Weld Fatigue Stress Evaluation of Pressure Vessel Nozzle Joint with FEA & Experimental Approach

Ankush A. Wagawad[1], Prof. S. B. Jadhav[2]

[12]*Department of Mechanical Engineering, Dr. D. Y. Patil School of engineering,Charholi(Bk), Pune-412105,Savitribai Phule Pune University.*

[1]ankush.wagawad@gmail.com

[2]hisantosh2009@gmail.com

Received xx/xx/xxxx;Revised xx/xx/xxxx; Accepted xx/xx/xxxx

***Abstract*— In the piping industries the pressure vessel-nozzle joints need to go for regular check-ups due to continues varying load. Weld joints are failed regular in equipment due to fatigue load. But investigation of joints manually is expensive and time consuming because of huge numbers of joints are present. In this regard FEA solution gives better results so that critical stresses can be found out in time. In this research using FEA approach critical stresses can be found out atwelded joint so that proper maintenance can be taken before failure. Two codes ASME and IIW standards for investigation of joints and tried to find out which approach will give the better results where one can correlate FEA results with experimental investigation. For FEA results and simulation ANSYS software is used.**

***Keywords*—ASME Code, IIW Standard, Finite element approach, weld fatigue stress analysis.**

I. INTRODUCTION

Safe and functional products are essential within the process industry. Failing to deliver such products can result in serious accidents and large expenses. As a supplier of process equipment for example, to a Power station initial care and regular checkup of the equipments and their accessories are very important and therefore efforts are made during the design process to ensure the functionality of its products. To guide and restrict the design of pressure vessels, regional regulations have been developed by standardizing organizations providing requirements on materials, design, manufacturing, inspections, etc. Traditionally formulas within these codes are used to determine critical dimensions, but also Finite Element Analysis (FEA) can be used. Today specialized commercial software are available that integrates FEA with the codes, but during regular checkup at a defined interval one need to take care at the local checks. As a solution, FEA of a parameterized model that easily can be adapted to each specific project would be interesting. By examining the obtained stresses from the analysis using the codes, weak areas can be identified early on in the product development process and the product during use, allowing necessary design changes. Alternatively, the design can be further optimized by reducing the amount of material used, which would reduce costs.

Cylindrical vessels with nozzles are common structural components in many industries, such as power, petrochemical, chemical etc. Under the applied pressure and piping loads, a high stress concentration is caused by the geometric discontinuity.Fatigue failure is a complex and progressive form of local damage which is significantly influenced by many factors such as magnitude and frequency of the loads causing the fluctuating stress, temperature, environment, geometrical complexities, material imperfections and discontinuities. The geometrical complexities and irregularities as well as load transfer conditions in steel structures can cause difficulties to estimate correctly the load effects on the fatigue strength of structure components. In the case of large steel structures with complex details, such as welded joint components in orthotropic material, an accurate estimation of the load effects in its welded details is often difficult to obtain applying a global life assessment method. A refined or local failure assessment method, which takes into account of the stress raising effects due to the geometry and/or loading conditions, might provide an accurate estimate of these effects. An accurately and precisely calculated fatigue design stress by that the evaluation of loading effects on the fatigue strength of complex welded steel structures might be obtained using advanced fatigue life evaluation techniques.
The connection region of the vessel shell and nozzle will become the weakest location of the entire structure. Therefore, it is necessary to analyze the junctions of intersection of Shell & Nozzle. Finite Element methods is to evaluate stresses at Intersection Junction by application of various load cases in terms of forces, moments & design pressure, The calculated stresses are evaluated according to the EN 13445-3 Annex C Design by analysis – Method based on stress categories. No nonlinear analysis is evaluated as this method is based on linear calculation models.

In this paper the fatigue stress analysis calculations are made as per ASME VIII div2. However, IIW standards are also referred for weld joint fatigue stress analysis. Both the codes are generalized codes and can be used for weld joint stress analysis. With the aid of FEA, it is tried to compare the analysis results by both codes which could help us to have a better process of how to get stresses at the weld joint using Finite element approach. The results got from FEA approach then examine with experimental approach. The analysis is done onto the equipment which already in use but need to go under observations and checkups for further necessary modifications.

II. LITERATURE REVIEW

Nozzle-FEA CHALMERS by ERIK BJÄRKBY[3] is a thesis for the Master of Science degree in Mechanical Engineering at Chalmers University of Technology. During the project a parameterized stress calculation tool for pressure vessels with large nozzles has been created. The tool can be used to evaluate the stresses using FEA and a method called stress categorization, which is described in the pressure vessel codes ASME VIII div 2 Part 5 Design by analysis requirements – Protection against plastic collapse.

Stress Analysis of Piping Systems by C. Basavaraju, P.E. and William Saifung Sun, P.E.[6] discusses several aspects of piping stress analysis. Integral Welded Attachments are often used to support piping systems. The local stresses in the piping at IWA locations are commonly evaluated using the Welding Research Council (WRC) Bulletin approach, which is based on Bijlaard’s work.

Determination of Weld Loads and Throat Requirements Using Finite Element Analysis with Shell Element Models - A Comparison with Classical Analysis by M. A. Weaver[10] defines Weld size requirements based on throat shear against electrode allowable were calculated with loads derived from FEA shell element results.

Design and Analysis of Welded Joints of Pressure Vessel” by Kadhim Hussein Mukhirmesh and Dr. P. Laxminarayana[13]defines in this article that pressure vessel is designed according to the weld efficiency and analyzed for its strength using Finite Element analysis with help of Ansys software. Mathematical correlations are considered for the design of pressure vessel whose design parameters are specified by a company according to the required weld efficiency. Modeling is done in Pro/Engineer. Structural analysis is done in Ansys on the welded joint of pressure vessel for different weld efficiencies.

III. PROBLEM STATEMENT

At weak section chances of crack propagation is more due to fatigue and improper weld joint. Investigating the joint using FEA tool with various types of loads is the main purpose of this project.

Going for traditional approach for every possible failing joint at pressure vessel weld junction is time consuming and expensive as par engineering work is concerned. FEA tool applied hence to check for every possible failure. But estimating weld stress is challenging task with FEA tool as every analyst has to give average stress value at weld junction. Problem arises that which value in critical zone should we define as critical stress during analysis.

IV. Objectives

1) Investigating the joint with respect to fatigue life with ASME and IIW standards.
2) Analysing the model for critical location with the aid of FEA tool.
3) Finding the variation of results with FEA and practical approach to reduce the error using FEA.

V. ANALYSIS

A.The dimensions of pressure vessel and nozzle

TABLE I
DIMENSIONS OF ROLLER

Properties	**Dimensions**
Outer diameter vessel	1722 mm
Inner diameter vessel	1700 mm
Outer diameter nozzle	323.9 mm
Inner diameter nozzle	298.5 mm
Weld fillet	11 mm
Nozzle flange thickness	49.2 mm
Nozzle flange diameter	520.5 mm

B .The material properties and loads

TABLE III
MATERIAL PROPERTIES

Component	Material	Young’s Modulus (GPa)	Poisson’s ratio	Yield Strength (MPa)
Shell	H11-1.0425	193	0.3	215.4
Nozzle Pipe	ASTM A106 Gr.B	193	0.3	209.8
Nozzle Flange	ASTM	192	0.3	195.0

The design pressure inside the pressure vessel is 0.85 MPa. There are three dimensional forces and moments which extract at 180^0C are also applied. 15% Brown field effect is considered for allowable stress check.

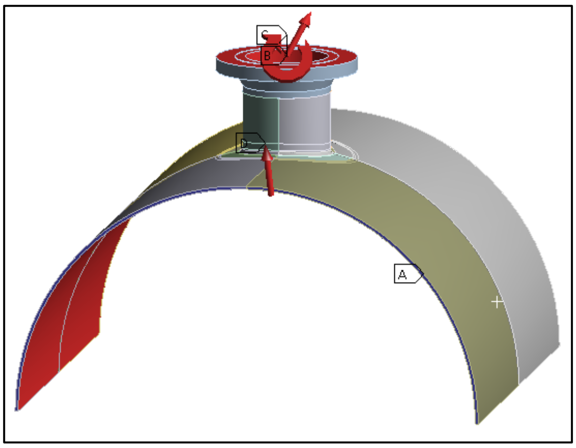

Fig.1 Loads and Boundary Conditions

For material ASTM - Handbook of Comparative Weld Steel Standards is used.

C. FE modelling

ANSYS Spaceclaim and Design modeller is used for CAD purpose and ANSYS Workbench is used for simulation. The equipment is modeled using Structural element ‘Solid 185’. It is 1^{st} order Hexa element. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. Both in-plane and normal loads are permitted. The element has three degrees of freedom at each node: translations in the nodal x, y, and z directions. Stress stiffening and large deflection capabilities are included.

The remote point is created at the flange top surface to apply remote forces and moment.

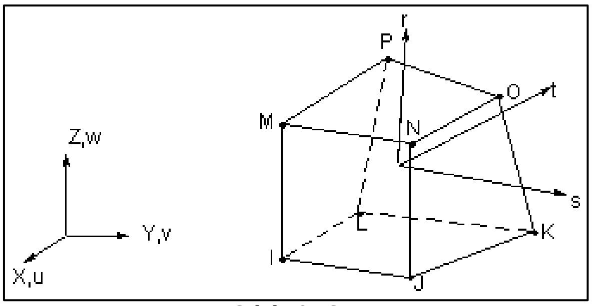

Fig.2 Solid-185 Structure

D. Solid modeling

Solid model of required pressure vessel nozzle assembly is created in ANSYS Spaceclaim software which makes modeling so easy and user friendly. Fig.3shows a solid model of half cut pressure vessel with nozzle and flange assembly.

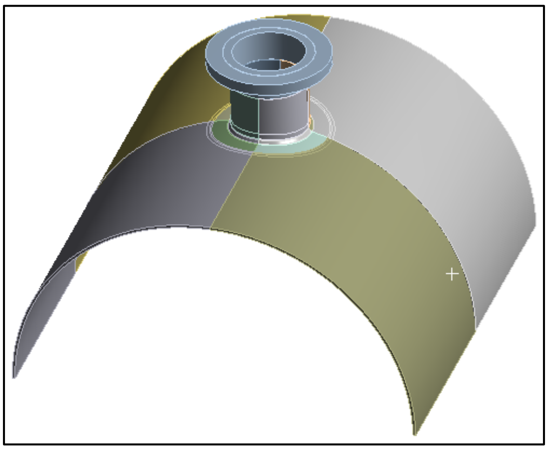

Fig.3 Solid Model of Pressure Vessel, Nozzle and Flange

*E. Stress*analysis

Solid model of roller follower is created by Spaceclaim software then this model is saving in SETP format and export into the FEA software ANSYS 15. Assembly without weld and with weld is analyzed in FEA software as per ASME and IIW standard to calculate weld joint stress.

F. Mesh generation

The meshing is done in ANSYS 15 (Workbench) software. Fig.4 shows themeshing of vessel nozzle assembly without weld and with weld

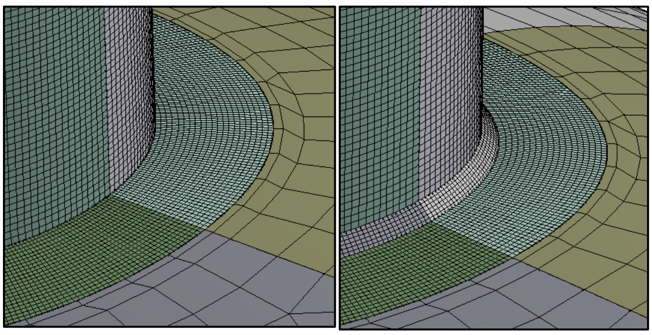

Fig. 4 Mesh ModelWithout Weld and With Weld

G. Boundary Conditions

Static structural analysis was performed to determine equivalent (von-Mises) stresses and total deformation of existing and modified roller follower by ANSYS software. For this above boundary conditions are used: Fixed support is given at the cut portion of pressure vessel.

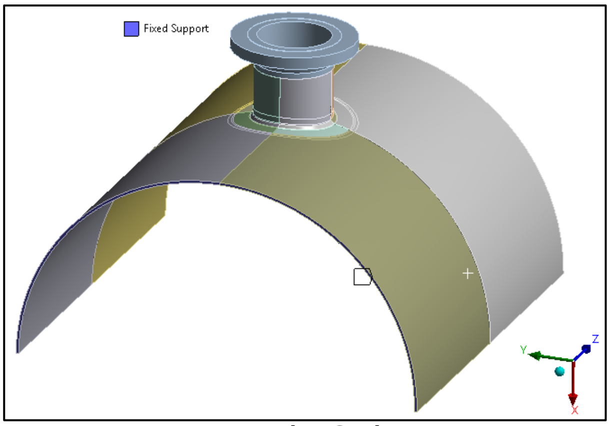

Fig. 5 Boundary Conditions

H. FE Analysis

FE analysis is shown below is the assembly with weld because it is the real scenario the results of which can be cross verified with practical approach.

For all materials the service factor 1.5 is considered which gives the maximum allowable stress limit to check the model for failure criteria under high stress.

To calculate the joints stress under fatigue load a procedure is followed as per ASME and IIW standards and calculations are done.

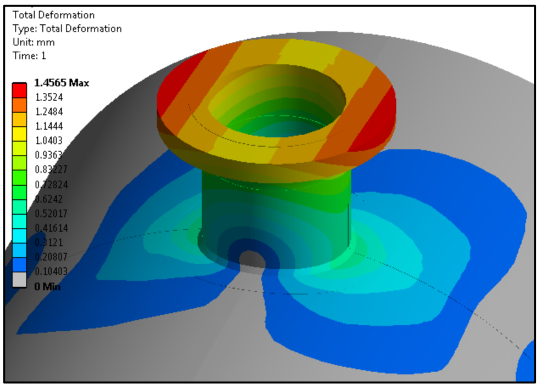

Fig. 6 Total Deformation Plot

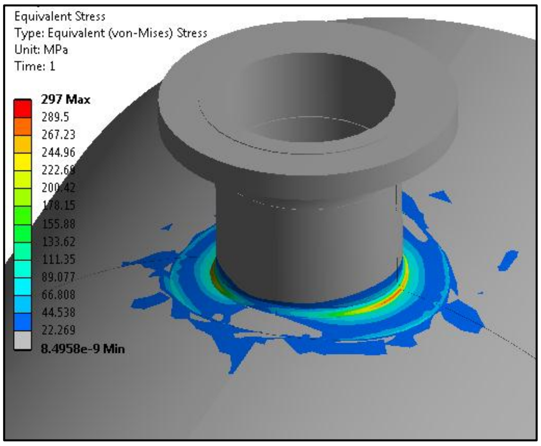

Fig. 7 von-Mises Stress Plot

a) As per ASME Standard

As per ASME section VIII Division 2[1] the analysis to be done for weld stress calculations under fatigue are as follows; As per ASME the stress calculations can be done using SIF concept that is stress intensity factor.

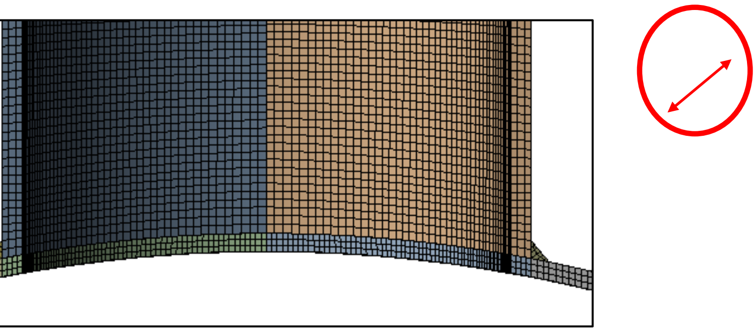

Fig. 8 Stress Linearization Path

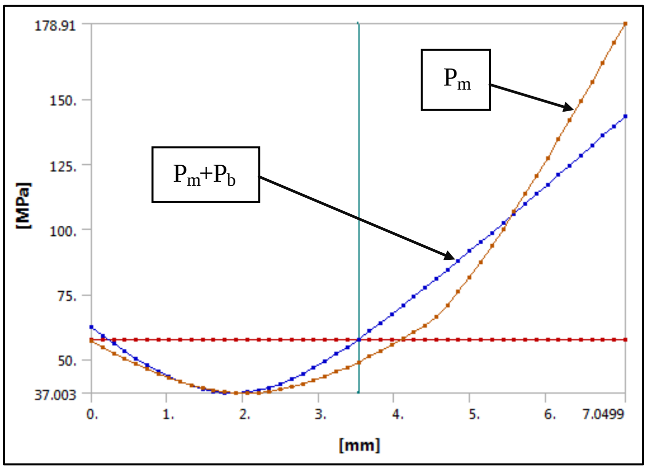

Fig. 9 Stress Linearization Plot

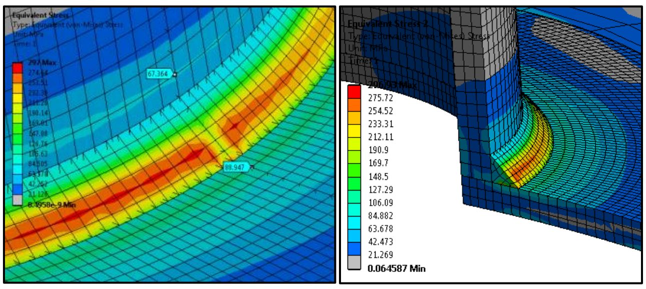

Fig. 10 von-Mises Stress Plot on Weld

For shell,

S_Y= 215.4 MPa

Considering service factor = 1.5 x S_Y

= 323.1 MPa

Considering brown field effect with service factor

= 274.64 MPa

Considering brown field effect without service factor

= 183.09 MPa

Hence,

P_m= 57.71 MPa

P_b= 94.25 MPa

$P_m + P_b$= 143.39 MPa

$P_m < S_Y$= 183.09 MPa

$P_m + P_b < 1.5\ S_Y$= 274.64 MPa

Where

P_m = Primary Membrane Stress &

P_b = Primary Bending Stress

Note: Weld material is considered as lower yield value of the base metals which are joined as weld joint.

b) IIW Standard

As per IIW standards [7] weld fatigue stress check is done considering hot spot method. Note, the tubular joints calculations have to be done for such types of joints. The stress point on FE model had to be taken at specific distance from weld zone as shown in figure 11.

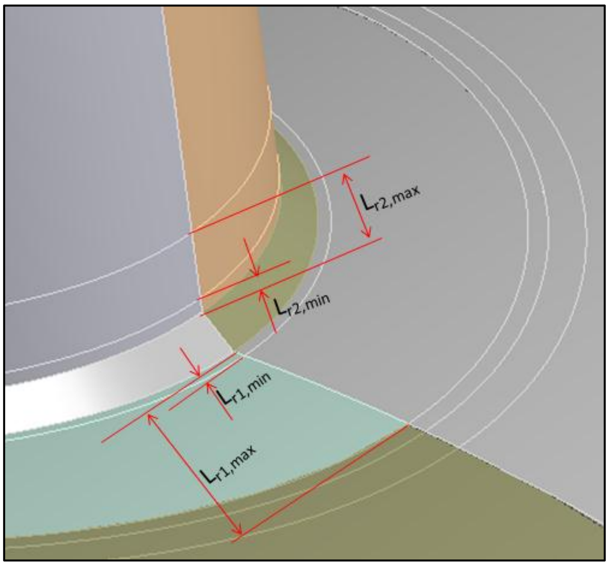

Fig. 11 Chord and Brace Distance Representation

For chord,

$L_{r1,min}$ = 0.4 x t_0

= 0.4 x 11 = 4.4 mm

$L_{r1,max}$ = 0.09 x r_o

= 0.09 x 861 = 77.5 mm

For brace,

$L_{r2,min}$ = 0.4 x t_1

= 0.4 x 11 = 4.4 mm

$L_{r2,max}$ = 0.65 x (r_1 xt_1)^0.5

= 0.09 x (161.95 x 12.7) = 29.48 mm

The stresses on the node at distance Lr1 and Lr2 are shown in figure 12.

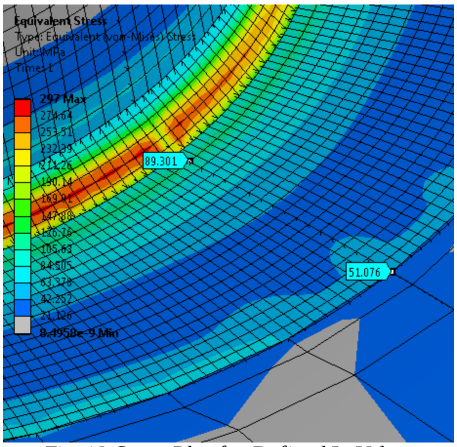

Fig. 12 Stress Plot for Defined L_r Values

VI. EXPERIMENTAL ANALYSIS

Once analysis is done and the model is finalized considering meshing and respective nodal stress values as per ASME and IIW standards the pressure vessel nozzle joint has to go under experimental investigation. Below figure13 is proposed setup defined for experimental analysis. For practical analysis the scale modal is to be taken. Two strain gauges are going to use. One is on tension side and one is on compression side of a joint. Likewise, there are two setups that are as ASME and for IIW.

As in figure13 shown the assembled modal is fixed with another flange which will help in constraining purpose. The flange at nozzle end is roped at the angle to resultant force and moment. The pulling force is applied at the nozzle end and at defined resultant load the strain will be noted down from strain gauges. Maximum strain will be considered for stress calculation purpose.

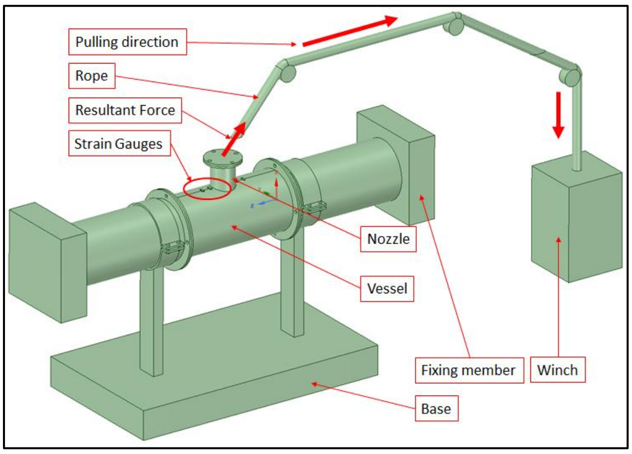

Fig. 12 Proposed Experimental Setup

A.Experimental results

From experimentation, we will get the value of strain at the location where strains are glued. From strain we can have stress at the location. Furthermore, the experimental stress can be verified with FEA results for specified location and nodes.

VII. CONCLUSION

Experimental analysis takes a lot of time in calculating stress at such junctions where junction number is more and huge number of data and time is required, the FEA calculations will take very less time in analyzing the same. Correlating the experimental data with FEA can also be done in easy way for further analysis to be done through finite element approach.

ACKNOWLEDGMENT

I take this opportunity to thanks Prof S. B. Jadhav and Prof. A. N. Patil for valuable guidance and for providing all the necessary facilities, which were indispensable in completion of this work. Also I sincerely thank to Citec India Pvt. Ltd. Pune and all the authors who worked on pressure vessel nozzle weld joints analysis.

REFERENCES

1. ASME Section VIII, Division 2
2. Material Data Sheet EN Std. for HYDRA
3. Nozzle-FEA CHALMERS by ERIK BJÄRKBY
4. 2004 ASME Boiler & Pressure Vessel Code - An International Code
5. Euro Inox – The European Stainless Steel Development Association
6. "Stress Analysis Of Piping Systems" by C. Basavaraju, P.E. and William Saifung Sun, P.E. - Senior Engineering Specialist Bechtel Power Corporation Frederick, Maryland
7. IIW document IIW-1823-07ex XIII-2151r4-07/XV-1254r4-07
8. ASTM - Handbook Of Comparative Weld Steel Standards
9. ASME IX QW - Welding General Requirements
10. "Determination of Weld Loads and Throat Requirements Using Finite Element Analysis with Shell Element Models" — A Comparison with Classical Analysis - by M. A. Weaver
11. Presentation on "Fatigue of welded pressure vessels" – World center for material joining technology by Steve Maddox (The welding Institute TWI) and Jim Wood (University of Strathclyde)
12. 2009 COADE Conference "Local Stress Analysis" by Chris Hinnant Paulin Research Group Huston, TX
13. "Design and Analysis of Welded Joints of Pressure Vessel" by Kadhim Hussein Mukhirmesh (College of Engineering, Wassit University, Ministry of Higher Education & Scientific Research, Iraq), Dr. P. Laxminarayana (Prof, Dept. of Mechanical Engineering, University College of Engineering, Osmania University, Hyderabad, India.)
14. "Pressure Vessel Limit Load Estimation by FEM and Experimental Method" by Dilip M. Patel Dr. Bimlesh Kumar Mechanical Department & NMU University