© August 2015 | IJIRT | Volume 2 Issue 3 | ISSN: 2349-6002

Torsional Vibration Analysis of Aluminium6061Sic Metal Matrix Composite

R.Manoharan[1], C.Parthasarathy[2]
[1]*Prist University/Tamilnadu*
[2]*Asst Proffessor/Prist University/Tamil Nadu*

Abstract*- Structural members subjected to external load achieve extra strength by an initial permanent deformation. During the initial stages of elasto-plastic deformation, increasing stress is needed to cause plastic strain in the member, due to strain hardening. In practice, the stresses imposed by service loadings are likely to be complex. When safety factors are reduced to save weight the stresses in the material will approach the yield condition. In most conventional engineering alloys, plasticity will offer the safety margin required. The present study examines the critical elasto-plastic load carrying capacity of AlSiC MMC rod under combined axial and shear stress states. The Aluminium alloy composite materials consist of high specific strength, high specific stiffness, more thermal stability, more corrosion and wear resistance, high fatigue life. Aluminium alloy materials found to the best alternative with its unique capacity of designing the materials to give required properties. In this project torsional strength experiments have been conducted by varying mass fraction of SiC (3%, 7%, 10%, and 20%) with Aluminum. to attain maximum torsional strength. Various mechanical behavior also has to investigated.

***Index Terms*- Aluminium, hardening, high specific strength, high specific stiffness,**

I. INTRODUCTION

Composite materials are important engineering materials due to their outstanding mechanical properties. Composites are materials in which the desirable properties of separate materials are combined by mechanically or metallurgically binding them together. Aluminum (Al) is a silvery white and ductile member of the poor metal group of chemical elements. Silicon carbide (SiC) is composed of tetrahedral of carbon and silicon atoms with strong bonds in the crystal lattice. This produces a very hard and strong material. The high thermal conductivity coupled with low thermal expansion and high strength gives this material exceptional thermal shock resistant qualities. SiC ceramics with little or no grain boundary impurities maintain their strength to very high temperatures, approaching 1600oC with no strength loss. Properties of silicon carbide are low density, high strength, low thermal expansion, high hardness, and high elastic modulus. Metal matrix composites (MMCs), such as SiC particle reinforced Al, are one of the widely known composites because of their superior properties such as high strength, hardness, stiffness, wear and corrosion resistance. They can be widely used in the aerospace, automobiles industry such as electronic heat sinks, automotive drive shafts, or explosion engine components. The physical and chemical compatibility between SiC particles and Al matrix is the main concern in the preparation of SiC/Al composites. In this study, the casting method is carried out to prepare SiC particle reinforced Al MMC. The effect of weight percentage of the reinforced particles on mechanical behavior such as hardness and corrosion of the composites can be investigated.

Aluminium silicon composite

II. MATERIAL

SILICON CARBIDE

Silicon carbide (**SiC**), also known as **carborundum**, is a compound of silicon and carbon with chemical formula SiC. It occurs in nature as the extremely rare mineral moissanite. Silicon carbide powder has been mass-produced since 1893 for use as an abrasive. Grains of silicon carbide can be bonded together by sintering to form very hard ceramics which are widely used in applications requiring high endurance, such as car brakes, car clutches and ceramic plates in bulletproof vests.

Electronic applications of silicon carbide as light emitting diodes (LEDs) and detectors in early radios were first demonstrated around 1907, and today SiC is widely used in high-temperature/high-voltage semiconductor electronics. Large single crystals of silicon carbide can be grown by the Lely method; they can be cut into gems known as synthetic moissanite. Silicon carbide with high surface area can be produced from SiO_2 contained in plant material.

PROPERTIES

Properties of major SiC polytypes

Polytype	**3C (β)**	**4H**	**6H (α)**
Crystal structure	Zinc blende (cubic)	Hexagonal	Hexagonal
Space group	T^2_d-F43m	C^4_{6v}-$P6_3mc$	C^4_{6v}-$P6_3mc$
Pearson symbol	cF8	hP8	hP12
Lattice constants (Å)	4.3596	3.0730; 10.053	3.0730; 15.11
Density (g/cm^3)	3.21	3.21	3.21
Bandgap (eV)	2.36	3.23	3.05
Bulk modulus (GPa)	250	220	220
Thermal conductivity (W cm^{-1}K^{-1}) @ 300K (see [28] for temp. dependence)	3.6	3.7	4.9

III. EXPERIMENTAL METHODOLOGY

MIXING RATIO

In our project Aluminum and silicon carbide mixed below mentioned categories

Sample1:95%AL203 5%Sic
Sample1:90%AL203 10%Sic
Sample1:85%AL203 15%Sic
Sample1:80%AL203 20%Sic

MATERIAL REQUIREMENT FINDING METHOD

Specimen size-25 mm dia-
Length-300 mm
Volume-3.14/4*25^2*300*percentage of composite*density*percentage of excess of material

Sample1:95%AL203 5%Sic- 30% Excess for Runner,Riser&Slag
Aluminium-510gm
Silicon crbide-30.8gm
Sample2:90%AL203 10%Sic- 30% Excess for Runner,Riser&Slag
Aluminium-483gm
Silicon crbide-61gm

Sample3: 85%AL203 15%Sic-30% Excess for Runner,Riser&Slag

Aluminium-453gm
Silicon crbide-91.91gm
Sample4:80%AL203 20%Sic-30% Excess for Runner,Riser&Slag
Aluminium-429gm
Silicon crbide-122.5gm

CASTING PROCESS

INTRODUCTION

- Casting is a manufacturing process by which a liquid material is usually poured into a mold, which contains a hollow cavity of the desired shapes, and then allowed to solidify.
- The solidified part is also known as a casting, which is ejected or broken out of the mold to complete the process. Casting materials are usually metals or various cold setting materials that cure after mixing two or more components together; examples are epoxy, concrete, plaster and clay.
- Casting is most often used for making complex shapes that would be otherwise difficult or uneconomical to make by other methods.

- In this project we have used sand mold casting for produce the requirement size. **Sand casting**, also known as **sand molded casting**, is a metal casting process characterized by using sand as the mold material.
- It is relatively cheap and sufficiently refractory even for steel foundry use. A suitable bonding agent (usually clay) is mixed or occurs with the sand. The mixture is moistened with water to develop strength and plasticity of the clay and to make the aggregate suitable for molding.

  The term "sand casting" can also refer to a casting produced via the sand casting process. Sand castings are produced in specialized factories called foundries. Over 70% of all metal castings are produced via a sand casting process.

BASIC PROCESS

There are six steps in this process:

1. Place a pattern in sand to create a mold.
2. Incorporate the pattern and sand in a gating system.
3. Remove the pattern.
4. Fill the mold cavity with molten metal.
5. Allow the metal to cool.
6. Break away the sand mold and remove the casting.

DESTRUCTIVE TEST

TENSILE TEST & ELONGATION

Tensile Test

Friction processed joints are evaluated for their mechanical characteristics through tensile testing. A tensile test helps determining tensile properties such as tensile strength, yield strength, percentage of elongation, and percentage of reduction in area and modulus of elasticity. The welding parameters were randomly chosen within the range available in the machine. The joints were made with random parameters and evaluate tensile strength and burn off. Then the joints were made and evaluate the mechanical and metallurgical characteristics. The friction welded specimens were prepared as per the ASTM standards. The test was carried out in a universal testing machine (UTM) 40 tones FIE make.

Elongation

Deformation in continuum mechanics is the transformation of a body from a reference configuration to a current configuration. A configuration is a set containing the positions of all particles of the body. Contrary to the common definition of deformation, which implies distortion or change in shape, the continuum mechanics definition includes rigid body motions where shape changes do not take place A deformation may be caused by external loads, body forces (such as gravity or electromagnetic forces), or temperature changes within the body.

In a continuous body, a deformation field results from a stress field induced by applied forces or is due to changes in the temperature field inside the body. The relation between stresses and induced strains is expressed by constitutive equations, e.g., Hooke's law for linear elastic materials. Deformations which are recovered after the stress field has been removed are called elastic deformations.

In this case, the continuum completely recovers its original configuration. On the other hand, irreversible deformations remain even after stresses have been removed. One type of irreversible deformation is plastic deformation, which occurs in material bodies after stresses have attained a certain threshold value known as the elastic limit or yield stress, and are the result of slip, or dislocation mechanisms at the atomic level. Another type of irreversible deformation is viscous deformation, which is the irreversible part of visco elasticdeformation

TENSILE STRESS(Dia of the Rod-12MM**)-** BREAKING LOAD

Sample 1 Composite 1(Al95-Si5%):17.50KN
Sample 2 Composite 2(Al95-Si5%):9.5KN
Sample 3 Composite 3(Al95-Si5%):9.3KN
Sample4 Composite 4 (Al80-Si20%): 5.2 KN

TENSILE STRNGTH

RESULT

ELONGATION

Sample 1 Composite 1(Al95-Si5%):10mm
Sample 2 Composite 2(Al95-Si5%):8mm
Sample 3 Composite 3(Al95-Si5%):5mm
Sample4 Composite4 (Al80-Si20%): 3mm

DOUBLE SHEAR STRENGTH

"Double shear" would mean that there are two shear planes through the fastener, or that the total shear force is "split" into two locations on the fastener body. It's most common to see hitch pins loaded in double shear - a good example of this would be the cross pin for your trailer hitch.

Identification	Dia mm	CSA mm^2	DSL KN	DSS N/mm^2
T1	12	113	13.70	121.23
T2	12	113	15.25	134.95
T3	12	113	13.60	138.05
T4	12	113	17.30	153.09

You know how you put the shank of the ball mount into the receiver, and then retain it with a large (~5/8") pin? That pin is loaded in "double shear" - one plane on either side of the ball mount shank. Loading a fastener in "double shear" allows you to either not quite double the load on it, or to use a smaller fastener (since the load is split.)

You'll still want to select for the shear load to be placed on the shank instead of the threads (where the available shear strength is rather lower!) While I like to leave a good safe margin for anything (without testing, shear strength can be estimated as being 75% of tensile strength, and tensile strengths for ISO and SAE fasteners are easy to find,) this still gives guidelines for not wandering too far into overkill.

Note that there is a fairly significant reduction in strength when loading a fastener in shear over the threads (due to reduced cross-sectional area.) I don't have it to hand, but I've been planning a "fastener primer" that will explain just how you can figure the reduction in strength due to threading.

There is a similar reduction in strength when loading in tension - same reason. You use the minimum cross-sectional area at the thread root for strength calculations in tension, and you should be able to use the full nominal diameter in shear (unless you screwed up, and loaded it in shear across the threads...)

Identification	Dia mm	CSA mm^2	TL KN	TS N/mm^2
12	T1	113	17.50	154.86
12	T2	113	9.5	84.07
12	T3	113	9.3	82.30
12	T4	113	5.2	46.01

.2 DOUBLE SHEAR TES

RESULT

DOUBLE STRESS TEST (Dia of the Rod-12MM)

Sample 1 Composite 1(Al95-Si5%):13.70KN
Sample 2 Composite 2(Al95-Si5%):15.25KN
Sample 3 Composite 3(Al95-Si5%):15.60KN
Sample4 Composite4 (Al80-Si20%): 17.30KN

IMPACT TEST

A method for determining behavior material subjected to shock loading in bending, tension, or torsion. The quantity usually measured is the energy absorbed in breaking the specimen in a single blow, as in the Charpy impact Test, Izod impact Test, and Tension Impact tests also are performed by subjecting specimens to multiple blows of increasing intensity, as in the drop ball impact test, and repeated blow test.

Impact resilience and scleroscope hardness are determined in nondestructive impact tests. Further complication is offered by the choice of failure modes: ductile or brittle. Brittle materials take little energy to start a crack, little more to propagate it to a shattering climax.

Other materials possess ductility to varying degrees. Highly ductile materials fail by puncture in drop weight testing and require a high energy load to initiate and propagate the crack. Many materials are capable of either ductile or brittle failure, depending upon the type of test and rate and temperature conditions. They possess a ductile/brittle transition that actually shifts according to these variables

S.No	Material	IMPACT STRENGTH JOULES
1	95Al-5Si%	8
2	90Al-10Si%	9
3	85Al-15Si%	13
4	80Al-20Si%	15

EXPERIMENTAL CALCULATION OF MACHINING TIME & MATERIAL REMOVEL RATE

S.No	Material	Trial 1	Trial 2	Trial 4	Mean dia	BHN
1	95Al-5Si%	4.62	4.69	4.57	4.6	21
2	90Al-10Si%	4.47	4,39	4,46	4.4	24
3	85Al-15Si%	4.32	4.34	4.32	4.3	26
4	80Al-20Si%	3.67	3.61	3.52	3.6	38

Depth of cut=0.5
Length=60mm
Dia=25
Spindle speed-200 Rpm

HARDNESS TEST

BRINELL TEST

The Brinell scale characterizes the indentation hardness of materials through the scale of penetration of an indenter, loaded on a material test-piece. It is one of several definitions of hardness in material science.

Proposed by Swedish engineer Johan August Brinell in 1900, it was the first widely used and standardized hardness test in engineering and metallurgy. The large size of indentation and possible damage to test-piece limits its usefulness.

The typical test uses a 10 millimeters (0.39 in) diameter steel ball as an indenter with a 3,000 kgf (29KN; 6,600 lbf) force. For softer materials, a smaller force is used; for harder materials, a tungsten carbide ball is substituted for the steel ball. The indentation is measured and hardness calculated as:

Where:
P = applied force (kgf)
D = diameter of indenter (mm)
d = diameter of indentation (mm)

$$\mathrm{BHN} = \frac{2P}{\pi D\left(D - \sqrt{D^2 - d^2}\right)}$$

RESULT

ANSYS INTRODUCTION

S.No	Material	Machining Time sec	Material Removal Rate gm
1	95Al-5Si%	56.31	6.40
2	90Al-10Si%	60.12	5.45
3	85Al-15Si%	60.28	5.45
4	80Al-20Si%	80.01	4.73

ANSYS is a general-purpose finite-element modeling package for numerically solving a wide variety of mechanical problems. These problems include static/Dynamic; structural analysis (both linear and nonlinear), heat transfer, and fluid Problems, as well as acoustic and electromagnetic problems may be analyzed with finite element methods. The origin of the modern finite element method may be traced back to early 1900s when some investigators approximated and modeled elastic continua using discrete equivalent elastic bars. However, courant (1943) has been credited with being the first person to develop the finite element method.

In a paper published in the early 1940s, courant used piecewise polynomial interpolation over triangular sub regions to investigate torsion problems. The next significant step in the utilization of the finite element method was taken by Boeing in the 1950s when Boeing, following by others, used triangular stress elements to model airplane wings.

Yet it was not until 1960 that Clough (1975) made the term "finite element" popular. During the

1960s, investigators began to apply the finite element method to other area of engineering, such as heat transfer and seepage flow problems. Sienkiewcz& Cheung (1967) wrote the first book entirely devoted to the finite element method in 1967. In 1971, ansys was released for the first time.

Ansys is a comprehensive general purpose finite element computer program that contains over 100,000 lines of codes. Ansys is a capable of performing static, dynamic heat transfer, fluid flow, and electromagnetism analyses. Ansys has been leading FEA program for well over 20 year. The current version of ansys has been completely new look, with multiple window incorporating a graphic user interface(GUI), pull down menus, dialog boxes, and a tool bar.

Today, ANSYS can be found in used in many engineering field, including aerospace, automotive, electronics, and nuclear. In general, a finite-element solution may be broken into the following three stages.

PREPROCESSING: DEFINING THE PROBLEM

The major steps in preprocessing are

(i) Define key points/lines/areas/volumes,

(ii) Define element type and material/geometric properties, and

(iii) Mesh Lines/areas/ volumes as required.

SOLUTION: ASSIGNING LOADS, CONSTRAINTS, AND SOLVING

Here, it is necessary to specify the loads (point or pressure), constraints

(Translational and rotational), and finally solve the resulting set of equations.

POST PROCESSING: FURTHER PROCESSING AND VIEWING OF THE RESULTS

In this stage one may wish to see

(i) Lists of nodal displacements,

(ii) Element Forces and moments,

(iii) Deflection plots, and.

(iv) Stress contour diagrams or Temperature maps.

TERMS USED IN ANSYS

ELEMENT

In this finite element material the geometry is divided up into much like basic building block. Each element has node associated with it the behaviors of the element is define in terms of the freedom at the nodes.

FINITE ELEMENT MODELING (FEM)

The process of setting up a model for analysis typically involving graphical generation of the model geometry meshing it into finite elements definition material properties and applying loads and boundary condition.

FIELD PROBLEMS

Problems that can be defined by a set of partial deferential equation are field problems any such problem can be solved approximately by the finite element method.

DISCRETIZATION

The process of dividing geometry into smaller piece (finite element) to prepare for analysis that is meshing.

DEGREES OF FREEDOM

The number of equations of equilibrium for the system In dynamics the number of displacement quantities which must be considered in order to represent the effect of all of the significant martial forces.

Degrees of freedom define the ability of a given node to move in any direction in space.

There are six types of DOF for any given nodes:

- 3 possible translations (one each in the X, Y, Z direction)
- 3 possible rotations (one rotation about each of the X, Y and Z axis)

DOF are defining and restricted by the elements and constrains associated with each node.

DEFINITION OF STRUCTURAL ANALYSIS

Structural analysis is probably the most common application of the finite element method. The term structural implies civil engineering structures such as bridges and buildings, but also naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as crankshaft, machine parts, and tools.

MODEL GENERATIONS

The ultimate purpose of a finite element analysis is to re-create mathematically the behavior of as actual engineering system. In other words, the analysis must be an accurate mathematical model of a physical prototype. In the broadest sense, this model comprises all the nodes, elements, material properties, real constants, boundary conditions, and other features that are used to represent the physical system.

DIRECT GENERATION

On the plus side, direct generation

- It convenient for small or simple models.
- Provides you with complete control over the geometry and numbering of every node and every element.

IMPORTING SOLID MODELS CREATED IN UNIGRAPHICS SYSTEMS

As an alternative to creating your s
olid model within ANSYS, you can create them in your favorite UNIGRAPHICS system and then import them into ANSYS for analysis.

This has the following advantages:

- You can avoid a duplication of effort by using existing UNIGRAPHICS models to generate solid models for analysis.
- Engineers can use familiar tools to create their model.

ELEMENT INPUT

Many features are common to all ANSYS element library.

1. Element name.
2. Nodes.
3. Degrees of freedom.
4. Real constants.
5. Material properties.
6. Surface loads.
7. Body loads.
8. Special features.
9. Key points.

MESHING THE MODEL

The procedure for generating a mesh of nodes and elements consists of three main steps:

- Set the element attributes.
- Set mesh controls, ANSYS offers a large number of mesh controls, which you can choose from to suit your needs.
- Generate the mesh.

FREE OR MAPPED MESH

Before meshing the model, and even before building the model, it is important to think about whether a free mesh or a mapped mesh is appropriate for the analysis. A free mesh has no restrictions in terms of element shapes, and has no specified pattern applied to it.

Compared to a free mesh, a mapped mesh is restricted in terms of the element shape it contains and the pattern of the mesh. A mapped area mesh contains either only quadrilateral or only triangular elements, while a mapped mesh typically has a regular pattern, with obvious rows of elements. If you want this and or areas that can accept a mapped meshed.

MESH CONTROLS

The default mesh controls that the ANSYS program uses may produce a mesh that is adequate for the model you are analyzing. In these cases, you will not need to specify any mesh controls. However, if you do use mesh controls, you must set them before meshing your solid model.

Mesh controls allow you to establish such factors as the element shapes, mid side node placement, and element size to be used in meshing the solid model. This step is one of the most important of your entire analysis, for the decisions you make at this state in our model development will profoundly affect the accuracy and economy of our analysis.

The main goal of a finite element analysis is to examine hoe a structure or component responds to certain loading conditions. Specifying the proper loading conditions is, therefore, a key step in the analysis. You can apply loads on the model in a variety of ways in the ANSYS program. Also with the help of load step options, you can control how the loads are actually used during solution.

LOADS

The loads in ANSYS terminology include boundary conditions and externally or internally applied forcing functions.

STRUCTURAL

Displacements, forces, pressures, temperatures (for thermal strain), gravity.

THERMAL

Temperatures, heat flow rates, convection, internal heat generation, infinite surface.

Steps in FEA analysis

There are a number of steps in the solution procedure using finite element methods.

1) Specifying Geometry - First the geometry of the structure to be analyzed is defined. This can be done either by entering the geometric information in the finite element package through the keyboard or mouse, or by importing the model from a solid modeler like Pro/ENGINEER.

2) Specify Element Type and Material Properties - Next, the material properties are defined. In an elastic analysis of an isotropic solid these consist of the Young's modulus and the Poisson's ratio of the material.

3) Mesh the Object - Then, the structure is broken (or meshed) into small elements. This involves defining the types of elements into which the structure will be broken, as well as specifying how the structure will be subdivided into elements (how it will be meshed). This subdivision into elements can either be input by the user or, with some finite element programs (or add-ons) can be chosen automatically by the computer based on the geometry of the structure (this is called automeshing).

4) Apply Boundary Conditions and External Loads - Next, the boundary conditions (e.g. location of supports) and the external loads are specified.

5) Generate a Solution - Then the solution is generated based on the previously input parameters.

6) Postprocessing - Based on the initial conditions and applied loads, data is returned after a solution is processed. This data can be viewed in a variety of graphs and displays.

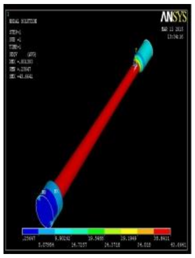
7)Refine the Mesh - Finite element methods are approximate methods and, in general, the accuracy of the approximation increases with the number of elements used. The number of elements needed for an accurate model depends on the problem and the specific results to be extracted from it. Thus, in order to judge the accuracy of results from a single finite element run, you need to increase the number of elements in the object and see if or how the results change.

8) Interpreting Results - This step is perhaps the most critical step in the entire analysis because it requires that the modeler use his or her fundamental knowledge of mechanics to interpret and understand the output of the model. This is critical for applying correct results to solve real engineering problems and in identifying when modeling mistakes have been made (which can easily occur).

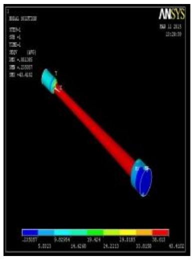
The eight steps mentioned above have to be carried out before any meaningful information can be obtained regardless of the size and complexity of the problem to be solved. However, the specific commands and procedures that must be used for each of the steps will vary from one finite element package to another. The solution procedure for ANSYS is described in this tutor. Note that ANSYS (like any other FEM package) has numerous capabilities out of which only a few would be used in simple beam problems.

DISPLACEMENT AND STRESS ANALYSIS

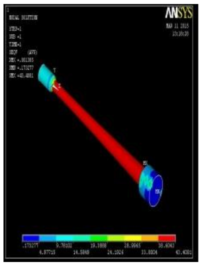
(i)95Al-5Si%



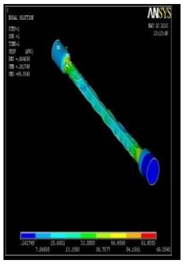
(ii)**90Al-10Si%**



(iii)85Al-15Si%



(iv) 80Al-20Si%



VALUE OF DISPLACEMENT AND STRESS

S.No	Material	Displacement	MAX STRESS
1	95Al-5Si%	0.001393	43.6641
2	90Al-10Si%	0.001385	43,4102
3	85Al-15Si%	0.001385	43.4081
4	**80Al-20Si%**	**0.004058**	**69.5545**

IV. CONCLUSIONS

Aluminium &Siliconcarbide alloy composite materials are widely used for a many number of applications like engineering structures, aerospace and marine application, automotive bumpers, sporting goods and so on. Based on our work we have found that the weight to strength ratio for aluminium silicon carbide of various proportional's has been analyze in the ANSYS and testing was carried out for the various proportional's. from these results that we have concluded 85%AL 15 %Sic is more effective when we compared to the other proportions .

REFERENCES

[1] H.E. Boyer, T.M. Gall, Metals Handbook, Desk Ed., American Society
for Metals, Metals Park, OH, 1991, pp. 20.16–20.21.
[2] G. Gusmano, A. Bianco, R. Polini, J. Mater. Sci. 36 (2001) 901–907.
[3] J.L. Johnson, R.M. German, Int. J. Powder Metall. 30 (1) (1994) 91–102.
[4] W.F. Wang, Powder Metall. 40 (4) (1997) 295–300.
[5] R. Jedamzik, A. Neubrand, J. Rodel, J. Mater. Sci. 35 (2000) 477–486.