Journal of Materials Science & Surface Engineering Vol. 3 (4), 2015, pp 298-302

Contents lists available at http://www.jmsse.org/

Journal of Materials Science & Surface Engineering

# Mixed Mode Fracture of Through Cracks In Nuclear Reactor Steam Generator Helical Coil Tube

Ajay Krishna V. S.[1*] and H. V. Lakshminarayana[2]

[1]*Lecturer, Department of Mechanical Engineering, A. P. S. College of Engineering, Somanahalli, Bangalore – 560082, India.*
[2]*Professor, Department of Mechanical Engineering, Dayananda Sagar College of Engineering, Bangalore-560079, India.*

Article history

Received: 15-June-2015
Revised: 07-Aug-2015
Available online: 01-Oct-2015

**Keywords:**
Mixed Mode Fracture,
Helical Coil Tube,
Finite Element Analysis,
3MBSIF,
Membrane and Bending SIF,
Strain Energy Density Theory

## Abstract

Helical coil tubes are widely used in nuclear reactor steam generators. The main advantage of helical coil tubular configuration is its high compactness, adaptibility to cylindrical shape and higher heat transfer coefficient than straight pipe shell and tube steam generator. A critical assessment of structural integrity(strength, stiffness and durability) is often based on fracture mechanics analysis. Fracture is a failure mode due to unstable crack propagation. Fracture mechanics provides a methodology for prediction, prevention and control of fracture in materials, components and structures. This paper presents a refined finite element model and a special purpose subprogram to determine mixed mode membrane and bending stress intensity factors for arbitraily located and oriented crack in helical coil tube. The proposed finite element model is implemented using commercial FEA software ANSYS and stress intensity factors are evaluated using 3MBSIF. The methodology is validated using bencmarks, a set of standard test problems with known target solutions. Parametric studies are carried out to study the effect of crack location, orientation on the stress intensity factor values. Strain Energy Density theory of fracture is used to predict mixed mode fracture i.e., direction of crack growth and pressure load at which fracture occurs for a specified location and crack length for all possible orientations of crack.

*The work had been presented at an international conference* ***Fatigue Durability India 2015****, 28-30th May 2015, JN TATA AUDITORIUM, Indian Institute of Science, Bangalore.*

## Introduction

A steam generator in a nuclear power plant is a heat exchanger specifically designed to transfer heat from a coolant to water, producing steam that is used for power generation. Presently, helical coil tubes are being used as secondary flow paths where water gets converted to steam. The main advantage of using this type of configuration is its high compactness, adaptability to cylindrical shape and an excellent behaviour in the presence of thermal expansions. The helical coil tube has a 16-43% higher heat transfer coefficient than a straight pipe shell and tube heat exchanger[1-3]. A critical assessment of the structural integrity (stiffness, strength and durability) of the steam generator helical coil tube is of prime importance. Damage tolerance design methodology based on fracture mechanics is the only design methodology to predict and avoid the failure of the structure[4].

Cracks are unavoidable in structures. The fracture of a component or structure begins at crack tips. A crack can trigger a local failure at lower load level[5-6]. Fracture mechanics enables design engineers to approach the problem of fracture safe design in a more rational manner. In design, consideration is now given to likelihood that a new structure contains flaws introduced during the processing of the basic material, during fabrication of components and during assembly process. Fracture mechanics analysis involves the determination of stress intensity factors at the crack tip. Therefore, an accurate determination of crack tip stress intensity factors in a given structure and application of strain energy density theory to predict fracture and critical pressure at which the structure fail for a particular location and orientation of crack is essential to the development of safe and reliable designs[7-9].

The main objective of this study is to develop a refined finite element model and new post processing subprogram to determine mixed mode membrane and bending stress intensity factors for arbitrarily located and oriented through cracks in a helical coil tube subjected to internal pressure loading. To accomplish this, finite element modelling using ANSYS, a commercial FEA software and development of new post processing sub program 3MBSIF to compute the stress intensity factors posteriori is presented. The methodology is validated using benchmarks. Parametric study for arbitrarily oriented and cracks are presented and discussed.

## Finite Element Modelling

Finite Element Modeling is defined here as the analyst’s choice of material models, finite elements, meshes, constraint equations, analysis procedures, governing matrix equations and their solution methods, specific pre- and post-processing options available in a chosen commercial FEA software for determination of mixed mode membrane and bending stress intensity factors for shell structures with arbitrarily located and oriented cracks under different types of applied loads and boundary conditions. In this study, ANSYS is used for FE modeling. A fine mesh of singular Isoparametric curved shell elements (STRIA6), triangular in shape and quadratic in order with six nodes and six engineering degrees of freedom at each node with user specified number NS from one crack face to another and size Δa is created around each crack tip. The rest of the domain under consideration is discretized using a compatible mesh of 8-noded curved shell element, quadrilateral in shape and quadratic in order (QUAD8) and 6-noded curved shell

Corresponding author. Tel./fax: +917829488394
E-mail address: ajaykrishna12@gmail.com

element of triangular shape (TRIA6). A brief description of these elements is given below.

The QUAD8 element is shown in figure 1. The TOP, BOTTOM and MIDDLE surfaces of the element are curved, whereas the sections across the thickness are generated by the straight lines. The geometric modeling requires specification of two vectors at each of the eight mid surface nodes. One is the position vector $R_i$ of the node, with the three global Cartesian components $X_i$, $Y_i$, $Z_i$ where the subscript identifies the node number. The other is the unit normal vector along with the wall thickness of the same nodes. The QUAD8 element carries six engineering degrees of freedom ($U_i$, $V_i$, $W_i$, $\theta_{xi}$, $\theta_{yi}$, $\theta_{zi}$) at each of the eight mid surface nodes. The nodal degrees of freedom are illustrated in Figure 1.

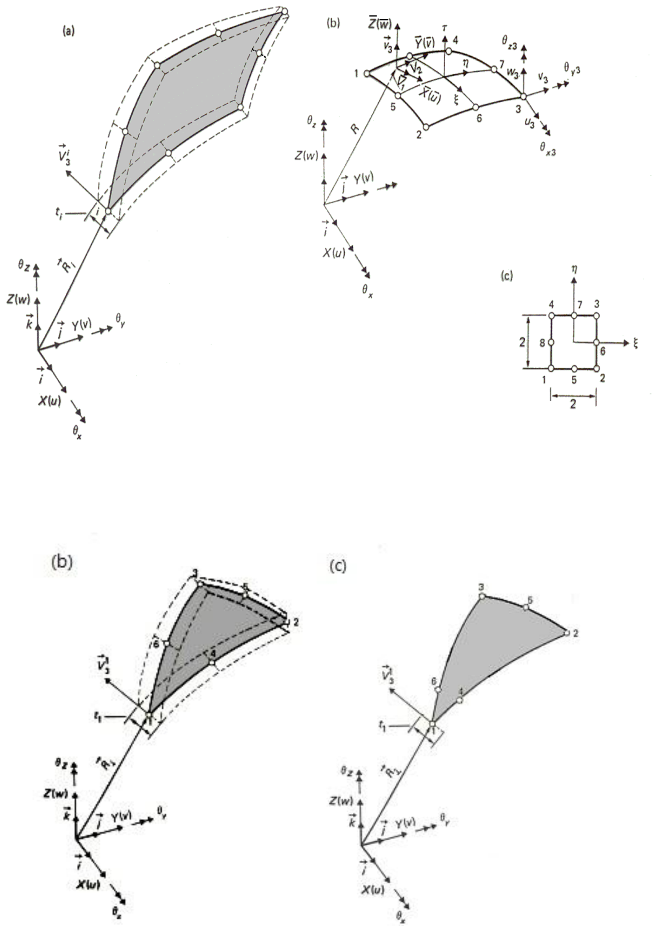

**Figure 1**: (a) Isoparametric quadrilateral shell element QUAD8; (b) Isoparametric triangular shell element TRIA6 and (c) singular Isoparametric triangular shell element STRIA6

The TRIA6 element shown in fig. 1 has six nodes and six engineering degrees of freedom at each node. The matrices and vectors for this element are computed as follows: The edge 1-4-8 of the QUAD8 element is collapsed and nodes 4 and 8 are co-located with node 1. Nodes 1, 4 and 8 are tied together to have the same degrees of freedom using multipoint constraint equations. The Singular Isoparametric Triangular Shell element (STRIA6), shown in fig. 4, has six nodes and six engineering degrees of freedom at each node. The matrices and vectors for this element are computed as follows: The nodes 4 and 6 which are normally located at mid side positions in the TRIA6 element are moved to the quarter point locations close to node 1. Node 1 in turn is located at a crack-tip. An analysis of the displacement, Strain and Stress field at any point within this element shows that the membrane and bending stress components exhibit the well-known $1/\sqrt{r}$ singularity. The number of STRIA6 elements used around a crack-tip can be progressively increased and their length reduced till accurate stress intensity factor solution is achieved. This demands a specific pre-processing capability. The pre-processing capability in ANSYS enables the creation of progressively refined mesh of STRIA6 element around each crack-tip with user specified $NS$ and $\Delta a$. A compatible mesh of regular elements (QUAD8 and TRIA6) then completes the FE Model. Consistent with this FE Model, the stress intensity factors have to be calculated posteriori. A critical assessment of post-processing options for Computational Fracture Mechanics in ANSYS identified the need for development and validation of a special purpose post-processing sub-program for computation of mixed mode membrane and bending stress intensity factors. This program is called 3MBSIF and an overview of this is given in next section.

A Post Processing sub program 3MBSIF to calculate posteriori Stress Intensity Factors $K_I^m$, $K_{II}^m$, $K_I^b$ and $K_{II}^b$ and out put their normalized values is developed in this study. 3MBSIF is a post-processing subprogram to compute crack-tip Stress Intensity Factors for shell type structures. It can output Mode-I and Mode-II components of MEMBRANE and BENDING Stress Intensity factors individually. It can be used with any commercial general-purpose Finite Element Analysis program that has the modeling capability. ANSYS has the required capability. ANSYS with 3MBSIF therefore is an efficient Computational Fracture Mechanics Tool. The nodal displacements and rotations will be extracted from properly flagged SINGULAR elements. There is a need to automate the orientation of the crack tip coordinates $\overline{X},\overline{Y},\overline{Z}$. $\overline{X}$ is along the crack plane, $\overline{Z}$ is along the crack front and $\overline{Y}$ is perpendicular to both $\overline{X}$ and $\overline{Z}$. The direction cosines of $\overline{X},\overline{Y},\overline{Z}$ enter into the transformation matrix $\lambda$. Using the properly flagged elements, their nodes and their global Cartesian coordinates, one should compute the $\lambda$ matrix. Transform them to a new Cartesian coordinate system originated at the crack tip. Using appropriate 3MBSIF-evaluation formulae compute, normalize, and output Stress Intensity Factor at points along the crack front. These points are located at TOP, MIDDLE and BOTTOM surfaces of the shell along the crack front. Explicit formulae to compute $K_I^{(m)}$, $K_{II}^{(m)}$ , $K_I^{(b)}$ , and $K_{II}^{(b)}$ are derived and used. These formulae enables user of FEM system to compute surface Stress Intensity Factor (TOP/ MIDDLE/ BOTTOM) using standard output namely nodal displacements and rotations.

## *Benchmarks*

A benchmark is a standard test problem with known target solution in the form of formulae/graphs/tables. These are used to validate finite element models developed using ANSYS and stress intensity factors calculated using 3MBSIF.

*Test Problem:*

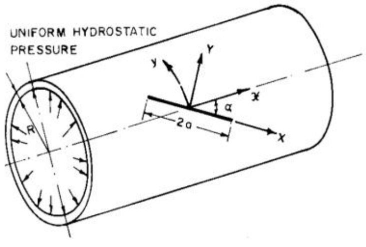

**Figure 2**: Pressure loaded cylinder with an arbitrarily oriented and located crack

A cylindrical shell of radius R, length 2L, wall thickness t, with arbitrarily located and oriented crack 2a is subjected to internal pressure P. R=1000mm, L=1000mm, t=10mm, P= 1MPa, E=200GPa and ν = 0.3 are used in the computation.

**Table 1**: Input Parameters

| Material Properties | Geometric Details | Applied Load |
|---|---|---|
| Young's modulus<br>$E = 2x10^5$ N/mm $^2$<br>Poisson's Ratio<br>$v$ =0.3 | R = 1000 mm<br>L = 1000 mm<br>ß = 0.6<br>α = 0 to $90^0$<br>t = 10 mm | P = 1 MPa<br>Reference Stress<br>$\sigma_o$ = PR/t |

*Result Comparison:*

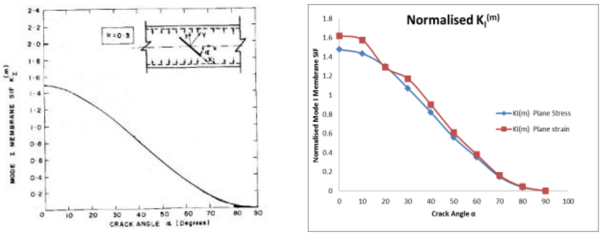

**Figure 3:** Mode I membrane SIF-Target solution and Present Analysis

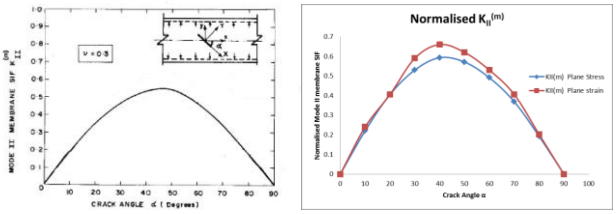

**Figure 4:** Mode II membrane SIF-Target solution and Present Analysis

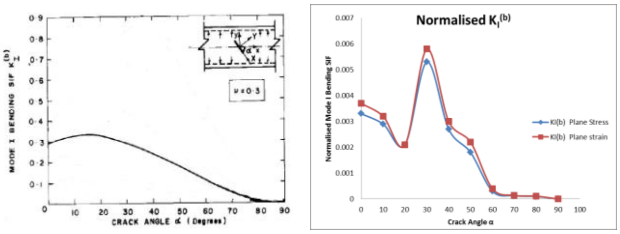

**Figure 5:** Mode I bending SIF-Target solution and Present Analysis

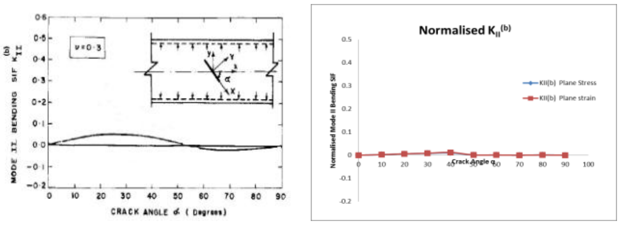

**Figure 6:** Mode II bending SIF-Target solution and Present Analysis

It can be observed that Normalized Mode I and Mode II components of Membrane Stress Intensity Factors obtained from 3MBSIF for plane stress and plane strain assumptions very closely matches with the target solution and the difference in Normalized Mode I and Mode II components of Bending Stress Intensity Factors obtained from 3MBSIF for plane stress and plane strain assumptions can be attributed to the Shear Deformation Theory used in present analysis. The target solution is based on classical shallow thin shell theory.

*Case Study*

The geometric modeling of the helical coil tube was done using CATIA V5R20. The geometric dimensions used in the computation are: Coil Radius= 1450mm, Coil Pitch=650mm, Tube diameter= 10.285mm, Tube thickness= 0.685. The applied internal pressure P= 100MPa. The material properties are E= 200GPa and Poisson's ratio ν= 0.3.

The finite element model was generated using ANSYS 12.0. The model was meshed suitably using Shell93 element in ANSYS. The geometric model of the tube is shown in figure 7. Finite element modeling for arbitrarily oriented and located crack is presented in figure 8. A refined mesh of singular elements (STRIA6) with a compatible mesh of regular elements (QUAD8 and TRIA6) used in the present study is illustrated. Rigid link elements are used to constrain the helical tube. These rigid link elements enforce kinematic relationships between the displacements at two or more nodes in the analysis. The helical tube with arbitrarily located and oriented crack is subjected to internal pressure. The graphical post processing capability in ANSYS is demonstrated. A refined mesh of 64 singular elements was generated around the crack tip.

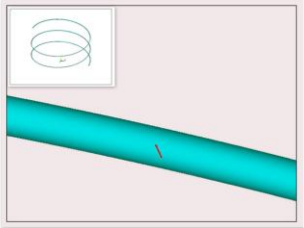

**Figure 7**: Geometric Model

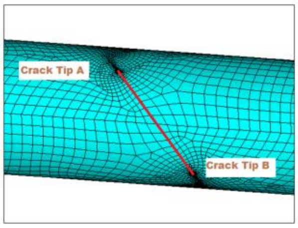

**Figure 8:** Finite Element Model

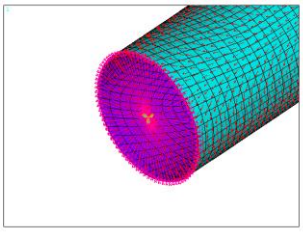

**Figure 9:** Boundary Conditions

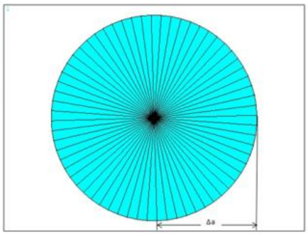

**Figure 10:** Singular Mesh(crack tip)

The helical coil tube with arbitrarily oriented crack is analyzed with crack angle varying from 0 to 90 degrees. The length of the crack is kept constant. The stress intensity factors for varying crack angles are presented below.

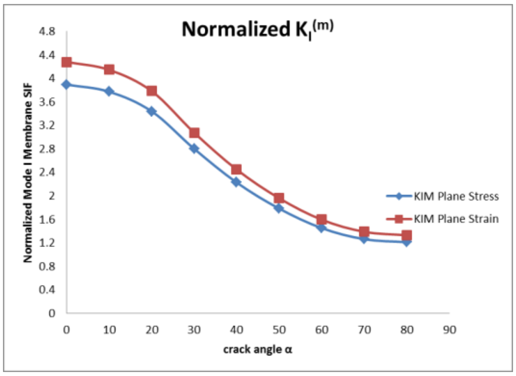

**Figure 11:** Normalized $K_I^{(m)}$ versus crack angle α

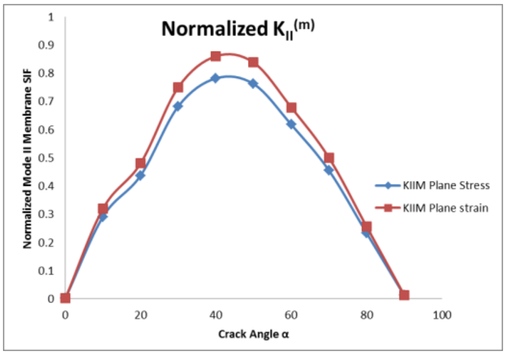

**Figure 12:** Normalized $K_{II}^{(m)}$ versus crack angle α

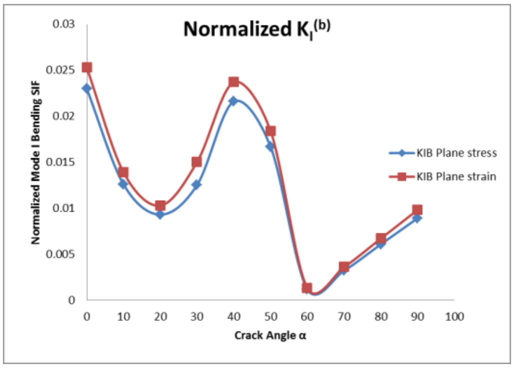

**Figure 13:** Normalized $K_{I}^{(b)}$ versus crack angle α

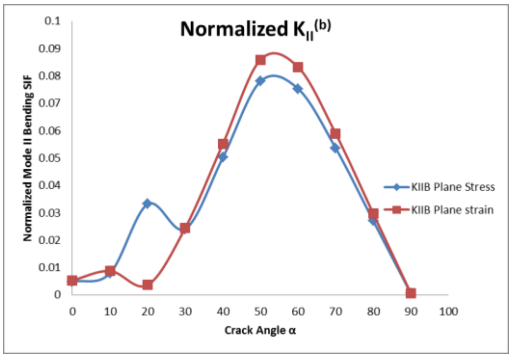

**Figure 14**: Normalized $K_{II}^{(b)}$ versus crack angle α

### *Mixed Mode Fracture Prediction*

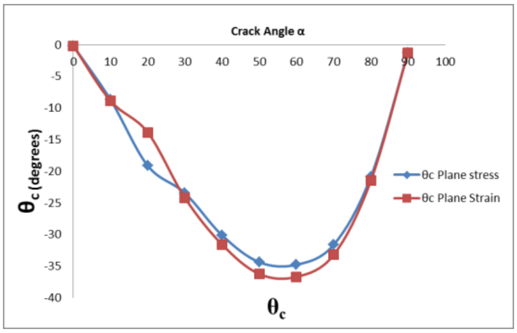

**Figure 15:** Critical angle $\theta_c$ versus crack angle α

Prediction of mixed mode fracture of a helical coil tube is done using the Minimum Strain Energy Density criteria. The crack growth direction and the critical pressure at which the crack growth occurs are evaluated. The fracture toughness of the material is $K_{IC}$= 1264.911 $MPa\sqrt{mm}$. Figure 15 shows the variation of critical crack angle $\theta_c$ for various orientations. $\theta_c$ is maximum for crack oriented at 60 degrees.

Figure 16 shows the critical pressure $P_{critical}$ variation for various crack orientations. From the graph we can observe that crack oriented at 60 degrees has the least value of $P_{critical}$.

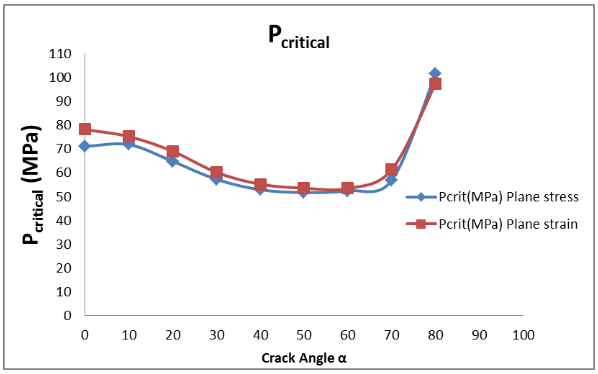

**Figure 16:** Critical pressure versus crack angle

## Conclusions

Finite element modelling using ANSYS software and the use of special purpose post processing sub program 3MBSIF to provide accurate mixed mode membrane and bending stress intensity factor solutions to the most complex problem of a helical coil tube under internal pressure with arbitrarily located and oriented through cracks of varying lengths is demonstrated in this investigation. Using the strain energy density theory of fracture as candidate mixed mode fracture criterion, predicted direction of crack growth and fracture pressure are presented for a specific case study. However, choice of other available mixed mode fracture criteria may lead to different results. This demands verification studies using experimental fracture test results.

## References

1. Nathan V. Hoffer, Nolan A. Anderson, Piyush Sabharwall, Development and Transient Analysis of a Helical-coil Steam Generator for High Temperature Reactors. The Journal of Young Investigators, Vol. 22(2), 2011.
2. Shivashankar R. Srivatsa, H.V. Lakshminarayana, Pramod R., Stress intensity factors for arbitrarily oriented cracks in a cylindrical shell with tori-spherical end closures subjected to internal pressure, International Journal of Engineering Research and Applications, Vol.2(3), 2012, pp.3057-3071.
3. Ajay Krishna V. S., Mahesh G. S., Lakshminarayana H. V., Mixed Mode Fracture of a Nuclear Reactor Steam Generator Helical Coil Tube with Arbitrarily Oriented and Located Through wall crack, Journal of Mechanical Engineering Research and Technology, Vol.2(1), 2014, pp. 59-66.
4. H.V. Lakshminarayana, M.V.V. Murthy, On stresses around an arbitrarily oriented crack in a cylindrical shell, International Journal of Fracture, Vol. 12(4), 1976, pp. 547-566.
5. H.V. Lakshminarayana, M.V.V. Murthy, L.S. Srinath, On an analytical-numerical procedure for the analysis of cylindrical shells with arbitrarily oriented cracks, International Journal of Fracture, Vol. 19, 1982, pp. 257-275.
6. Shivashankar R. Srivatsa, Lakshminarayana H. V., Sudhakar V. H. Finite Element Models for Fracture Mechanics Analysis of Complex Shell Structures, International Conference on Total

Engineering and Manufacturing Technologies, Indian Institute of Science, 2008.

7. Structural analysis guide, Release 12.1, ANSYS Inc. Canonsburg, PA. Nov 2009.
8. Prashanth Kumar, Elements of Fracture Mechanics, McGraw Hill Education (India) Ltd., New Delhi, 2009.
9. H. V. Lakshminarayana, Finite Element Analysis: Procedures in Engineering, Universities Press, Hyderabad, 1st edition, 2004.