# Explicit Dynamic Analysis of Tensional & Torsional Propagations on Composite Material with Dog Bone Shaped Testing Specimen

Yedla Gopala Rao[1] | M R S Nataraj[2] | P Srinivas[3]

[1]Department of Mechanical Engineering, Srinivasa Institute of Engineering and Technology, Cheyyeru, Amalapuram, India.
[2]Assistant Professor, Department of Mechanical Engineering, Srinivasa Institute of Engineering and Technology, Cheyyeru, Amalapuram, India.
[3]Associate Professor, Department of Mechanical Engineering, Srinivasa Institute of Engineering and Technology, Cheyyeru, Amalapuram, India.

## ABSTRACT

*In general the composite are made by percentage of mixed or arrangement or concentration of material properties to increase the strength and advantages of the materials. Toeliminate the holding and gluing problems making the direct tensile strength test hard and torsional momentum loads to be applied, a new method of testing specimens prepared using 3d designing CATIA software to make the dog bone shape is assessed whether it could be applied to determine accurate direct tensile strength values and torsional loads over al alloy materials. In the project, i here considered 3 conditions, High percentage of aluminum alloy and low percentage of the silicon, Low percentage of aluminum alloy and high percentage of the silicon and Balanced percentage of aluminum alloy and silicon A series of modeling and analysis was performed using finite element method (ansys workbench) to investigate the deformation and stress to change of effect by material properties. A proper mechanical material property of the specimens was suggested and ideal failure of the dog bone shaped specimens was determined according to the results obtained from this study.*

***KEYWORDS:*** *material, composite, aluminum, silicon, dog bone, tensile, torsional*

## I. COMPOSITE MATERIALS AND LAMINATES

Composites were first considered as basic materials somewhat more than 50 years prior. From that opportunity to now, they have gotten expanding consideration in all parts of material science, producing innovation, and hypothetical examination.

Composites may sound somewhat "geek" and new, however they're incredibly regular in our general surroundings. Bats for ball sports (tennis rackets, golf clubs, and hockey sticks) are frequently produced using them. Vehicles, planes, and pontoons have for quite some time been produced using composites, for example, fiberglass, since they're lighter than metals yet regularly similarly as solid. What's more, on the off

chance that you think composites sound super-present day, reconsider: solid, wood, and bone are on the whole composite materials. Overlays are composites in which layers of various materials are reinforced together with glue, to give included quality, strength, or some other advantage.

### 1.1 What Are Composites

A composite is made by joining at least two different materials so they improve each other however keep particular and separate characters in the last item. So a composite is definitely not a compound (where particles or atoms tie together synthetically to make something very extraordinary), a blend (where one material is mixed into another), or an answer (where something like salt breaks up in water and successfully vanishes). A composite is something like solid, where stones of different sizes are specked in the middle of concrete. Reinforced cement is additionally a composite produced using steel strengthening bars set inside wet solid—which makes it, in actuality, a composite of a composite. Fiberglass is a composite of minor glass shards stuck inside plastic. In concrete, reinforced cement, and fiberglass, the first fixings are still simple to spot in the last material. So in solid, you can plainly observe the stones in the concrete—they don't vanish or break down.

### 1.1.1 Why Do We Need Composites

In at any rate one significant way, a composite must be superior to anything the materials from which it's made—generally there's no good reason for it. Considering concrete once more, it's solid on the off chance that you use it in vertical shafts to take the heaviness of a structure or a structure pushing down—as it were, on the off chance that you use it squashed (in pressure). Yet, it's very powerless and will in general bow and afterward snap in the event that you use it on a level plane, where it's extended (in pressure). That is clearly going to be a noteworthy issue in a structure that has heaps of flat bars. An extraordinary arrangement is to pour wet cement around tight steel bars (called rebars) with the goal that it sets into a composite material called reinforced cement. The steel pulls on the solid and stops it snapping when it's in strain, while the solid shields the steel from rust and rot. What we end up with is a composite material that functions admirably in both pressure and pressure.

### 1.1.2. How Are Composites Made

Composites are commonly made of two fundamental materials (however there might be different added substances too): there's a "foundation" material called a grid (or lattice stage) and, to this, we include a changing material called the fortification (or strengthening stage). In spite of the fact that we will in general think about the support as being comprised of filaments (as in fiberglass), that is not generally the situation. In reinforced cement, the "strands" are huge scale, wound steel poles; in fiberglass, they're modest stubbles of glass. At times the support is made of granules, particulates, or bristles, however it can likewise be made of collapsed materials.

## II. UNIVERSAL TESTING MACHINE

A general testing machine (utm), otherwise called a widespread analyzer, materials testing machine or materials test outline, is utilized to test the rigidity and compressive quality of materials. A previous name for a tractable testing machine is a tensometer. The "all inclusive" some portion of the name mirrors that it can perform numerous standard pliable and pressure tests on materials, segments, and structures (at the end of the day, that it is adaptable).



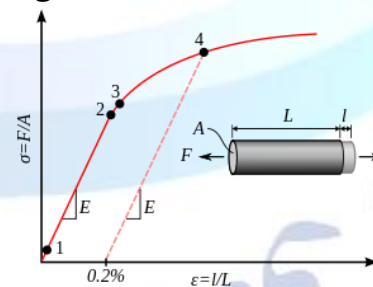**Figure 2.1 A Modern Universal Testing Machine**



**Figure 2.2 Stress Strain Graph**

Stress–strain curve showing typical yield behavior for nonferrous alloys.

1. True elastic limit
2. Proportionality limit
3. Elastic limit
4. Offset yield strength

The set-up and use are point by point in a test technique, frequently distributed by a measures association. This indicates the example arrangement, fixturing, measure length (the length

which is under examination or perception), investigation, and so forth.

The example is put in the machine between the holds and an extensometer whenever required can naturally record the adjustment in check length during the test. On the off chance that an extensometer isn't fitted, the machine itself can record the relocation between its cross heads on which the example is held. In any case, this technique not just records the adjustment long of the example yet additionally all other expanding/versatile parts of the testing machine and its drive frameworks including any slipping of the example in the holds.

When the machine is begun it starts to apply an expanding load on example. All through the tests the control framework and its related programming record the heap and expansion or pressure of the example.



**Figure 2.3 Dog Bone Sample**

Dog bone ductile test tests are fundamentally utilized in malleable tests. The example has a shoulder at each end and a check area in the middle. The shoulders are more extensive than the measure segment which makes a pressure focus happen in the center when the example is stacked with an elastic power. This pressure fixation guarantees a higher likelihood that the example will crack away from the finishes.

A dog bone test can be made out of any material that might be tried in strain, however are typically produced using plastics, polymers, material textures or elastic. The slicing technique is utilized to make tests from adaptable material as it is simpler to drive the shaper through them without breaking the example because of their malleable nature. Metals and earthenware production are commonly increasingly weak as are shaped into test tests. A metal is warmed until it condenses and is then filled the formed, while, pottery are shaped into dog bone shapes before they are relieved. Pursue this connect to see our adaptable dogbone malleable test cutting kick the bucket.

Tensile: tensile quality estimates the power required to pull something, for example, rope, wire, or an auxiliary pillar to the point where it breaks. The elasticity of a material is the most extreme measure of pliable pressure that it very well may be exposed to before disappointment.

Tensile estimated: ultimate tensile unit. Rigidity is characterized as an estimation of stress, which, thusly, is estimated as power per unit zone. The si unit of uts is pascal or pa. It's normally communicated in super pascals, so the uts is ordinarily communicated in uberpascals (or mpa).

Tensile test done: tensile testing: what is it? Pliable tests are utilized to decide how materials will act under pressure load. In a basic ductile test, an example is commonly dismantled to its limit to decide a definitive rigidity of the material.

### III. CHOOSING OF MATERIAL

*3.1. Aluminum*

*3.1.1. Aluminum-Silicon Alloys*

Silumin is a general name for a gathering of lightweight, high-quality aluminum alloys dependent on an aluminum–silicon framework. Aluminum-silicon alloys regularly contain 3 to 25% silicon content. Throwing is the essential utilization of aluminum-silicon alloys, however they can likewise be used in quick hardening procedures and powder metallurgy. Alloys utilized by powder metallurgy, as opposed to throwing, may contain significantly more silicon, up to half. Silumin has a high protection from consumption, making it helpful in moist situations.

*3.2. 8079 (8079-H14, Alfe1si) Aluminum*

8079 aluminum is a something else unclassified alloy of aluminum. Refered to properties are suitable for the h14 temper. To accomplish this temper, the metal is strain solidified to a quality that is generally somewhere between strengthened (o) and full-hard (h18). 8079 is the aluminum association (aa) assignment for this material. In european guidelines, it will be given as en aw-8079. Alfe1si is the en synthetic assignment. Moreover, the uns number is a98079.

It initially gotten its standard assignment in 1969.

The chart bars on the material properties cards underneath contrast 8079 aluminum with: generally unclassified aluminums (top), all aluminum alloys (center), and the whole database (base). A fully stocked bar implies this is the most elevated an incentive in the significant set. A

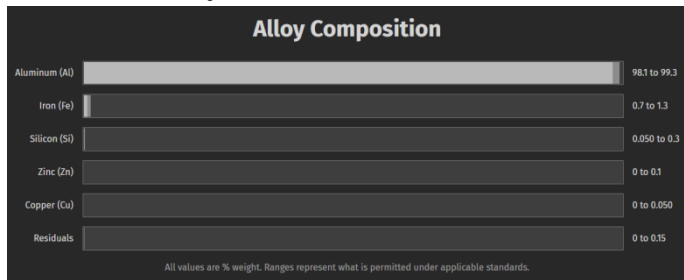half-thoroughly stocked bar implies it's half of the most noteworthy, etc.



**Figure 3.1 Material Composition**

## IV. LITERATURE REVIEW

A point by point far reaching writing audit on normal filler reinforced polymer composite material, and sugar stick bagasse fiber reinforced polymer composite material is displayed including distinctive kind of polymer, filler measurements, applications, and so on. Jayaramudu, agwuncha, ray, sadiku, and rajuluet. Al. Contemplated with regular polyalthiacerasoide woven textures blending with epoxy composite. The woven textures separated from bark of the tree to make half and half composites. The hand lay-up method was utilized to manufacture of half breed composite at room temperature. The surface alteration of woven fiber was finished by the procedure of antacid treatment. The microstructure and morphology contemplated was finished utilizing fourier changes infrared spectroscopic (ftir) and examining electron tiny techniques separately. The ftir investigations speak to minimal estimation of hemi-cellulose and lignin substance of antacid treated woven texture. The half and half composite recommended for different applications in structure and development ventures as boards for apportioning, flooring, stockpiling tanks and table taps, and so forth.

Barnasree, kumar, and bhowmiket. Al. Were contemplated wood dust molecule reinforced in epoxy based composite for examination of mechanical conduct. The sundy wood dust molecule utilized as support and ly556 epoxy for gum. The six distinctive level of filler molecule utilized in study. Malleable and flexural test were done utilizing utm and test size dependent on astm standard. The diverse structure parameters like as filler substance and speed for stacking with elastic and flexural quality utilizing gra were optimized.optimizationbygra has the upside of choosing best and most noticeably awful choices. Grg demonstrates that trial number 13 is the most appropriate and trial number 3 is the least significant. Epoxy composite with 10 fillercontents (wt%) at relating pace of 1 mm/min shows best execution and then again with 0 filler substance (wt%) at the speed of 3 mm/min demonstrates theworst execution.

Dinesh and jagdish et. Al. Research concentrated on wear investigation of sisal fiber reinforced epoxy based composite materials. Ly-556 and hy 951 utilized as sap and hardener individually. 10%, 20%, and 30% sisal fiber utilized as fortification during creation of composite by had lay-up technique. By expanding the level of the sisal fiber in manufacture work improve the weight reduction of the example of wear test. Sfrecm can be utilized as substitute materials for human orthopedic implants.

Brien, chin, long, and wetzelet. Al. Was worked with advancement of composites having polymer–lattice reinforced by polymer lace monofilaments. This material is precisely strong and straightforward composite materials. The result speaks to that these polymer–polymer composites offer great straightforwardness with wide temperature range, and unrivaled ballistic scattering obstruction contrasted and solid straightforward polymers.

Rathankar and shivanandet. Al. Took a shot at impact of fiber direction of epoxy covered composite. Three-point static bowing test/flexural test led by astm guidelines. Hand lay-up strategy is embraced for creation of tests. In composite creation fiber direction on 00, 450, 900 and covered composite plate thickness 2mm and 4mm. Graphite fiber cover 450 display higher flexural quality than glass fiber overlay with same direction.

## V. METHODOLOGY

*Step 1 : Choosing A Material*

- The projects deal the testing of the composite material, the which is made by using layer by layer method (laminated materials). In general the laminated material are used to made by the fiber and no-fiber. In this project i here consider the a real composite material of aluminum alloy (aluminum silicon carbide) which have a high composite of aluminum, silicon and cast iron.

**Figure 5.1 Material Composition**

| S.no | Material | | High % Of silicon | High % Of al alloy | Balance |
|---|---|---|---|---|---|
| | | | Default | Composition 1 | Composition 2 |
| 1 | Aluminum | Al | 98.1 | 98.35 | 98.225 |
| 2 | Iron + Other metals | Fe + Zn + Cu | 1.6 | 1.6 | 1.6 |
| 3 | Silicon | Si | 0.3 | 0.05 | 0.175 |
| | Percentage | | 100 | 100 | 100 |

**Table 1 Material Composition**

- In general the composite are made by percentage of mixed or arrangement or concentration of material properties to increase the strength and advantages of the materials.
  In the project, i here considered 3 conditions,
  1. High percentage of aluminum alloy and low percentage of the silicon
  2. Low percentage of aluminum alloy and high percentage of the silicon
  3. Balanced percentage of aluminum alloy and silicon
As per the percentages as shown in above figure.

*Step 2: Creating A Modal By Using 3d Software*

- In the step i here consider a 3d software catia to create a simple dog bone shape to test the laminated material.

*Step 3: Executing The Analysis Over The Material*

In this step i here consider ansys software, with any application of the tensional and torsional load over it. By using the analysis software i can define the deformation, stress and strain of the material and its valves.

## VI.  INTRODUCTION OF CATIA

Catia (/kəˈtiːə/, an abbreviation of pc supported three-dimensional intelligent application) is a multi-stage programming suite for pc helped structure (cad), pc helped producing (cam), pc supported designing (cae), plm and 3d, created by the french organization dassaultsystèmes.

*6.1. Mechanical Engineering*

*6.2. Creating Of Modal:*

To start the modal: to windows and type catia. The click over it it will start the software. Now go to the start and select mechanical design domain. In mechanical design domain select part design.

In part design module, go to insert and select sketch based feature. In sketch based feature-select pad tool. Now in part tool definition, go to sketcher tool and start creating i shape as shown in below.
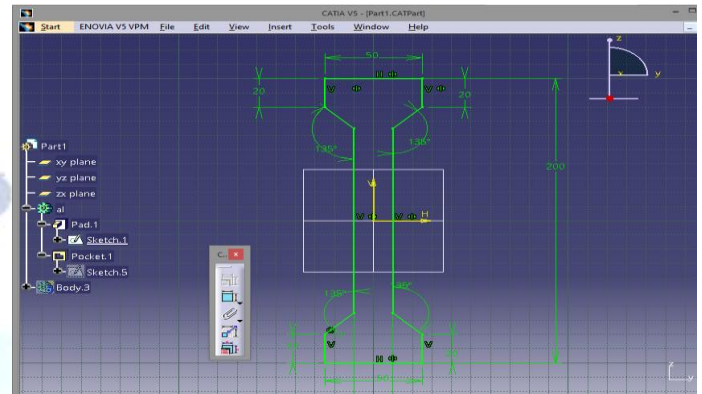


**Figure 6.1 Dog Bone Shape Sketch In Catia**

After the completion of the sketch as shown in above, go to exist workbench to complete the sketching process. Then we will enter into the part design module again. In pad definition specify the depth of the i section as shown in below to covert the 2d layout sketch to 3d modal.
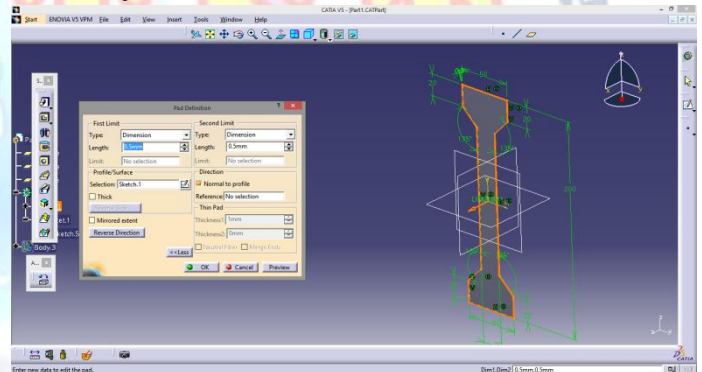


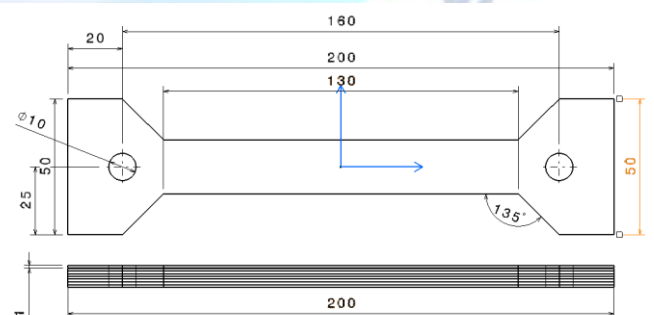**Figure 6.2 Dog Bone Shape 3d Shape InCatia For Laminated**



**Figure 6.3 Drafting And Dimension Of Dog Bone Shape**

## VII.  ANSYS SOFTWARE

Ansys offers an exhaustive programming suite that traverses the whole scope of material science, giving access to for all intents and purposes any

kind of building reproduction required in the structure procedure. Associations around the globe trust ansysfea and cfd programming to convey the best an incentive for their building reenactment programming speculation.

*7.1. Introduction To Ansys Mechanical (Workbench)*

Successful completion will give understudies the capacity get ready and upgrade their geometry for examination and to utilize simulation to perform explicit dynamics limited component investigations, just as give a concise prologue to warm and modular examinations..

*7.1.1. Immediate Benefits*

•effectively and productively make, adjust or potentially import geometry for certifiable limited component applications.

•create and examine definite, certifiable limited component explicit dynamics models utilizing the ansys workbench endless supply of the class.

•knowledge in performing limited component investigation and understanding the key factors in making quality cross sections, applying stacking and limit conditions that copy certifiable applications, and translating results.

In this section, we will almost certainly characterize:

1. Understand the sorts of framework
2. Understand various sorts of cells
3. Understand the realistic ui of the workbench window
4. Start another undertaking in ansys workbench windows
5. Add the first and consequent examination framework to a venture
6. Set units for the venture

Ansys workbench, created by ansysinc., usa, is a pc supported limited component demonstrating and limited component examination device (cafem and cafea). In the graphical uigui of ansys workbench the customer can deliver 3-dimensional and fea models, perform examination and make outcomes of assessment. We can play out a combination of assignments running from diagram evaluation to constrained segment examination to complete thing streamlining assessment by using ansys workbench. Ansys also enable the mix of autonomous examination system into an endeavor and to manage the assignment work process.

*7.1.2. Beginning Ansys Workbench 19.2*

The workbench windows help streamline a whole task to be completed in ansys workbench 19.2 . In this window, one can make, oversee, and see the work process of the whole task make by utilizing standard examination framework. The workbench

windows for the most part comprise of the menu bar, standard toolbar, the toolbar windows, venture schematic windows, and the status bar.
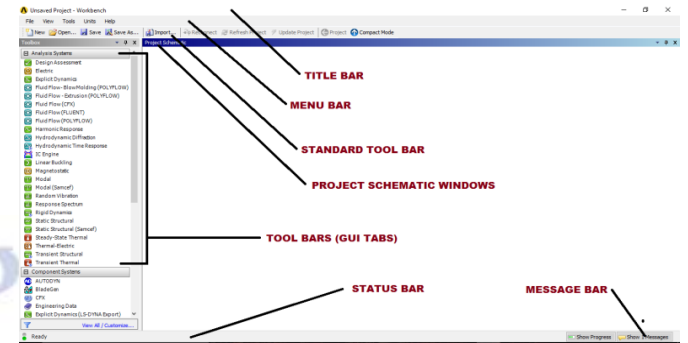


**Figure 7.1 The Component Of The Workbench Windows**

The undertaking schematics windows help deal with a whole venture. It shows the work process of whole examination venture. To add an investigation framework to the undertaking schematic windows, drag the examination framework from tool stash windows and drop into the green shaded box showed in the task schematic windows.
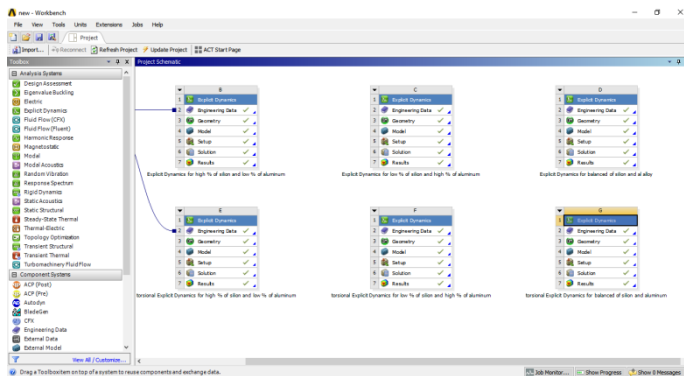
## VIII.   EXPLICIT DYNAMICS ANALYSIS

### a)   *Project Objective*

In this project, we will be able to define total deformation and stress, etc

➢ Create the explicit dynamics analysis system
➢ Apply different types of materials
➢ Applying of boundary conditions
➢ Apply a different type of constraints
➢ Apply different loads
➢ Generate the results as per required
➢ Generate project reports

The explicit dynamics investigation is one of the significant examinations in ansys workbench. It is accessible as explicit dynamics investigation framework under the examination framework tool kit in the tool compartment window, this framework investigations the auxiliary segments for removals (disfigurement), stresses, strains, and powers under various stacking conditions. The heaps in this examination framework are accepted not to have damping qualities (time subordinate). Enduring stacking and damping conditions are expected in this kind of examination framework.

**The Explicit Dynamics Analysis Added To The Project Schematic**
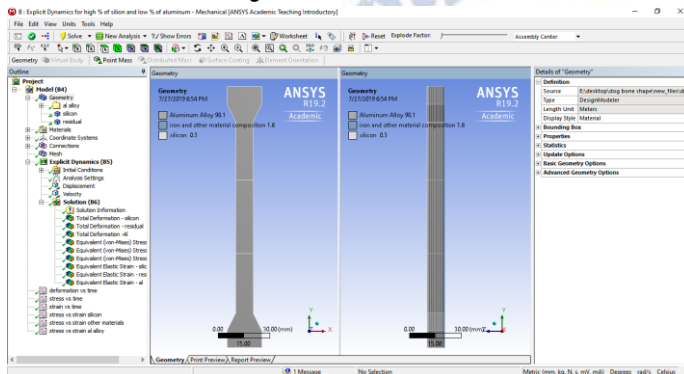


**The Mechanical WindowWith Modal In It**

* the first layer is made upto of silicon and last layer in above figure is made up of cast iron (carbide materials). The rest layer 2nd ,3rd ,4th ,5th ,6th ,7th ,8th and 9 th are consider al aluminum alloys.

*8.1. Project Overview*

The dimensions to create the model and its boundary and loading conditions are also given in the same figure. Run a explicit dynamics analysis on the model and evaluate the total deformation and stress.

A. Start the project
B. Create the model.
C. Generate the mesh.
D. Set the boundary
E.  Loading conditions.
F. Solve the model.
G. Evaluate results.
H. Save the project.

*A Starting A New Project And Creating The Model*

The first step is to start a new project in the workbench window.

1. Start ansys  workbench.
2. Double-click on explicit dynamics in the toolbox window; the explicit dynamics analysis system is added in the project schematic window.
3. Double-click on the geometry cell; the design modeler window along with the ansys  workbench dialog box is displayed.

4. In the ansys  workbench dialog box, set the unit to millimeter.
5. Now, go to file and select dog bone.stp file from catia software then click over generate.
6. Exit the design modeler window to display the workbench window.

*B Adding The Material To The Engineering Data Figure Workspace*

After creating the holes in the model, you now need to apply the material to it. The material to be applied is stainless steel.

1. Double-click on the engineering data cell of the spring plate analysis system; the engineering data workspace is displayed in the workbench window.
2. Choose the engineering data sources toggle button from the standard toolbar; the engineering data sources window is added to the engineering data workspace.
3.  In the engineering data sources window, select the general materials library to display the outline of general materials window.
4. In the outline of general materials window, choose the plus symbol corresponding to aluminum alloy, silicon and cast iron; the material is added to the engineering data in the outline window of the engineering data workspace.
5. Again, choose the engineering data sources toggle button from the standard toolbar to switch to the default view of engineering data workspace.

| S.no | Material | | High % Of silicon | High % Of al alloy | Balance |
|------|----------|------|------|------|------|
| | | | Default | Composition 1 | Composition 2 |
| 1 | Aluminum | Al | 98.1 | 98.35 | 98.225 |
| 2 | Iron + Other metals | Fe + Zn + Cu | 1.6 | 1.6 | 1.6 |
| 3 | Silicon | Si | 0.3 | 0.05 | 0.175 |
| | Percentage | | 100 | 100 | 100 |

**Table 2Material composition**
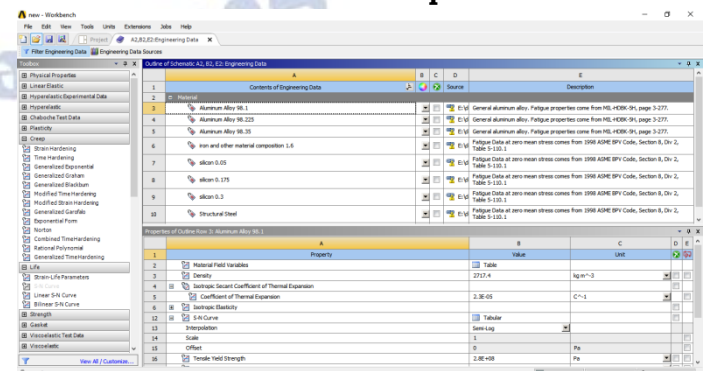


**Specifying A Material**

## 8.2 Composite Data 1 OfSilicon 0.3 And Al Alloy 98.1

### 8.2.1 Silicon 0.3

Constants

| Density | 2.5802e-006 kg mm^-3 |
|---|---|
| Isotropic secant coefficient of thermal expansion | 1.2e-005 c^-1 |
| Specific heat constant pressure | 4.34e+005 mj kg^-1 c^-1 |
| Isotropic thermal conductivity | 6.05e-002 w mm^-1 c^-1 |
| Isotropic resistivity | 1.7e-004 ohm mm |

Compressive ultimate strength

| Compressive ultimate strength mpa |
|---|
| 365 |

Compressive yield strength

| Compressive yield strength mpa |
|---|
| 280 |

Tensile yield strength

| Tensile yield strength mpa |
|---|
| 280 |

Tensile ultimate strength

| Tensile ultimate strength mpa |
|---|
| 365 |

### 8.2.2. Aluminum Alloy 98.1

Constants

| Density | 2.7174e-006 kg mm^-3 |
|---|---|
| Isotropic secant coefficient of thermal expansion | 2.3e-005 c^-1 |
| Specific heat constant pressure | 8.75e+005 mj kg^-1 c^-1 |

Compressive yield strength

| Compressive yield strength mpa |
|---|
| 280 |

Tensile yield strength

| Tensile yield strength mpa |
|---|
| 280 |

Tensile ultimate strength

| Tensile ultimate strength mpa |
|---|

### 8.2.3. Iron And Other Material Composition 1.6

Constants

| Density | 1.152e-007 kg mm^-3 |
|---|---|
| Isotropic secant coefficient of thermal expansion | 1.2e-005 c^-1 |
| Specific heat constant pressure | 4.34e+005 mj kg^-1 c^-1 |
| Isotropic thermal conductivity | 6.05e-002 w mm^-1 c^-1 |
| Isotropic resistivity | 1.7e-004 ohm mm |

Compressive ultimate strength

| Compressive ultimate strength mpa |
|---|
| 840 |

Compressive yield strength

| Compressive yield strength mpa |
|---|
| 240 |

Tensile yield strength

| Tensile yield strength mpa |
|---|
| 240 |

Tensile ultimate strength

| Tensile ultimate strength mpa |
|---|
| 840 |

Isotropic secant coefficient of thermal expansion

| Zero-thermal-strain reference temperature c |
|---|
| 22 |

Strain-life parameters

| Strength coefficient mpa | Strength exponent | Ductility coefficient | Ductility exponent | Cyclic strength coefficient mpa | Cyclic strain hardening exponent |
|---|---|---|---|---|---|
| 920 | -0.106 | 0.213 | -0.47 | 1000 | 0.2 |

Isotropic elasticity

| Young's modulus mpa | Poisson's ratio | Bulk modulus mpa | Shear modulus mpa | Temperature c |
|---|---|---|---|---|
| 1.1e+005 | 0.28 | 83333 | 42969 | |

Choose the return to project button from the standard toolbar to display the project schematic window.

### 8.5. Generating The Mesh

After the model is made in the plan modeler window, you have to create the work for the model in the mechanical window.

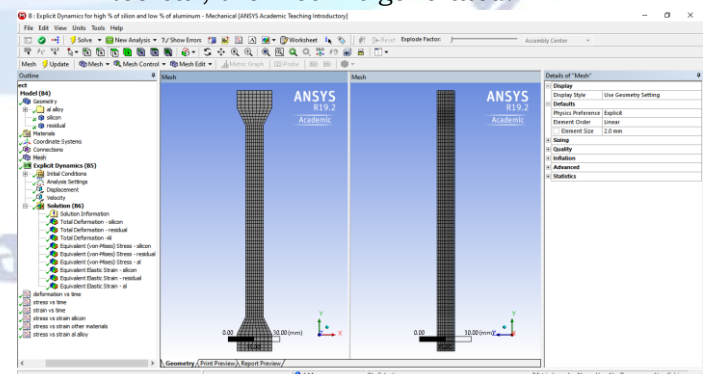1. Choose the generate mesh tool from the mesh drop-down in the mesh contextual toolbar; the mesh is generated.



**Figure 8.1 Mesh Generated With Default Mesh Controls**

Expand the statistics node in the details of "mesh" window to display the total number of elements created. On doing so, you will find that the total number of elements.

## 8.6. Specifying The Boundary Conditions

After you mesh the model, it is required to specify the boundary and loading conditions.

1. In the mechanical window, select the explicit dynamics node from the tree outline; the details of "explicit dynamics" window is displayed along with the environment contextual toolbar.

2. In mechanical window, on left side we can see the explicit dynamic setup as shown figure. Then right over it and select displacement and choose the face as shown in figure
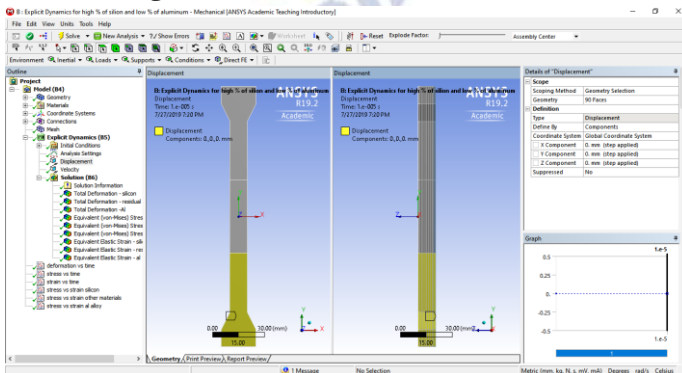


**Figure 8.2 Choosing The Displacement Tool From The Supports Drop-Down**

3. In mechanical window, on left side we can see the explicit dynamic setup as shown figure. Then right over it and select velocity and choose the face as shown in figure

*2) A On First Condition: Tensile Test*

4. In details of displacement, specify x= 0, y= 20 mm and z=0. That means in y direction the body is elongated 20 mm.
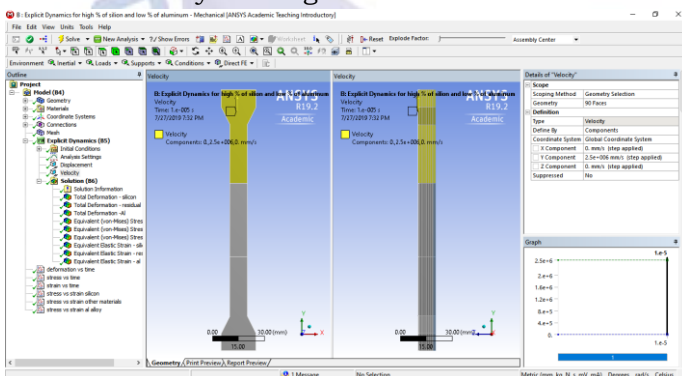


**Figure 8.3 Choosing The Velocity Tool From The Supports Drop-Down**

*B On Second Condition: Torsional Test*

5. In details of displacement, specify x= 0, y= 6.28 rad/sec (360 deg/sec) and z=0. That means the all degree freedom is zero.
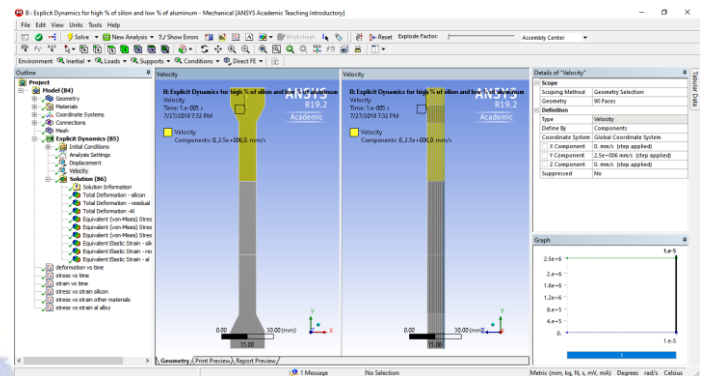


**Figure 8.4 Choosing The Angular Velocity Tool From The Supports Drop-Down**

## 8.7. Solving The Fe Model And Analyzing The Results

After the boundary and load conditions are specified for the model, you need to solve the analysis. After solving, you will get the total and directional deformations due to the given condition. Also, you will get equivalent stress, maximum principal, and minimum principal stresses.

1. Select the solution node in the tree outline; the solution contextual toolbar is displayed. Also, the details of "solution" window is displayed.

2. Choose the total tool from the deformation drop-down of the solution contextual toolbar; total deformation is added under the solution node.
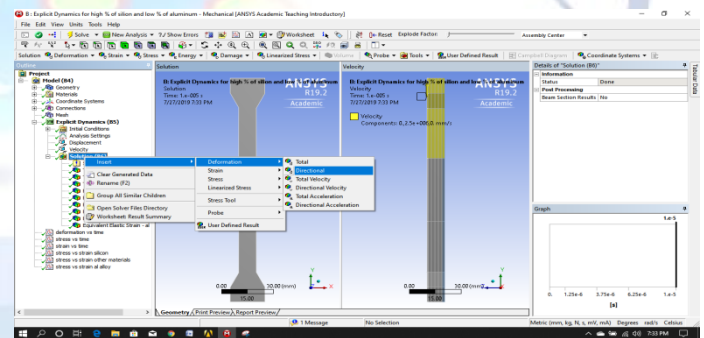


**Figure 8.5 Choosing The Total Tool From The Deformation Drop Down**

3. Now, choose the directional tool from the deformation drop-down; directional deformation is added under the solution node.

4. Choose the equivalent (von-misses) tool from the stress drop-down in the solution contextual toolbar;

## IX. RESULTS FROM EXPLICIT DYNAMICS

The consider material of the project is aluminum silicon (8079-h14, alfe1si) which having a percentage of aluminum alloy form 98.1 to 9.35 %

and silicon 0.05 to 0.3 % and residual materials 1.6 %.

A. *9.1. Results And Graphs For Tensional Loads:*
B. *9.1.3. For Material Composite 3:*
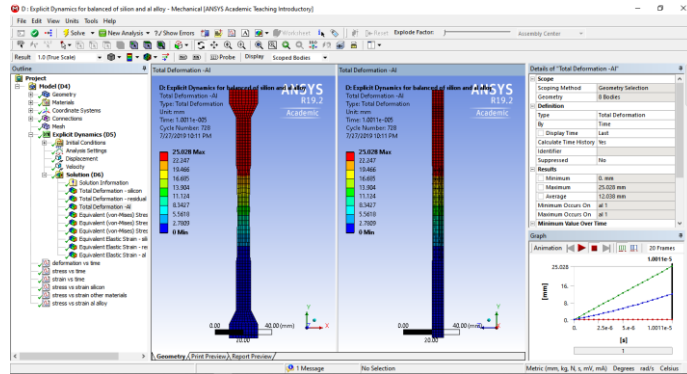
Balanced percentage of silicon and aluminum alloy



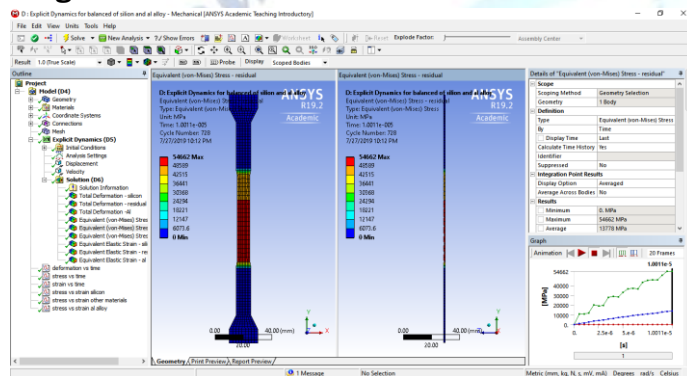**Figure 9.1 The Values Of Total Deformation.**
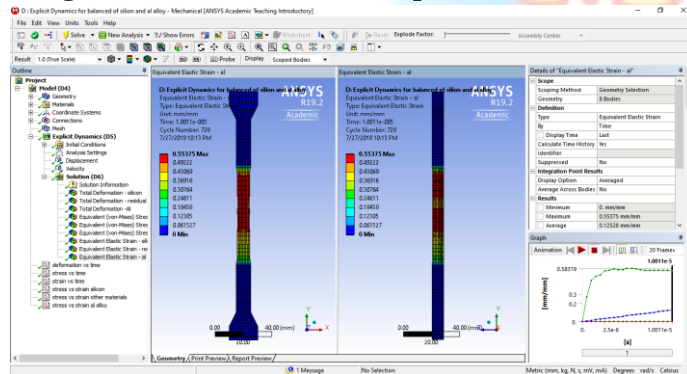


**Figure 9.2 The Values Of Equivalent Stress.**



**Figure 9.3 The Values Of Equivalent Strain.**

*1) J Deformation Vs Time Or Step*

**Deformation vs time**

| Steps | Time [s] | [b] total deformation - silicon (max) [mm] | [d] total deformation - residual (max) [mm] | [f] total deformation -al (max) [mm] |
|---|---|---|---|---|
| 1 | 1 | 0. | 0. | 0. |
| | 2 | 1.2643 | 1.2643 | 1.2643 |
| | 3 | 2.5334 | 2.5334 | 2.5334 |
| | 4 | 3.7661 | 3.7661 | 3.7661 |
| | 5 | 5.0352 | 5.0352 | 5.0352 |
| | 6 | 6.268 | 6.268 | 6.268 |
| | 7 | 7.5008 | 7.5008 | 7.5008 |
| | 8 | 8.7698 | 8.7698 | 8.7698 |

| | 9 | 10.003 | 10.003 | 10.003 |
|---|---|---|---|---|
| | 10 | 11.272 | 11.272 | 11.272 |
| | 11 | 12.504 | 12.504 | 12.504 |
| | 12 | 13.783 | 13.783 | 13.783 |
| | 13 | 15.025 | 15.025 | 15.025 |
| | 14 | 16.268 | 16.268 | 16.268 |
| | 15 | 17.515 | 17.515 | 17.515 |
| | 16 | 18.758 | 18.758 | 18.758 |
| | 17 | 20.028 | 20.028 | 20.028 |
| | 18 | 21.269 | 21.269 | 21.269 |
| | 19 | 22.509 | 22.509 | 22.509 |
| | 20 | 23.762 | 23.762 | 23.762 |
| | 21 | | | |
| N/a | 22 | 25.028 | 25.028 | 25.028 |

**Table 3 Deformation Vs Time Or Step**
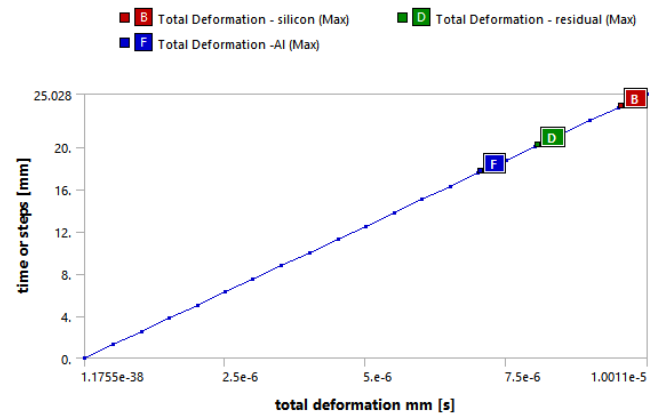


**Graph 1** Deformation Vs Time

*2) K Stress Vs Time Or Step*

| Steps | Time [s] | [b] equivalent (von-mises) stress - silicon (max) [mpa] | [d] equivalent (von-mises) stress - residual (max) [mpa] | [f] equivalent (von-mises) stress - al (max) [mpa] |
|---|---|---|---|---|
| 1 | 1 | 0. | 0. | 0. |
| | 2 | 26375 | 10671 | 18763 |
| | 3 | 38675 | 10774 | 30309 |
| | 4 | 40731 | 11624 | 33359 |
| | 5 | 44844 | 20120 | 35716 |
| | 6 | 46310 | 18864 | 38606 |
| | 7 | 46697 | 19525 | 38909 |
| | 8 | 46789 | 27815 | 40057 |
| | 9 | 47357 | 28663 | 39272 |
| | 10 | 47419 | 28381 | 39412 |
| | 11 | 47016 | 32931 | 40482 |
| | 12 | 46121 | 36487 | 40669 |
| | 13 | 45975 | 37282 | 40116 |
| | 14 | 46414 | 36870 | 39505 |
| | 15 | 46435 | 43198 | 39014 |
| | 16 | 46270 | 45439 | 38971 |
| | 17 | 45825 | 46071 | 39269 |
| | 18 | 45639 | 45807 | 39256 |

| | 19 | 47146 | 50793 | 39018 |
|---|---|---|---|---|
| | 20 | 51947 | 54610 | 38956 |
| | 21 | | | |
| N/a | 22 | 58037 | 54662 | 38616 |

**Table 4 Stress Vs Time Or Step**
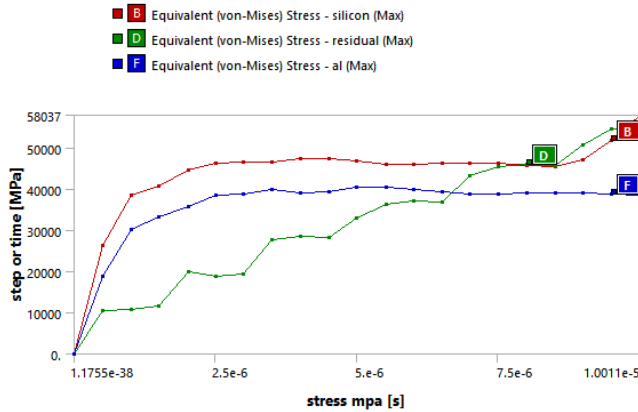


**Graph  2**Stress Vs Time

*3)  L Stress Vs Strain Over Aluminum Alloy*

| Steps | Time [s] | [b]  equivalent (von-mises) stress - al (max) [mpa] | Equivalent  elastic strain - al  (max) [mm/mm] |
|---|---|---|---|
| 1 | 1 | 0. | 0. |
| | 2 | 18763 | 0.27215 |
| | 3 | 30309 | 0.44897 |
| | 4 | 33359 | 0.49455 |
| | 5 | 35716 | 0.51305 |
| | 6 | 38606 | 0.55436 |
| | 7 | 38909 | 0.55978 |
| | 8 | 40057 | 0.57502 |
| | 9 | 39272 | 0.56403 |
| | 10 | 39412 | 0.56542 |
| | 11 | 40482 | 0.58052 |
| | 12 | 40669 | 0.58319 |
| | 13 | 40116 | 0.57537 |
| | 14 | 39505 | 0.56673 |
| | 15 | 39014 | 0.55952 |
| | 16 | 38971 | 0.559 |
| | 17 | 39269 | 0.56335 |
| | 18 | 39256 | 0.56299 |
| | 19 | 39018 | 0.55959 |
| | 20 | 38956 | 0.55868 |
| | 21 | | |
| N/a | 22 | 38616 | 0.55375 |

**Table 5 Stress Vs Strain Over Aluminum Alloy**
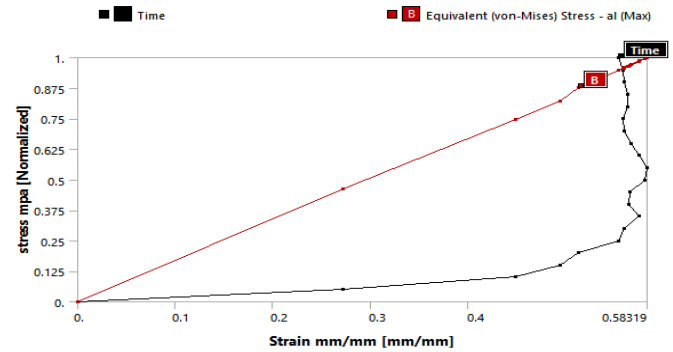


**Graph  3**Stress Vs Strain Al Alloy

*9.2. Results And Graphs For Torsional Loads:*

*9.2.3.  For Material Composite 3:*
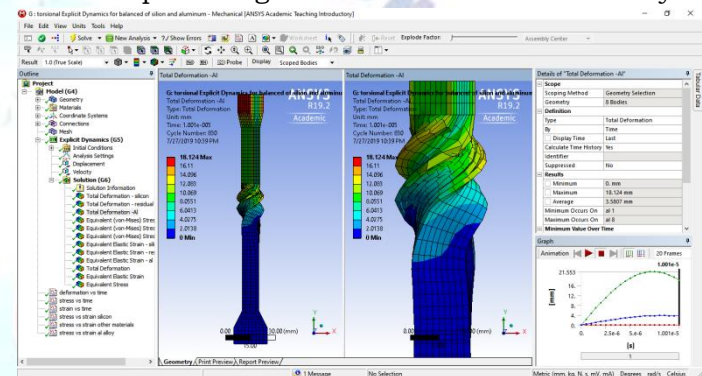
Balanced percentage of silicon and aluminum alloy



**Figure 9.4 The Values Of Total Deformation.**



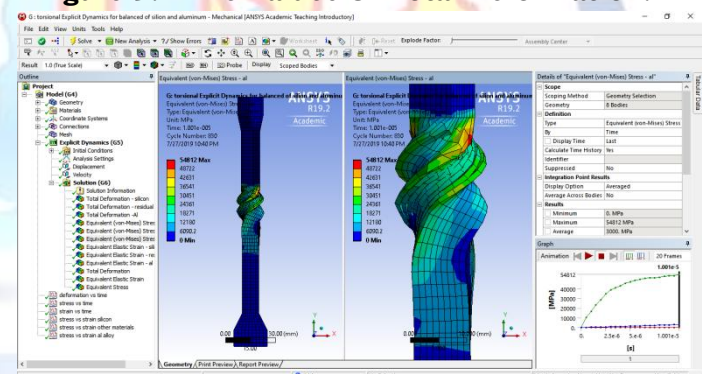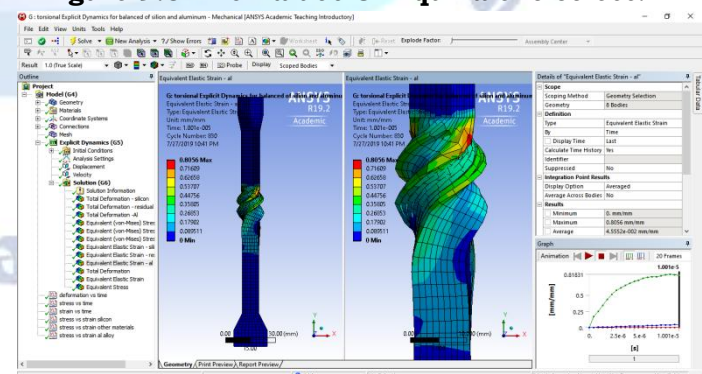**Figure 9.5 The Values Of Equivalent Stress.**



**Figure 9.6 The Values Of Equivalent Strain.**

*J Deformation Vs Time Or Step*

| Steps | Time [s] | [b]  total deformation | [d]  total deformation | [f]  total deformation |
|---|---|---|---|---|

| | | - silicon (max) [mm] | - residual (max) [mm] | -al (max) [mm] |
|---|---|---|---|---|
| 1 | 1 | 2.5121e-015 | 2.5121e-015 | 2.6645e-015 |
| | 2 | 2.4129 | 2.4129 | 2.323 |
| | 3 | 4.815 | 4.815 | 4.6356 |
| | 4 | 7.0942 | 7.0942 | 6.83 |
| | 5 | 9.3136 | 9.3136 | 8.9667 |
| | 6 | 11.449 | 11.449 | 11.022 |
| | 7 | 13.447 | 13.447 | 12.946 |
| | 8 | 15.281 | 15.281 | 14.712 |
| | 9 | 16.942 | 16.942 | 16.311 |
| | 10 | 18.412 | 18.412 | 17.726 |
| | 11 | 19.684 | 19.684 | 18.951 |
| | 12 | 20.706 | 20.706 | 19.935 |
| | 13 | 21.503 | 21.503 | 20.703 |
| | 14 | 22.051 | 22.051 | 21.23 |
| | 15 | 22.348 | 22.348 | 21.515 |
| | 16 | 22.386 | 22.386 | 21.553 |
| | 17 | 22.173 | 22.173 | 21.347 |
| | 18 | 21.702 | 21.702 | 20.894 |
| | 19 | 20.979 | 20.979 | 20.198 |
| | 20 | 20.031 | 20.031 | 19.285 |
| | 21 | | | |
| N/a | 22 | 18.825 | 29.548 | 18.124 |

**Table 6 Deformation Vs Time Or Step**
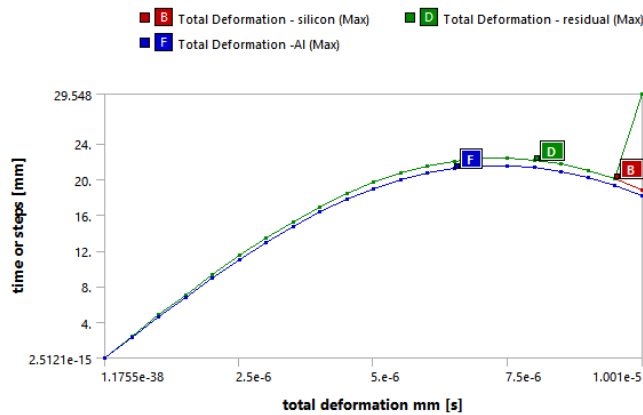


**Graph 4** Deformation Vs Time

*4) K Stress Vs Time Or Step*

| Steps | Time [s] | [b] equivalent (von-mises) stress - silicon (max) [mpa] | [d] equivalent (von-mises) stress - residual (max) [mpa] | [f] equivalent (von-mises) stress - al (max) [mpa] |
|---|---|---|---|---|
| 1 | 1 | 0. | 0. | 0. |
| | 2 | 17032 | 10808 | 10289 |
| | 3 | 29924 | 15597 | 16777 |
| | 4 | 37511 | 19098 | 23082 |

| | 5 | 46371 | 22060 | 28695 |
|---|---|---|---|---|
| | 6 | 54514 | 24366 | 34781 |
| | 7 | 60834 | 26960 | 38671 |
| | 8 | 64993 | 29045 | 40995 |
| | 9 | 69003 | 28232 | 43106 |
| | 10 | 71934 | 25425 | 45264 |
| | 11 | 73868 | 28179 | 46908 |
| | 12 | 74566 | 29044 | 48465 |
| | 13 | 74544 | 30603 | 49880 |
| | 14 | 74818 | 34082 | 50752 |
| | 15 | 75355 | 33469 | 51220 |
| | 16 | 75252 | 32239 | 51154 |
| | 17 | 74152 | 36281 | 51632 |
| | 18 | 72094 | 41104 | 52502 |
| | 19 | 69267 | 45157 | 53180 |
| | 20 | 65158 | 45664 | 53474 |
| | 21 | | | |
| N/a | 22 | 62082 | 47583 | 54812 |

**Table 7 Stress Vs Time Or Step**
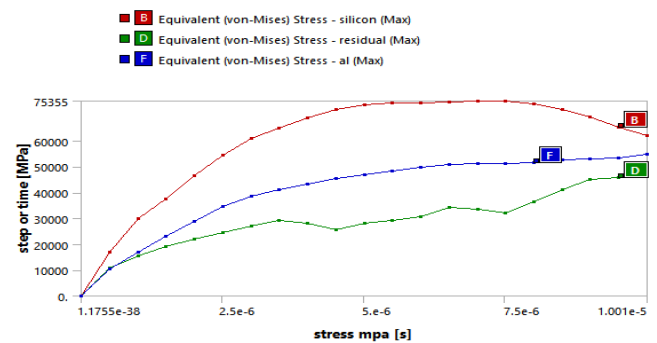


**Graph 5** Stress Vs Time

*L Stress Vs Strain Over Aluminum Alloy*

| Steps | Time [s] | [b] equivalent (von-mises) stress - al (max) [mpa] | Equivalent elastic strain - al (max) [mm/mm] |
|---|---|---|---|
| 1 | 1 | 0. | 0. |
| | 2 | 10289 | 0.16756 |
| | 3 | 16777 | 0.24728 |
| | 4 | 23082 | 0.34255 |
| | 5 | 28695 | 0.43318 |
| | 6 | 34781 | 0.51284 |
| | 7 | 38671 | 0.56927 |
| | 8 | 40995 | 0.61118 |
| | 9 | 43106 | 0.64548 |
| | 10 | 45264 | 0.67801 |
| | 11 | 46908 | 0.70789 |
| | 12 | 48465 | 0.72847 |
| | 13 | 49880 | 0.75282 |
| | 14 | 50752 | 0.76902 |
| | 15 | 51220 | 0.77785 |
| | 16 | 51154 | 0.78174 |
| | 17 | 51632 | 0.79186 |
| | 18 | 52502 | 0.80757 |

|  | 19 | 53180 | 0.81831 |
|  | 20 | 53474 | 0.81237 |
|  | 21 |  |  |
| N/a | 22 | 54812 | 0.8056 |

**Table 8 Stress Vs Strain Over Aluminum Alloy**
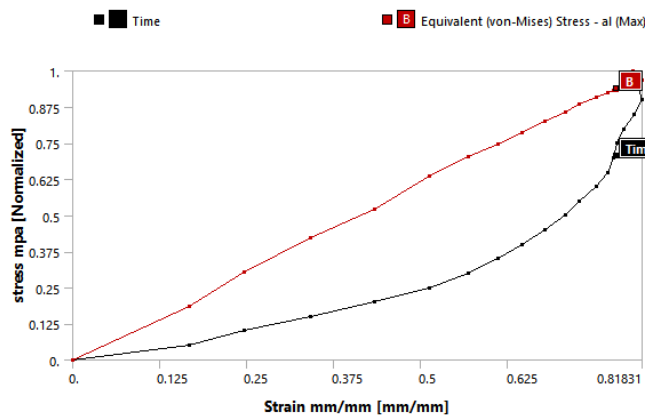


**Graph 6 Stress Vs Strain Over Aluminum Alloy**

Select all other parameters from the solution node; the respective view is displayed in the graphics screen. The table given next lists all the results obtained from the analysis. Show the corresponding graphical representation of values obtained.

Close the existing mechanical window; the workbench window is displayed.

A body is called to be deformed if its shape is changed temporarily or permanently. The temporary change of shape is known as elastic deformation and a permanent change of shape is known as plastic deformation. In ansys workbench, you can determine deformation in terms of total and directional deformations.

Total deformation is the total change of shape in a given working condition. You can view the total deformation induced in any component by using the total tool from the deformation drop-down in the solution contextual toolbar. Directional deformation is the total change of shape in a particular axis, due to given working conditions. You can view directional deformation by using the directional tool from the deformation drop-down in the solution contextual toolbar.

Total deformation is the summation of all directional deformations produced in a certain region of the model. The following equation describes the total deformation:

If deformation in the x-axis $u_x$

Deformation in the y-axis $u_y$

Deformation in the z-axis $u_z$

Then total deformation. U will be given as follows:

$$U = (u_x^2 + u_y^2 + u_z^2)^{1/2}$$

The legend has colors arranged in a band from top to bottom. Depending upon the type of analysis and the parameters evaluated, each color will indicate a different value. A typical legend displayed when total deformation is selected from the tree outline.

The blue color in the legend indicates the minimum value of total deformation. In this case, it displays 0 which means there is no deformation at that region.

The value that is displayed next to each color is the total deformation in the region which is depicted by that particular color in the model. The blue color in the model represents the lowest value of the total deformation, whereas the red color denotes the maximum value of the total deformation.

To view the transparent mode of the model, select the show undeformed model option from the edges drop-down list. Similarly to show the elements that are created after meshing in the deformed shape of the model, select the show elements option from the edges drop-down list, shows the undeformed model when the show undeformed model option is selected. Displays the elements when the show elements option is selected.

Exit the explicit dynamics analysis - mechanical window

## X. CONCLUSION

| S.No | Material | Percentage | Deformation | Stress | Strain | Strength | Force |
|---|---|---|---|---|---|---|---|
|  |  | % | MM | MPA | MM/MM | MPA | KN |
| 1 | Composite 1 | 98.1 % Al + 0.35 % Si + 1.6 % Other | 25.028 | 38567 | 0.55375 | 69646.95 | 385.67 |
| 2 | Composite 2 | 98.35 % Al + 0.05 % Si + 1.6 % Other | 25.028 | 38660 | 0.55367 | 69824.99 | 386.6 |
| 3 | Composite 3 | 98.225 % Al + 0.175 % Si + 1.6 % Other | 25.028 | 38666 | 0.55375 | 69825.73 | 386.66 |

As we study in this project, i here consider a sample material of aluminum silicon combination material which having a percentage range of aluminum alloy form 98.1% to 90.35% and silicon 0.05% to 0.35%.

The study of the material is done by considering low aluminum alloy and high silicon percentage, high aluminum alloy and low silicon percentage which are limited based on the composite from AISI standards. But in study i also consider a equally balanced aluminum alloy and silicon percentage. By using a 3d design CATIA software i created an

3d modal geometry for simulation or analysis process. In ansys software in here used explicit dynamic analysis. In simulation process, i here consider 2 method of analysis by application of boundary condition 1. Tensional load and 2.Torsional load or angular displacement load.

After the study of examination over the percentage of material and boundary conditions, i found that the material which is will balanced percentage having more strength and required more tensile load to break the body.

In final, the adjust the percentage or balancing the percentage of material composition will give a effect in strength and generate more withstanding limited to work longer time.

## REFERENCES

[1] Application of various unique vibration protections to diminish nvh threats brought about by elective half shaft plan (2017-03-28 technical paper)

[2] Design of experiment of dynamic vibration absorber for drive shaft (2015-04-14 technical paper 2015-01-0631)

[3] Geometrical optimization of an automotive halfshaft 2017-03-28 technical paper (2017-01-1125) victor baumhardt, valdineiszibor

[4] Jin kook kim. Dai gillee, and durkhyuncho, 2001, "assessment of adhesively bonded joints for composite propeller shafts", journal of composite materials, vol.35, no.11, pp.999-1021.

[5] Development of a vehicle drive shaft telemetry system 2016-04-05 technical paper 2016-01-0410 joseph v. Gabiniewicz, douglas m. Batter puncher, michaeltestani

[6] Design and analysis of composite material drive shaft chaitanya g rothe, a. S. Bombatkaret., al.

[7] Review of design of cream aluminum/composite drive shaft for vehicle bushan k. Suryawanshi, g. Damle international journal of innovative technology and exploring engineering issn:2278-3075, vol-2, issue-4, march-2013.

[8] "plan and analysis of composite drive shaft using ansys and ga" an essential review international journal of ebb and flow building research (ijmer) www.ijmer.com vol.3, issue.1, jan-feb 2013 pp 490-496 issn:2249-6645 sagar r. Dharmadhikari, 1 sachin g. Mahakalkar, 2 jayant p. Giri, 3 nilesh d. Khutafale4.

[9] Mcdermott, j.; fiberglass recycling; composite fabri-cation, composites fabrication association; arlington, va;may 1996, pp. 9-13.

[10] Jones r.m., 1990, mechanics of composite materials, 2e, mcgraw-hill book company, new york.

[11] Belingardi.g, calderale.p.m. Moreover, rosetto.m, 1990, "diagram of composite material drive shafts for vehicular applications", int. J. Of vehicle design, vol.11, no.6, pp. 553-563.

[12] Pollard. A, 1989, "polymer matrix composites in driveline applications", journal of composite structures vol.25, pp.165-175.

[13] Osama mohammedelmardisuleimandeflection and stress examination of sinewy composite covers int j adv res computsci software eng (ijarcsse), 6 (august (8)) (2016), pp. 105-115

[14] David roylance-a prologue to composite materials, branch of material science and designing massachusetts institute of technology, cambridge (2000)-

[15] Stephen w. Tsai, hahn h. Thomas-introduction to composite materials-technomic distributing organization (1980)

[16] G.j. Turvey, i.h. Marshall-buckling and post clasping of composite plates-t.j press ltd, padstow, cornwall, great britain (1995)

[17] Naruoka m. Book reference on hypothesis of plates. Gihodo, tokyoe

[18] Th.a. Winterstetter, h. Schmidt-stability of roundabout tube shaped steel shells under consolidated stacking thin—walled struct, 40 (2002), pp. 893-910-m. Pircher, r. Scaffold

[19] The impact of circumferential weld—actuated defects on the clasping of storehouses and tanks-j constr steel res, 57 (5) (2001), pp. 569-580

[20] M. Deml, w. Wunderlichdirect assessment of the most noticeably awful defect shape in shell - clasping comput methods applmech eng, 149 (1–4) (1997), pp. 201-222

[21] Articledownloadpdfview record in scopusgoogle scholar

[22] J. Arbocz, j.h. Starnes-future bearings and difficulties in shell dependability examination thin—walled struct, 40 (2002), pp. 729-754

[23] B. Geier, h. Klein, r. Zimmermann-buckling tests with pivotally compacted unstiffened round and hollow shells produced using cfrp-proceedings, universal colloquium on clasping of shell structures ashore, in the ocean, and noticeable all around, elsevier connected sciences, london and new york (1991), pp. 498-507

[24] Davis, joseph r. (2004), tensile testing (second ed.), asm international, p. 2, isbn 978-0-87170-806-9.

[25] Annappa, c h (july 2012), "use of value engineering for cost reduction – a case study of universal testing machine", international journal of advances in engineering and technology, 4 (1): 618–629, recovered 1 december 2017

[26] Nist, large scale structure testing facility, filed from the first on 5 june 2010, recovered 4 may 2010.

[27] Kirstein (1971). All inclusive testing machine of 12-million-lbf capacity, nbs pub 355 (pdf) (report). Nist. Recovered 22 may 2017