Available online at www.elixirpublishers.com (Elixir International Journal)

Mechanical Engineering

Elixir Mech. Engg. 93 (2016) 39506-39510

ISSN: 2229-712X

Functional Design for the Manufacture of Quality and Cost Effective Assembly Components of an Automobile Car using FEA

Mohammed Yunus[1,*], Mohammad S. Alsoufi[1] and Mahaboob Tabriz Basha[2]

[1]Department of Mechanical Engineering, Umm Al-Qura University, Makkah, KSA.

[2]Department of Mechanical Engineering, H.K.B.K. College of Engineering, Bangalore, India.

ART IC LE INF O

Article history:
Received: 29 February 2016;
Received in revised form:
3 April 2016;
Accepted: 8 April 2016;

Keywords
Computational materials science,
Finite Element Method, Materials,
Composites,
Engineering polymers,
Biomaterials.

ABSTRA C T

The Finite Elements Analysis (FEA) is the powerful tool for solving a wide variety of engineering problems with applications covering different fields of modern science and manufacturing industries such as material engineering, mechatronics, biochemistry, medicine, aircraft, space, ship building, electrical and computer industry. FEA simulation is utilized to obtain better, accurate and wider results than experimental methods in strain analysis of joints and assembly structures of vehicles. It allows the kind of stresses occurring in the vehicles and the location their occurrence to be determined. This paper highlights the complete functional design, methodology, approach for manufacturing of an automobile car keeping in view the life expectancy, quality, reliability and cost effectiveness, using the FEA analysis and virtual prototyping to predict the final shape of an assembled structure. Methodology of sequentially simulating steps in the manufacturing process of an assembly of automobile is proposed. Each step of the proposed methodology is described followed by a validation performed by making a comparison with a physically manufactured assembly. The trial and error process at development stage may be shortened by replacing most of the physical try out with FEA simulation of the manufacturing process. The automotive industry is constantly finding ways and means of reducing costs and remaining profitable while delivering on time. In developing indigenous design and manufacturing technologies to emerge successful in the world market and able to sustain and innovate at a faster clip than the rest of the competition, companies require huge investments.

Introduction

Finite Elements analysis (FEA) is useful in solving a wide range of engineering problems. During the first fifteen years of its existence, this method was the subject of intensive studies and as a result its general rules and mathematical basis were formed. By the middle of the seventies the method had already been widely used in engineering design and its more intensive development has been held back merely by the insufficient calculative powers of computers [1]. It appeared in the form of FEA tool on which programs of large calculation units were replaced as an effective tool in solving problems by carrying out analyses and designing of geometric models and made commercially available. Emphasizing the huge possibilities and usability of FEM, special attention shall be paid to the necessity for cautious and thought-out application of this method [2]. It must be underlined that it is an approximation method. Solutions obtained from the results of analysis do not represent the real constructional system, but their models. The difference is between a real problem and the solution of a model calculation. There are shown multistage steps and simplifications made from the reality to the final solution [3].

From wide range of FEA applications (engineering to medical field), it is drawing attention of the manufacturing industries [4]. Both technological and economic pressure in product engineering require the introduction of continuous optimization of existing manufacturing processes, including those at the design stage. Recently, the FEA has been emerging as successful design tool in field of medicine and the manufacturing industry. FEA simulation provides more accurate and extensive data than experimental methods as we find in strain analysis of joints [6]. It allows determination of the kind of stresses occurring in the car, the location of its main occurrence and the influence of the stress pattern in developing the final shape of car body [7]. With the construction of the rib model using FEM simulation, the analysis can be related with rib behaviour subjected to the high impact force, e.g. during an automobile accident. Finite Element Method is also applied in creation of biostatic models of the mandible or one-layer aortal valve, additionally, there are realized works connected with coronary, urological or oesophageal stents. Considering the prerequisites mentioned above, FEM appears to be the natural choice for successful simulation of the internal prosthesis of the oesophagus [8].

An increasing number of components in automotive structures are today made from Advanced High Strength Steel (AHSS). Since AHSS demonstrates more severe spring back behaviour than ordinary mild steels, it requires more effort to meet the design specification of the stamped parts.

*Tele: 00919141369124, 00966535459243
*e-mail address: yunus.mohammed@rediffmail.com

Consequently, the physical fine tuning of the die design and the stamping process can be time consuming. FEA handles such situation with ease (e.g. forming process including spring back behavior) [9]. Still it is difficult to identify when a stamped part will lead to an acceptable assembly with respect to the geometry and the residual stress state. In parts, the assembling process itself will distort the components. To resolve this matter it is here proposed to extend the FE simulation of the stamping process to also include the first level sub-assembly stage. In this study, a methodology of sequentially simulating each step in the manufacturing process of an assembly is proposed[1]. The work presented here demonstrates that by using virtual prototyping it is possible to predict the final shape of an assembled structure.

Increasing consumer demands for a high degree of safety, fuel-efficiency and low emission vehicles, forces the automotive industry to increase its use of high strength materials in various structural parts. This may cause problems during the manufacturing of such parts and their subsequent assembly. Virtual prototyping of the dietry-out phase, based on FE simulations of the forming process, is an efficient methodology to manage these difficulties [11].

In the field of geometry in relation to the prediction of assemblies, the main work has been carried out in the analysis of dimension variations. These analyses are performed during the early design stages to avoid variation-related production problems. A deterministic FE simulation of the manufacturing process chain is another approach to evaluate the manufacturing process and the properties of the final assembly [12].

Much information about the manufacturing process may be gained from FE simulations of the entire manufacturing process chain, i.e., from forming of the individual sub-components to the assembling process. Different fixture designs as well as clamping and welding sequences can be explored. Predictions can be made as to whether the final assembly will meet the design specification, or if some alteration of the assembling procedure or the stamping process of the individual components must be made [13].

In this contribution, a methodology to predict the properties of a sheet metal assembly is presented. By sequentially simulating each step of the manufacturing process, the methodology can be applied to describe the evolution of deformations and residual stresses throughout the entire manufacturing process chain. The virtual prototyping process starts with the creation of FE models for forming and trimming operations, including the associated spring back, of the individual components. The FE components obtained are then positioned and clamped to the assembly fixture, and their flanges are clamped together. The flanges are joined with spot welds and, finally, a spring back analysis of the complete assembled structure is carried out. The geometry of the assembled part obtained from the FE simulations is compared with the geometry of the part obtained from physical tests.

Challenges of Automotive Industry

The automotive industries have many requirements to be addressed for developing indigenous design and technologies to emerge successful in the world market are possible on account of the following capabilities [14]:

Delivering products faster with up-front engineering design validation as a part of the product development process while compressing the cycle time.

Lean design by incorporating value engineering as a part of the design process ensuring evaluation of least cost alternatives before even the first prototype is developed.

Increasing reliability by design – an initiative that will eliminate hidden costs associated with product re-call, re-design and/or replacement of parts.

Value facture of a product ensures product acceptance by the customer while assuring improvement in bottom-line profitability. Design engineering had been identified as a profit centre and there has been continued investment in tools and technologies that augment design validation with its assured return on investment. Figure 1 shows the finite element mesh of an Automotive Body using FEA.

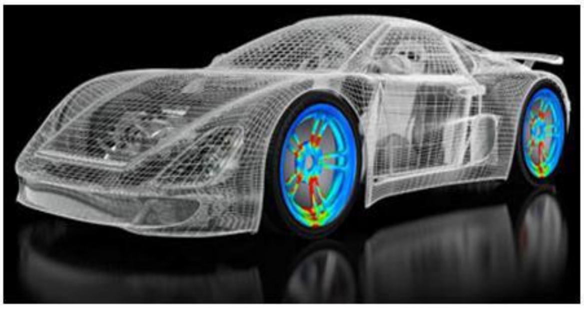

Figure 1. FEA of an automotive body.

Increasing functional complexity, predatory pricing, compulsive cost reduction commitments, aggressive deadlines, field failures, warranty costs, product recalls are all too familiar situations in the automotive sector. Almost all automotive tier I and II suppliers and their vendors share these common issues. All cross-functional teams in every organization deal with one thing in common – drawings. As a reflection of perfection in design, drawings affect the profitability of any organization. Perfection in design is achieved only by validating the design concept and continuously refining the design to enhance value and optimize many performance and reliability aspects of their design and manufacturing processes. Benefits include: greater innovation, process automation, shorter development cycles, design for durability, reduced physical prototypes, improved quality and innovation.

The Automotive Industry Routinely Addresses

- Design and simulation of full vehicle modeling as well as component level such as chassis, engine, driveline, body, etc.
- Engine operating environment under structural and thermal loading to assess durability.
- Transmission performance, including non-linear and frictional effects.
- Component design.
- Power train flexural and torsional dynamic response. Noise and vibration of the Body-in-White.
- Thermal cycling.
- Mechanism analysis.
- System and component performance and durability.
- Assembly analysis of mechanical joints such as bolts, spot welds, and seam welds.
- Low speed impact analysis.

Functional Design Validation for Automotive Components and Systems

Figure 2 illustrates the automotive body parts design and engineering, including cowl, wheel house, pedal, roof rail, locker, floor structure, seat frame, bumper reinforcement, bumper, side member, fender shield and instrument panel reinforcement.

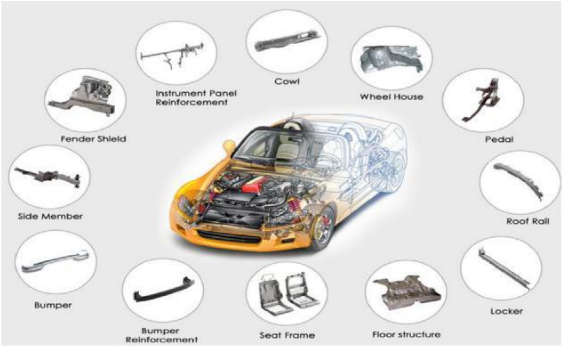

Figure 2. Design and engineering of automotive body parts.

Stiffness and Deflection Analyses

Stiffness of automotive parts is vital to meet performance criteria, especially in parts such as brackets, mounts, body structure components, machined castings among others (see Figure 3). Stiffness studies find immediate relevance in the following areas:

- Anti-vibration mounts design.
- Suspension attachment to body structure. Instrument panel support structure.
- Brackets used in mounting sub-systems such as water pump, oil pump, radiator, cross-member reinforcements, steering attachment among others.
- Machining of casings – to maintain tolerance of straightness, flatness, location and orientation.
- Tyre deflection due to inflation and loading – for ride comfort and force transmissibility studies.

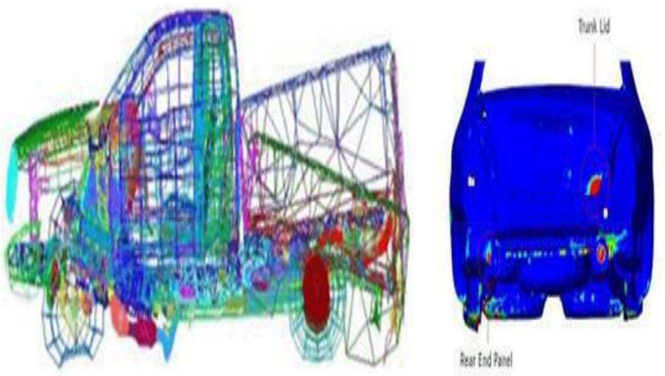

Figure 3. Stiffness and deflection analysis results at f = 11.2 Hz.

Frequency and Buckling Analyses

It is a well-known fact that resonant conditions reduce the life of components. Systems designed for limit loads in axially-loaded members need to have buckling calculations done to ascertain safety and reliability. Figure 4 shows the results of frequency and buckling analysis some of the benefits of frequency and buckling analyses for automotive systems include:

- Natural frequency calculations for body structures, mounts, support brackets, rotating components including fans, transmission elements, power-train among others.
- Limit load calculations using linearized buckling analysis for steering systems, suspension components, engine components, brackets and body structure sub-systems.
- Frequency calculations are a pre-requisite for performing response estimations based on harmonic and random excitations.

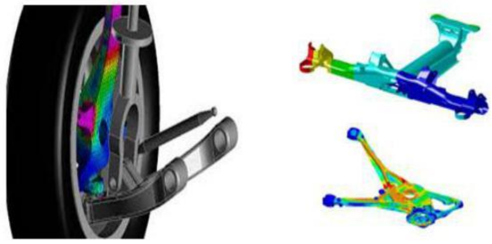

Figure 4. Results of frequency and buckling analysis.

Durability and Fatigue Analyses

A vehicle is tested on a 4-post shaker system using road-loads as input. At the end of testing components are found to have failed. Since loads transmitted to the failed components would not be known, re-design to overcome the failure is a daunting task. A system level analysis with fatigue calculations would throw light on vulnerable areas in the system and help address the failure issues. This is a powerful approach to solving such issues in addition to predicting possible modes of failure. Components are subjected to cyclical load testing on test rigs to ensure required numbers of cycles are completed without evidence of failure. This requirement can be easily tested on a digital prototype virtually to ascertain zones of failure, cycles of failure and modes of failure. When this exercise is carried out at the design stage, it is possible to evaluate alternate designs in order to arrive at least cost designs for higher profitability.

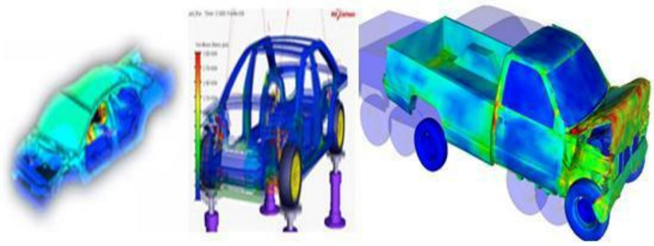

Figure 5. Results of durability and fatigue analyses.

Vibration and Dynamic Analyses

Accelerated durability testing is conducted by placing automotive subsystems on shaker tables and subjecting the same to varying base excitations such as fixed-amplitude sine-sweep or random excitations. Life is consumed in these tests rapidly leading to a failure. If no failures are evident after a stipulated duration, the component(s) are said to have passed the criteria. This can be easily verified on digital designs using a simulation approach first by extracting the required number of modes and then performing a post-dynamic analysis in order to compute the response characteristics. The advantage of digital simulation is enhanced by performing a combination of what-if scenarios that is generally unavailable in test rigs unless an expensive investment is made for the special purpose of testing equipment. Figure 6 shows the results of modal and harmonic analysis using FEA.

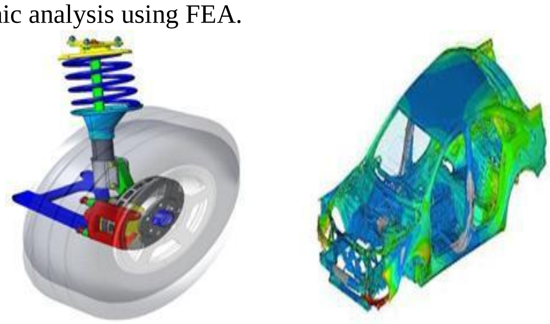

Figure 6. Results of modal and harmonic analysis using FEA.

Thermal and Fluid Flow Analyses

Estimation of temperature, pressure, velocity among other parameters can be computed using Computational Fluid Dynamics Software (CFD) in combination with Finite Element Analysis (FEA) for thermal deformation and effect of pressure on structural side. Be it electronic thermal management of engine control systems, climate control devices, radiator circuit, fan studies, brake rotor cooling, hot- and cold- chamber testing, simulation has come a long way in addressing critical performance challenges and optimization of design. Several correlations between test and analysis data have revealed the powerful technological advantage a manufacturer enjoys by implementing these solutions effectively. CFD is very useful in estimating characteristics of water pump, oil pump, fuel delivery devices and systems among other sub-systems. Figure 7 shows the results of CFD for thermal deformation and pressure on structure.

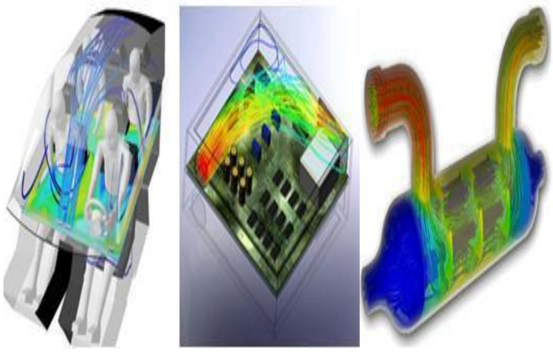

Figure 7. Results of CFD for thermal deformation and pressure on structure.

Non-linear Analysis

Critical sealing systems, lightweight composites, manufacturing process simulations require sophisticated non-linear analytical treatment. Effectiveness of sealing systems under assembled conditions involving initial deformation can be found using Non-linear Simulation Analysis (see Figure 8) [18]. Large deformation analysis and large displacement analysis ensure that product safety and performance are not affected during function. Seals, gaskets, bellows, hoses, weather beading, tire inflation and loading are some of the areas in which Non-Linear Analysis finds application in the automotive sector.

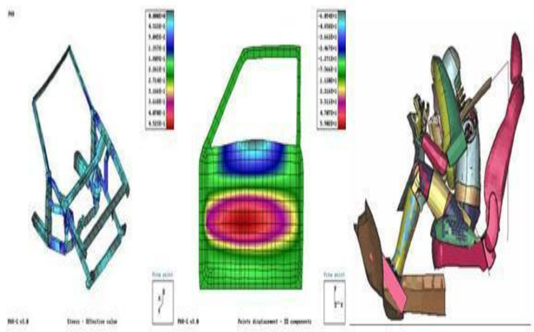

Figure 8. Results of large displacement and non-linear analysis of automotive components.

Kinematic Analysis

Mechanisms find use in the automotive industry extensively. Door closure, windows, accelerator, braking, fuel-trap-door opening, transmission gear selection, hood/deck-lid opening, seat actuation are some of the areas where kinematic analysis is important to ensure fail-safe actuation and performance. Forces calculated using kinematic analysis will be used in FEA to estimate stresses and deflections (see Figure 9) [12].

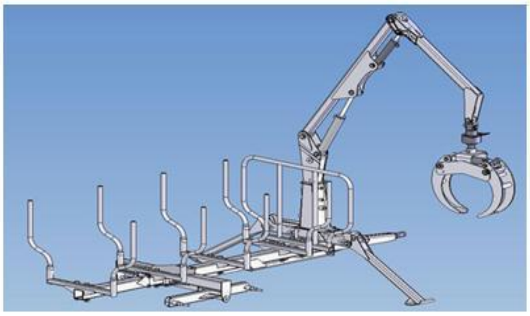

Figure 9. Mechanisms of automotive body.

Conclusion

FEA has been widely accepted design and analysis tool. The applications of FEA and the essential advantages resulting from FEA are presented. Description of the importance and utility of FEM during solving the problems dealing with the very complicated geometrically complex state of loadings, various boundary conditions and/or various materials have been provided. Challenges imposed on automotive industries meeting deadlines have been addressed especially in developing expensive cars. FEA has capability to perform design and simulation of full vehicle as well as component level and provides an extensive data of car for the type of analysis, type of loading, material engineering, mechanism involved, etc. The objective of this work is to present selected problems concerning the application of FEA in developing expensive cars in relation to the example of a chosen program which makes the most of this method to simulation. A 3D model in FEA however requires in depth knowledge of technologists and engineers in Stress-Strain analysis, stress tensors/matrixes, material properties, isotropy and anisotropy, stress states, residual stresses, von-Mises, Mohr circle and basic mechanics evaluations criteria. Even with the software advantages, the output in the FEA models still remains as the engineering duty. The obtained results of FEA calculations can however be used to solve many problems in the initial stages of design with success.

References

[1] Liu, J. P., Ye, B. Y. and Li, G. W. (2011). Application of FE analysis in automotive wheel design. Key Engineering Materials, 455:350-354.

[2] Fagen, M. J. (1992). Finite Element Analysis Theory and Practice. Longman Scientific and Technology.

[3] Bathe, K. J. and Dvorkin, E. N. (1986). A formulation of general shell elements-the use of mixed interpolation of tensorial components. International Journal for Numerical Methods in Engineering, 22: 697-722.

[4] Bathe, K. J. 1996. Finite Element Procedures. Prentice Hall. Englewood Cliffs, NJ.

[5] Nitin Gokhale, S., Sanjay Deshpande, Sanjeeve Badekar, V., Anand Thithe, N. (2008).Practical finite element analysis, 1st ed. Finite to Infinite Publishers.

[6] Abhijeet Shinde and Prof. DhananjayThombare. (2014). Finite element method for stress analysis of passenger car floor. International Journal of Engineering Research and Applications, 4(4 (8)): 38-42.

[7] Zheng, G. J., Zhang, J. W., Hu, P. and Shi, D. Y. (2015). Optimization of hot forming process using data mining techniques and finite element method. International Journal of

Automotive Technology, 16(2) :329−337.
[8] Yildiz,Y. and Duzgun, M. (2010). Stress analysis of ventilated brake discs using the finite element method. International Journal of Automotive Technology, 11(1):133−138.
[9] Gyu Ha Kim, Kyu Zong Cho, In Bum Chyun and Gi Seob Choi. 2003. Dynamic stress analysis of vehicle Frame using a nonlinear finite element method, KSME International Journal, 17(10): 1450 – 1457.
[10] Fernandez, J., Pernia, A., Martinez-De-Pison, F. J. and Lostado, R. (2010). Prediction models for calculating bolted connections using data mining techniques and the finite element method. Engineering Structures, 32(10):3018−3027.
[11] Ravikiran Duggirala, (1989). Design of Forging Dies for Forming Flash less Ring Gear Blanks Using Finite Element Methods. Journal of Materials Shaping Technology, 7: 33-47.
[12] Malagi, R., Kurbet, S. and Gowrishenkar, N. (2011). Finite element study on piston assembly dynamics in multi-cylinder internal combustion engine. SAE Technical Paper, 1:1075. [13] Long, A., Ge, R., Zhang, Y. S. and Wei, X. (2012). Numerical simulation of B-Pillar's hot press forming process and its shape optimization. Applied Mechanics and Mechanical Engineering, 1 - 2, 138-139, 749− 753.
[14] Zheng, G. J., Men, Y. X., Hu, P. and Zhao, F. Q. (2011). Key technologies of CAE data process procedure based on knowledge engineering. Chinese Journal of Mechanical Engineering, 47:1-17.
[15] Ghoo, B., Umezu, Y., Watanabe, Y., Ma, N. and Averill, R. (2010). An optimization study of hot stamping operation. 2, 1252, pp. 537−544.