International Journal of Scientific and Technical Research in Engineering (IJSTRE)
www.ijstre.com Volume 4 Issue 6 ǁ Nov-Dec 2019.
ISSN: 2581-9941

A Concise Review on Applications and Challenges of Stress Analysis

Michael Kwabena Boadu[1], Ebenezer Omari Abankwa[2], Listowel Abugri Anaba[3]
[1],[2]Department of Mechanical and Industrial Engineering, University for Development Studies, Ghana
[3]Department of Agricultural Engineering, University for Development Studies, Ghana

Abstract: *Stress analysis is highly necessary to investigate the reliability of an engineering structure. A concise literature review was carried out to find the applications and challenges of stress analysis. This was done by reviewing papers published by researchers whose areas of interest were related to the course of study. It was concluded that stress analysis can be used to determine the strength of structures and to monitor the failure progression of composite materials among others. It was found that the complexity of structural geometry and localized or diffused structural damage are some of the challenges of stress analysis.*
Keywords: *Applications; Challenges; Concise; Review; Stress Analysis;*

I.Introduction

Stress analysis is an important aspect in engineering science, since failure of most engineering structures are caused by stresses. Members of a machine such as an engine, blades of a turbine, fuselage of a jumbo jet aircraft and others have to be subjected to rigorous stress analysis to ensure they are safe to be used for their intended purpose. Rankine became famous for his contributions in the field of mechanical engineering by concluding that stress concentration zones in mechanical parts were the main cause of structural failures [1]. Wöhler conducted some investigations on the axes of axles of carriages for the railway. The tests were performed in laboratory fatigue under repeated stresses subjected to bending, torsion and axial loads [2]. The studies showed that the fatigue life increased with decreasing applied stress field so that below a certain threshold of tension, the member seemed to have infinite life. On the other hand, he observed that the fatigue life was drastically reduced by the presence of notches. Other contributions were introduced particularly by Fairbairn [3] and Gerber [4], which developed the design methods for different fatigue stress cycles. Similar studies was performed by Goodman [5], and he observed that the elastic limit of metals under cyclic elastic limit was different from the monotonic regime, which came to confirm the results published by Wöhler. It looks clear that many people have contributed immensely to the field of stress analysis.

The determination of reliable and unique stress quantity required for fatigue analyses has been attempted by utilizing the hot spot or structural stress, σ_{hs} , employed previously in the offshore structures sector [6, 7]. Webster et al. [8] performed three dimensional finite element analysis of a high-speed diesel engine connecting rod. For this analysis they used the maximum compressive load which was measured experimentally, and the maximum tensile load which is essentially the inertia load of the piston assembly mass.

Hippoliti [9] reported design methodology in use at Piaggio for connecting rod design, which incorporates an optimization session. However, neither the details of optimization nor the load under which optimization was performed were discussed. Two parametric FE procedures using 2D plane stress and 3D approach developed by the author were compared with experimental results and shown to have good agreements. The optimization procedure they developed was based on the 2D approach. Athavale and Sajanpawar [10] modeled the inertia load in their finite element model. An interface software was developed to apply the acceleration load to elements on the connecting rod depending upon their location, since acceleration varies in magnitude and direction with location on the connecting rod. They fixed the ends of the connecting rod, to determine the deflection and stresses.

Yoo et al. [11] used variational equations of elasticity, material derivative idea of continuum mechanics and an adjoint variable technique to calculate shape design sensitivities of stress. The results were used in an iterative optimization algorithm, steepest descent algorithm, to numerically solve an optimal design problem. Sarihan and Song [12], for the optimization of the wrist pin end, used a fatigue load cycle consisting of compressive gas load corresponding to maximum torque and tensile load which also correspond to maximum inertia load. Evidently, they used the maximum loads in the whole operating range of the engine. To actually design for fatigue, modified Goodman equation with alternating octahedral shear stress and mean octahedral shear stress was used. For optimization, they generated an approximate design surface, and performed optimization of this design surface. The objective and constraint functions were updated to obtain precise values. This process was repeated till convergence was achieved. They also included constraints to avoid fretting fatigue. The mean and the alternating components of the stress were calculated using maximum and minimum values of octahedral shear stress. Their exercise reduced the connecting rod weight by approximately 27%.

Niemi [13] did some work to come by a proposition to decompose the linearized through thickness stress field into the uniformly distributed membrane (axial) stress field, σ_{hs}^{m} and the anti-symmetric bending stress, σ_{hs}^{b}. The stress distribution, required for determining the stress intensity factor K by the weight function method, can be found by utilizing the universal stress distributions proposed by Monahan [14]. The combination of appropriate weight function [14, 15, 16] and the through thickness stress distribution renders it feasible to determine the stress intensity factor and to subsequently simulate the fatigue crack growth in any welded member without extensively relying on FE numerical analyses of cracked bodies. A special coarse mesh FE modeling technique was proposed [17] to enable the determination of membrane and bending hot spot stresses, σ_{hs}^{m} and σ_{hs}^{b}. It is however, necessary to review the applications and challenges of stress analysis.

II.Applications Of Stress Analysis

Nowadays there are many computer tools which can be used purposely to investigate the strength and dynamic structural features of individual bodies or complex mechanical systems made up of large number of bodies [18, 19, 20, 21]. Finite Element Method is the most commonly used numerical technique based on the continuum mechanics [22, 23] allowing mainly the investigation and determination of the stresses and strains in materials [24, 25, 26, 27] and structures subjected to forces, torques etc. [28, 29]. Oterkus et al. [30] established a principle for stress analysis of composite materials/structures. Failure analysis of composite materials is quite complex and challenging. Finite element analysis was utilised as a numerical tool to determine the failure progression in composite materials/structures. There are various techniques which were specifically developed for this purpose. One of the common techniques was Virtual Crack Closure Technique [31, 32]. This technique is based on Linear Elastic Fracture Mechanics (LEFM) and it is based on the idea that the amount of energy required to form a crack surface is equal to the amount of energy needed to close that same crack. Since the principle is based on Linear Elastic Fracture Mechanics, an initial crack is required to perform the analysis. The pseudo stress output can be used together with a multi-axial notch analysis technique to estimate local true stresses or strains at stress concentration zones for fatigue damage evaluation. The fixed reactive stress analysis is a common, fundamental solution for a constrained structure that is subjected to a set of constant or time independent actions such as forces, moments, torque, and temperatures [33].

Stress analysis is significant to evaluate the damage accumulation and remaining life of structures because of the flexibility to predict remaining life if the operating pattern contains wide variation in the combination of operation time and start-up cycles [34]. Darshit et al. [35], performed a static stress analysis on the upper surface of a truck chassis. Critical parts that will lead to failure were observed. A 3-D finite element model of the truck chassis was made using ProE before analyzed through ANSYS software. Numerical results showed that the critical part was at the mounting bracket of the tire and also at the front part of the chassis. Some design modifications were presented to reduce the stress and to improve the strength of the truck chassis. MSC Nastran is a multidisciplinary structural analysis application utilized to perform static, dynamic, and thermal analysis across the linear and nonlinear domains, supported by an automated structural optimization and high performance fatigue analysis technologies, all enabled by powerful computing [36].

Masahiro et al. [37] predicted the strength of main frames of dump truck based on static stress analysis where dynamic loads obtained by measurement with actual machines were replaced with static loads. For the articulated dump truck that was developed recently, it was needed to precisely predict the stresses that act on the frames during travel. They introduced elastic characteristic of main frame into kinematical analysis models using kinematical analysis software ADAMS and finite element method software NASTRAN so as to compute frame stress that occurs during travel, which was applied to the rear frame of the articulated dump truck. Stress analysis is one of the most important steps in any structural design practice. A desirable stress analysis method should give reasonably accurate, reliable results in terms of the magnitude and distribution of stresses or strains in the structure of interest which is subjected to a definite load and boundary condition. The information from the analysis will enable the engineer to predict the strength of the structure [38]. Karaoglu and Kuralay [39] conducted a research on the stress analysis of a truck chassis with riveted joints utilizing FEM. The numerical results showed that stresses on the side member can be decreased by increasing the side member thickness locally. If the change in thickness is not feasible, then increasing the connection plate length may be a good alternative. This result may lead to optimal design of the truck chassis.

While investigating a connecting rod failure that led to a disastrous failure of an engine, Rabb [40] performed a detailed FEA of the connecting rod. He modeled the threads of the connecting rod, the threads of connecting rod screws, the pre-stress in the screws, the diametric interference between the bearing sleeve and the crank end of the connecting rod, the diametric clearance between the crank and the crank bearing, the inertia load acting on the connecting rod, and the combustion pressure. The analysis clearly indicated the failure location at the thread root of the connecting rod, caused by improper screw thread profile. The connecting rod failed at the location indicated by the FEA.

Mahmoodi et al. [41] discussed the stress and dynamic analysis of truck ladder chassis. At the first stage, in order to design a chassis for self-weight reduction, material type and cross section profiles of chassis were chosen according to a maximum normal stress and maximum strain theories. Then, the stress analysis of truck chassis was carried out by ABAQUS software to determine maximum transverse deflection and stress distribution. The stress/strain distribution was computed along the chassis. Maximum stress and strain levels are found in the front section of chassis, where engine and transmission are installed. Moreover, results showed that open U-shaped profiles are sufficient for weight reduction which can endure loads.

Roslan Abd Rahman et al. [42] conducted stress analysis using ABAQUS to locate the critical point of stress. They used ASTM Low Alloy Steel A 710 C material which has 552 MPa of yield strength and 620 MPa of tensile strength. The simulation result showed that the critical point of stress occurred at the opening of chassis which is in contact with the bolt. The stress magnitude for the critical point was 386.9 MPa. They reported that the critical point is an initial to probable failure since fatigue failure started from the highest stress point. They also asserted that the magnitude of the stress can be used to predict the life span of the truck chassis. The knowledge of elastic peak stress, σ_{peak}, makes it feasible to assess the fatigue crack initiation life by employing the local strain-life technique [43, 44].

III.Challenges Of Stress Analysis

Accurate stress analysis of real structures containing diffuse or localized damage is a very difficult task even in the elastic state. The task is further complicated if the material is in the plastic state and if the structural geometry is complex. This task cannot be accomplished analytically in all but the most simple structures [45]. Stress analysis of adhesive joints is one of the most difficult engineering tasks due to the presence of bimaterial interfaces and geometrical discontinuities which might cause stress singularities and consequently uncertainty in evaluations of stresses or strains [38]. FEM analysis of local stress-strain state of micro-heterogeneous composite material is actually a difficult task [46]. The FEA software which is basically used for stress analysis does not give accurate results when it comes to stress concentration testing. Finite element analysis as a tool for stress analysis is also a complex process and requires higher time for compilation as compared with other similar methods [47]. When dealing with contact problems, stress analysis becomes difficult because the boundary for the analysis is unknown and thus, the problem becomes part of the solution [48]. There are difficulties when working on bodies with non-straight boundaries which subsequently makes the member difficult to impose boundary conditions. Non-uniform and non-rectangular meshes obtained when using ANSYS to do stress analysis is also a challenge [49]. Determining infinite stress at a sharp corner can become a difficult task in stress analysis. Thus, singularities can be confusing because they cause an accuracy problem inside the model, which implies a problem with visualization because singularities extend the range of the stresses. This means that smaller stresses can appear to be negligible [50].

IV.Conclusions

The literature review provided us with the following relevant information:

1. Stress analysis enables us to monitor the failure progression of composite materials
2. Stress analysis helps us to predict the strength of structures
3. Stress analysis helps us to determine the magnitude and distribution of stresses and strains in a structure of interest
4. Stress analysis allows the evaluation of damage accumulation and remaining life of structures
5. Stress analysis enables us to optimize the design of an engineering structure
6. Stress analysis is affected when the structure of interest contains diffuse or localized damage patterns
7. Stress analysis is also affected when the part or structure of interest have a complex geometry
8. Stress analysis is affected when you are dealing with structures with adhesive joints
9. It is time consuming
10. The stress analysis of local stress-strain state of micro-heterogeneous composite material is difficult
11. When dealing with contact problems stress analysis becomes difficult
12. Determination of infinite stress at a sharp corner can be difficult

References

[1] Rankine, W. J. M. (1843). "On the causes of the unexpected breakage of the journals of railway axles, and on the means of preventing such accidents by observing the law of continuity in their construction". Minutes of the Proceedings of the Institution of Civil Engineers, 2, pp 105–107.

[2] Wohler, A. (1860). Versuche uber die Festigkeit der Eisenbahnwagenachsen, Zeitschrift fur Bauwesen,10; English summary (1867), Engineering, Vol.4, pp.160-161.

[3] Fairbairn, W. (1864). ''Experiments to Determine the Effect of Impact, Vibratory Action, and Long-Continued Changes of Load on Wrought-Iron Girders'', Philosophical Transactions of the Royal Society of London Vol (154) pp. 311-325

[4] Gerber W., (1874). ''Bestimmung der zulossigne Spannungen in eisen Constructionen, Z. Bayer Arch. Ing. Ver., 6

[5] Goodman, J., (1899), Mechanics Applied to Engineering, New York, Longmans Green and Co.

[6] Marshall, P. W., (1992). Design of welded tubular connections, Elsevier, Amsterdam.

[7] Dong, P., (2001). ''A structural stress definition and numerical implementation for fatigue analysis of welded joints'', International Journal of Fatigue, 23, 865-876.

[8] Webster, W. D., Coffell R., and Alfaro D., (1983). "A Three Dimensional Finite Element Analysis of a High Speed Diesel Engine Connecting Rod," SAE Technical Paper Series, Paper No. 831322.

[9] Hippoliti, R., (1993). "FEM Method For Design and Optimization of Connecting Rods for Small Two-Stroke Engines," Small Engine Technology Conference, PP. 217-231

[10] Athavale, S. and Sajanpawar, P. R., (1991). "Studies on Some Modelling Aspects in the Finite Element Analysis of Small Gasoline Engine Components," Small Engine Technology Conference Proceedings, Society of Automotive Engineers of Japan, Tokyo, PP. 379-389.

[11] Yoo, Y. M., Haug, E. J., and Choi, K. K., (1984). "Shape Optimal Design of an Engine Connecting Rod," Journal of Mechanisms, Transmissions, and Automation in Design, Transactions of ASME, Vol. 106, PP. 415-419.

[12] Sarihan, V. and Song, J., (1990). "Optimization of the Wrist Pin End of an Automobile Engine Connecting Rod With an Interference Fit", Journal of Mechanical Design, Transactions of the ASME, Vol. 112, PP. 406-412

[13] Niemi, E., (1995). ''Stress determination for fatigue analysis of welded components'', The International Institute of Welding, Abington Publishing, Cambridge, UK, Doc. IIS/IIW-1221-93.

[14] Monahan, C. C. (1995). Early fatigue cracks growth at welds. Computational Mechanics Publications, Southampton, UK.

[15] Glinka, G. and Shen G., (1991). "Universal Features of Weight Functions for Cracks in Mode I", Engineering Fracture Mechanics, vol. 40, No. 6, pp. 1135-1146

[16] Wu, Z., Glinka G., Jakubczak, H., and Nilsson, L., (2003). "Calculation of Stress Intensity Factors for Cracks in Structural and Mechanical Components Subjected to Complex Stress Fields", Fatigue Testing and Analysis Under Variable Amplitude Loading Conditions, ASTM STP 1439, P. C. McKeighan and N. Ranganathan, Eds., ASTM international, West Conshohocken, PA.

[17] Glinka, G; Goyal, R; El-Zein, M. (2013). ''Stress data for fatigue analysis of welded structures'' Welding in the World, Le Soudage Dans Le Monde Vol 55 pp 7-8.

[18] Harusinec, J., Suchanek, A, Gerlici, J. & Lack, T. (2012). Locomotive brake unit modification for laboratory experimental tests. In Dynamics of rigid and deformable bodies 2012 (electronic source): Proceedings of international scientific conference, Usti nad Labem, Czech Republic, 10 - 12 October 2012 (9 pages). Ústí nad Labem, Czech Republic.

[19] Svoboda, M. Soukup, J. (2013). Verification of numeric solution by experiment for examination vertical oscillation of a mechanical system. Manufacturing Technology, 13(4), 559-563.

[20] Hauser, V., Stastniak, P., Gerlici, J. & Lack, T. (2014). Design solution of stanchions mechanism for goods mounting during transport on the Shimmns wagon. In: Innovations in conception, design, production and testing the freight wagons I.: Proceedings of conference, Žilina, Slovak Republic, pp. 47-53. Žilina, Slovak Republic.

[21] Suchanek, A., J. Harusinec, J., Gerlici, J. & Lack, T. (2013). Analysis of models for simulation computations and experimental detection of stress in braked railway wheel In: TRANSCOM 2013: 10th European conference of young researchers and scientists: Žilina, pp. 283 - 287. Slovak Republic. Žilina, University of Žilina

[22] Klimenda, F., Skocilasova, B. (2015). Rollers vibration of pipe conveyor. Manufacturing Technology, 15(6), 991-995.

[23] Bathe, K. J. (1996). Finite Element Method. Prenice-Hall, Inc. New Jersey.

[24] Lack, T., Gerlici, J. & Manurova, M. (2016). Freight car bogie properties analysis by means of simulation computations. Manufacturing Technology, 16(4), 667-672

[25] Štastniak, P., Gerlici, J., Lack, T. & Harusinec, J. (2013). Computer aided simulation analysis for computation of modal analysis of the freight wagon. In: TRANSCOM 2013: 10th European conference of young researchers and scientists: Žilina, June 24-26, 2013, (pp. 297-300). Slovak Republic. Žilina, University of Žilina.

[26] Gerlici, J., Lack, T. (2014). Rail vehicles brake components test bench utilisation. Applied Mechanics and Materials, 486, 379-386.

[27] Gerlici, J., Lack, T. & Harusinec, J. (2014). Development of test stand prototype for rail vehicles brake components testing. Komunikacie, 16(3A), 27-32.

[28] Melnik, R., Sowinski, B. (2014). The analysis of rail vehicle model eigenvalues for suspension fault detection method. In: 14th Mini Conference on Vehicle system Dynamics, Identification and Anomalies, VSDIA 2014, Budapest University of Technology and Economics, Hungary, 10 - 12 November, 2014, (pp. 39-46). Hungary, Budapest.

[29] Steisunas, S., Bureika, G. (2014). Study of freight wagon running dynamic stability taking into account the track stiffness variation. Transport Problems, 9(4), 131-143.

[30] Oterkus, E; Diyaroglu, C; De Meo, D; Allegri, G (2016). 'Marine Applications of Advanced Fibre-Reinforced Composites', Pages 79-102.

[31] Krueger, R. (2002). "The virtual crack closure technique: history, approach and applications", NASA CR-2002-211628.

[32] Rybicki, E. and Kanninen, M. (1977). "A finite element calculation of stress intensity factors by a modified crack closure integral", J. Eng. Fract. Mech., Vol 9, pp. 931-938

[33] Yung-Li Lee, Mingchao Guo (2012). 'Metal Fatigue Analysis Handbook

[34] Fujiyama, K. (2016). ''Damage Informatics and Materials Engineering for Optimizing Operation and Maintenance of Power Plant Components'', in Reference Module in Materials Science and Materials Engineering

[35] Darshit Nayak, Dr. Pushpendra Kumar Sharma, Ashish parkhe, (2014). "Modelling and analysis of existing and modified chassis in Tata truck "International Journal of Advanced Technology in Engineering and Science, Volume No. 02, Issue No. 05.

[36] NX Nastran Basic Dynamic Analysis User's Guide, 2003.

[37] Masahiro Koike, Sanshirou Shimoda, Toshihide Shibuya, Hirofumi Miwa, (2004). "Development of Kinematical Analysis Method for Vehicle". Komatsu technical report, Vol. 50, No. 153

[38] Haghani, R. (2014). ''Finite element modelling of adhesive bonds joining fibre-reinforced polymer (FRP) composites to steel'', in Rehabilitation of Metallic Civil Infrastructure Using Fiber Reinforced Polymer (FRP) Composites

[39] Cicek Karaoglu, Sefa Kuralay, N. (2000). "Stress analysis of a truck chassis with riveted joints" Elsevier Science Publishers B.V. Amsterdam, the Netherlands, Vol 38, pp. 1115-1130

[40] Rabb, R., (1996). "Fatigue Failure of a Connecting Rod", Engineering Failure Analysis, Vol. 3, No. 1, PP. 13-28

[41] Mehdi Mahmoodi-k, Iraj Davoodabadi, Vinko Višnjić, Amir Afkar, (2014). "Stress And Dynamic Analysis Of Optimized Trailer Chassis" Tehnički vjesnik 21, 3, 599-608

[42] Roslan Abd Rahman, Mohd Nasir Tamin, Ojo Kurdi, (2008). "Stress analysis of heavy duty truck chassis as a preliminary data for its fatigue life prediction using FEM" Jurnal Mekanikal, No. 26, 76-85

[43] Bannantine, J., Corner, Handrock, (1990). Fundamentals of Metal Fatigue Analysis, Prentice-Hall.

[44] Dowling, N.,(2011). Mechanical Behaviour of Materials, Prentice Hall,.

[45] Milne, I; Karihaloo, B; Ritchie, R. O., (2003). Comprehensive Structural Integrity, Volume 1, Pages 1-24

[46] ' 'What type of problems Finite element methods FEM cannot solve?'' www.researchgate.net/post. Retrieved May 18, 2019

[47] Admin (2018). ''4 major challenges with finite element analysis''
www.zcads.com.au. Retrieved May 19, 2019

[48] Nam-Ho Kim, ''Finite Element Analysis of Contact Problem''
www2.mae.ufl.edu. Retrieved May 19, 2019

[49] shodhganga.inflibnet.ac.in/bitstream/10603/139421/13/13_chapter%205.pdf
Retrieved 19-05-2019

[50] ''What's the difficulty in the finite element method for problems with singularity?''
Quora.com. Retrieved May 18, 2019