# A Study on Static, Dynamic and Vibration Analysis of Industrial Robot

***Rohit Kumar****
*Department of Mechanical Engineering, Rajshahi University of Engineering & Technology, Rajshahi, Bangladesh*
***Corresponding Author**
**Email Id: kumarrohit198900@gmail.com**

## ABSTRACT

*Purpose of the present work is to perform modeling and static, dynamic and vibration analysis of mechanical systems like Industrial Robots. A Mechanical System is a system of elements that interact on mechanical principles. The contemporary technical problems are lashed with high work demands such as high speeds of mechanisms, using lower density materials, high precision of work, etc. The main objective of this work is the dynamical analysis with taking into consideration the interaction between main motion and local vibrations during the model is simulated.*

*Taking the basics of Finite Element Analysis and FEM ; using the CATIA program gives the possibility to generate models; the HYPERMESH is for individual meshing of various components assembled in the robot; at last but not the least ANSYS is used for Static and Dynamic Analysis, on the whole the study is in a phased manner . By bringing in certain program working conditions, many types of numerical data that represented the examined system had to be determined at that time directly by user through a suitable interface system. As a future scope of the project on such a complex system of meshed elements, Vibration analysis using MATLAB might be possible in future by completely understanding the results of the above static and dynamic analysis in finite number of cases of operation of the robot. The present work deals with the analysis of the robot in two different cases of operation.*

***Keywords**: Static Analysis, Dynamic Analysis, Hypermesh, Ansys, FEm, Catia.*

## INTRODUCTION

### Finite Element Analysis

FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. FEA is a group of numerical methods for approximating the solution of governing equations of any continuous system. It is used in new product design, and existing product refinement. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition [1].

The need for numerical methods arises from the fact that for most practical engineering problems analytical solutions do not exist. While the governing equations and boundary conditions can usually be written for these problems, difficulties introduced by either irregular geometry or other discontinuities render the problems intractable analytically. To obtain a solution, the engineer must make simplifying assumptions, reducing the problem to one that can be solved, or a numerical procedure must be used. In an analytic solution, the unknown quantity is given by a mathematical function valid at an infinite number of locations in the region under study, while numerical methods provide approximate values of the unknown quantity only at discrete points in the region. In the finite element method, the region of interest is divided up into numerous connected sub-regions or elements within which approximate functions (usually polynomials) are used to represent the unknown quantity [2].

There are generally two types of analysis that are used in industry: 2-D modelling and 3-D modelling. While 2-D modelling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modelling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modelling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture [3].

**Example of Problems that can be treated by Finite Elements:**

1) Structural analysis
2) Heat Transfer
3) Fluid Flow
4) Mass Transport
5) Electromagnetic Potential
6) Acoustics
7) Bioengineering

**How FEA works**

FEA uses a complex system of points called node which make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress.

Points of interest may consist of: fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many elements [4].

A wide range of objective functions (variables within the system) are available for minimization or maximization:

1) Mass, volume, temperature
2) Strain energy, stress strain
3) Force, displacement, velocity, acceleration
4) Synthetic (User defined)

There are multiple loading conditions which may be applied to a system:

1) Point, pressure, thermal, gravity, and centrifugal static loads
2) Thermal loads from solution of heat transfer analysis

3) Enforced displacements
4) Heat flux and convection
5) Point, pressure and gravity dynamic loads

Each FEA program may come with an element library, or one is constructed over time. Some sample elements are:
1) Rod elements
2) Beam elements
3) Plate/Shell/Composite elements
4) Shear panel
5) Solid elements
6) Spring elements
7) Mass elements
8) Rigid elements
9) Viscous damping elements

Many FEA programs also are equipped with the capability to use multiple materials within the structure such as:
1) Isotropic, identical throughout
2) Orthotropic, identical at 90 degrees
3) General anisotropic, different throughout

## TYPES OF ENGINEERING ANALYSIS

**Structural** analysis consists of linear and non-linear models. Linear models use simple parameters and assume that the material is not plastically deformed. Non-linear models consist of stressing the material past its elastic capabilities. The stresses in the material then vary with the amount of deformation as in Figure 4.

**Vibration** analysis is used to test a material against random vibrations, shock, and impact. Each of these incidences may act on the natural vibration frequency of the material which, in turn, may cause resonance and subsequent failure.

**Fatigue** analysis helps designers to predict the life of a material or structure by showing the effects of cyclic loading on the specimen. Such analysis can show the areas where crack propagation is most likely to occur. Failure due to fatigue may also show the damage tolerance of the material (Figure 5).

**Heat Transfer** analysis models the conductivity or thermal fluid dynamics of the material or structure. This may consist of a steady-state or transient transfer. Steady-state transfer refers to constant thermal properties in the material that yield linear heat diffusion [5].

## PROCEDURE OF FINITE ELEMENT ANALYSIS

1) Computer modelling or mesh generation
2) Definition of properties
3) Assembly of elements
4) Specification of boundary conditions and definition of loads.
5) Solution using the required solver and display results summary and graphs etc.

## TYPICAL STEPS INVOLVED

1) Divide or discretize the structure or continuum into finite elements. This is typically done using mesh generation program called Pre-processor.
2) Formulate the properties of each element. For example nodal loads that are associated with all elements, Deformation states that are allowed.
3) Assemble the elements to obtain the FEA model.
4) Specify the load, boundary conditions, force and known temperatures.
5) Solve simultaneous linear algebraic equations to obtain the solutions.

## RESULTS OF FINITE ELEMENT ANALYSIS

FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested [6].

### Project Phases

1) Modeling in CATIA
2) Meshing in HYPERMESH
3) Static Analysis in ANSYS
4) Dynamic Analysis in ANSYS
5) Vibration Analysis in MATLAB

### CATIA

CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Systemes and marketed worldwide by IBM. Written in the C++ programming language, CATIA is the cornerstone of the Dassault Systemes product lifecycle management software suite.

### HYPERMESH

Altair HyperMesh is a high-performance finite element pre- and post-processor for major finite element solvers, allowing engineers to analyze design conditions in a highly interactive and visual environment. HyperMesh's user-interface is easy to learn and supports the direct use of CAD geometry and existing finite element models, providing robust interoperability and efficiency.

Advanced automation tools within HyperMesh allow users to optimize meshes from a set of quality criteria, change existing meshes through morphing and generate mid-surfaces from models of varying thickness.

### ANSYS

ANSYS Mechanical software offers a comprehensive product solution for structural linear or nonlinear and dynamics analysis. The product provides a complete set of elements behavior, material models and equation solvers for a wide range of engineering problems.

In addition, ANSYS Mechanical offers thermal analysis and coupled-physics capabilities involving acoustic, piezoelectric, thermal–structural and thermoelectric analysis.

**MATLAB**

MATLAB is a high-level language and interactive environment that enables you to perform computationally intensive tasks faster than with traditional programming languages such as C, C++, and Fortran [6].

## SPECIFICATIONS

**ABB IRB 4400/60 Robot [7]**

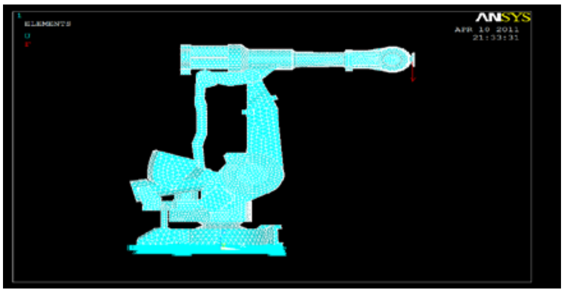
**Features**

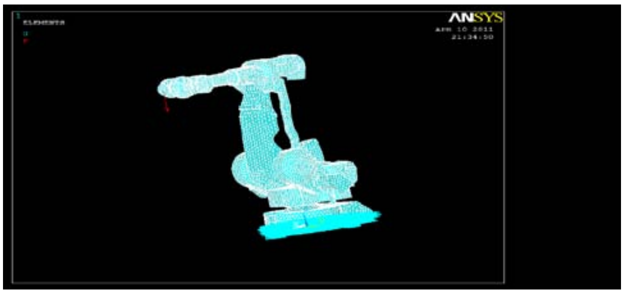
1) Axes: 6
2) Payload: 60 kg
3) H-Reach: 1960 mm
4) Repeatability: ±.19 mm
5) Robot Mass: 900 kg
6) Mounting: floor

**ANSYS**

**Static Analysis**

Statics is the branch of mechanics concerned with the analysis of loads (force, torque/moment) on physical systems in static equilibrium, that is, in a state where the relative positions of subsystems do not vary over time, or where components and structures are at a constant velocity. When in static equilibrium, the system is either at rest, or its center of mass moves at constant velocity. Static Analysis is performed on the meshed model .When the robot is holding a mass of 60kg is simulated by applying force equivalent to 60kgf (i.e 588.6 N) [8-10].




## RESULTS

The maximum deformation is 0.06 mm in negative z-axis which tells us that it is well beyond failure limit.The models above show the deformation amplified 1400 times to give a proper visualization to deformation.

**Dynamic Analysis**

Dynamics includes the study of the effect of torques on motion. In the field of physics, the study of the causes of motion and changes in motion is dynamics**.** In this case we simulate that the robot is swinging with a mass of 60kg and to simulate the movement a force is applied in the direction of movement and the force applied here is 100N.The main constraint is that the force is applied for 1 second, starting from 0.001s with 100N force (Ramped

loading) till 0.7s loading (stepped loading) and then to 1s with no force at the end to simulate the end of motion. The sub steps of 1, 10, 10 are taken for load steps of time ranges of 0.001s, 0.7s and 1s. An analysis is performed on 74987 elements of the meshed robot.
Total number of elements: 74987 elements
Minimum Element Size : 10

## FUTURE SCOPE AND CONCLUSION

The next part is the Vibration analysis which comes under future scope of the analysis as all working positions of the robot are to be taken into consideration for generating equations required for calculation the residual and local vibrations. The reason for not carrying out Vibration analysis is that, the model we meshed is highly complex and has more than 70000 meshed elements; so generation of equations and solving for that many elements is similar to an industrial CAD/CAE project. Basing on the size of the .lst files generated in ANSYS analysis mounting up to 5GB of DATA, the dynamic analysis itself is complex and time consuming. Proper analysis of static and dynamic loading is to be done for better understanding of vibration characteristics of that robot. Hence we leave to the coming batches of Mechanical Engineering, the great working of Vibration analysis and hoping that they may succeed with better futuristic Workstations.

## REFERENCES

1) Physical and geometrical data acquiring system for vibration analysis software by J.´ Swider, P. Michalski, G. Wszołek (Journal of Materials Processing Technology 164–165 (2005) 1444–1451)
2) Vibration analysis of the excavator model in GRAFSIM program on the basis of a block diagram method by G. Wszołek (Journal of Materials Processing Technology 157–158 (2004) 268–273)
3) Analysis of complex mechanical systems based on the block diagrams and the matrix hybrid graphs method by G. Wszołek (Journal of Materials Processing Technology 164–165 (2005) 1466–1471)
4) Modeling of mechanical systems vibrations by utilization of GRAFSIM software by G. Wszołek (Journal of Materials Processing Technology 164–165 (2005) 1466–1471)
5) Vibration analysis software based on a matrix hybrid graph transformation into a structure of a block diagram method by J.´Swider, G. Wszołek (Journal of Materials Processing Technology 157–158 (2004) 256–261)
6) Vibration analysis of mechanical systems with utilization of GRAFSIM and CATGEN software by J. Świder , G. Wszołek ,*, K. Foit , P. Michalski , S. Jendrysik (Journal of Achievements in Materials and Manufacturing Engineering VOLUME 23 ISSUES 1 July 2007)
7) Analysis and modeling of rotational systems with the Modyfit application by S. Żółkiewski (Journal of Achievements in Materials and Manufacturing Engineering VOLUME 30 ISSUES 1 September 2008).
8) Graphs Application in Computer Analysis of Mechanical Systems. Monograph by J. Świder , G. Wszołek (Silesian University Publishing Company, Gliwice 2002, (in Polish)
9) Dynamic analysis of the mechanical systems vibrating transversally in transportation by A. Buchacz, S. Żółkiewski (Journal of Achievements in Materials and Manufacturing Engineering 20 (2007) 331-334).
10) Mechanical Vibrations. Introduction by C. Cempel (Poznan University Publishing Company, Poznan,1984 )