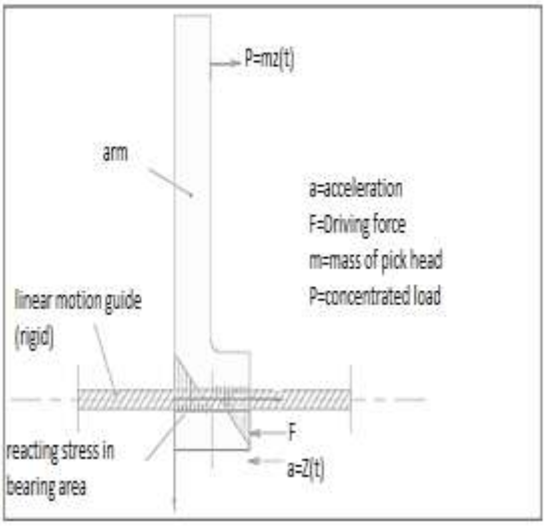


Figure 2.1
Effect of stress and displacement on transfer arm

2.2 PROBLEM SOLUTION

To reduce the impact of this problem alloy 6061 replaced as duralumin and extra super duralumin whose yield strength are 280MPa and 620MPa respectively. From this we identified the stress and displacement will be considerably reduced as compared to alloy 6061 for the same mass and acceleration.

2.3 MATERIALS DISCOVERED AND ITS PROPERTIES

2.3.1 Duralumin.

Alloy of aluminum (over 90%) with copper (about 4%), magnesium (0.5%-1%), and manganese (less than l%). Before a final heat treatment the alloy is ductile and malleable; after heat treatment a reaction between the aluminum and magnesium produces increased hardness and tensile strength.

Duralumin (2017) had yield strength of 280MPa, and the historical development of alloys with progressively increasing strength levels that have been used for upper wing skins of commercial aircraft is depicted in Fig. below. These improvements have come from the development of new alloy systems, modifications to compositions within particular systems, and from the use of a range of multi-stage ageing treatments (tempers).

2.3.2 Extra super Duralumin A7075

- The strongest aluminium alloy.
- Zn5.6 Mg2.5 Cu1.6 Cr0.25 and Al.

Al-Zn-Mg-(Cu) alloys

In 1923, Sander and Meissner in Germany found that some ternary Al-Zn-Mg alloys showed a greater response to age hardening than any other compositions investigated at that time. The potential of these alloys for aircraft materials was recognized in several countries but their adoption was delayed because they proved very susceptible to stress corrosion cracking (SCC). In Japan, an alloy known as ESD (Extra Super Duralumin) was developed that was successfully introduced to lower the weight of the Zero fighter aircraft in 1938 . When this innovation was revealed to the Allies through chemical analysis of a crashed aircraft during World War 2, similar alloys were quickly used for the manufacture of military aircraft in the USA and England.

The best known was 75S (later 7075: Al- 5.6Zn-2.5Mg-1.6Cu-0.23Cr), which was used for the skin and stringers of the American B- 29 "Super Fortress" bomber aircraft, and provided an immediate weight saving of 180kg . The SCC resistance was improved due primarily to the inclusion of Cu as an alloying element and the production of extrusions, forgings and plate soon followed.

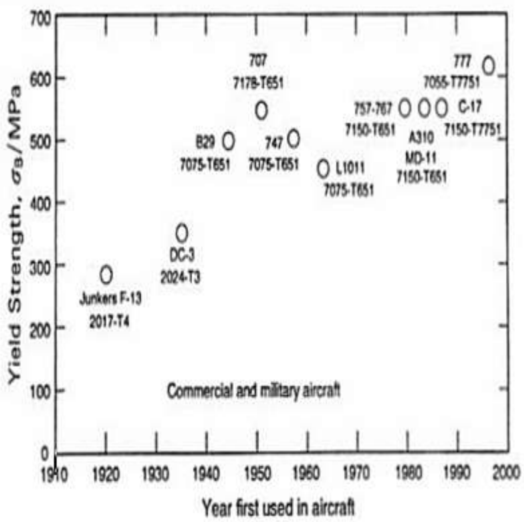


Figure 2.2
Al-Zn-Mg-(Cu) system

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 8 Aug. 2022, pp: 1603-1619 www.ijaem.net ISSN: 2395-5252

Table2.1: Materials Discovered and its Properties

MATERIAL	Young's modulus E (MPa)	Yield strength S_y (MPa)	S_y/E x1000
Teflon (PTFE)	345	23	66.7
Delrin	2068	69	33.4
Nylon (Type 6)	2620	81	30.9
E-glass (73,3vol%) in epoxy	56000	1640	29.3
Polypropylene	1400	34	24.4
Polyethylene (HDPE)	1400	28	20.0
Steel (Sandvick11R15)	186000	1950	10.5
Titanium Ti-13 Heat Treated	114000	1170	10.3
Aluminium-7075Heat Treated	71100	503	7.1

The alloy 7075 was widely adopted for post-war passenger and military aircraft including the widely used Boeing 707 and the ill-fated Comet. Since then, continued demands for materials with higher and higher strength: a weight ratio has led to the development of a family of alloys based on the Al-Zn-Mg-Cu system and this progressive trend is shown in fig above. A recent alloy, 7055 (Al-8Zn-2.05Mg-2.3Cu-0.16Zr), has a yield strength that may exceed 620MPa and the estimated weight saving attributed to its use for components in the Boeing aircraft 777 is 635kg . Crucial to the introduction of these more highly alloyed compositions has been the parallel development of complex multi-stage ageing tempers, some incorporating cold or warm working, that allow adequate ductility, fracture toughness and a satisfactory resistance to SCC to be maintained.

III. TRANSFER ARM MODULE

In this project, the problem of the Transfer Module is simplified as to analysis the stress distribution of the Transfer Arm, the critical part of a Transfer Module is designed (refer to figure 3.1), based on the assumptions that are,

1) The material of Transfer Arm is linear elastic, Homogeneous and isotropic;
2) The Linear Motion Guide and the Ball screw mechanism are rigid;
3) The speed change rate of the Transfer Arm is constant;
4) The period of speed changing is infinitely long;
5) Friction forces are neglected;
6) Inertia force from Pick Head is Point load.

3.1 BASIC GRASP THEORY

A grasp - set of contacts on the surface of the object. The forces or torques the manipulator can exert in these contacts depend on the contact model and on the abilities of the manipulator. Here we only consider precision grasps, where only the fingertips are in contact with the object, so we can focus on the contact model and neglect manipulator constraints. After defining the contact model one can formalize commonly used closure properties of a grasp and we give a quality measure for grasps proposed by Ferrari and Canny. These are all prerequisites for our statistical analysis of grasps on a set of generic objects.

3.2GRASP CONTACT

The mostly used fingertip contact models are hard-finger contacts with and without friction and soft-finger contacts. All these contacts can be modeled as single point contact. In the first case, hard-finger without friction, only forces against the surface normal in the contact point can be exerted on the object (fig. 3.1 A). With friction all forces that lie within the friction cone around the surface normal can be exerted. The cone angle is defined as $\alpha = \tan(\mu)$ where μ is the friction coefficient. (fig. 3.1 B). With soft-finger contacts a torque around the normal can also be applied in the contact point (fig.3.1 C).
There are also more realistic but more complicated models, especially for soft-finger contacts. For the following considerations the simple ones are sufficient.

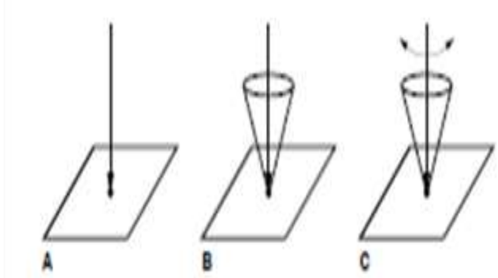


Figure 3.1 Finger contact models: A - hard-finger frictionless, B - hard-finger with friction, C - soft-finger.

In the following sections we assume to have hard finger contacts with friction. The other contact types either unrealistic, as the one without friction, or unnecessary complex as the soft finger contact where the effect of the torque around the surface normal can almost be neglected for more than two contacts.

3.3 MODELING AND ANALYSIS OF A ROBOT FINGER

The dynamic force of the transfer arm is limited by the mechanical strength of its links and

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 8 Aug. 2022, pp: 1603-1619 www.ijaem.net ISSN: 2395-5252

bearings. If the transfer arm strength can be improved, a much dynamic force can be expected. The present study aims at improving the rigidity of the transfer arm by improving its mechanical strength. In order to improve the mechanical strength of the transfer arm, vast study has been done on the materials used to make transfer arm and connections to the other components. To carry out analysis in ANSYS, preliminary data's such as model of the transfer arm, material of the transfer arm and its properties, force calculations, boundary conditions are required. From the literature collected, it's been evident different property materials can used to make transfer arm instead of with single material in order to improve its mechanical strength.

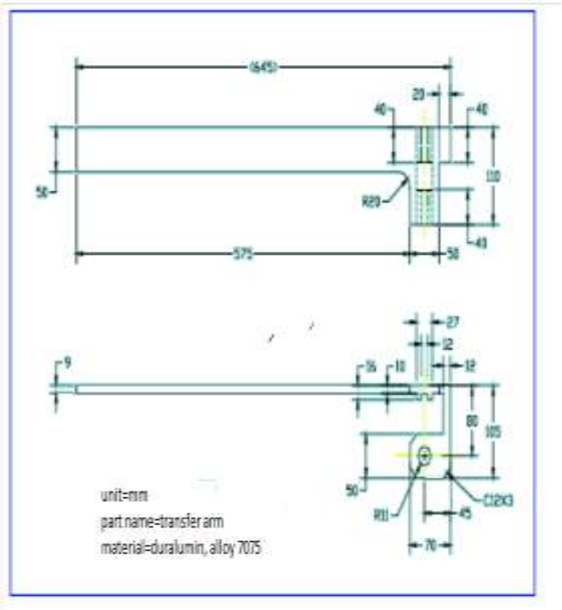


Figure 3.2
Model of the robot finger and its dimensions

3.4 DETAILED VIEW OF A TRANSFER ARM

The simplified mathematical model for the Transfer Arm is shown in figure 3.3.

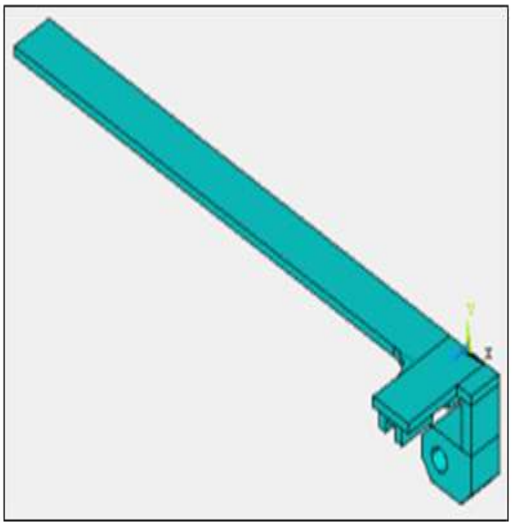


Figure 3.3

Detailed view of a Arm

Form the detailed view of the transfer arm of the pick and place transfer module, it is clearly understood that the operating mechanism is simple one based on ball and screw mechanism. The transfer arm moves in a linear fashion over the ball screw mechanism. Various forces acting on the transfer arm are shown in the figure above. As the transfer arm is moving with acceleration, the arm is subjected to dynamic loading which in turn produces time varying stresses at the critical regions of the arm. Mass of the pick head plays a crucial factor in deciding the magnitude of the dynamic forces. By varying the mass of the pick head the magnitude of the time varying stresses in the transfer arm would be controlled and limited to the minimum value.

IV. FINITE ELEMENT ANALYSIS

4.1 INTRODUCTION

The finite element method is numerical analysis technique for obtaining approximate solutions to a wide variety of engineering problems. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in almost every industry. In more and more engineering situations today, we find that it is necessary to obtain approximate solutions to problem rather than exact closed form solution. It is not possible to obtain analytical mathematical solutions for many engineering problems. An analytical solutions is a mathematical expression that gives the values of the desired unknown quantity at any location in the body, as consequence it is valid for infinite number of location in the body. For problems involving complex material properties and boundary conditions, the engineer resorts to numerical methods that provide approximate, but acceptable solutions.

The finite element method has become a powerful tool for the numerical solutions of a wide range of engineering problems. It has been developed simultaneously with the increasing use of the high-speed electronic digital computers and with the growing emphasis on numerical methods for engineering analysis. This method started as a generalization of the structural idea to some problems of elastic continuum problem, started in terms of different equations.

4.2 PROCEDURE FOR ANSYS ANALYSIS

Static analysis is used to determine the displacements stresses, stains and forces in structures or components due to loads that do not induce significant inertia and damping effects. Steady loading in response conditions are assumed. The kinds of loading that can be applied in a static analysis include externally applied forces and pressures, steady state inertial forces such as gravity or rotational velocity imposed (non-zero) displacements, temperatures (for thermal strain). A static analysis can be either linear or non linear. In our present work we consider linear static analysis.

The procedure for static analysis consists of these main steps

- ❖ Building the model.
- ❖ Obtaining the solution.
- ❖ Reviewing the results.

4.3 APPLY FEM TO SOLVE THE PROBLEMS

Finite Element Method (FEM) is a numerical technique for solving any ordinary or partial, linear or non-linear, differential equations, which govern the behavior of physical systems. It approximates the complicated problem with acceptable and adjustable calculating accuracy by a collection of simple coupled algebraic problems that can easily be solved on digital computers. In this project, the problems addressed above are solved by applying the following eight procedures, which are described in detail in the succeeding paragraphs.

Procedures in Apply FEA Method
1. Simply the problems by making assumptions;
2. Build geometric model in ANSYS;
3. Select suitable type of finite element;
4. Assign material properties to finite element;
5. Mesh domain into contiguous set of finite elements jointed by common nodes;
6. Apply boundary conditions;
7. Solve finite element system of equations;
8. Access analysis results and interpret solution.

4.4 BUILD THE MODEL

In this step we specify the job name and analysis title use PREP7 to define the element types, element real constants, material properties and model geometry element type both linear and non- linear structural elements are allowed. The ANSYS elements library contains over 80 different element types. A unique number and prefix identify each element type.

E.g. BEAM 94, PLANE 71, SOLID 96 and PIPE 16

4.5 MATERIAL PROPERTIES

Young's modulus (EX) must be defined for a static analysis. If we plan to apply inertia loads (such as gravity) we define mass properties such as density (DENS). Similarly if we plan to apply thermal loads (temperatures) we define coefficient of thermal expansion (ALPX).

4.6 SOLUTION

In this step we define the analysis type and options, apply loads and initiate the finite element solution. This involves three phases:

- ❖ Pre-processor phase
- ❖ Solution phase
- ❖ Post-processor phase

4.6.1 Pre-processor

Pre processor has been developed so that the same program is available on micro, mini, super-mini and mainframe computer system. This slows easy transfer of models one system to other. Pre processor is an interactive model builder to prepare the FE (finite element) model and input data. The solution phase utilizes the input data developed by the pre processor, and prepares the solution according to the problem definition. It creates input files to the temperature etc. on the screen in the form of contours.

A) Geometrical definitions

There are four different geometric entities in pre processor namely key points, lines, area and volumes. These entities can be used to obtain the geometric representation of the structure. All the entities are independent of other and have unique identification labels.

B) Model Generations

Two different methods are used to generate a model:

- ❖ Direct generation.
- ❖ Solid modeling

With solid modeling we can describe the geometric boundaries of the model, establish controls over the size and desired shape of the elements and then instruct ANSYS program to generate all the nodes and elements automatically.

By contrast, with the direct generation method, we determine the location of every node and size shape and connectivity of every element prior to defining these entities in the ANSYS model. Although, some automatic data generation is possible (by using commands such as FILL, NGEN, EGEN etc) the direct generation method essentially a hands on numerical method that requires us to keep track of all the node numbers as we develop the finite element mesh. This detailed book keeping can become difficult for large models, giving scope for modeling errors. Solid modeling is usually more powerful and versatile than direct generation and is commonly preferred method of generating a model.

C) Mesh generation

In the finite element analysis the basic concept is to analyze the structure, which is an assemblage of discrete pieces called elements, which are connected, together at a finite number of points called Nodes. Loading boundary conditions are then applied to these elements and nodes. A network of these elements is known as Mesh.

D) Finite element generation

The maximum amount of time in a finite element analysis is spent on generating elements and nodal data. Pre processor allows the user to generate nodes and elements automatically at the same time allowing control over size and number of elements. There are various types of elements that can be mapped or generated on various geometric entities. The elements developed by various automatic element generation capabilities of pre processor can be checked element characteristics that may need to be verified before the finite element analysis for connectivity, distortion-index etc. Generally, automatic mesh generating capabilities of pre processor are used rather than defining the nodes individually. If required nodes can be defined easily by defining the allocations or by translating the existing nodes. Also on one can plot, delete, or search nodes.

E) Boundary conditions and loading

After completion of the finite element model it has to constrain and load has to be applied to the model. User can define constraints and loads in various ways. All constraints and loads are assigned set ID. This helps the user to keep track of load cases.

F) Model display

During the construction and verification stages of the model it may be necessary to view it from different angles. It is useful to rotate the model with respect to the global system and view it from different angles. Pre processor offers this capability. By windowing feature pre processor allows the user to enlarge a specific area of the model for clarity and details. Pre processor also provides features like smoothness, scaling, regions, active set, etc for efficient model viewing and editing.

G) Material defections

All elements are defined by nodes, which have only their location defined. In the case of plate and shell elements there is no indication of thickness. This thickness can be given as element property. Property tables for a particular property set 1-D have to be input. Different types of elements have different properties for e.g.
Beams: Cross sectional area, moment of inertia etc
Shell: Thickness
Springs: Stiffness
Solids: None

The user also needs to define material properties of the elements. For linear static analysis, modules of elasticity and Poisson's ratio need to be provided. For heat transfer, coefficient of thermal expansion, densities etc. are required. They can be given to the elements by the material property set to 1-D.

4.7 SOLUTION

The solution phase deals with the solution of the problem according to the problem definitions. All the tedious work of formulating and assembling of matrices are done by the computer and finally displacements are stress values are given as output. Some of the capabilities of the ANSYS are linear static analysis, non linear static analysis, transient dynamic analysis, etc.

4.8 POST- PROCESSOR

It is a powerful user- friendly post-processing program using interactive colour graphics. It has extensive plotting features for displaying the results obtained from the finite element analysis.
One picture of the analysis results (i.e. the results in a visual form) can often reveal in seconds what would take an engineer hour to assess from a numerical output, say in tabular form. The engineer may also see the important aspects of the results that could be easily missed in a stack of numerical data.
Employing state of art image enhancement techniques, facilities viewing of:

- ❖ Contours of stresses, displacements, temperatures, etc.
- ❖ Deform geometric plots
- ❖ Animated deformed shapes
- ❖ Time-history plots

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 8 Aug. 2022, pp: 1603-1619 www.ijaem.net ISSN: 2395-5252

- Solid sectioning
- Hidden line plot
- Light source shaded plot
- Boundary line plot etc.

The entire range of post processing options of different types of analysis can be accessed through the command/menu mode there by giving the user added flexibility and convenience.

4.9 GEOMETRIC MODEL AND MESHING

The fingertip force of the robot finger is limited by the mechanical strength of its links and bearings. If the finger strength can be improved, a much greater finger tip force can be expected. The present study aims at improving the rigidity of the robot finger by improving its mechanical strength. In order to improve the mechanical strength of the finger, vast study has been done on the materials used to make robot finger and links of the robot hands. To carry out analysis in Ansys, preliminary data's such as model of the finger, material of the finger and its properties, force calculations, boundary conditions are required. From the literature collected, its been evident different property materials can used to make single robot finger instead with single material in order to improve its mechanical strength**.** Using the commands such as key points, lines, extrude, shell, extrude cut, mirror, both the 2D and 3D models are created both in ANSYS and Pro/E which are shown in the figure below,

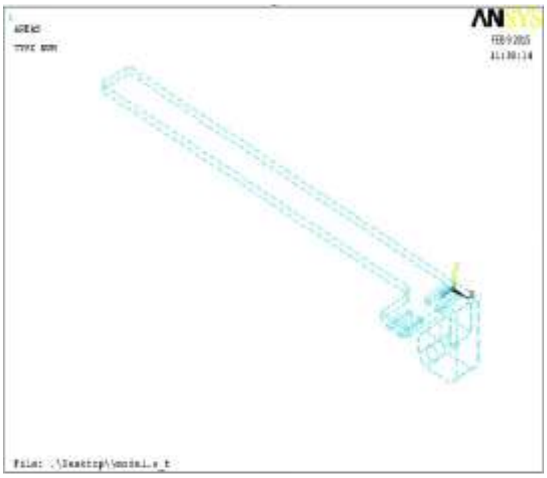


Figure 4.1
Model of the finger

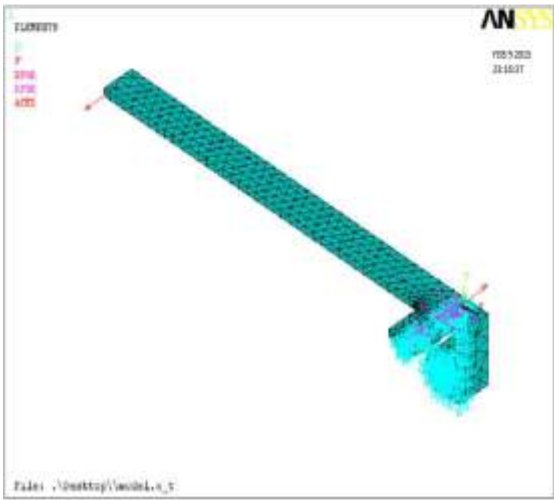


Figure 4.2
Transfer Arm model with mesh

The analysis of the robot finger has been done in Ansys by discretizing the finger by the 2D elastic beam element. The deformation of the finger under the gripping force with the appropriate boundary conditions is shown above. The deformation under different gripping force - which is based on the mass and the nature of the object – for two different materials of the robot finger is studied and compared the strength aspect among it. There are two methods to create the finite element model: solid modeling and direct modeling. With solid modeling, you describe the geometric shape of your model, and then instruct the ANSYS program to automatically mesh the geometry with nodes and elements. With direct modeling, you "manually" define the location of each node and the connectivity of each element. Creating a 3-D solid analysis model by direct generation method usually requires considerable effect.

Comparatively speaking, the solid modeling method is more powerful and versatile than the direct generation method. Based on the simplifications made in last procedure, the geometric model of Transfer Arm is created as in figure 3.3 and its sizes are shown in figure 3.2

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 8 Aug. 2022, pp: 1603-1619 www.ijaem.net ISSN: 2395-5252

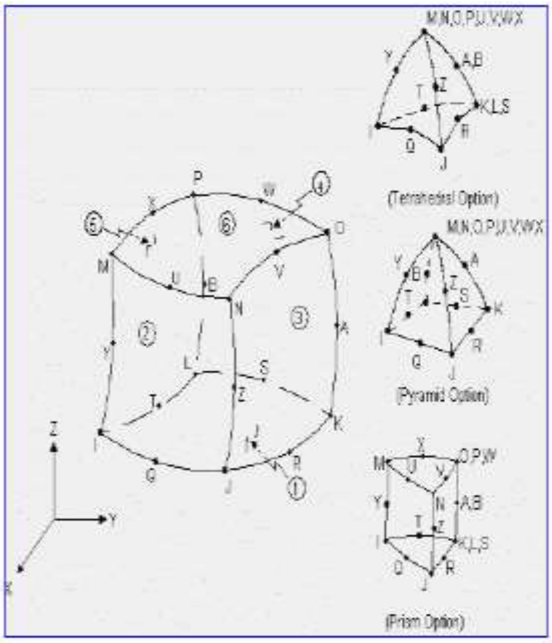


Figure 4.3
3-D structure stress analysis

For the 3-D structure stress analysis of the Transfer Arm, Solid95, figure 4, is the most suitable one that can tolerate irregular shapes without much loss of accuracy. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. SOLID95 elements have compatible displacement shapes and are well suited to model curved boundaries. In figure 8 is the diagram of Solid95 the 3-D 20-node structural solid element.

There have two methods in domain meshing, one is the smart sizing, and the other is the mapped meshing. Smart sizing is a meshing feature that creates initial element sizes for free meshing operations; it gives the mesher a better chance of creating reasonably shaped elements during automatic mesh generation. In this project, the smart sizing method is applied. Refinement is made at the inner corner of the Transfer Arm. One of the mesh results is shown in figure 4.2.

4.9.1 Solid 95

3-D 20-Node Structural Solid

SOLID95 is a higher order version of the 3-D 8-node solid element SOLID45. It can tolerate irregular shapes without as much loss of accuracy. SOLID95 elements have compatible displacement shapes and are well suited to model curved boundaries.

The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element may have any spatial orientation. SOLID95 has plasticity, creep, stress stiffening, large deflection, and large strain capabilities.

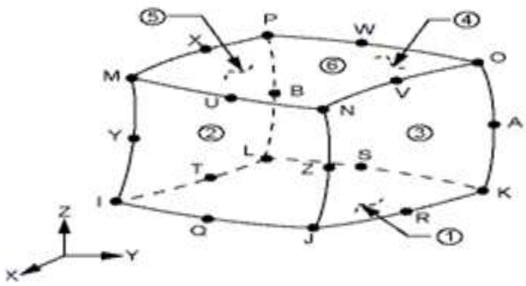


Figure 4.4
3-D 20-Node Structural Solid

1) SOLID95 Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom

UX, UY, UZ

Real Constants

THETA - x-axis adjustment (used only when KEYOPT(1) = 1)

Material Properties

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ or THSX, THSY, THSZ), PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),
face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

Temperatures

T(I), T(J), ..., T(Z), T(A), T(B)

Special Features

- Plasticity
- Creep
- Swelling
- Stress stiffening
- Large deflection
- Large strain
- Birth and death
- Adaptive descent
- Initial stress import

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 8 Aug. 2022, pp: 1603-1619 www.ijaem.net ISSN: 2395-5252

SOLID95 Assumptions and Restrictions

- The element must not have a zero volume.
- The element may not be twisted such that the element has two separate volumes. This occurs most frequently when the element is not numbered properly.
- An edge with a removed mid side node implies that the displacement varies linearly, rather than parabolically, along that edge.

4.10 MATERIAL PROPERTY

The material properties of Transfer Arm are listed in the table below. When assigning the material properties to the solid95 element, it is assumed that the Transfer Arm is linear elastic, homogeneous and isotropic.

- Modulus of Elasticity
- Tensile Strength
- Yield Strength
- Density
- Poisson's ratio

Table4.1 Properties of materials

MATERIAL	DURALUMIN	ALLOY 7075
Modulus of Elasticity, E(GPa)	73	82
Tensile Strength, σsut (MPa)	420-500	510-538
Yield Strength, σsyt (MPa)	280	620
Density, ρ (g cm^{-3})	2.8	2.8
Poisson's ratio	0.32	0.33

4.11 BOUNDARY CONDITIONS

The system of simultaneous equations of the problem wouldn't have any solutions if there are no correct boundary conditions to constraint them and give them initial values.

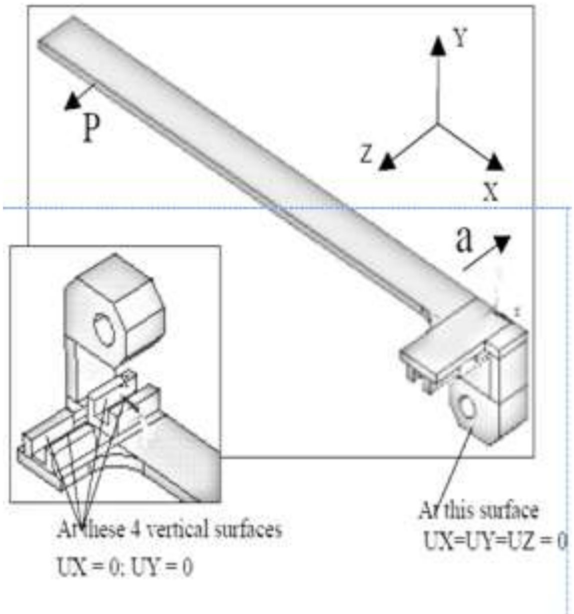


Figure 4.5

Boundary conditions

Based on the assumptions made in step 1, the displacement constraints of Transfer Arm come from Linear Motion Guide and Ball screw mechanism. Linear motion guide only allows Transfer Arm to move along its guiding directions i.e. Z-Axis. Thus the displacements on the guiding surfaces attached to Transfer Arm are zero, i.e. UX=UY=0. Ball screw mechanism drives Transfer Arm, there will not have any displacements between them, thus the DOFs of the surface contacting with Ball screw mechanism are zero, i.e. UX=UY=UZ=0.

During the sharp acceleration period, Pick Head with its considerable mass will produce noteworthy inertia force to resist the acceleration, its absolute value of the inertia force P = Mass of Pick Head Rate of Acceleration. For simplifying the calculation at the same time not losing much accuracy in interesting area, P can be simplified as a point load act on one node at the far end side of Transfer Arm. The inertia force is significant in the particular situation of this project, the acceleration field will be applied to the overall body of Transfer Arm. Together acting on Transfer Arm is the gravity field.

4.12 SOLVE FINITE ELEMENT SYSTEM OF EQUATIONS

Several methods of solving the system of simultaneous equations are available in the ANSYS program. In this project, the default sparse direct solution is applied. In this step, the computer will take over and solve the system of simultaneous

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 8 Aug. 2022, pp: 1603-1619 www.ijaem.net ISSN: 2395-5252

equations that the finite element method generates. Nodal DOF values and field derived values will be calculated and stored inside the computer.

4.13 ANALYSIS RESULTS

The most interesting items of the project are maximum Von Misses (SMX), maximum deformation (DMX), their locations, and their relation to the boundary conditions.

ANSYS RESULTS

The analysis is done on the transfer arm made up of Duralumin, Alloy 7075 and the results are given below. The figures show the different effect of stress distribution and displacement on transfer arm by different load condition and acceleration.

V. ANSYS RESULTS

5.1.1. Material 1&2-Gravity alone

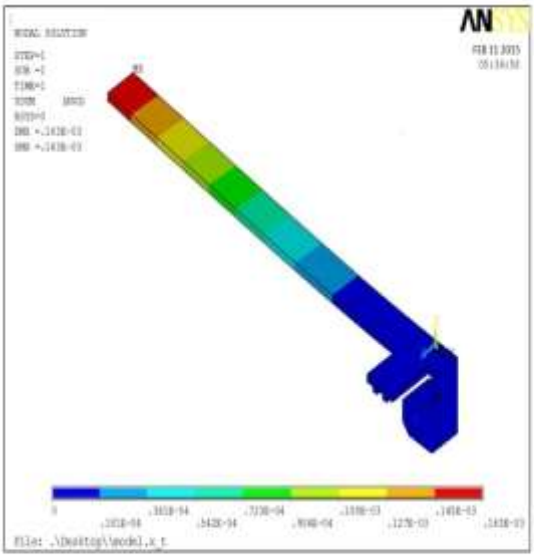


Figure 5.1
Displacement of Material 1 -Gravity alone

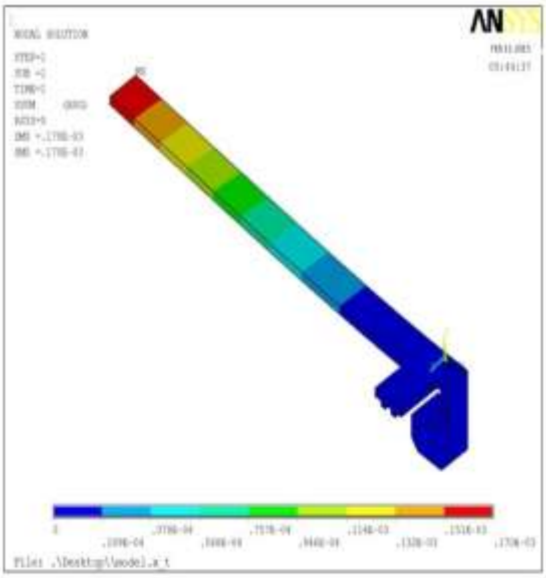


Figure 5.2
Displacement of Material 2 -Gravity alone

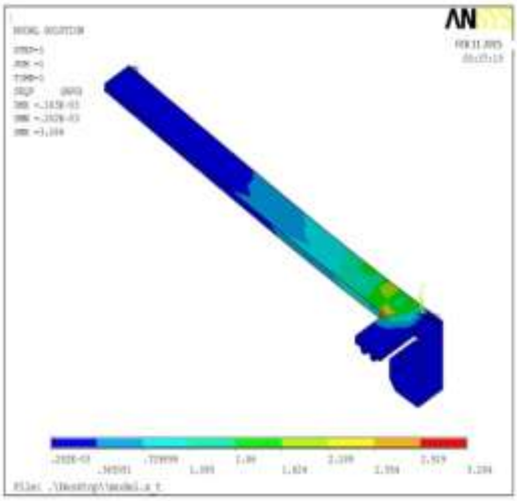


Figure 5.3
Stress of Material 1 -Gravity alone

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 8 Aug. 2022, pp: 1603-1619 www.ijaem.net ISSN: 2395-5252

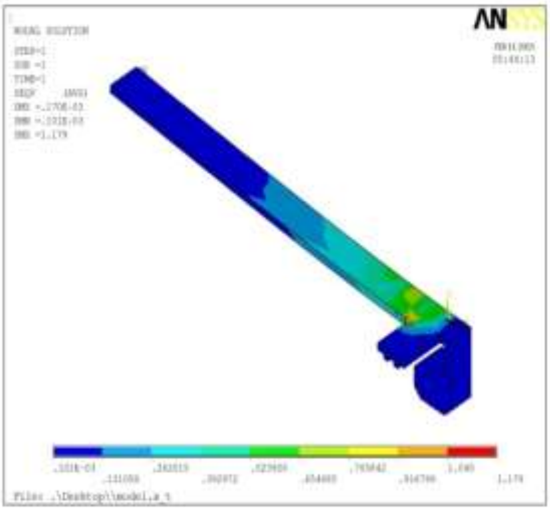


Figure 5.4
Stress of Material 2 -Gravity alone

5.1.2 Material 1&2 – [Gravity+ Acceleration (10g)+ Point Load]

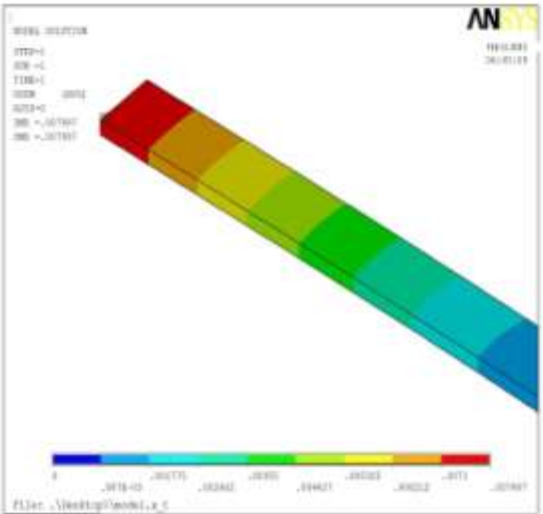


Figure 5.5(Material 1)
Displacement of Mass 6 kg & Acceleration 10g

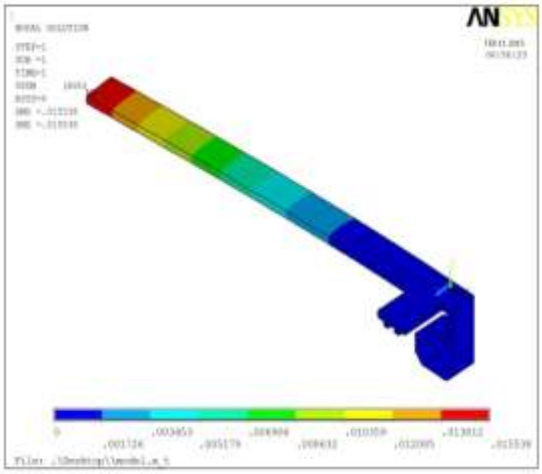


Figure 5.6(Material 2)
Displacement of Mass 6 kg & Acceleration 10g

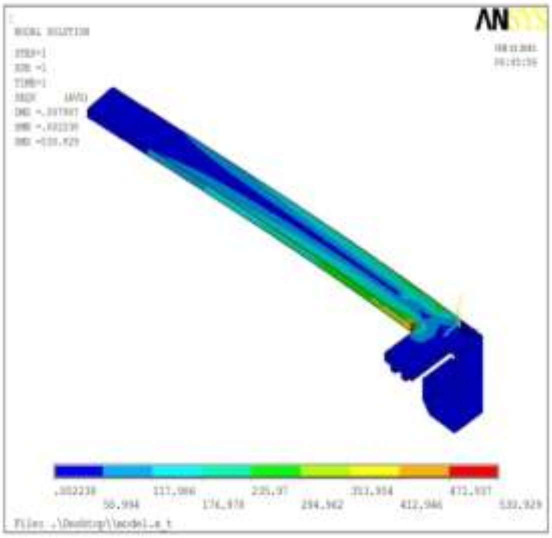


Figure 5.7(Material 1)
Stress of Mass 6 kg & Acceleration 10g

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 8 Aug. 2022, pp: 1603-1619 www.ijaem.net ISSN: 2395-5252

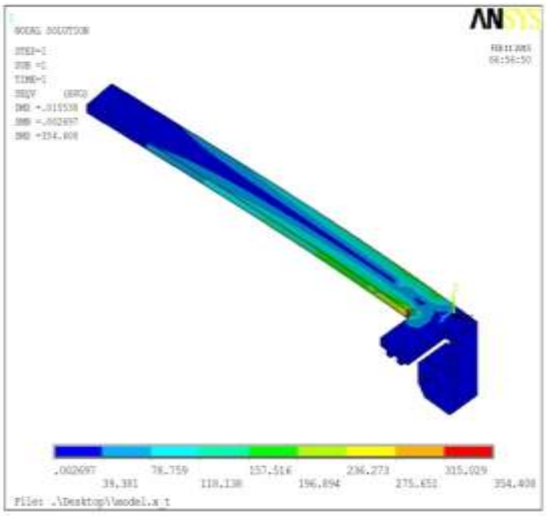


Figure 5.8(Material 2)
Stress of Mass 6 kg & Acceleration 10g

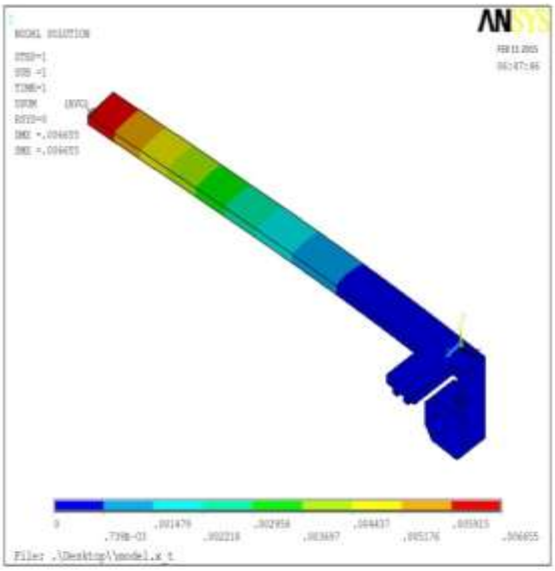


Figure 5.9 (Material 1)
Displacement of Mass 3 kg & Acceleration 10g

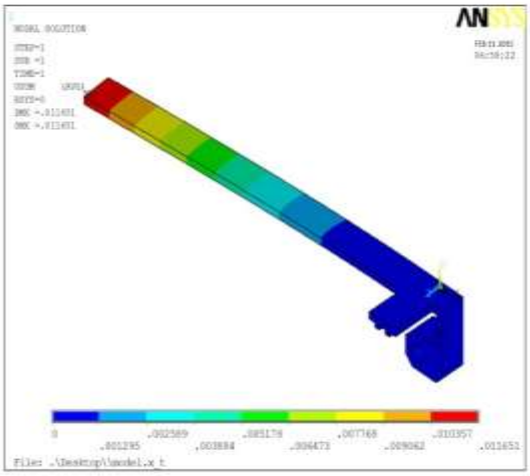


Figure 5.10 (Material 2)
Displacement of Mass 3 kg & Acceleration 10g

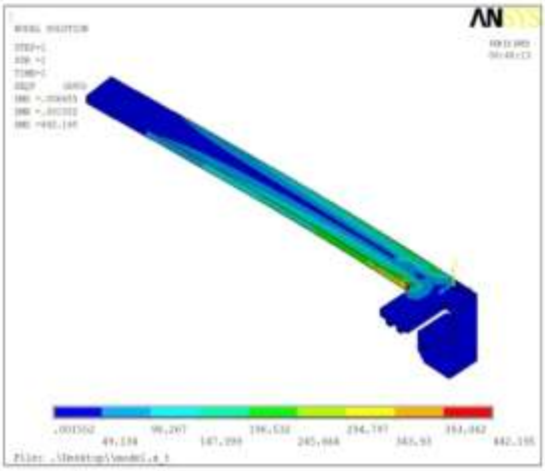
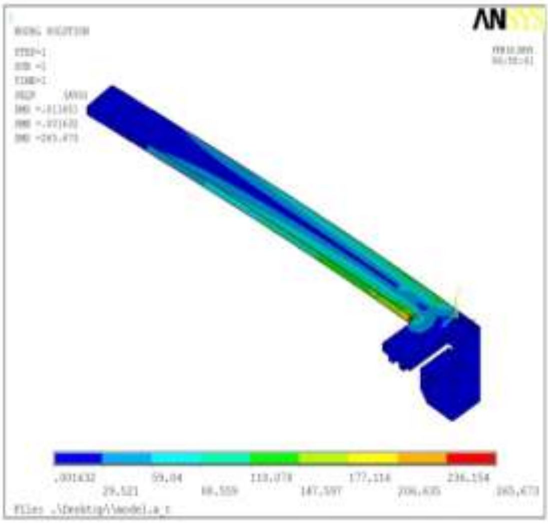


Figure 5.11 (Material 1)
Stress of Mass 3 kg & Acceleration 10g



Figures 5.12 (Material 2)
Stress of Mass 3 kg & Acceleration 10g

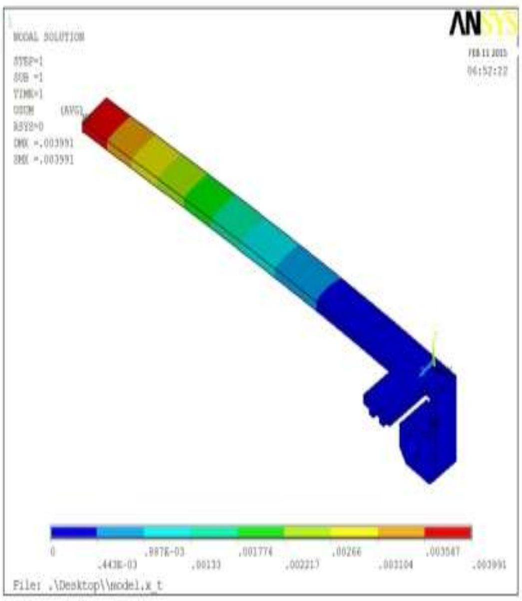


Figure 5.13 (Material 1)
Displacement of Mass 1.5 kg & Acceleration 10g

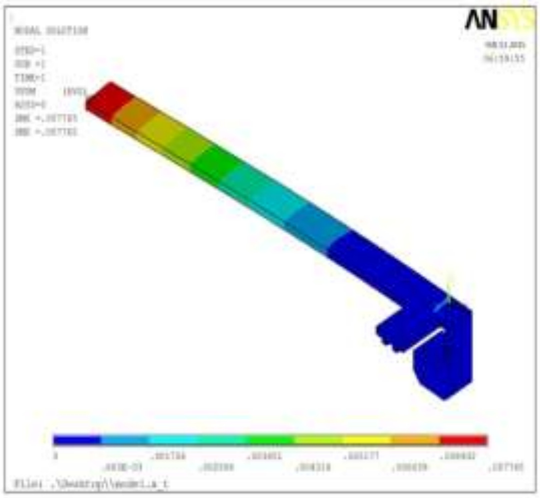


Figure 5.14 (Material 2)
Displacement of Mass 1.5 kg & Acceleration 10g

VI. DISCUSSION

The results in figures 5.5 to 5.16 shows that the stress and displacement decreases along with the addition of 10g acceleration and the mass of 6kg, 3kg, 1.5kg respectively. But the resulting stress produced by 6kg pick is high of material-1 when compared to material-2.It suggest that attain effective stress distribution, the mass of pick head should be reduced for material-1 as compared as material-2.Also the displacement of both materials under varying load condition within the safe limit which significantly reduce the machine vibration during acceleration and deceleration of transfer arm. Also the stress at far end portion considerably reduced which will not hazardous to the transfer arm.

In the same way the results in figures 5.17 to 5.28 shows that the stress distribution ,displacement decreases along with the addition of 5g acceleration and the mass of 6kg, 3kg, and 1.5kg respectively. It shows us that more stress distribution takes place in the same area, but which is less than the 10g acceleration. From the results we conclude that the stress distribution and displacement decreases 3 times lesser than 5g acceleration as compared with 10g acceleration. So the machine vibration also reduces 3 times less compared to 10g acceleration.

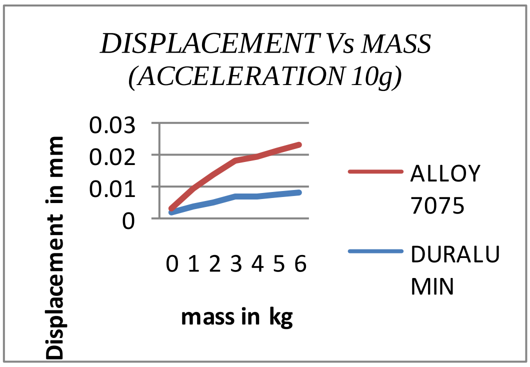


Figure 6.1
Displacement vs. Mass (Acceleration 10g)

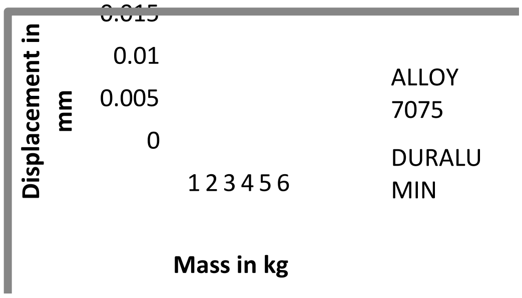


Figure 6.2
Displacement vs. Mass (Acceleration 5g)

Fig 5.29 and 5.30 shows the relationship between duralumin& alloy 7075 under 10g &5g acceleration.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 4, Issue 8 Aug. 2022, pp: 1603-1619 www.ijaem.net ISSN: 2395-5252

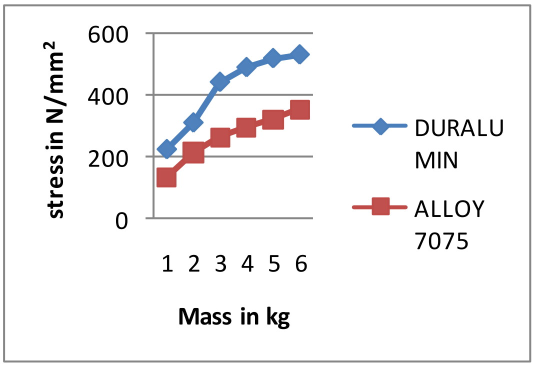


Figure 6.3
Stress vs. Mass (Acceleration 10g)

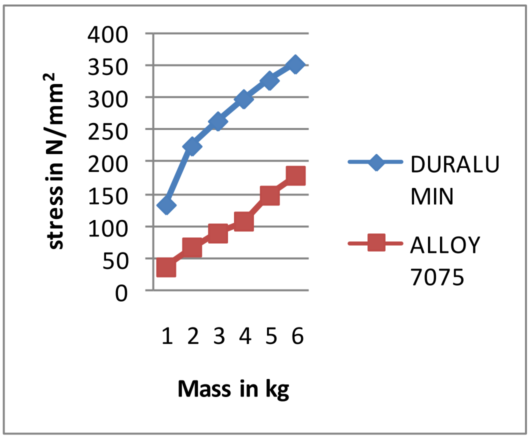


Figure 6.4
Stress vs. Mass (Acceleration 5g)

Fig 5.31 and 5.32 shows the relationship between duralumin& alloy 7075 under various mass of pick head. So the obtained result is, under 10g acceleration running condition 1.5kg pick head transfer arm is strong enough for duralumin. The Transfer Arm made by Alloy 7075 is till rigid enough, Even though when it is running at 10g acceleration with 6kg Pick Head that the settling time of it will increases due to extensive vibration.

COMPARISON OF RESULTS

Table 6.1 Comparison of Results

S.No	Mass kg	Acceleration	Material I Duralumin		Material II Alloy 7075	
			Displacement mm	Stress N/mm^2	Displacement mm	Stress N/mm^2
1	1.5	10 g	0.0039	264.73	0.0077	176.94
2	3		0.0066	442.19	0.0116	265.67
3	6		0.0079	530.93	0.0155	354.41
4	1.5	5 g	0.0027	175.92	0.0019	43.84
5	3		0.0039	264.72	0.0038	88.20
6	6		0.0053	353.46	0.0077	176.94

***Yield Stress for the Alloy 7075 = 215 N/mm^2**
***Yield Stress for the Duralumin = 165 N/mm^2**

VII. CONCLUSION

The advantages of FEM are displayed in dealing with complex structure stress analysis where normal mechanic methods falling short of their wishes. From the results, we conclude that the maximum stress distribution and displacement decreases along with the addition of 10g and 5g acceleration and the mass of 6kg, 3kg, and 1.5kg respectively. But the resulting stress produced by 6kg pick is high of duralumin as compared to alloy 7075.it suggest that attain effective stress distribution, the mass of pick head should be reduced for duralumin as compared as alloy 7075.also the displacement of both materials under varying load condition within the safe limit which significantly reduce the machine vibration during acceleration and deceleration of transfer arm. Also the stress at far end portion considerably reduced which will not hazardous to the transfer arm.

It conclude that even running at 10g acceleration with 6kg Pick Head, the Transfer Arm is till rigid enough though the settling time of Transfer Arm made up of alloy 7075 will hugely increase caused by extensive vibration.

REFERENCES

[1]. Analysis of Potential Failure in Pick & Place Transfer Module - Tiefu Shao , Term Project-Mechanical and Industrial Engineering

Department University of Massachusetts, Amherst May 15, 2002.
[2]. 100g-100N Finger Joint with Load-Sensitive Continuously Variable Transmission - Takeshi Takaki and Toru Omata, Proceedings of the 2006 IEEE International Conference on Robotics and Automation Orlando, Florida - May 2006.
[3]. Simulation of Multifinger Robotic Gripper for Dynamic Analysis of Dexterous Grasping - s. s. ohol, s. r. kajale, Proceedings of the World Congress on Engineering and Computer Science 2008, WCECS 2008, October 22 - 24, 2008, San Francisco, USA
[4]. Overloading Failures in Robot Manipulators- Damanbir Singh Virk, V. K. Banga - International Conference on Trends in Electrical, Electronics and Power Engineering (ICTEEP'2012) July 15-16, 2012 Singapore.
[5]. Analysis on the fundamental deformation effect of a robot soft finger and its contact width during power grasping - NatarajanElango& R. Marappan, International Journal of Advanced Manufactured Technology.
[6]. Aluminium Alloys – A Century of Age Hardening -I.J. Polmear, materials forum volume 28 - Published 2004,
[7]. J.M. Waller and W.A. Anderson, Aluminum Company of America, Unpublished work, (1963).
[8]. R.E. Sanders, Jr., S.F. Baumann, and H.C. Stumpf, Aluminum Alloys: Their Physical and MechanicalProperties, Vol III, Ed. E.A. Starke, Jr. and T.H. Sanders, Jr., (1986)
[9]. G-M Raynaud, B. Grange, and C. Sigli, Proceedings 3rd International Conference on Aluminum Alloys,(1992).
[10]. E. H. Dix, Jr., W. A. Anderson, and M. B. Shumaker, "Development of Wrought Aluminum-MagnesiumAlloys,", Technical Paper No. 14, Aluminum Company of America, (1958).
[11]. R. Dif, J-C Ehrstrom, and G-M Raynaud, Aluminum Alloys: Their Physical and Mechanical Properties,Vol III, Ed. T. Sato et. al., (1998),
[12]. Craig J.J., Introduction to Robotics: Mechanics and control, PearsonEducation Asia, 2004.
[13]. Deb S.R., Robotics Technology and Flexible Automation. Tata Mc-GrawHill, 2002.
[14]. Galicki M., 2000. Time-optimal controls of kinematicallyredundantmanipulators with geometric constraints. IEEE Transactions on Roboticsand Automation 16(1),
[15]. Gemeinder M. and Gerke M., 2003. GA-based path planning for mobile robot systems employing an active search algorithm. Elsevier AppliedSoft Computing 3(2),
[16]. Fu K.S., Gonzalez R.C., Lee C.S.G., Robotics: Control, Sensing, Visionand Intelligence. Mc-Graw Hill International Editions, 1987.
[17]. K, Lingaiah, Machine Design Data Handbook, McGraw- Hill Inc., New York, 1994
[18]. Robert O. Parmley, HVAC Design Data Sourcebook,
[19]. Ian R. Grosse, MIE/CEE Introduction to Finite Element Modeling, Analysis and Applications, University of Massachusetts, Amherst, 2002
[20]. ANSYS 10.0 Manual.