

# Design and Analysis of Drill Jig at Variable Materials

**K.Rama Subba Reddy<sup>1</sup>,**

P.G. Scholar, Department of mechanical Engg, AVR & SVR College of Engg & Tech, Nandyal, Kurnool, A. P, India

**S.Ramesh Kumar Babu<sup>2</sup>**

Assistant Professor, Department of mechanical Engg, AVR & SVR College of Engg & Tech, Nandyal, Kurnool, A. P,  
India

**A.V. Hari Babu<sup>3</sup>**

Associate Professor & HOD, Department of mechanical Engg, AVR & SVR College of Engg & Tech, Nandyal,  
Kurnool, A. P, India

**Abstract** - This project chiefly treated modeling of drill jig and also the analysis over that. The modeling of the element was done by the advanced modeling software. The analysis was done by using one amongst the foremost necessary numerical strategies is FEA and also the software used is Ansys. The modeling of the individual components and also the assembly of the individual components was module by bottom up methodology. In the analysis, the behaviors of the drill jig are shown in shapes by modal analysis. The static structural analysis was done at totally different load conditions for various materials and results are tabulated were plot. From that the most effective material are chosen.

**Key words:** Ansys, Behavior, Drill jig static structural analysis, effective and model analysis

## 1. INTRODUCTION

Tool design cares with design and development of machines and special tooling, strategies and techniques needed by today's high speed mass manufacture at high potency and productivity [2]. Its main objective is to provide element at a competitive value, maintaining quality and accrued production. Design of tools to be proof with no chance of wrong operation, quality consciousness in any respect level. The utilization of jigs and fixtures is a vital facet of workshop engineering for the assembly of articles in giant quantities with a high degree of accuracy and changeability at a competitive value. . The first objective of the utilization of jigs and fixtures is to facilitate the holding and supporting of the element by using fixtures, position it properly and guide the cutters in order that each element are uniform.

## 1.1 PRINCIPLES OF JIGS AND FIXTURES:

### Locating Point:

The most necessary demand in jigs or fixture design square measure sensible facilities to be provided for the work, the article to be machined could simply be inserted and quickly taken out from the jig in order that no time is wasted in inserting the work piece within the position to perform operations. The position of labor piece ought to be correct with relevancy tool guiding within the jig or setting components within the fixture [1].

### Clamping Device:

It ought to be as straightforward as potential while not sacrificing effectiveness. The strength of the clamp ought to be such not solely to carry the work piece firmly in situ however additionally to require the strain of the cutting tools while not springing or giving once planning the jigs and fixtures. The clamps ought to be convenient, quickly operated and supply less fatigue to the operator.

### Material used for Jigs and Fixtures:

Jigs and fixtures are product of hardened material to avoid frequent injury and to resist wear. the fabric used for jigs and fixtures are soft-cast steel, cast iron, die steel, steel, high speed steel, nickel-chrome steel, bronze, plastic material etc.

## 1.2 FACTORS TO BE THOUGHT-ABOUT FOR DESIGN OF JIGS FIXTURES

- Variety of part or work-piece.
- Capability of the machine & Production needs.
- Location.

- Loading and un-loading arrangements.
- Clamping arrangements.
- Clearance between jig and part.
- Ejectors & Base or body construction.
- Tool guiding and cutter setting.
- Rigidity and vibration, Safety & Cost.
- Material list or bill of fabric.

### 1.3 LOCATING PRINCIPLES

To ensure precision in any machining operations the work-piece should be properly positioned with relevance the cutter [3]. The locating principle refers to the dimensional and point relationship between the work-piece and cutter.

### 1.4. CLAMPING DEVICES

Once the work-piece is found it's necessary to check it against the locating surfaces and hold it there against the force acting upon it.

Various clamping devices area unit as follows Screw clamp, C – clamp, Strap clamp, Swing clamp, Pivoted clamp, Hinged clamp, Wedge clamp, Latch clamp, Toggle clamp, Cam action clamp & Equalizer clamp

### 1.5. TYPES OF JIGS

- Plate jig. & Angle plate jig.
- Latch jig & Gas jig.
- Channel jig.
- Box jig.
- Post jig.
- Table jig or flip over jig.
- Angular post jig & Pot jig.

### CHANNEL JIG:

Plate jig in the main consists of one bushing plate with a provision for locating and clamping of work. The work profile is found by six location pins and clamped by 2 knurled screws against the situation pins. Sometimes, it's economical to form the jigs channel formed so as to change location and clamping.

### 1.6. ADVANTAGES OF JIGS

- To cut back the price of production.
- To assure high accuracy of components.
- To extend production.
- To avoid wasting labor value.
- To worth the standard management expense.

Because the world's one in every of the provider of software package, specifically meant to support a completely Integrated development method. production Technology Corporation (PTC) is recognized as a strategic partner which may facilitate a manufacturer to the flip a method into competitive advance, bigger market share and better profits and industrial and mechanical design to purposeful simulation producing and knowledge management.

"Feature based" implies that we tend to produce elements and assemblies by process feature like extrusion, sweeps, cuts, holes, spherical so on rather than specifying low level pure mathematics like lines, areas and circles.

"Parametric" implies that the physical form of the half as assembly is driven by the worth assigned to the attributes of its options. We tend to might outline or modify a feature dimension or alternative attributes at any time. Any changes can mechanically propagate through the model.

"Solid modeling" implies that the pc model we tend to produce is ready to contain all the data that a true solid object would have. it's volumes and thus, if you offer a price for the density of the fabric it's mass and inertia

### Components create by using pro e software Part module:

This starts by adding a group of default data point planes and making the primary solid and surface options, then still increase deliver the goods the designed form by making varied construction options and protrusions, slots and cuts, holes, shafts, ribs, chamfer, etc.

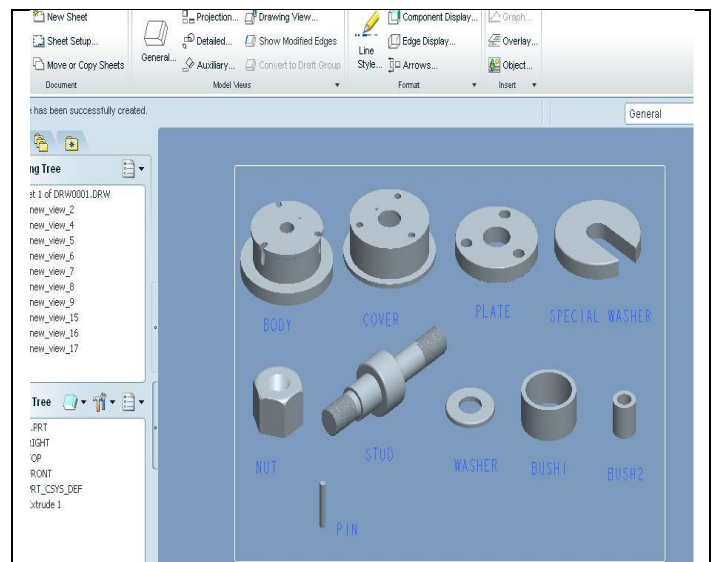


Fig 1: Part Module

## 2. INTRODUCTION TO PRO-E

### Assembly Module:

Assembly shows all the elements each commonplaces and non standard. a typical half is one, that we tend to merely purchase from manufacture and use because it is. A non-standard or custom half is one we tend to should design for the present project.

During this assembly to insert the elements one by one, we tend to use the desired constraints they are:

Default for the primary element and for the remaining all elements insert, mate and align constraints square measure used and created the element absolutely constraint.

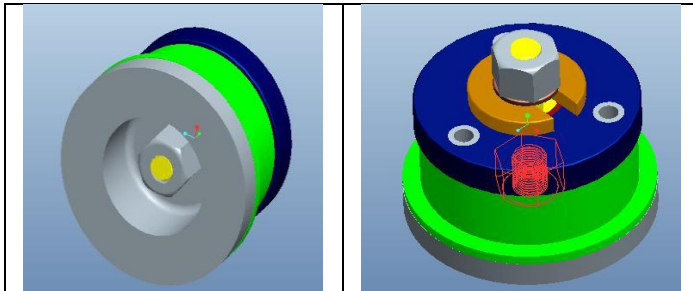


Fig.2. Final Assembly of Drill Jig

## 3. INTRODUCTION TO FINITE ANALYSIS METHOD

The finite part method represents associate degree extension of the matrix strategies for the analysis of framed structures to the analysis of the time structures. The fundamental philosophy of the strategy is to interchange the structure of the time having an infinite or infinite range of unknowns at bound chosen separate points. The strategy is extraordinarily powerful because it helps to accurately analyze structures with advanced geometrical properties and loading conditions. within the infinite technique, a structure or time is discretized and perfect by employing a mathematical model that is an assembly of subdivisions or separate components, referred to as finite part, are assumed to be interconnected solely at the joints referred to as nodes.

- Equations of equilibrium
- The compatibility of displacements
- The material organic relationship

The fundamental plan within the finite part technique is to search out resolution of an advanced drawback by exchange it by less complicated one. Since an easier one find the answer replaced the particular drawback, we are going to be able to realize solely Associate in nursing approximate resolution instead of

actual resolution. In finite part technique, it'll usually be doable to boost or refine the approximate resolution by causing a lot of machine effort.

### 3.1. ENGINEERING APPLICATIONS OF FEM:

The final nature of its theory makes it applicable to a large form of boundary price issues in engineering. A boundary price drawback is one amongst that an answer is sought-after in domain (region) of a body subject to the satisfaction of the prescribed boundary (edge) condition of the variable of their derivatives. Largely all engineering issues that are illustrated within the table four.1 of the finite part technique comes underneath 3 classes of boundary issues, namely

1. Equilibrium or steady state or time freelance drawback,
2. Eagan price drawback,
3. Transient or propagation programs.

### 3.2. GENERAL DESCRIPTION OF FEM

The stepwise procedure for static drawback will be declared as follows:

Step 1: Discretization of time

Step 2: choice of correct Interpolation model

Step 3: by-product of part stiffness matrices (Characteristic matrices) and cargo vectors

Step 4: assemblage of part equation to get the general equilibrium equations

$$[K] = \phi [P]$$

Wherever  $[K]$  is termed assemblage stiffness matrix,

$\phi$  is termed nodal displacement vector and

$P$  is termed nodal vector for the entire structure.

Step 5: resolution of systems equations to search out nodal values of the displacement (field variable):

### 3.3. BENEFITS OF FEM:

- Its ability to use varied size and form and to modal a structure of absolute pure mathematics.
- Its ability to accommodate absolute boundary conditions, loading, together with thermal loading.
- Its ability to modal composite structures involving completely different structural elements appreciate stiffening member on a shell and combination of plates, bars and solids, etc.,
- The finite component structure closely resembles the particular structure rather than being quite completely different obstruction that's arduous to check.
- The FEM is tested with success in representing varied forms of sophisticated material properties and material behavior (nonlinear, aeolotropic, time dependent or fixed support dependent material behavior).

- It pronto accounts for non-homogeneity of the fabric by distribution completely different properties to different components

### 3.4. FINITE COMPONENT ANALYSIS:

Finite component analysis was initial developed for the employment of region and nuclear industries wherever the protection of structure is important. Nowadays growth within the usage of methodology is directly thanks to the speedy advances in engineering. As a result business finite component packages exist that square measure capable of resolution the foremost subtle issues, not simply in structural analysis, except for big selection of phenomena appreciate steady state and dynamic fixed support distributions, fluid flow and producing processes appreciate injection molding and metal forming.

Finite component analysis is employed in new product design, and existing product refinement. Modifying AN existing product or structure is used to qualify the merchandise or structure for a brand new service condition. Just in case of structural failure, Finite component Analysis is also wont to facilitate in determinant the look modifications to satisfy the new conditions.

#### Types of Analysis:

There square measure differing kinds of study that square measure employed in industry: Structural, Modal, Harmonic, Transient and Spectrum.

Structural Analysis consists of linear and non-linear models. Linear models square measure easy parameters and assume that material isn't plastically malformed. Non-linear models comprise stressing the fabric past its elastic capabilities. The stresses within the material then vary with the quantity of deformation.

Vibration analysis is employed to check the fabric against random vibrations, shock and impact. Every of those incidents might act on the natural vibration frequency of the fabric that successively, might cause resonance and succeeding failure. Thus analysis is finished on the fabric to predict the lifetime of the fabric.

Heat-Transfer Analysis models the thermal physical phenomenon or thermal fluid dynamics of the fabric or structure. This could comprise a gentle state or transient transfer. Steady-state transfer refers to constant thermo properties in material that yield linear heat diffusion.

## 4. INTRODUCTION TO ANSYS SOFTWARE

The ANSYS program is self-contained general purpose finite component program developed and maintained by Gloria May Josephine Svensson Analysis Systems Iraqi National Congress. The program contains several routines, all reticulated and every one for main purpose of achieving a solution to an engineering drawback by Finite component methodology.

ANSYS provides an entire resolution to design issues. It consists of powerful design capabilities like full constant quantity solid modeling, design optimization and automotive vehicle meshing, which provides engineers full management over their analysis.

The following square measure the special options of ANSYS software:

- It includes linear components.
- Heat flow analysis, fluid flow and component flow analysis is done. Graphic package and intensive preprocessing and post process.

The following shows the transient description of steps followed in every phase:

### 4.1. PRE-PROCESSOR

The pre-processor stage in ANSYS package involves the following:

1. Specify the title that is the name of the matter.
2. Set the kind of the analysis to be used, i.e., structural, thermal, fluid, or electro-magnetic, etc.,
3. Define the part sort, this might be 1D, 2D or 3D, and specify the analysis sort being meted out.
4. Apply mesh – Mesh generation is the method of dividing the analysis time in to variety of separate elements or finite parts.
5. Assign the properties – Material properties (Young's Modulus, Poisson's magnitude relation, density, and if applicable constant of growth, friction, thermal physical phenomenon, damping result, heat, etc.,) have to be compelled to be outlined.

### 4.2: RESOLUTION

Apply the masses. Some variety of loads is really applied to the analysis model. The loading could also be within the kind of some extent load, pressure or a displacement during a stress analysis, a fixed supporter total deformation during a thermal analysis and a fluid pressure or speed during a fluid analysis. FE convergent thinker may be logically divided in to 3 main elements, the pre-solver, the mathematical-engine and therefore the post-solver.



## MESHING

### Manual Meshing:

In manual meshing the weather square measure smaller at joint. This is often called mesh refinement, and it allows the strain to be captured at the geometric separation. Manual meshing is long and tedious method for models with any degree of geometric complication, however with great tool rising in pre-processes; the task is changing into easier.

### Meshing controls:

Good component filler (Smart sizing) may be a meshing feature that makes initial part sizes free meshing operations. Good filler provides the mesher a more robust probability of making fairly formed parts throughout automatic mesh generation.

### 4.3: POST-PROCESSOR

During this module, the results of the analysis square measure browse and understood. All post-processor embody the calculation of stress and strain altogether of the X, Y, or Z directions, or so within the direction at an angle to the coordinate axes. The principle stress and strain may be premeditated.

### Overview of steps in a static analysis:

The process for static analysis consists of 3 main steps:

**1. Build the Model:** - to create the model, specify the task name, analysis title then outline the component varieties, component real constants, material properties, and also the model pure mathematics. The structural parts are often linear or nonlinear. Material properties are often linear or nonlinear, identical or writing, and constant or temperature-dependent. The Young's modules ought to even be outlined.

**2. Apply the loads get the solution:** - during this step, outline the analysis kind and choices, apply loads, specify load step choices, and start the finite component resolution. the loads that may be applied square measure : Displacements- degree of freedom, constraints typically given at modal boundaries to outline rigid support points.

- Forces- targeted loads typically given on the model exterior, moments.
- Pressures- surface loads typically applied on the model exterior, temperatures.
- Fluencies-Applied to stuffy the consequences of swelling or creep.

**3. Review the results:** - Results from a static analysis include the nodal displacements, nodal and component

stresses, nodal and component strains, component forces, nodal reaction forces etc.

## 5. ANALYSIS PROCEDURE

### 5.1 MODAL ANALYSIS:

Use import tool to import the geometry with STEP or IGES file format. After importing the geometry file we do the required analysis. Now on our model analysis was done. From the tool box select modal analysis tab and drop on gui. Now link the imported geometry to the modal analysis tab and double click on model to open analysis window as shown below in the **project schematic window**.

After importing the model into project schematic window drag and drop the MODEL tab on to the screen from the toolbox window and link the geometry to geometry both will be linked together. Double click on the model it opens the mechanical window with object.

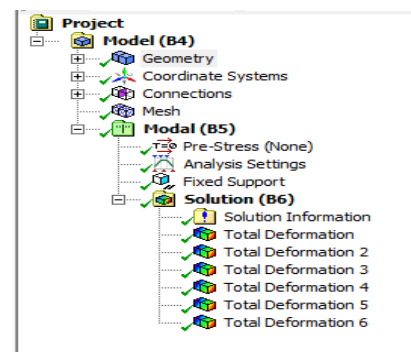
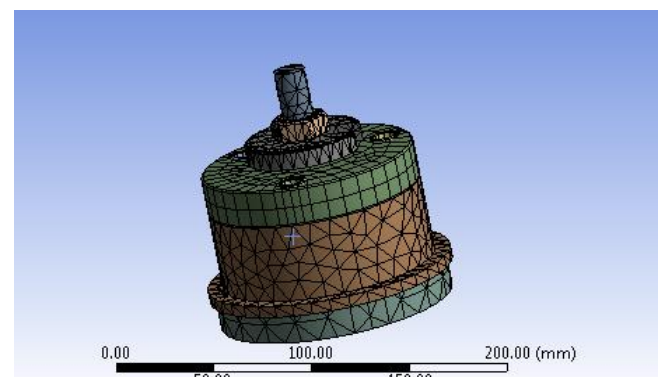


Fig. 3. Model with Analysis tools

Mesh: To generate the meshing, there are two methods one is automatic mesh generation and the other is with required size meshing. In this we used auto meshing with medium meshing.



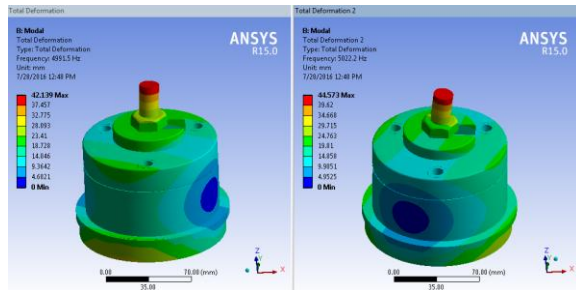
**Fig.4.** Meshed Geometry

**Assigning of material:**

From the outline tab – select the geometry – Part – from the bottom detailed window- material –Assignment – Select required material. Here we are considering the four materials Right click on Analysis settings – Insert – Fixed support–and select as shown below

**Solution:**

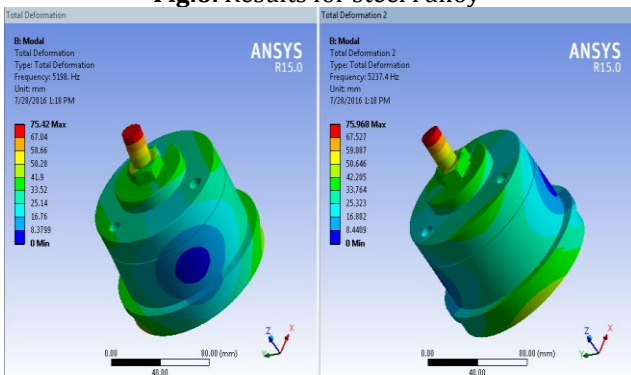
Right Click on solution – Insert total deformation support- Right Click on solution – Solve. The results window looks as below.



**Fig.5.** For material structural steel alloy

Results	Minim...	Maximum	Units	Reported Frequency (Hz)
Total Deformation	0.	42.139	mm	4991.5
Total Deformation 2	0.	44.573	mm	5022.2
Total Deformation 3	0.	24.823	mm	6149.5
Total Deformation 4	0.	74.687	mm	6464.9
Total Deformation 5	0.	74.697	mm	6467.5
Total Deformation 6	0.	93.707	mm	7534.7

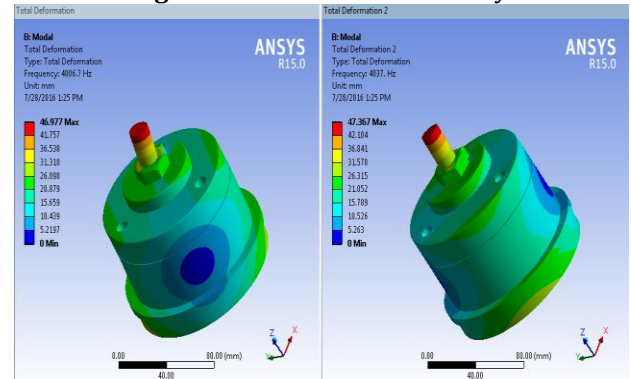
**Fig.6.** Results for steel l alloy



**Fig.7.** For material aluminum alloy

Results	Minim...	Maximum	Units	Reported Frequency (Hz)
Total Deformation	0.	75.42	mm	5198.
Total Deformation 2	0.	75.968	mm	5237.4
Total Deformation 3	0.	39.505	mm	6351.2
Total Deformation 4	0.	184.91	mm	8231.
Total Deformation 5	0.	135.57	mm	8315.4
Total Deformation 6	0.	75.295	mm	9047.9

**Fig.8.** Results for aluminum alloy



**Fig.9.** For material cast iron

Results	Minim...	Maximum	Units	Reported Frequency (Hz)
Total Deformation	0.	46.977	mm	4006.7
Total Deformation 2	0.	47.367	mm	4037.
Total Deformation 3	0.	24.495	mm	4988.1
Total Deformation 4	0.	123.85	mm	6376.3
Total Deformation 5	0.	89.845	mm	6455.
Total Deformation 6	0.	46.015	mm	7017.6

**Fig.10.** Results for cast iron

**5.2. STATIC STRUCTURAL ANALYSIS:**

After importing the model into project schematic window drag and drop the static structural tab on to the screen from the toolbox window and link the geometry to geometry both will be linked together. Double click on the model it opens the mechanical window with object.

Mesh: To generate the meshing, there are two methods one is automatic mesh generation and the other is with required size meshing. In this we used auto meshing with medium meshing.

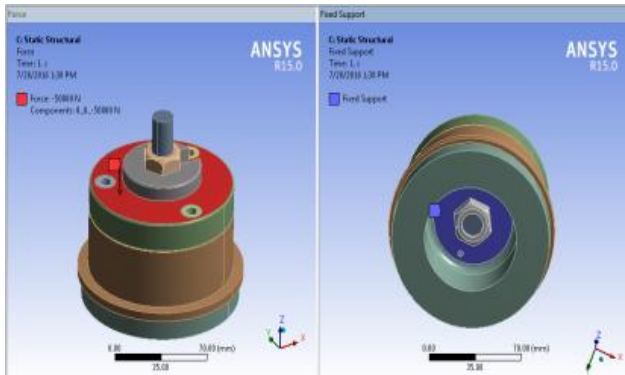


Fig.11. Defining the fixed point

**5.3 IN STATIC STRUCTURAL:**

Set the initial fixed support

Right click on Analysis settings – Insert – Fixed support–50 kN force or pressure and select as shown below

**Solution:**

Right Click on solution – Insert Fixed support–Right Click on solution – Solve. The results window looks as below.

**Project schematic**

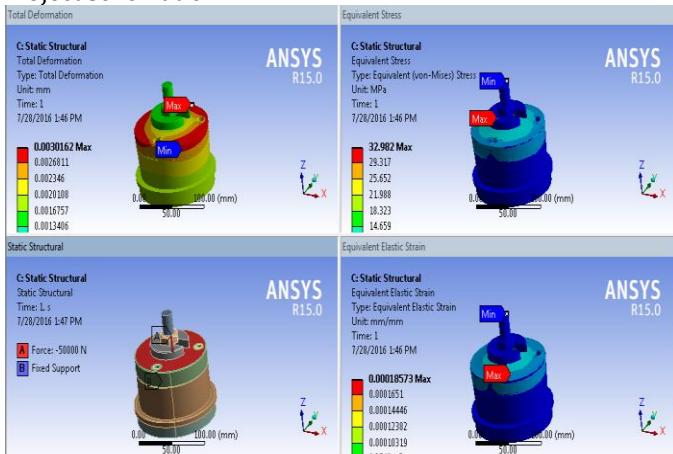


Fig.12. For structural steel

Results	Minimum	Maximum	Units	Time (s)
Total Deformation	0.	3.0162e-003	mm	1.
Equivalent Stress	2.188e-004	32.982	MPa	1.
Equivalent Elastic Strain	1.3027e-009	1.8573e-004	mm/...	1.

Fig.13. Results for structural steel



Fig.14. For aluminum alloy

Results	Minimum	Maximum	Units	Time (s)
Total Deformation	0.	8.5231e-003	mm	1.
Equivalent Stress	2.1905e-004	31.48	MPa	1.
Equivalent Elastic Strain	3.704e-009	5.0327e-004	mm/...	1.

Fig.15. Results for aluminum alloy



**Fig.16.** For cast iron

Results	Minimum	Maximum	Units	Time (s)
Total Deformation	0.	5.4677e-003	mm	1.
Equivalent Stress	2.2239e-004	33.94	MPa	1.
Equivalent Elastic Strain	2.3889e-009	3.4562e-004	mm/...	1.

**Fig.17.** Results for cast iron

## REFERENCES

- [1]. P H Joshi, "Jigs and fixtures" Tata McGraw-Hill Education, 2010
- [2]. Edward Hoffman "Technology & Engineering" Cengage Learning, 21-Aug-2003
- [3]. Cyril Donaldson, V. C. Goold Machine-tools -Tata McGraw-Hill Education, 1976.

## RESULTS:

**Table 1:** Modal Analysis

S.No	Material	Total Deformation (mm)					
		1	2	3	4	5	6
1	Steel	42.14	44.57	24.82	74.69	74.70	93.71
2	Al Alloy	75.42	75.96	39.50	184.91	135.57	75.30
3	Cast Iron	46.97	47.36	24.49	123.85	89.84	46.01

**Table 2:** Static Structural Analysis: F= 50 KN

S.No	Material	Total Deformation x 10 <sup>-3</sup> mm	Equivalent Stress M.Pa	Equivalent Strain x 10 <sup>-4</sup>
1	Steel	3.016	32.98	1.857
2	Al Alloy	8.5231	31.48	5.032
3	Cast Iron	5.4677	33.94	3.456

## CONCLUSION

Modeling and analysis was completed by using advanced modeling software system Catia and Ansys respectively. Static structural and MODAL analysis was done on DRILL JIG for 3 completely different materials and results are achieved. in line with the results stress is nearly shut for 3 materials however considering deformation is a smaller amount for steel compared with grey forged iron and aluminum alloy. Thus the steel is best material for the producing.