International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075 (Online), Volume-8 Issue-5, March 2019

A Structural Displacement and Material Variation in Fiber Optic Sensor for Acoustic Sensitivity

Prashil M. Junghare, Cyril Prasanna Raj P., T. Srinivas

Abstract – In this article, the fiber optic acoustic hydrophone made with a composite structure design. The structure is designed with different material layers and structural properties. The sensing plastic spool is outlined certain to existing harmonic frequency ranges at 0.2 to 2.5 KHz. In this research paper, the major pivot was to prototyped the mandrel which remained at a gap ranging from 20 to 200 meter underwater accompanied by varying constructional and material properties having non-identical friction coefficient. A design has much rapport with two fiber optic layers which wounded over center of length of mandrel. However the magnitude of sensitivity is determined to be -20.172dB for the applied static force of 0.2 Mega Pascal.

Pre-managing of sensing composition is performed using Hyper Mesh model, inspection is performed in ABAQUS CAE software and conceptualization of results in Hyper View model appliance. On any occasion, the force is registered on the mandrel, phase variation in light together with path length of fiber change which measures sensitivity mathematically.

Keywords—Hydrophone, Friction Coefficient, Static Pressure, Hyper Mesh, Hyper View

I. INTRODUCTION

Interferometry is the one of the key type to design a hydrophone mandrel. In interferometry, with a medium of optical waves is sourced and superimposed for extracting the knowledge about waves. Optical fiber medium uses the effect of interference. Here an input beam split in two sub beam having equal intensities with the use of beam splitter or coupler and some of these beams are exposed to random external influences like as path length change or core refractive index change in fiber medium. The beams are recombined by a beam coupler. In the design two symmetric mandrels are used, one is considered as reference plastic spool which provides results under ideal environment and another is a sensing device, whose output is to be measured. Reference mandrel is designed such that, it does not respond to any of the environmental effects. Sensing device is proposed to place in underwater region to listen the acoustic sound; thus it is familiar as hydrophone device. The motive of planting sensing insulated plastic spool in underwater is to detect the pressure alterations known as acoustic pressure.

Manuscript published on 30 March 2019.
***Correspondence Author(s)**
Prashil Manohar Junghare, M.Tech, (Communication Engineering Branch) VIT University, Vellore. Ph.D. (pursuing) Communication Engineering PRIST University,
Cyril Prasanna Raj P. Ph.D. Degree from Conventry University, UK (United Kingdom).
T. Srinivas, Professor Indian Institute of Science in ECE department. Ph. D., IISc, Bangalore.

Different elasto-optic materials are considered in designing hydrophone they are Nylon, Aluminum, Polystyrene, Optic fiber and polyurethane. The design has to be comfortable to suspend the sensing mandrel under the water at a depth of 200m. With requirements, the depth may be varied. The final output required from the hydrophone in terms of sensitivity is expressed in decibels. To get the prime sensitivity, a measurable extent of parameters such as effective length,

diameter and thickness are varied. Finite element investigation of plastic mandrel is performed to know the recommended design [1].

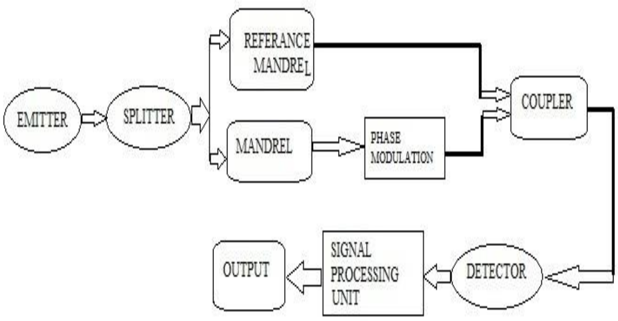

Fig. 1. Phase change detection process

Basic hydrophone mandrel is shown Fig. 1. Light gleam from transmitter is move through optic fiber that is bending in to Polystyrene and Polyurethane elastic coating. As light circulate through optic fiber, once the load is applied on the sensor, change in angle is takes place in sensing part and signal through reference mandrel without change are coupled using a coupler and passed to the demodulator where the output is translated to electrical form. Signal processing unit, thus it processed in signal processing unit to detect the acoustic pressure via sensing the optical phase difference between both mandrel to get the characteristics of incoming signal [2].

Highlight a section that you want to designate with a certain style, and then select the appropriate name on the style menu. The style will adjust your fonts and line spacing. **Do not change the font sizes or line spacing to squeeze more text into a limited number of pages.** Use italics for emphasis; do not underline.

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)

II. FINITE ELEMENT ANALYSIS

Eventually each and every phenomenon in nature related to biological, geological or mathematical can be defined with aid of laws of physics. Analysis of phenomenon consists of two major tasks,

- Mathematical formulation of physical process.
- Numerical analysis of mathematical model.

Historical background related aspects are required for the mathematical formulation of physical process. Formulation results in form of mathematical statements, often differential equations relating quantities of interest in understanding and design of physical process. Assumptions related to proceedings of work carried are needed for the development of physical process. Derivation of governing equation for complex problem is not so difficult. Finding their solution by exact analysis is a time consuming job. For idealized and simplified situations analytical solution can be easily obtained [3].

The problems with complex geometry, complex material internal properties and boundary conditions finding analytical solution will be difficult. For such situations analysis with approximate methods provides alternative way to find solution for the problem. Among the approximate methods finite difference methods and variational methods are frequently in use. Different variational methods are Rayleigh-Ritz method and Galerkin method. These two methods approach solution in different manner. They differ from each other in the choice of integral from weight function and approximation method.

The traditional variational methods are replaced by finite element method by providing symmetric procedure for the derivative of the approximation function over sub-regions of the field. This method is highlighted with three basic functions which makes this method unique over other methods. First one is geometrically complexity of problem is divided into number of geometrically simple problems called as finite element. Second is an approximation function is derived from continuous function which represents linear combination of algebraic polynomials.

Third one is algebraic relations among undetermined coefficients, i.e. nodal values are obtained by satisfying the governing equation over each element. Thus Rayleigh-Ritz method provides an application in form of finite element method. This approach covers from entire complex problem to small part of the problem and from small part of the problem to entire complex problem. The analysis procedure is thus simplified and the quantity of data to be handled will not depend upon the number of smaller bodies into which the original body is divided.

This method finds majority of applications in the field of solid mechanics, thus problems solved by using this method are tackled by three approaches.

- Displacement method: Here displacement is considered as primary unknown quantity and given first preference.
- Equilibrium method: Stresses are considered as primary unknown and first priority is given to stresses.
- Mixed method: Here both displacement and stresses are considered as unknown quantities.

Mainly there are three steps in analysis

1. Preprocessor
2. Processor
3. Post processor

III. PREPROCESSOR WITH HYPERMESH 12.0

3.1 Preprocessing Structure

The motive of preprocessing is to evolve an accurate finite element meshing procedure with worthy material layers properties and proper boundary conditions in terms of pressure. The geometry is subdivided into elements by using finite element mesh that provides accuracy in the analysis. There are two kinds of elements used for meshing one is 2D elements and 3D elements, 2DS elements can be of plane stress, asymmetric and plane strain conditions, and where for 3D solid analysis the element will have physical thickness in all three dimensions. For 2D elements we have to assign thickness value after creating the element, but for 3D element is having thickness when element is created. The degree of freedom at each node is assigned. Usually solid elements have three translation degrees of freedom at every node [4].

Type of analysis will decide the assignment DOF at every node. In FEA meshing, it will consume more time than for analyzing. For linear static analysis requires following data young’s modulus (elastic modulus), Poisson’s ratio and density. When the output is required in the form of strain the yield stress at initial position and yield stress and plastic strain at final position is assigned as per the material used.

3.1.2 Model Meshing by Altair’s HYPER MESH 12.0

HYPERMESH is among the one high-performance finite elements pre-processor, post-processor for major finite element technique solvers, which allows engineers and designers to accomplish the design considerations and condition in a visual and highly predictive interactive environment. HYPERMESH’S tool is easy to learn and supports the direct use of CAD design and existing finite element models, providing robust interoperability and efficiency.

Advanced tools in automation that allows users to design optimistic flow of mesh for free geometry can be achieved. Quality criteria help to clear and satisfy almost all requirements so that compromising with quality is avoided. Generate mid-surfaces for all kind of models that have varying thickness. Hyper Mesh 12.0 provides multi-features to strengthen the design tool competitiveness in specific domain to improve performance analysis. Automation of Computer Aided Engineering (CAE) design model build and simulation processes.

New modified and advanced features related engineering design for elements have been added to enhance Hyper Mesh’s shell element (2D) and solid element (3D) meshing abilities that results in significantly avoids the time required for building the model. For further reduction of time new type of connections are available to provide better flow of elements, that are used to gain a high-quality mesh flow of fully assembled model. The output file consists of two types data such as model data another is history data. Model data gives information on geometric parameter and history data contains loads and load steps.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

3.1.2 C3D20 elements features capabilities

In case of 3D modeling with solid elements C3D20 elements are used to define the geometry for finite element analysis. From Fig. 2, C3D20 element consists of 20 nodes that have 3 degrees of freedom at each node. Here in this type, accuracy will be less in the calculation part of contour integrals for bending stress in shell. If contour integrals values are needed where the bending stress are important, we can use second-order solid elements (C3D20) at the crack tip region where the integral of shell is evaluated instead of shell element [5].

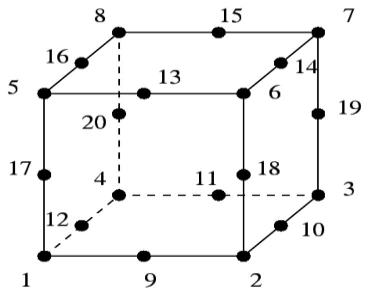

Fig. 2. Quadratic element (C3D20, 20 nodded brick)

3.1.3 Meshing

1. The Mesh tool helps in creating 2D shell mesh directly geometry and thicknesses can be assigned to the relevant elements.
2. Solids mesh provides the auto hexahedron and/or pentagonal mesh for sheet metal or plastic surfaces parts.
3. Mesh is allowed to project, reflect and rotate an existing mesh to the specified length.
4. Mesh translates helps in shifting the same mesh pattern another specified location.
5. Quality Index cleanup tools can be used to swap and clear war-page and Jacobins.

3.2 Processing with ABAQUS 6.10

In the present trend ABAQUS is one of the powerful engineering simulation program suites. Using this suite one can solve wide range of problems ranging from relatively small and simpler linear analyses to complicated non-linear analysis [16]. ABAQUS consists of a vast element library that can be used while studying different (other than stress/ displacement) problems. Analysis such as mass diffusion, heat transfer, and thermal management of electrical component (coupled thermal-electrical analysis), acoustics, solid mechanics (coupled pure fluid-stress analysis) and piezoelectric analyses can be done with the help of ABAQUS. In handling of highly complicated nonlinear analysis the user is supposed to provide only engineering data such as material behavior, geometry of the structure, boundary conditions and loading conditions. While dealing with nonlinear analysis ABAQUS automatically choose appropriate load increments and convergence tolerance [6].

3.2.1 ABAQUS 6.10 Capabilities

Structural analysis:

Different types of structural analyses can be done with ABAQUS suites are

Static analysis:

This type of analysis is used to determine the displacement, stresses strains etc. under static loading condition. Here both linear and nonlinear static analyses can be performed. Plasticity stress stiffening, large deflection, large strain, contact surface and hyper-elasticity etc. will come under nonlinear static analyses.

Modal analysis:

This type of analysis is used to determine the natural frequencies and mode shapes of a structure. Different modes extractions are available to solve the problems.

Dynamic analysis:

This type of analysis is performed to determine the response of a structure that varies arbitrarily with respect to the time-varying loads. Plasticity stress stiffening, large deflection, large strain, contact surface and hyper-elasticity etc. will come under nonlinear static analyses.

Thermal analysis:

This kind of analysis plays an important role in design of many engineering application, including internal combustion engines, turbine, heat exchangers, turbo chargers, fluid temperature variations and heat transfer in electrical components. In many cases many engineers follow thermal analysis with a stress analysis to calculate thermal stresses (i.e. stress caused by thermal expansion or contraction).

Modeling fracture mechanics:

This type of analysis is performed to know and study the crack parameters in structure. Here crack in a quasi-static model by selecting regions from the model that will be used to compute contour integral estimates. In history output, output database ABAQUS/standard writes the values of contour integrals during analysis only. There are different types of contour integral calculation

- J-integral (for Elastic-Plastic)
- C_t-integral (for creep)
- T-stress (for linear materials)
- Stress intensity factor for linear homogeneous materials and for interfacial cracks lying on the surface between two linear homogeneous materials.

IV. MODELING AND DESIGN

Steps Involved in Modeling and Design

- Preparation of CAD model using CATIA V5 R20.
- Meshing the model by using Altair's HYPERMESH 12.
- DECK prepared by using Altair's HYPERMESH 12.
- Analysis and synthesis is performed in ABAQUS CAE 6.10 solver.
- Required output such as strain values are viewed either using Altair's HYPERVIEW or ABAQUS CAE 6.10.

Following table 1 gives every detail related to discretization the model. The details such as total number of elements, material used, area occupied by the elements and thickness for the same.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

Fig. 2, suggested the mesh assembly which contains 46266 solid elements. The total area covered by these elements pixel is 76036.56mm^2 and the radial thickness is 41.72mm excluding the hollow space. The volume occupied by these solid elements is 31117.624mm^3 and the mass of the assembly is 4.205*10^{-4} Ton [7].

TABLE 1. Element and Area Details

Sl/no	Material	Number of elements	Area occupied (mm^2)	Thickness (mm)
1	Nylon	744	4430.123	1.5
2	Aluminum	6138	10102.437	10
3	polystyrene	26784	26635.680	20
4	Optic fiber	3600	13901.519	0.22
5	polyurethane	9000	20966.297	10
Total Bench Assembly quantity		46266	31117.624	

For a finite element model it is necessary to assign material properties. Fig. 2 shows the structural Material properties such as Density, Elastic modulus, Poisson's ratio, yield stress and plastic strain at two steps. Material properties structural dimensions are plotted with the following values and are assigned.

Now groups are to be created, i.e. surface elements are to be created for individually as required. Surface elements are created to provide contact between adjacent surfaces, so that results accuracy will increase.

TABLE 2. Material Properties

Sl/no	Material	Density (ton/mm^3)	Elastic modulus (ton/mm^2)	Poisson's ratio (ton/mm^2)
1	Nylon	1.15*10^{-9}	2*10^3	0.4
2	Aluminum	2.705*10^{-9}	0.69*10^5	0.33
3	polystyrene	1.05*10^{-9}	0.8*10^3	0.22
4	Optic fiber	2.2*10^{-9}	0.60*10^5	0.19
5	Polyurethane	1.1*10^{-9}	1*10^3	0.22

Usually surface elements contact the tolerance of 0.0001mm is obtained. There after contact pairs are created with friction properties shown in table 2.

After creating contact pairs with the above mentioned friction properties, now contact controls are to be created with interface properties which are having automatic tolerance and stabilize factor of 0.01. Now element sets and node set are to be created, because rather than applying load on each node it is better to create an element set. When the loads are applied on element set the time taken during the analysis will be reduced to maximum extent. While constraining the boundary, it is will be preferred to go with constraining the node set that reduces complexity which is the main purpose of designing [8].

TABLE 3. Friction coefficient and contact pairs

Sl/no	Contact pair	Contact between the surfaces	Friction coefficient
1	Contact pair 01	Nylon outer layer and Aluminum inner layer	0.2
2	Contact pair 02	Aluminum outer layer and Polystyrene inner layer	0.45
3	Contact pair 03	Polystyrene outer layer and Optic fiber inner layer	0.33
4	Contact pair 04	Optic fiber outer layer and polyurethane inner layer	0.45

Creating the rigid will helps in constraining single node rather than all the nodes. Load collectors will give the brief information related to the type of load applied and location. Here in this load is applied on the surface element on the outermost part of polyurethane surface.

V. RESULTS AND DISCUSSION

In result section, static load analysis with steady pressure is observed to the mandrel places at underwater surrounding.

5.1. Discussion of Results with Respect to Pressure Load

The displacements under periodic specific loads with respect to time increments are plotted in Fig. 3.

TABLE 4. Displacement v/s pressure load

Sl/no	Pressure load (MPa)	Displacement (mm)
1	0.2	3.699*10^{-3}
2	0.4	7.940*10^{-3}
3	0.7	1.431*10^{-2}
4	1.15	2.386*10^{-2}
5	1.825	3.818*10^{-2}
6	2	4.190*10^{-2}

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)

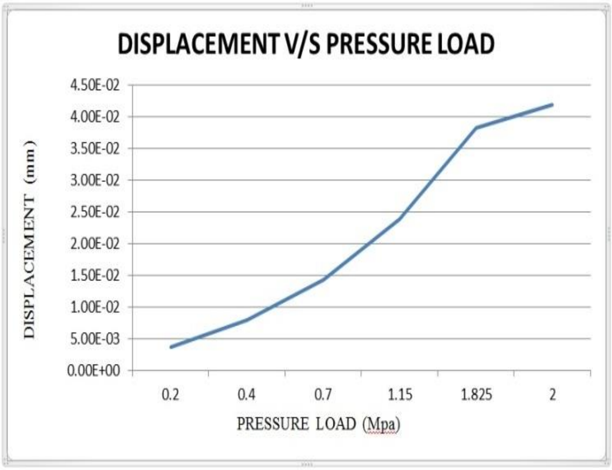

Fig. 3. Displacement v/s pressure load

The plot indicates that when the maximum load of 2MPa is applied displacement of $4.19*10^{-2}$ is obtained and also periodic increments are tabulated in table 4.

5.1.1 Stress strain distribution

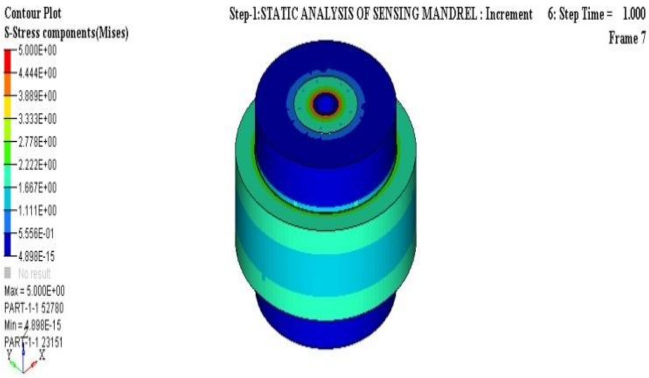

Fig. 4. von mises stress distribution structure

From the static analysis of sensing mandrel, von misses stress distribution at pressure load of 2Mpa is shown in Fig. 4. Also Fig. 5 represent the plot of stress v/s step increments, it is seen that stresses are spaced linearly with respect to step increments. The maximum von-misses stress observed is 5MPa [9]-[10].

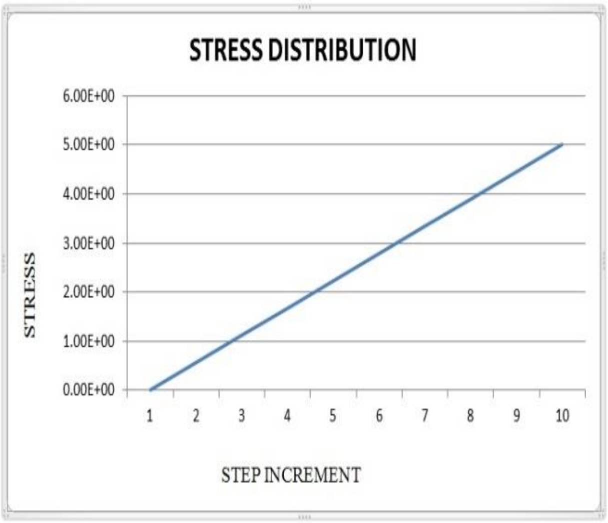

Fig 5. Stress v/s step increments

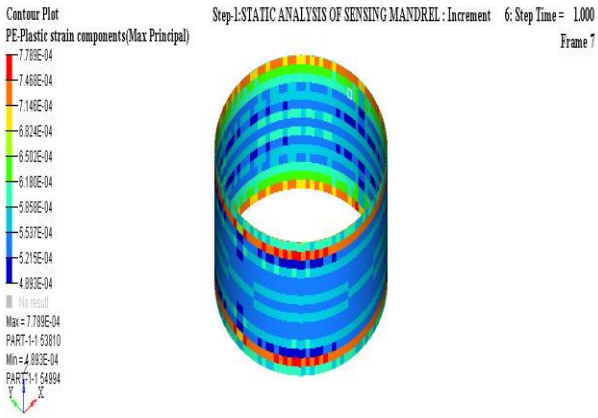

Fig. 6. Maximum principal strain distribution

TABLE 5. Stress strain distributions

Sl/no	Step increment	Von mises stress (MPa)	Maximum principal strain
1	1	$4.898*10^{-15}$	0
2	2	$5.556*10^{-1}$	$8.655*10^{-5}$
3	3	1.111	$1.731*10^{-4}$
4	4	1.667	$2.596*10^{-4}$
5	5	2.222	$3.462*10^{-4}$
6	6	2.778	$4.327*10^{-4}$
7	7	3.333	$5.193*10^{-4}$
8	8	3.889	$6.058*10^{-4}$
9	9	4.444	$6.924*10^{-4}$
10	10	5.000	$7.789*10^{-4}$

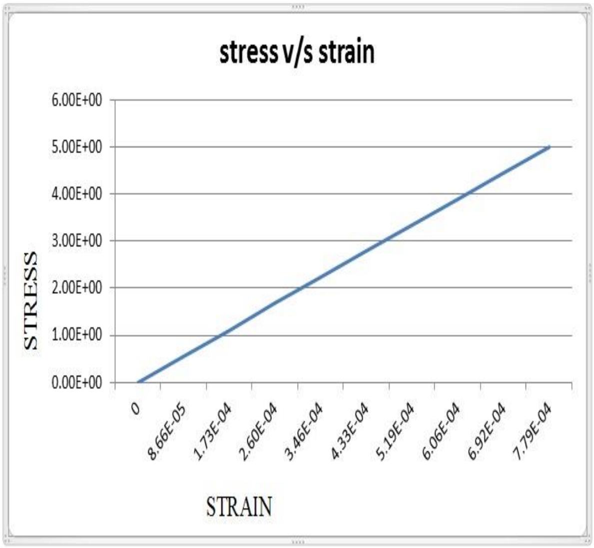

Fig. 7. von misses stress v/s max principal strain

Also maximum principal strain at pressure load of 2Mpa is shown in Fig. 7 and Fig. 8, where the highest strain produced is $7.789*10^{-4}$ which can be seen from table 5.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

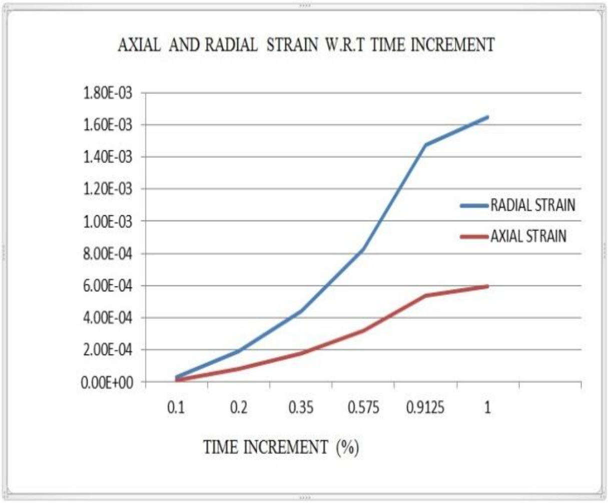

Fig. 8. Axial and Radial strain v/s time increment

Whenever the pressure is applied on the outer surface of mandrel (on surface element), phase changes in light happens. In addition to this phase change of light, strain is produced. By using these strain values, sensitivity of mandrel is calculated mathematically. Results considered in this model are Displacement model, axial strain model, radial strain model and stresses in the model [11]-[12].

VI. CONCLUSION

In this paper, fiber optic sensing mandrel with Michelson interferometry technique has studied. By changing the material and structural properties, still more scope of improvement is applicable for low pressure acoustic sensitivity.

REFERENCES

1. Yingying Wang et.al, Atlantis Press,Simulation, "High-Sensitivity Hydrophone Based on ANSYS", International conference, MEMS 2012 Published page no. 797-699.
2. S. Africk et. Al, "Design studies with fiber optic hydrophones," Report No. 4658, Bolt, Beranek & Newman, Inc., Cambridge, Mass, 1981.
3. R. Hughes,"Static pressure sensitivity amplification process in interferometric fiber optic hydrophones, "Applied Optics, 19(1), 1 (1980)
4. Peter Shajenko, James P. Flatley "On fiber optic hydrophone sensitivity" Naval Underwater system Center, New London Laboratory, New London, Connecticut 0632
5. M. Young, The Technical Writer's Handbook. Mill Valley, CA: University Scie,. June 1999. nce, 1989.
6. G. B. Hocker, "Fiber Optic sensing of pressure and temperature" Applied Optics, 18(9), 1445 .
7. Zefeng Wang, College of Photoelectric Science and Engineering, National University of Defense Technology, 410073, P. R. China)
8. N. Stockbridge, "Planar Fiber Optic Hydrophones, Winfrith Technology Centre, Winfrith Newburgh, Dorchester, vol 6619, 661907-2, 2007.
9. M Shahrieel and M Aras, "Development of hydrophone sensor system for autonomous underwater vehicle application", Second International Conference on Engineering and ICT, February 2010.
10. Mohd Rizal Arshad, Indian Journal of Marine Sciences, vol 38, 267-273, September 2009.
11. Jong-in Im, "A Finite Element Analysis of an Interferometric optical fiber hydrophone", American Journal of Engineering Research, June 1999.
12. A.N. Lagakos and T.R. Hickman, "Planar flexible fiber-Optic acoustic sensors", Proceedings of the 34th Midwest Symposium on Circuits and Systems, vol 56, 666-671,May 1991.

AUTHORS PROFILE

Prashil Manohar Junghare has completed his M.Tech from VIT University, Vellore in Communication Engineering. He is currently pursuing Ph.D. in Communication Engineering Branch at PRIST University, Vallam, Thanjavur. He has published several research papers in reputed journals. He has received Best researcher award two times.

Cyril Prasanna Raj P. has completed his Ph.D. Degree from Conventry University, UK (United Kingdom). He has published several research papers in reputed journals. Also He has published two books and awardees for research work

T. Srinivas is a Professor from Indian Institute of Science in ECE department. He has completed his Ph. D. From IISc, Bangalore. He has published many papers in Scopus and SCI Journals. He is also awarded by many institutions for his outstanding contribution in research and academics.