MODELLING AND SIMULATION OF COMPOSITE MATERIAL PROPELLER SHAFT FOR ASHOK LEYLAND ENGINE,TRUCK MODEL 6DT120 USING FEM

Manish Rathore

Assistant Professor, mechanical engineering Dept. of Dr. APJ Abdul kalam UIT,RGPV Jhabua M.P.

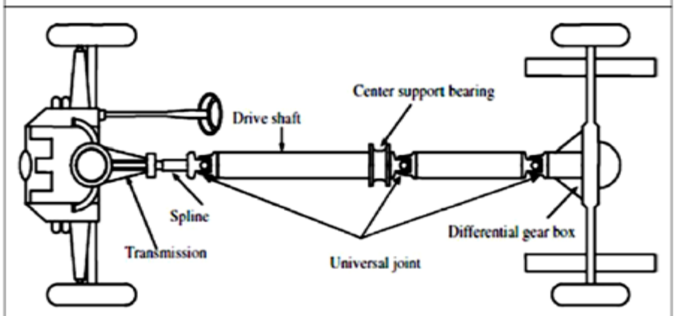
Abstract -Propeller shaft is s very important component in automobile, which is used for the transmission of power generated by the engine to the wheels. The power transmission takes place from the engine to the wheels mainly depends upon the weight of the propeller shaft, as the weight increases transmission capacity of the shaft decreases. To overcome this, steel shaft is replaced with the single piece composite shaft. When we use composite material the weight of the shaft decreases as a result the weight of the automobile also decreases. It can be achieved without an increase in cost and a decrease in quality and reliability.

Keywords: composite propeller shaft, weight reduction E-glass/epoxy UD, carbon /epoxy UD ,

1 INTRODUCTION

The present research work focuses on the design of an light weight propeller shaft using composite materials. now a day all automobiles(which are having front engine, rear wheel drive)have the transmission shaft as shown in fig.1. two -piece propeller shaft increases the weight of propeller shaft which is not desirable in today's market. the reduction in weight of the propeller system is advantageous in overall weight reduction in automobiles. Which is a highly desirable goal of the design engineer.

Figure 1: Conventional Two-Piece Steel Drive Shaft



1.1 Objective of the research work

1. To decrease the weight.
2. To increase the torque transmission capacity.
3. To study induced stress in propeller shaft
4. To study dynamic behavior of propeller shaft.

To replace conventional steel material two piece three universal joint drive shaft with composite materials.

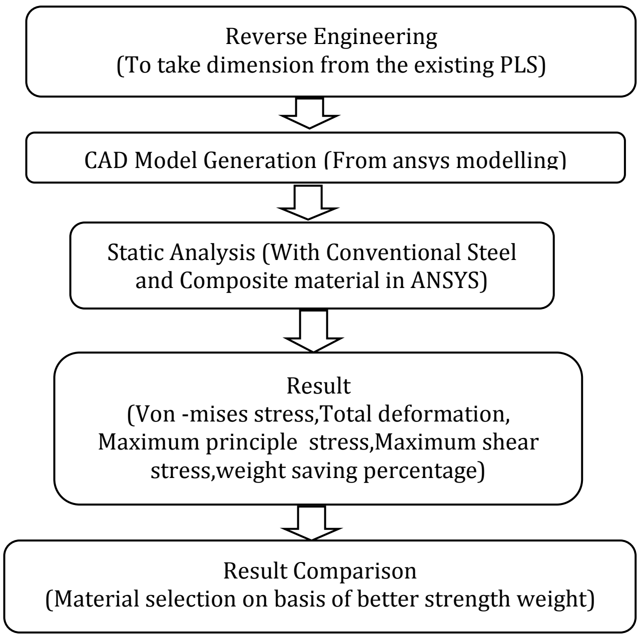
1.2 The specific properties of composites

- Low density
- High specific strength
- High specific modulus
- High thermal conductivity
- Good fatigue modulus
- Control of thermal expansion
- High abrasion and wear resistance

Composite are needed because modern applications require materials with strange combination of properties like low stiffness, high strength, abrasion and impact resistance.

2 Research work Methodology

For achieving the objective of the work a flow chart is prepared which shows various steps taken in to consideration.



3.Design data for design a CAD model of propeller shaft

S no.	Description	Notations	Value
1.	outer diameter	d	70 mm
2.	inner diameter	d	56 mm
3.	thickness of the propeller shaft	t	7 mm
4.	length of the propeller shaft	l	1800 mm

4 FEM Methodology

4.1 Preprocessing

4.1.1 CAD Modeling: Creation of CAD model by using any of the CAD modeling tools like Pro/Engineer, Catia, UG, Solid Edge, for creating the geometry of the part/assembly of which needs to perform FEA.

4.1.2 Importing the Geometry to FEA Package: Run the FEA package and import the CAD model/geometry in IGES format into the FEA package.

4.1.3 Defining Material Properties: In this step we have to choose the material which we are going to use for the FEA analysis of part/geometry. Material can be selected from FEA package engineering materials section, if particular is not available in the library than we can add the new materials with valid material properties in the material section.

4.2 Processing

4.2.1Meshing: Meshing is a critical and most important operation in FEA. In this operation the CAD model/geometry is divided into large numbers of small pieces called elements. The analysis accuracy and duration depends on mesh size of the geometry. With the increase in mesh size the finite element analysis speed increase but the accuracy decrease. Mesh convergence test can be performed to select the correct mesh size.

4.2.2 Defining Boundary Condition: Defining boundary condition is the most important process in FEA analysis because results of the FEA analysis is depend on the given boundary conditions, so this section require lot of study to giving correct boundary conditions. We have to tell the FEA package where we want to apply loads and where we want to fix the part.

4.2.3 Solution

This is the solution phase. In this section we tell the FEA package to solve the problem for the defined material properties, mesh size and boundary conditions. In this section we can select the parameters like stress, strain, deformation, contact tool, and can get the results for these parameters in single analysis.

4.3 Post Processing

Post-processing provides the tool with easy-to-use powerful result visualization Features. It provides an in depth view of data with visualization tools such as cutting planes, contouring, streamlines, line plots, data probes and animation. At the last finally the results could be studied on the post processing stage, where different physical variables such as stress, strain and pressure would be displaced and interpreted. can be viewed in various formats such as graph, value animations etc.

5 FEM Modeling of Composite material Propeller shaft

ANSYS is a feature-based, parametric solid modeling system with many extended design and manufacturing applications. Three dimensional model of propeller shaft are prepared by using 3D modeling software ANSYS as shown in fig.5.1

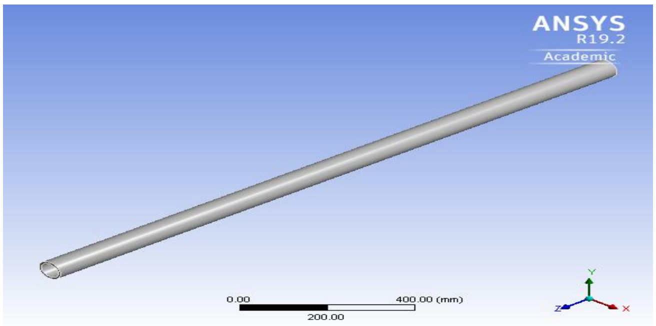


Fig 5.1 CAD Model of propeller shaft

5.1 FEM modeling of propeller shaft

5.1Introduction

In this section application of FE modelling is discussed, FEA tool is the mathematical idealization of real system. Is a computer based method that breaks geometry into element and link a series of equation to each, which are then solved simultaneously to evaluate the behavior of the entire system. It is useful for problem with complicated geometry, loading, and material properties where exact analytical solution are difficult to obtain. The static in FE analysis is carried out in this section.FEM general purpose software ANSYS -19.2 is used for the analysis of composite material propeller shaft.

5.2 Preprocessing

5.2.1 Composite propeller shaft Model

CAD modeling of the composite propeller shaft components is the first step of finite element analysis. CAD modeling of the composite propeller shaft using conventional steel EN45 and composite materials has been done according to the dimensions suggested by Ansys software.

5.2.2 Importing the Geometry to the ANSYS

CAD model of the composite propeller shaft is saved as WBPJ format in ansys modelling. Then, this WBPJ CAD file imported into ANSYS-19.2 for further analysis.

5.2.3 Material Properties

There are three composite propeller shaft on which the analyses are going to perform, one is structural steel propeller shaft and other two are composite propeller shaft. The mechanical properties of the structural steel material being used in this analysis are show in Table no-5.1 and the mechanical properties of composite material which can be taken as per ANSYS-19.2 material library.

Table 5.1 Mechanical properties of Structural steel

Material properties	**Value**
Density	7850 Kg/m^3
Young's modulus	200 Mpa
Poisson's ratio	0.3
Compressive yield Strength	250 Mpa
Tensile yield strength	250 Mpa
Tensile ultimate strength	460 Mpa

5.3 processing

5.3.1 Mesh convergence test

A check point is tested on the assembly by using mesh convergence test in order to simplify and justify the analysis result. In this process the von- mises stress level is tested on assembly by taking different size of element during meshing. With the assistance of ANSYS-19.2 software, the respective mesh sizes with corresponding von- mises stresses are given in the fig. 5.2. The moment value is same for each mesh size 9.3 mm. We observed that below the mesh size of 11mm there are small variations in the value of von- mises stresses. So the mesh size of 9.3mm is taken in this study for meshing of the composite propeller shaft.

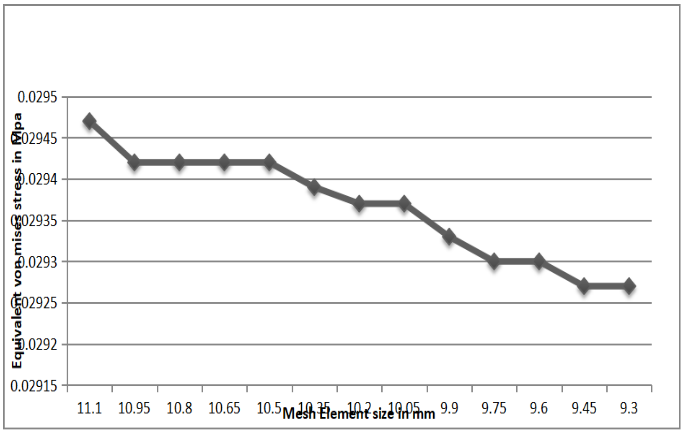


Fig -5.2 Variation in von Mises stress value with respect to the mesh size

linear elements are used for all the components of composite propeller shaft. linear elements better approximate the shape with minimum error as compared to brick elements. According to the mesh convergence test, Size of the linear elements are 9.3 mm for all the components of composite propeller shaft a total no. of 30885 nodes and 4900 elements are generated after the meshing. Meshed composite propeller shaft model is shown in fig. 5.3

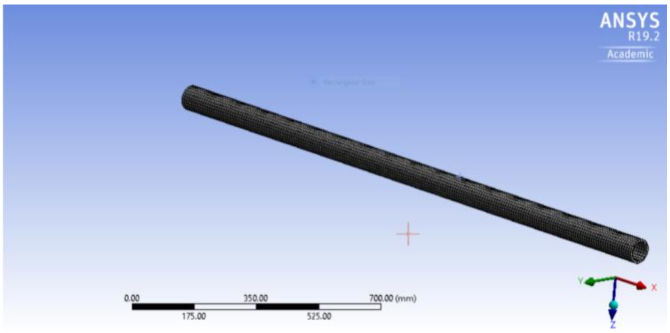


Fig.5.3 Meshed composite propeller shaft

5.3.2 Boundary conditions

1. Fixed support

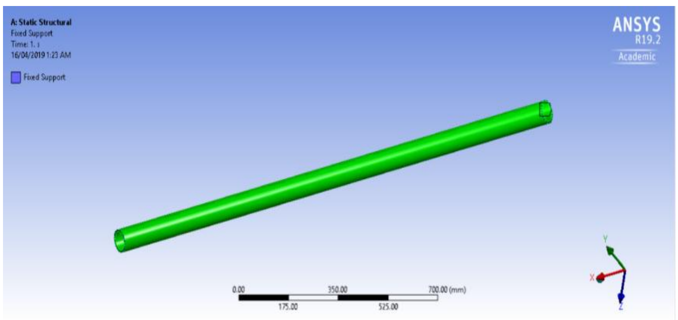


Fig-5.4 Applying fixed support one end of propeller shaft

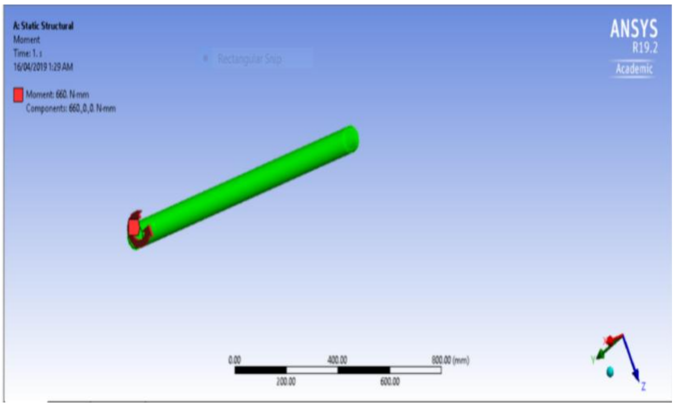


Fig. 5.5 Applying moment one end of propeller shaft

5.4 Post processing

STATIC ANALYSIS OF THE PROPELLER SHAFT

Static analysis of the composite propeller shaft (structural steel and composite material) is performed using ANSYS-19.2 software to find the von-mises stresses, maximum principal stresses, equivalent elastic strain and deflection. The Analysis involves the following discretization called meshing, boundary conditions . A virtual model of each propeller shaft is modeled separately, and then it is assembled together using ansys modelling (figure 5.1).This model is then imported into ANSYS- 19.2 for conducting static analysis. Same model is used for the static analysis with three different materials structural steel, E-glass/epoxy, and carbon/epoxy . These materials are selected in the Ansys-19.2 software by inserting the appropriate material properties.

In this section the structural steel and composite propeller shaft will be analyzed to see the various results from the static analysis The software used to perform the analysis is ANSYS -19.2,The different comparative results of steel propeller shaft and composite propeller shaft are obtained to predict the advantages of composite for a vehicle.

6 RESULTS AND DISCUSSIONS

STATIC ANALYSIS RESULTS

6.1Von-Mises Stress

Von-Mises stresses are widely used to check whether the design will withstand the given load conditions, using this information we can say design will fail, if the maximum value of von-Mises stress induced in the material is more than yield strength of the material. In this study it can be seen that the von-Mises stress is maximum towards the fixed end of the composite propeller shaft, and the value is less than yield point value of structural steel and considered composite materials. So, the design of composite propeller shaft is safe. Figure 6.1 shows the comparison of thickness verses stress for steel and composite propeller shaft. It is observed that the stresses increase linearly with the applied load maximum. Maximum stresses are exhibited by Carbon epoxy and minimum by the E-Glass epoxy.

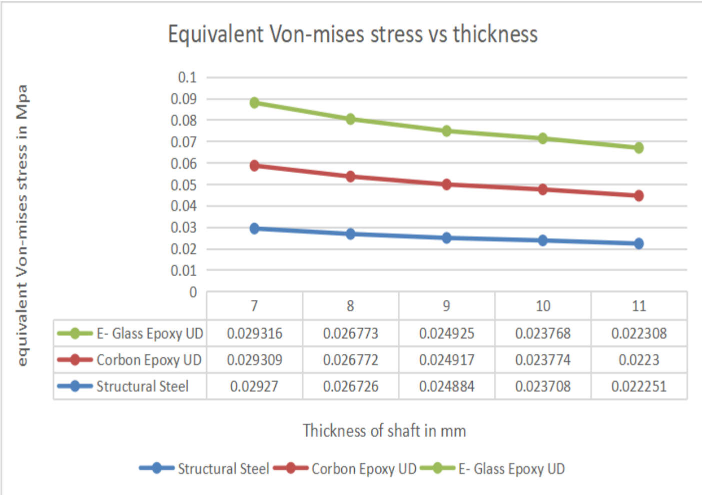


Figure- 6.1 Thickness vs equivalent (von-Mises) stress

6.2 Total Deformation

The rate of deformation is a function of the material properties, and the applied load depending on the magnitude of the applied stress and its duration. Figure-6.2 shows the comparison of thickness verses deformation of both steel and composite propeller shaft. It is found that the deformation in composite propeller shaft is higher than steel propeller shaft for the given thickness conditions. Maximum deformation is exhibited by E-Glass epoxy and minimum by the Structural Steel.

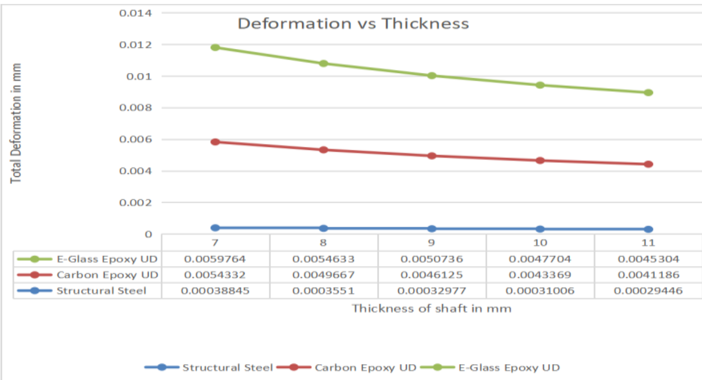


Figure 6.2 Thickness vs total displacement

6.3 Maximum principle stress

According to maximum principle stress criterion the material of the composite propeller shaft will be safe, if the ultimate tensile strength of the material is greater than maximum principle stress. Figure-6.3 shows the deviation of principle stress is very low at maximum thickness condition and the deviation of principle stress increases when the magnitude of applied thickness decrease. Maximum principal stresses are exhibited by Carbon epoxy and minimum by the Structural steel.

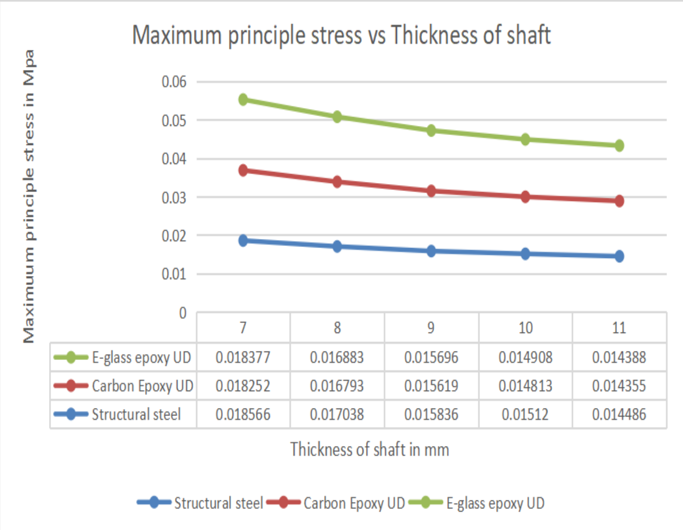


Figure 6.3 thickness vs maximum principal stress

6.4 Maximum Shear Stress

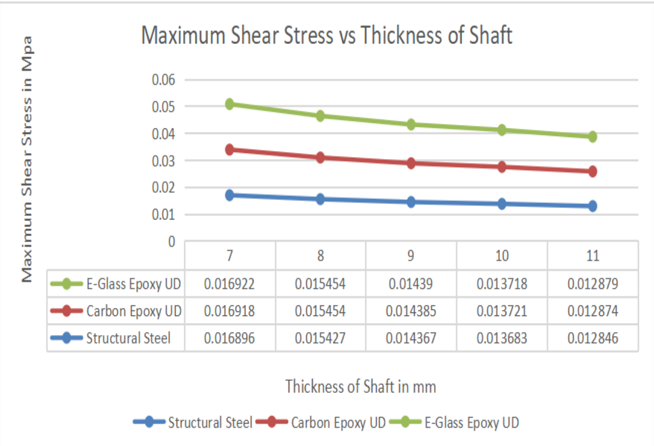


Figure 6.4 thickness vs maximum Shear stress

6.5 Percentage of weight saving for different propeller shaft materials.

Figure 6.5 shows the comparison of propeller shaft mass (Kg) for steel and other composite materials. From the fig. 6.5 it is easily observed that the weight reduction composite propeller shaft E-glass/epoxy mass is 4.9876 kg, carbon epoxy mass is 3.8404 kg. Table shows the % saving of weight by using composites instead off structural steel.

	Material	Mass of the propeller shaft	% weight saving
1	Steel EN45	19.576kg	-
2	E-glass/epoxy	4.9876 kg	74.52%
3	Carbon epoxy	3.8404 kg	80.38%

Table 6.1 % of weight saving for different materials

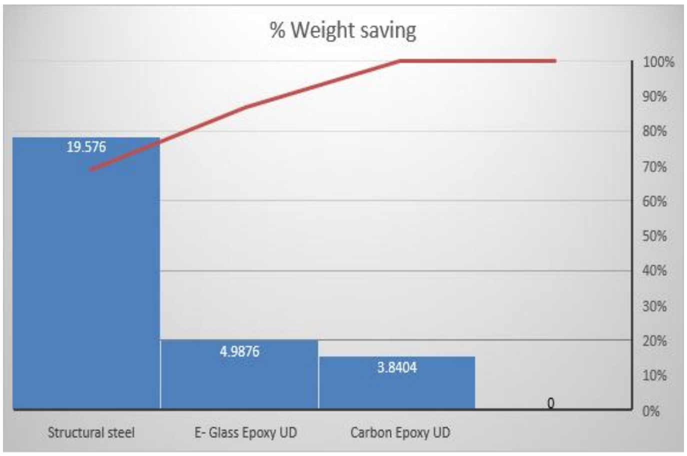


Fig.6.5 Comparison of % weigh saving of different propeller shaft

7 Conclusion and Future Scope

7.1 Conclusion

In this research work a propeller drive shaft is designed and analyzed for Ashok Leyland engine, truck model 6DT120.The propeller drive shaft is designed for the torque of 660N-mm. Theoretical calculations have been calculated for propeller drive shaft dimensions at varying thickness, by mathematical approach. In this work analysis has been done by taking materials Steel , E-Glass/Epoxy UD, Carbon/Epoxy UD. Static structural analysis are conducted on propeller drive shaft.

Following are the major outcomes:

-The generated stresses in the composite propeller drive shaft are much lower than that of the yield stresses and the stresses generated in structural steel.

-The composite propeller drive shaft is lighter and more economical than the conventional steel shaft with similar design specifications.

-It is observed that the weight of the composite propeller drive shaft made of E-Glass/Epoxy UD, is reduced by 74.52% compared to made of steel, by using material

Carbon/Epoxy UD it is reduced by 80.38% compared to made of steel.

-It is observed that the composite material shows more deflection and strain energy than that of steel, for less weight of the propeller drive shaft mechanical efficiency will be increased.

		Deformation in mm				
S. no.	Material	Thickness of shaft in mm				
		7mm	8mm	9mm	10mm	11mm
1	Structural steel	0.00038845	0.0003551	0.00032977	0.00031006	0.0029446
2	E-Glass /Epoxy UD	0.0059764	0.0054633	0.0050736	0.0047704	0.0045304
3	Carbon/ Epoxy	0.0054332	0.0049667	0.0046125	0.0043369	0.0041186

-Obtained results from the static analysis Carbon/Epoxy UD is better than E-Glass/Epoxy UD because its stresses are less than other.

-Using FEM software tool like ANSYS-19.2 Workbench prove the reliability of the validation methods based only on simulation, thereby saving time, and help to understand behavior of the composite propeller drive shaft.

7.2 Scope of future work

Following future recommendations can be added as an extension to this work:As analysis of composite propeller drive shaft and steel propeller drive shaft

is validated by comparing previously published data, so it is needed that one can perform experimental validation with manufacturing of actual prototype of composite propeller drive shaft. As this analysis is under static load condition, so one can go for dynamic loading condition. Dynamic , Model & Vibration analysis of propeller shaft may also be done.

REFRENCES:-

1. Atul Kumar Raikwar, Prof. Prabhash Jain & Raj kumari Raikwar -International Journal of Engineering Development and Research. Volume 4, Issue 4 ISSN: 2321-9939 -2016.

2. 6-Muni kishore, Jaligam Keerthi, Vinay kumar - International Journal of Engineering Trends and Technology, Volume 38 Number 6-August 2016

3. Sridhar, Dr. R. Mohan, R. Vinoth Kumar -International Journal of Scientific &Engineering Research, Volume 7, Issue 5,May-2016, ISSN 2229-5518.

4. Dr. R. Ganapathi, Dr. B. Omprakash, J.Vinay Kumar - International Journal of Latest Engineering Research and Applications, Volume – 02, Issue – 11,November 2017, PP – 24-28, ISSN:2455-7137.

5. Ashwani Kumar, Neelesh Sharma, Pravin P Patil - Proceedings of the World Congress on Engineering 2017 Vol II WCE2017, July 5-7, 2017, London, U.K.

6. Vinodh Kumar S, Sampath V and Baskar P - Research Journal of Recent Sciences. Vol. 4(9), 9-15, September (2015),ISSN 2277-2502