Finite Element Application to Estimate In-service Material Properties using Miniature Specimen

G. Partheepan, D.K. Sehgal, and R.K. Pandey

Abstract**—**This paper presents a method for determining the uniaxial tensile properties such as Young's modulus, yield strength and the flow behaviour of a material in a virtually non-destructive manner. To achieve this, a new dumb-bell shaped miniature specimen has been designed. This helps in avoiding the removal of large size material samples from the in-service component for the evaluation of current material properties. The proposed miniature specimen has an advantage in finite element modelling with respect to computational time and memory space. Test fixtures have been developed to enable the tension tests on the miniature specimen in a testing machine. The studies have been conducted in a chromium (H11) steel and an aluminum alloy (AR66). The output from the miniature test *viz.* load-elongation diagram is obtained and the finite element simulation of the test is carried out using a 2D plane stress analysis. The results are compared with the experimental results. It is observed that the results from the finite element simulation corroborate well with the miniature test results. The approach seems to have potential to predict the mechanical properties of the materials, which could be used in remaining life estimation of the various in-service structures.

Keywords**—**ABAQUS, finite element, miniature test, tensile properties

I. Introduction

For the reliable estimation of the remaining life of in-service process plant equipments, petroleum industry equipments, fossil and nuclear power plant equipments etc, the degraded material properties of these equipments must be accurately known. It is well known that these equipments do undergo degradation due to prolonged exposure to high temperature and adverse environment etc causing embrittlement. It is not practicable to assess the degraded properties of these equipments and components by conducting standard destructive mechanical tests, because this would mean damaging of equipments and rendering them non-functional in addition to long shut down [1]. Over the years the miniature specimen test technique has evolved to meet these requirements, which can be used to determine the relevant mechanical properties of the aged structures and components without affecting their functionality. It includes a wide type of tests as follows: tensile test, micro hardness test, creep test, impact test, bend test, fracture toughness test, punch test (ball, shear punch) etc. [2].

A variety of approaches have been employed to determine mechanical properties from small disks and coupons. The earliest use of the 3 mm diameter disk specimen was an attempt by Huang et al. [3] to assess the tensile ductility of a set of irradiated steels. They used simply supported TEM disks of 3 mm diameter, which were displaced by a spherical tipped indenter. Mao and Takahashi [4], Mao et al. [5] investigated the deformation behaviour using small punch test and also applied recrystallization-etch technique as well as semi analytical method to find equivalent fracture strain. Kullen et al. [6] employed a shear punch test technique to determine the mechanical properties of neutron irradiated 9Cr-1Mo and 12Cr-1Mo steels. Pandey and Bhowmick [7] further used the used the disk bend test specimen to predict tensile properties and fracture toughness (J_{IC}) for a number of steels. Recently Husain [8] employed small punch test on different materials having varieties of strength to establish a general relationship between the data obtained from small punch test and the yield strength.

The finite element method has considerably attracted the researchers for predicting the material properties with the use of miniature/sub size specimens. Recently Partheepan et al. [9] obtained the tensile properties of steels by performing inverse finite element simulation in conjunction with miniature specimen. Manahan et al. [10, 11] employed a hemispherical punch of 1 mm diameter to force through a 3 mm diameter, 0.25 mm thick simply supported disk. The load vs. load-line displacement was recorded and the deformation of the disk was also modeled with a finite element code. Manahan [12] performed finite element analysis to convert the miniature disk bend test load-deflection curve into useful stress-strain curve for obtaining information on ductility using ABAQUS. At that time, however, the boundary non-linearity was not adequately addressed in general purpose finite element code. A new finite element frictional contact boundary condition model was proposed by him to accurately analyze the miniature disk bend test using FEM. Cheon and Kim [13] used small punch test to estimate the yield stress from the initial deformation behaviour

Dr. G. Partheepan is with the Department of Civil Engineering, IIT Delhi, New Delhi, India 110 016 (email: partheepan@gmail.com).
Prof. D. K. Sehgal is with the Department of Applied Mechanics, IIT Delhi, New Delhi, India 110 016. (e-mail: profsehgal@gmail.com).
Prof. R. K. Pandey is with the Department of Applied Mechanics, IIT Delhi, New Delhi, India 110 016. (e-mail: rkpiitd@yahoo.com).

of heat treated SA508 Cl.3 nuclear pressure vessel steel and 12Cr turbine rotor steel. They also carried out FEM of small punch test using ABAQUS code. Husain et al. [14] performed an experimental and a computational (FE) study of small punch test using circular disk shaped miniature specimen through inverse finite element procedure for the determination of constitutive tensile behavior of materials.

From the review [15], it was noted that different test techniques have their own advantages and limitations. In the present study a novel dumb-bell shaped miniature specimen has been deigned and fabricated for predicting the properties of unknown materials using the experimental setup described elsewhere [16]. The specimen has special advantage especially with respect to application of finite element technique. The tensile properties such as the elastic modulus (E), yield strength (σ_y) and plastic flow properties of the material may be predicted in combination with the finite element simulation. The finite element simulated model of miniature test on dumb-bell shaped miniature specimen is carried out with the help of finite element based ABAQUS code (implicit scheme) [17].

II. Experimental Work

The miniature test technique has been applied in the present work in a chromium (H11) steel and an aluminum alloy (AR66) whose chemical compositions are given in Table I. Fig. 1 shows the configuration of the dumb-bell shaped miniature specimen of thickness 0.50 mm employed in the present study.

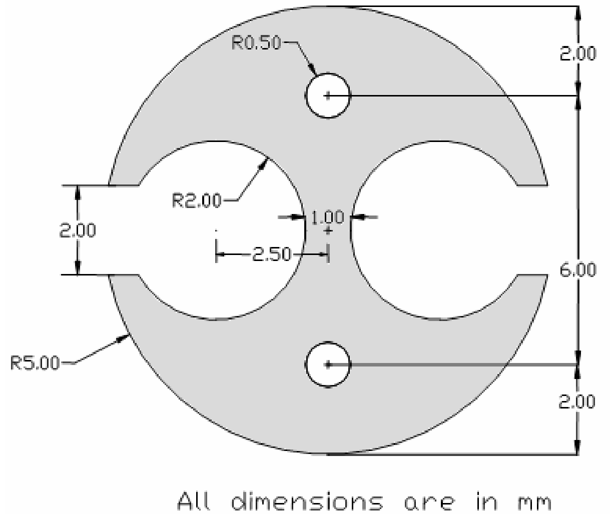

Fig. 1 Configuration of the miniature dumb-bell specimen

The above miniature specimen requires minimum number of machining operations. The specimens were mechanically polished to maintain uniformity in thickness within 1% and were marked with gauge length.

A special test fixture was used to hold the miniature specimen in the test machine as the small size of specimen does not permit the movable cross heads to come very close to the top cross head. The complete assembly consisting of fixture along with the specimen holders, test specimen and loading pins is shown in Fig. 2.

TABLE I THE CHEMICAL COMPOSITIONS (WT %) OF STEELS USED IN THE PRESENT STUDY

Material	C	Mn	Si	Zn	Ni	Cr	Mo	Al	Cu	Zr	Fe
H11 steel	0.36	0.40	1.00	0.00	0.00	5.00	1.10	0.00	0.00	0.00	Rest
AR66 alloy	0.00	0.00	0.00	6.30	0.00	0.00	0.00	89.70	1.55	0.14	0.00

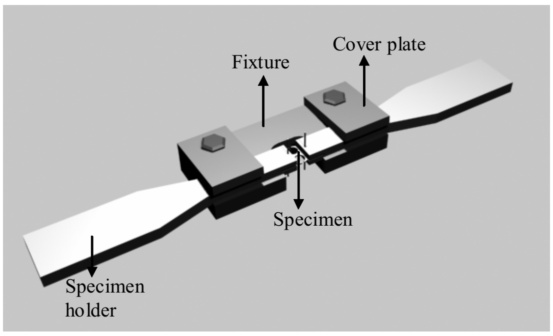

Fig. 2 Assembly of fixture attaching specimen holders and the specimen

Firstly, a specimen holder is fitted into the groove of the specially designed fixture. The groove in the fixture ensures proper alignment of specimen holder and the test specimen while fixing in the test machine. The upper specimen holder is tightly fixed with the help of cover plate and bolt. Subsequently, the miniature specimen is fixed with the help of a loading pin. Then the lower specimen holder is attached with the fixture from the other end in such a way that the miniature specimen experiences tensile load through loading pin. The complete assembly is installed in the Zwick testing machine having a load cell of 5kN capacity and testware named *testXpert*® interfaced with it. Then the fixture attached to the specimen holder is detached smoothly. An extensometer is attached with the testing machine for the measurement of elongation of the miniature specimen during the test with an accuracy of 1 μm. The sensor arm of the extensometer is placed close to the miniature specimen. The complete test setup is shown in Fig. 3.

All the dimensions such as thickness, width *etc.* of the miniature specimen were recorded before the test with an accuracy of 0.01mm. A crosshead speed of 0.05 mm/min was used during the test and the load-elongation diagram was recorded. The typical load-elongation diagrams as obtained from the test for the H11 and AR66 alloys are shown in Figs 4 and 5 respectively.

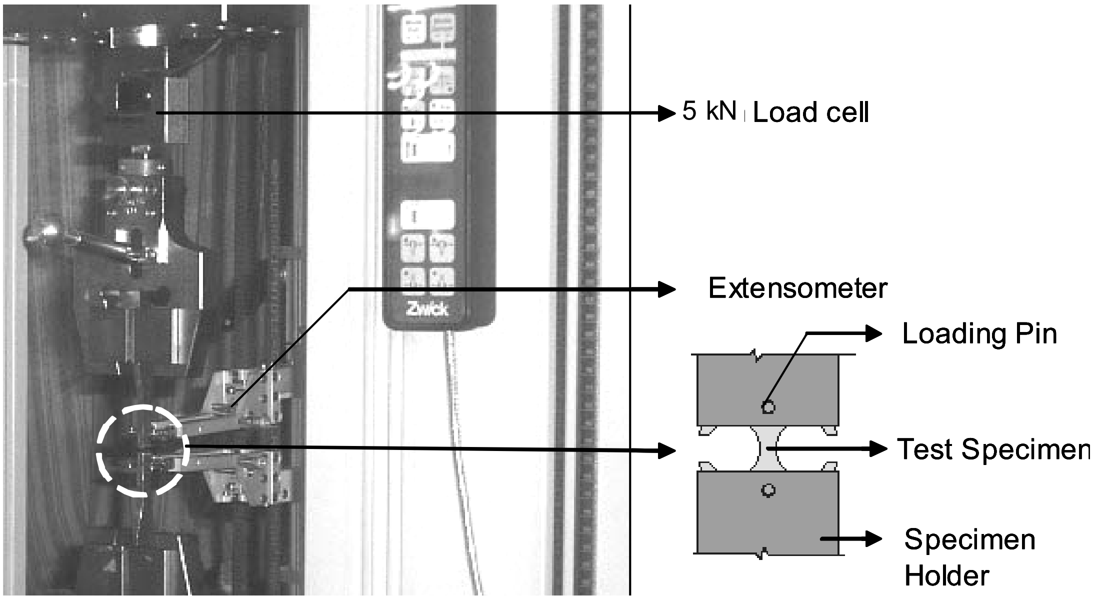

Fig. 3 Details of the experimental setup

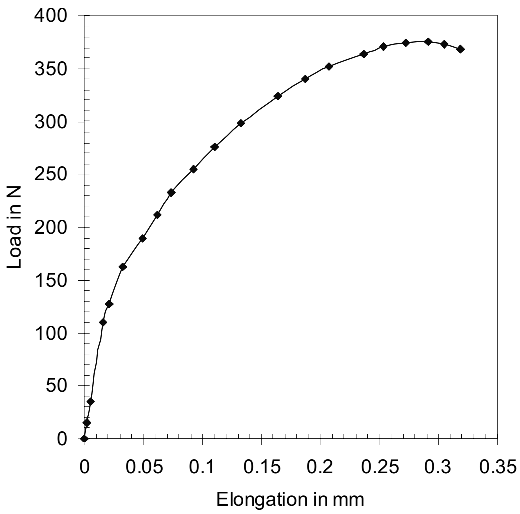

Fig. 4 Typical load-elongation diagram from the miniature test on H11 steel

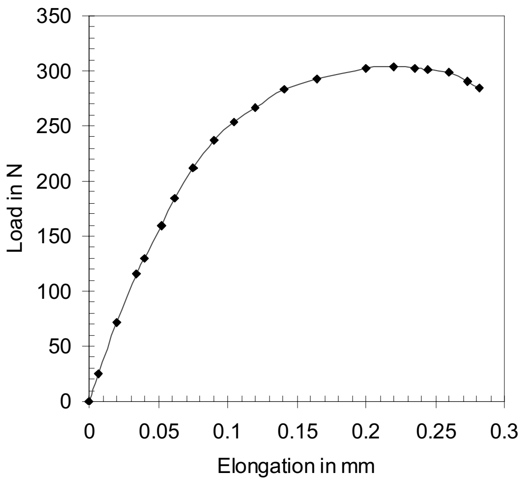

Fig. 5 Typical miniature test load-elongation diagram on AR66 alloy

III. Finite Element Modeling (FEM)

A. Geometrical modeling

The finite element modeling computations were conducted using the finite element code, ABAQUS. The 2-dimensional finite element model of dumb-bell shaped miniature specimen used for the simulation is shown in Fig. 6.

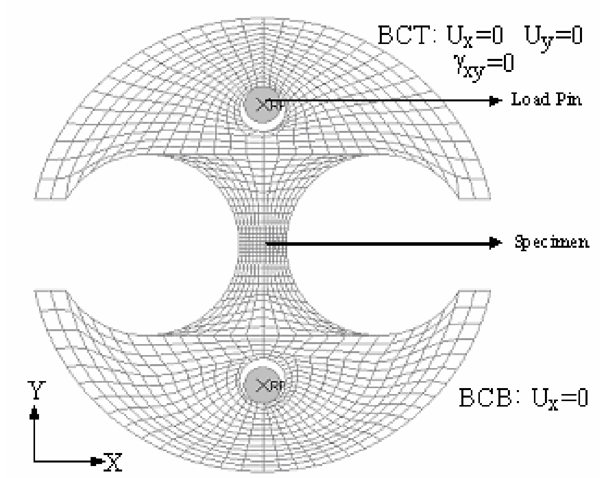

Fig. 6 Finite element model showing the mesh

The test specimen is modeled with eight noded quadratic quadrilateral plane stress elements because the thickness of the body or domain is small relative to its lateral (in-plane) dimensions. The stresses are functions of planar coordinates alone, and the out-of-plane normal and shear stresses are equal to zero. The model consists of 3977 nodes and 1258 quadrilateral plane stress elements. The free meshing option is used for the meshing of the test specimen. The sufficiency of the mesh fineness was confirmed by testing the finer one which is made of 6000 elements. In order to represent the situation exactly as in the case of the experimental setup, the full specimen together with the loading pins are modelled. The loading pins were treated as 2-D rigid bodies with a low friction coefficient of 0.01. The boundary condition used in the present case is as follows: The loading pin in the top fixture (BCT) is fixed, and the one in the bottom fixture (BCB) is constrained against translation in the *X* direction. However, it was allowed to experience displacement in the negative *Y* direction as in the experimental situation. A concentrated force was applied to the reference point of the bottom pin in the negative Y direction. The load is applied through the amplitude field in a linear fashion i.e. at time zero, the load value is zero and at time t=1 the load is maximum. All these prescribed conditions are applied in the respective reference node of load pins.

B. Material modeling

In the elastic analysis of simulation, the material properties of the columns were defined by density, elastic modulus and Poisson's ratio. In the nonlinear analysis stage, material nonlinearity or plasticity was included in the FEM using a mathematical model known as the incremental plasticity model. In the finite element simulation of miniature dumb-bell specimen, the plastic properties are defined together with the isotropic hardening rule. It means that the yield surface size changes uniformly in all directions such that the yield stress increases in all stress directions as plastic straining occurs. In the incremental plasticity model true stresses (σ_t) and true plastic strains (ε_t^p) were specified This is because, during the tensile test, the cross-section area of the sample decreases due to elongation, while each subsequent increase of sample length takes place over an already elongated sample length. The true stress-true strain defines the flow curve appropriately as they represent the basic flow characteristics of the material. The incremental plasticity model required the true stress-strain curve from the point corresponding to the last value of the linear range of the engineering stress-strain curve to the ultimate point of the stress-strain curve. The material properties used for the finite element simulation were determined through standard uniaxial tensile test and are shown in Table. II.

TABLE II MATERIAL PROPERTIES USED IN FE SIMULATION

H11 steel		AR66 alloy	
Density (kg/m^3)	7800	Density (kg/m^3)	2700
Young's modulus (GPa)	213	Young's modulus (GPa)	70.00
Poisson's ratio	0.30	Poisson's ratio	0.33
True stress (N/m^2)	True plastic strain	True stress (N/m^2)	True plastic strain
474.25	0.0000	506.90	0.0000
518.82	0.0128	512.32	0.0025
573.43	0.0173	520.04	0.0035
633.31	0.0258	526.58	0.0048
672.17	0.0352	537.57	0.0084
697.68	0.0443	552.50	0.0171
720.22	0.0555	568.66	0.0299
734.61	0.0649	591.39	0.0509
748.09	0.0761	606.47	0.0667
758.52	0.0866	612.65	0.0737
764.17	0.0932	620.25	0.0826
772.12	0.1438	626.75	0.0908
		632.01	0.0983
		634.52	0.1019

IV. 4 FINITE ELEMENT RESULTS

A concentrated load of 500 N is applied in the reference node of the bottom pin in a linearly varying manner i.e. at time t=0, the load was zero and at time t=1, the load applied is (500N) with the use of amplitude option in ABAQUS. For each time step both the load taken by the specimen and the corresponding elongation in the specimen is stored in ABAQUS. Fig. 7 shows the load-time diagram obtained by the FE simulation of specimen from H11 material. It is observed from Fig. 7 that the specimen from the H11 steel could withstand a load of 370 N only as it can be seen in the time scale showing the maximum value of 0.74 in the load-time diagram.

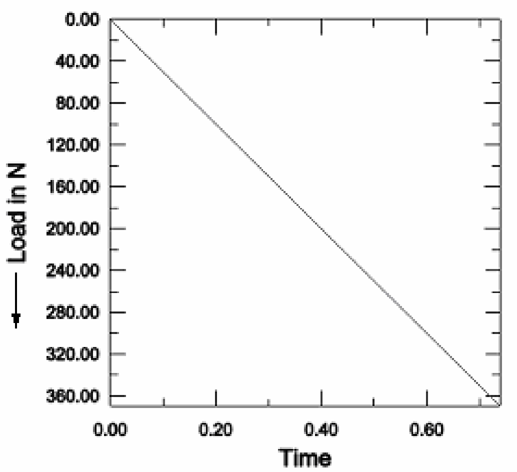

Fig. 7 FE predicted load Vs. time plot for H11 steel specimen

Similarly elongation-time curve is also obtained as shown in Fig. 8 for H11 material. From the elongation-time diagram maximum extension of the specimen observed is 0.282 mm. at this time the material in the central region obtained the maximum stress and could not carry any further load and the FE simulation got aborted. The corresponding load–elongation curves for the specimen from the H11 steel based on the finite element simulation is shown in Fig. 9 and it is compared with the miniature test load-elongation diagram. A fairly good agreement is observed between the load-elongation diagram of FE simulation and that of the miniature test.

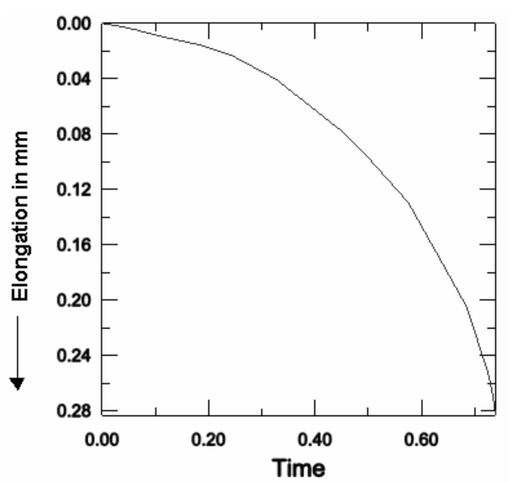

Fig. 8 FE predicted elongation Vs. time plot for H11 steel specimen

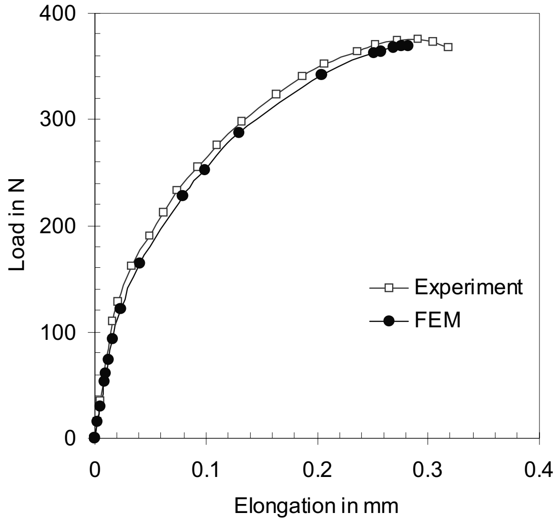

Fig. 9 Comparison of experimental and FE load–elongation curve for H11 steel

Finite element simulation was performed also for the AR66 Al-alloy. A concentrated force of 500 N was applied in the lower pin at the reference point. It is observed from Fig.10 that the specimen got elongated to a length of 0.228 mm. Also it could carry a load of 308 N as may be seen in Fig. 11. The load–elongation curves from the AR66 alloy using the finite element simulation and that from the miniature test are presented in Fig. 12. It may be observed that there is a fairly good agreement between the results of finite element simulation and the miniature test.

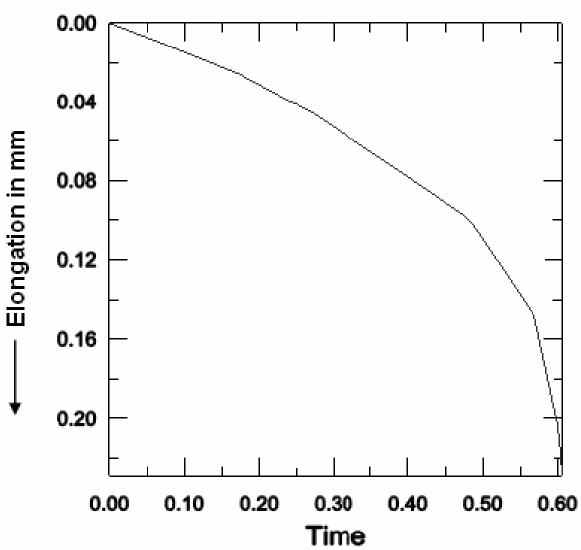

Fig. 10 FE predicted elongation Vs. time plot for AR66 alloy specimen

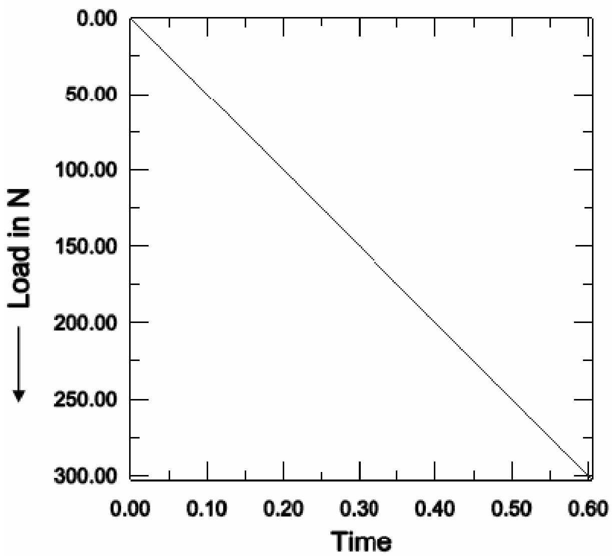

Fig. 11 FE predicted load Vs. time plot forAR66 alloy specimen.

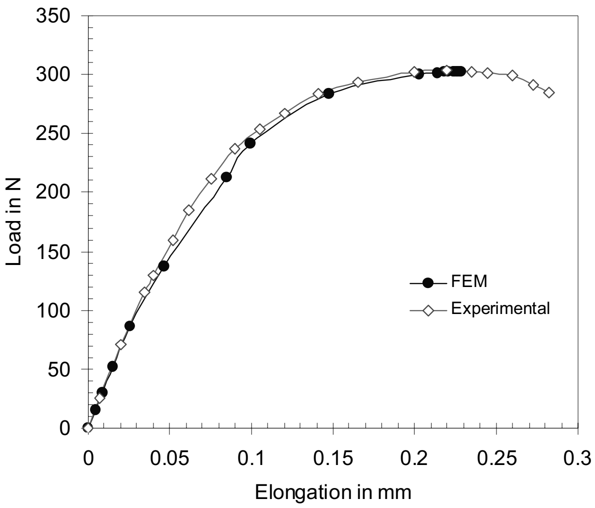

Fig. 12 Comparison of experimental and FEM load–elongation curve for AR66 alloy

A single run for the finite element analysis of the present problem consists of 3977 nodes, 1258 8-noded quadrilateral plane stress elements and requires about 30 incremental steps. Since the finite element model is in two dimensional space and the tensile loading is in in-plane direction, the complete simulation takes only around 240-300 seconds. The above time taken is much less as compared to the CPU time required for the simulation of small punch (SP) test, which some times takes more an hour too because the finite element modeling of SP test involves lot of contacts during the simulation.

Though the experimental investigation provides only the load–elongation curve, the finite element analysis can give varieties of information such as load-elongation behaviour, von-Mises stress contours, logarithmic strain and equivalent plastic strain etc. These results can be achieved at any time during the test and also the behaviour of the specimen can be studied.

V. Discussion

As may be seen from above the results from the finite element simulations are comparing well with the miniature test results. It is contented that one can easily create the data base of load-elongation curves for different materials using finite element technique. For any unknown material the load-elongation curve can be plotted using dumb-bell shaped miniature specimen test. The experimental load-elongation curve can be matched with the results in the finite element load-elongation data base. Once the experimental load-elongation curve is matched with the one available in data base, tensile properties of the unknown material can be easily predicted. A scheme for the prediction of material properties using the above technique is presented in Fig. 13.

The proposed miniature specimen and the designed loading geometry in the present investigation have some special advantages in finite element modeling. The two dimensional finite element modeling is sufficient and so it can be used in any basic finite element software. The computation time and memory space required for the analysis of this miniature test is considerably less (10-15 times) as compared to the conventional small punch test where lot of contacts involved in finite element simulation.

VI. Conclusions

The following conclusions can be made based on the present investigation:

1. The 2D finite element simulation may be performed on the dumb-bell shaped miniature specimen to generate load-elongation diagram for any material for which tensile properties and flow parameters are known.
2. The experimental load-elongation diagram obtained from the test conducted on the above miniature specimen may be used to determine properties of any unknown material by comparing with finite element simulated data base.
3. The proposed dumb-bell shaped specimen has some special advantages as compared to the other miniature test specimens.

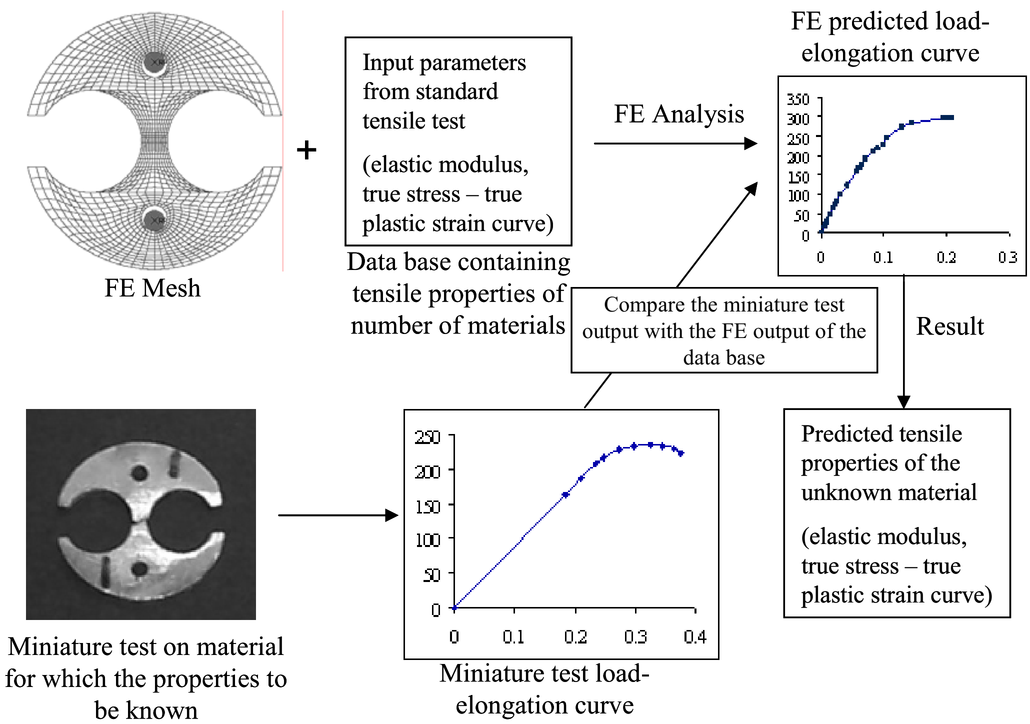

Fig. 13 The scheme of material property prediction

References

[1] J. R. Foulds, and Viswanathan, R, "Determination of the toughness of in-service steam turbine disks using small punch testing," *Journal of Materials Engineering and Performance*, vol. 10, no. 5, pp.614-619, 2001.

[2] G. E. Lucas, "Review of small specimen test techniques for irradiation testing," *Metallurgical Transactions A*, vol. 21A, pp. 1105-1119, 1990.

[3] F. M. Huang, M. L. Hamilton, and G. L. Wire, "Bend testing for miniature disks," *Nuclear Technology*, vol. 57, pp. 234-242, 1982.

[4] X. Mao, and H. Takahashi, "Development of a further-miniaturized specimen of 3 mm diameter for TEM disk small punch tests," *Journal of Nuclear Materials*, vol. 150, no. 1, pp. 42-52, 1987.

[5] X. Mao, T. Shoji, and H. Takahashi, "Characterization of fracture behavior in small punch test by combined recrystallization-etch method and rigid plastic analysis," *Journal of Testing and Evaluation*, vol. 15, no. 1, pp. 30-37, 1987.

[6] P. S. Kullen, H. H. Smith, and D. J. Michel, "The shear punch measurement of the mechanical properties of selected unirradiated and irradiated alloys," *Journal of Nuclear Materials*, vol. 58, pp. 57-63, 1987.

[7] R. K. Pandey, S. Bhowmick, "Studies on miniature test technique for the assessment of residual life," in *Proc. of Trends in mechanical engineering Education and Research*. 11th ISME conference, IIT Delhi, India, pp 542-547, 1999.

[8] A. Husain, "Determination of Mechanical Behaviour of Materials Using Miniature Specimen Test Technique and finite Element Method," PhD Thesis, Indian Institute of Technology, Delhi, India, 2003.

[9] G. Partheepan, D. K. Sehgal, and R. K. Pandey, "An inverse finite element algorithm to identify constitutive properties using dumb-bell miniature specimen," *Modelling And Simulation in Materials Science and Engineering*, vol. 14, pp 38-50, 2006.

[10] M. P. Manahan, A. S. Argon, and O. K. Harling, "Miniaturized disk bend test techniques-development and application," *ASTM STP* 888, pp. 17-49, 1981.

[11] M. P. Manahan, A. S. Argon, and O. K. Harling, "The development of a miniaturized disk bend test for the determination of post irradiation mechanical properties," *Journal of Nuclear Materials*, vol. 103-104, pp. 1545-1550, 1981.

[12] M. P. Manahan, "A new post irradiation mechanical behavior test-the miniaturized disk bend test," *Nuclear Technology*, vol. 63, pp. 295-315, 1983.

[13] J. S. Cheon, and I. S. Kim, "Evaluation of thermal aging embrittlement in CF8 duplex stainless steel by small punch test," *Journal of Testing and Evaluation*, vol. 24, no. 4, pp. 255–262, 1996.

[14] A. Husain, D. K. Sehgal, and R. K. Pandey, "An inverse finite element procedure for the determination of constitutive tensile behavior of materials using miniature specimen," *Computational Materials Science*, vol. 31, pp. 84–92, 1984.

[15] G. Partheepan, "Determination of mechanical properties of materials using newly developed dumb-bell shaped miniature specimen and numerical techniques," PhD Thesis, Indian Institute of Technology, Delhi, India, 2006.

[16] G. Partheepan, D. K. Sehgal, and R. K. Pandey, "Design and usage of a simple miniature specimen test setup for the evaluation of mechanical properties," *International Journal of Microstructure and Material Properties*, vol. 1, no. 1, pp 38-50, 2005.

[17] K. Hibbit and Sorensen Inc. ABAQUS Users Manual vol. I–II, version 6.3, 2003, USA.