MASS REDUCTION FOR STEERING KNUCKLE ARM IN A SUSPENSION SYSTEM THROUGH TOPOLOGY OPTIMIZATION IN CAE

[1]Kamlesh Lalasaheb Chavan,[2]S R Deodas, S.S.Kulkarni[3]
[1]ME (Design Engg), D.Y. Patil College of Engg., Akurdi, Pune, India
[2]Assistance Professor, BE (Mechanical),ME (Heat Power),P G coordinator (Heat Power Department) D Y Patil college of engineering, Akurdi , Pune, India
[3]Director, Able Technologies (I) Pvt. Ltd., Pune, India

ABSTRACT

The steering knuckle on your vehicle is a joint that allows the steering arm to turn the front wheels. The forces exerted on this assembly are of cyclic nature as the steering arm is turned to maneuver the vehicle to the left or to the right and to the centre again. For the proposed work over the `steering knuckle arm', the objective is to identify the nature and magnitude of stresses over the component while contemplating improvement in the design. Taking into consideration static and dynamic load conditions, structural analysis and modal analysis were performed. Finite element model was developed in Hypermesh10. 10 node tetrahedral elements were used for meshing, providing better results in less time. FEA/ CAE software like Abacus solver shall be engaged to simulate and predict the level of stresses for the given input loads. Optistruct is used as optimization tool for finding out the stress distribution, density distribution over the component. After analyzing the result from optistruct the component is modified by removing the material from low stress region. After performing the analysis on modified component it is observed that the stress and the displacement are reduced. Hence it is observed that the component is safe under given loading conditions and stresses are below yield stress. For validation of the project experimental setup was designed which consist of Universal testing machine for applying the loading conditions, strain gauge for measuring the displacement and suitable fixture for holding the component is used. After applying the loading the stress and displacement are recorded. It observed that part is safe under given loading conditions. And percentage variation of result with the FEM analysis is about 1.01.

KEYWORDS: *Steering knuckle component, topology optimization, experimental analysis*

I. INTRODUCTION

Steering knuckle is the most important part in the vehicle but not everyone knows what this for steering knuckle is a forging that usually includes the spindle and steering arm, and allows the front wheel to pivot. The knuckle is mounted between the upper and lower ball joints on a SLA suspension, and between the strut and lower ball joint on a MacPherson strut suspension.

The steering knuckle on your vehicle is a joint that allows the steering arm to turn the front wheels. The forces exerted on this assembly are of cyclic nature as the steering arm is turned to maneuver the vehicle to the left or to the right and to the centre again.

Figure1. Forged component of steering knuckle

The stress analysis is carried out for the extreme conditions, and the purpose is to investigate the structural strength. In actual use, the structure bears the unlimited loads mainly.

The knuckle exist three types of extreme conditions, namely:

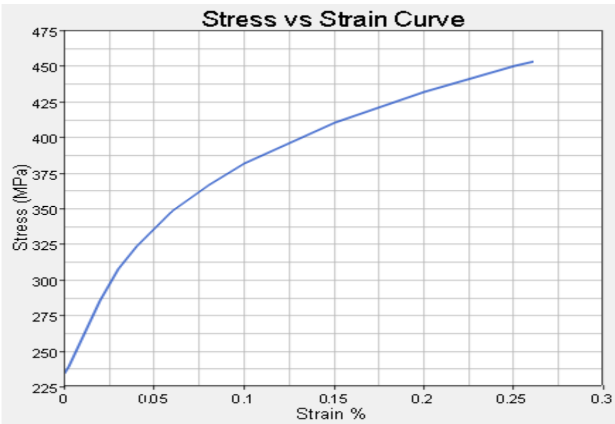
1) The uneven ground conditions
2) Emergency braking conditions.
3) And side slip conditions.

Material properties

The material used for the production of steering knuckle is S32-HSS/ EN42 (Ferrite Ductile iron). EN42is supplied in accordance with our ISO 9001: 2008 registrations.

Table 1 Composition of material

Content	**Percentage composition**	**Content**	**Percentage composition**
Carbon	0.42-0.44%	Silicon	0.10-0.35%
Nickel	3.90-4.30%	Manganese	0.40-0.60%
Chromium	1.10-1.40%	Phosphorous	0.050% max
Molybdenum	0.20-0.40%	Sulphur	0.050% max



Graph 1 Material Non linear curve

Graph 1 shows material non linear curve for the material EN42.

1. Need

It has been observed from the knuckle manufacturers that weight reduction and advanced materials are the real need for the current automobile industry.

The steering knuckle accounts for maximum amount of weight of all suspension components, which requires high necessity of weight reduction. Under operating condition is subjected to dynamic forces transmitted from strut and wheel. The weight reduction of steering knuckle is done such that the strength, stiffness and life cycle performance of the steering knuckle are satisfied. Topology optimization is carried out to produce lighter, less expensive & more efficient steering knuckle that exhibits precise dimensions, need less machining & requires less part processing.

2. **Problem Formulation**

The steering knuckle accounts maximum weight of all suspension components which requires high necessity of weight reduction.

The current design challenge for the Steering Knuckle Arm is to generate the most optimal configuration of the component design for the given input conditions of loading.

Find out stress distribution on the component after loading.

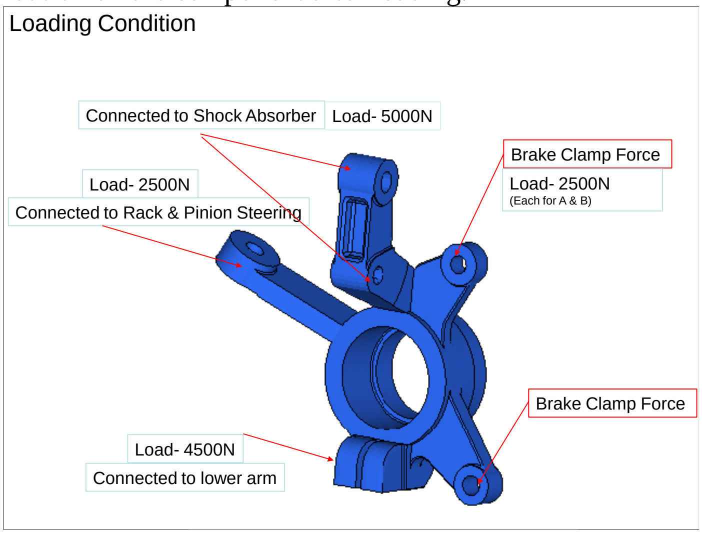


Figure2. Loading conditions for steering knuckle

Input conditions: geometry of the component, dimensions of the component, material specifications, material properties, loading conditions and weight of the component. The loading conditions are shown in above figure

It is subjected to 5000N load which is connected to suspension system, 2500N connected to rack and pinion of steering, 2500N connected to brake clamp. Weight of steering knuckle is 3.25 Kg and yield stress 235.3 Mpa. Material is EN 42 as per the ISO standards and it is homogeneous and isotropic properties. For these loading conditions optimize the steering knuckle using topology optimization and find out the location from where we can remove the material, to which extend we can remove the material and in what shape we can remove the material

II. Objectives Of The Work

This project aims to contribute to the development of structural design and mass reduction of vehicle components using topology optimization method. Optimization process for this work would be conducted using OptiStruct in order to reduce the mass of the component which will reduce the cost with respect to the mass production process.

- To conduct structural analysis using linear static method of FEA.
- To identify areas of concern for improvement.
- To suggest suitable alternatives to design for enhancement of strength or mass reduction.
- To perform experiment in an effort to validate the design.

III. Methodology

For the optimization various CAE tools are used

For generating the 3D model there is software like Catia, Pro-E, solid edge but Catia is preferred because our geometry containing the surface features and Catia is mostly used for surface features. Meshing can done by various software like Hypermesh, Ansys. We have selected Hypermesh because

it is having more user friendly interface for complex geometry features. For analysis software like Ansys, Abacus, Nastran, Radioss, Patran are used. But for our geometry Abacus is preferred because our geometry of the component is non linear. And also it is suggested by the sponsoring company.
Preliminary analysis will be conducted to the initial model using OptiStruct to get initial required information (stress distribution, displacement, maximum stress, etc). That information would be used as reference for optimization process setting and compared to the optimized model for assessing optimization process performance. There are some iterations and evaluations during the optimization process to achieve an optimized model.
After getting required stress distribution through the component we can find out minimum and maximum stress on the component. Hence we can remove the material from low stress region.
After optimizing the geometry run the analysis trough Abacus and view the result in Hyperview.
Validation process is important step in this design optimization. The optimized model's performance is compared with initial models. If the result is not satisfactory, shapes redefinition is required to explore others possible design space.
Shape optimization was applied to reduce volume of steering knuckle model. OptiStruct was used to perform the process. Furthermore, Hyperview and Hypergraph were used to display and plot the data for results interpretation.

IV. FEA ANALYSIS

- Analysis of existing component
- Analysis of optimized component

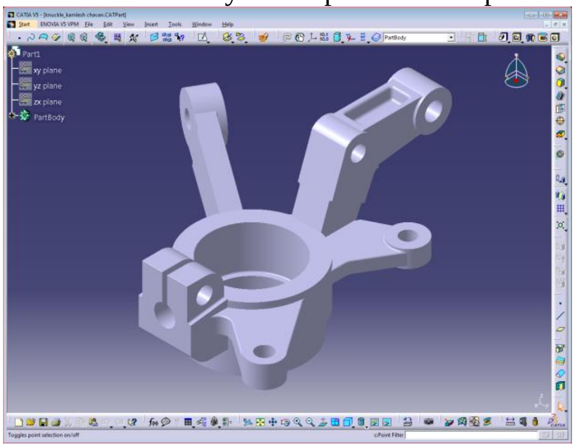


Figure 3 Catia model of existing component

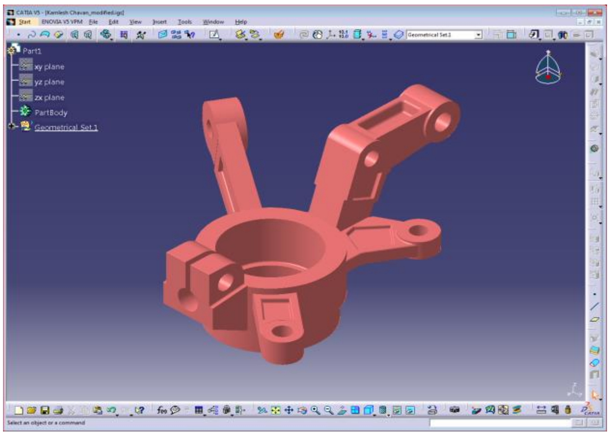
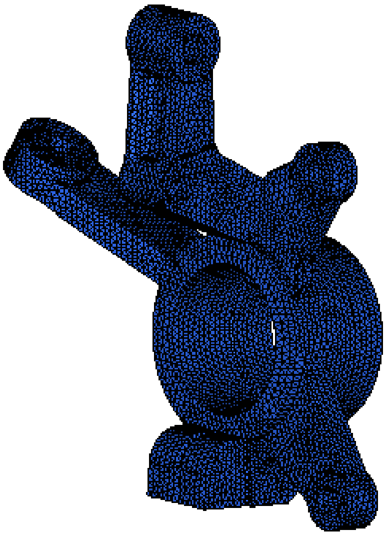


Figure 4 Catia model of optimized component

Table 2: Meshing details for Existing frame

Name of element	C3D10
No. of Elements	72,189
No. of Nodes	1, 17,457

International Journal of Scientific Research and Management Studies (IJSRMS)
ISSN: 23493771 **Volume 1 Issue 11, pg: 363-371**



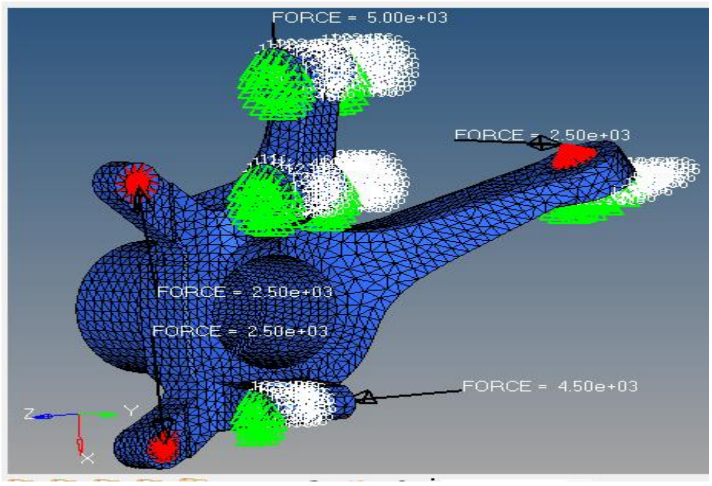
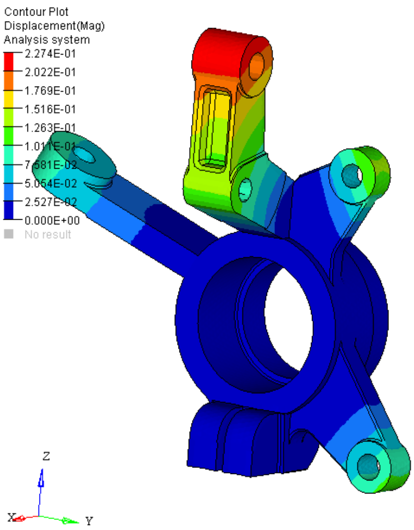


Figure 4 Meshed model of existing component; **Figure 5** Loading conditions on the component



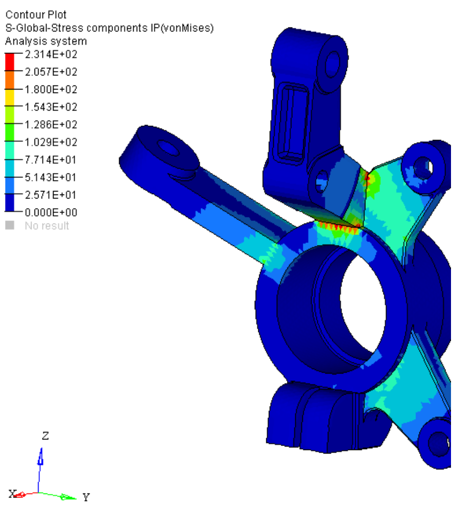


Figure6 displacement analysis of existing component, **figure 7** Stress analysis of given component

Table 3: Material Properties for Existing component

Sr.No	Parameter	Description
1.	Material	EN42
2.	Youngs moduus	210Mpa
3.	Density	7.85 E-09 Tonns/mm^3
4.	Poissons ratio	0.3
5.	Yield stress	235.4 MPa

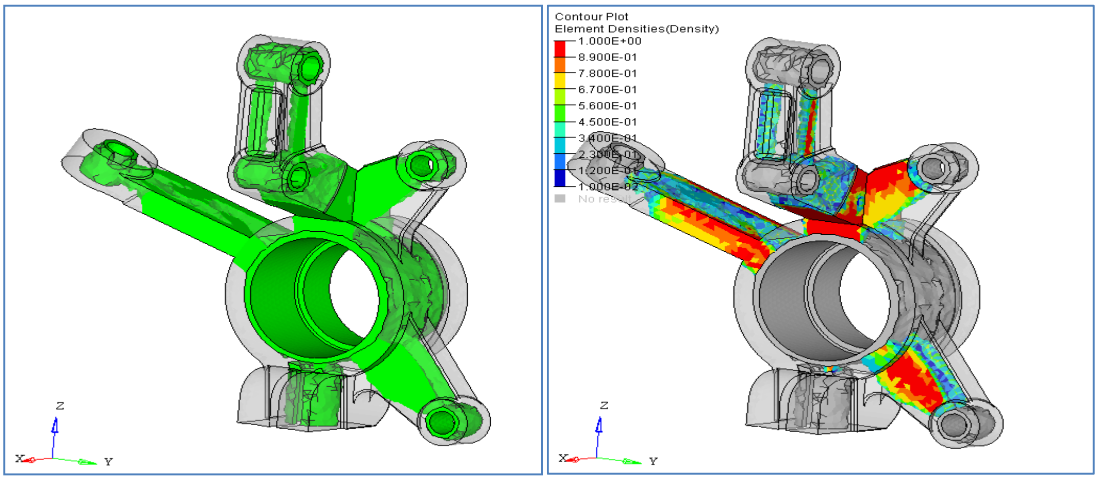


Figure 8 material stress density distribution

V. OPTIMIZATION

Topology optimization

Optimum material layout for a given design space which takes into any number of design constraints. By defining a design space that the engineer has to work in and applying boundary conditions such as predefined loads and fixture positions, topology optimization can suggest the ideal layout of material to meet defined performance targets.

Topology optimization can be used at the concept level of the design process to arrive at a conceptual design proposal that is then fine tuned for performance, weight and manufacturability. This process replaces time consuming and costly design iterations and hence reduces design development time and overall cost while improving design performance.

Optistruct

It uses highly advanced optimization algorithms; OptiStruct can solve the most complex optimization problems with thousands of design variables in a short period of time. OptiStruct's advanced optimization engine allows users to combine topology, topography, size and shape optimization methods to create better and more alternative design proposals leading to structurally sound and lightweight design. Manufacturing requirements can also be defined as input to the simulation to create design proposals that are easier to interpret and to manufacture.

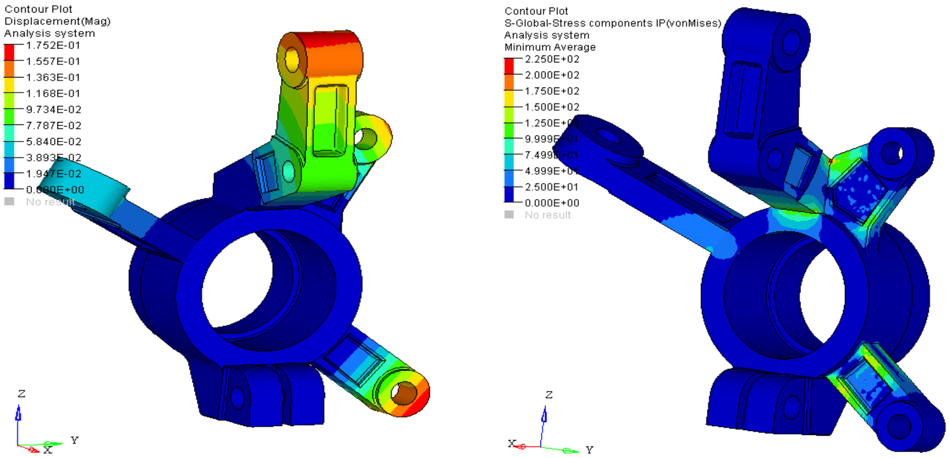


Figure9 displacement and stress analysis of optimized geometry

From the static analysis of the modified geometry it is observed that the maximum magnitude of the displacement is 0.1752mm and the maximum stress 225 MPa observed as shown in encircled region.

VI. RESULTS AND DISCUSSION

The result obtained from analysis of basic geometry are analysed and interpreted. It is observed that steering knuckle is safe under given loading condition and stresses are below the yield stress (235.3 Mpa) and the stresses are observed in local areas which is shown in encircled region in figure25. The maximum stress observed in encircled region is 231.4 Mpa. After carrying the topology optimization the stress level reduced up to 225 Mpa.

- Stress levels are reduced from 231.4 Mpa to 225 Mpa.
- Original Mass 3.25 Kg After modification 2.9 Kg
- Total 11.5 % material reduction
- Modal frequency for the steering knuckle is 1067.26 Hz.

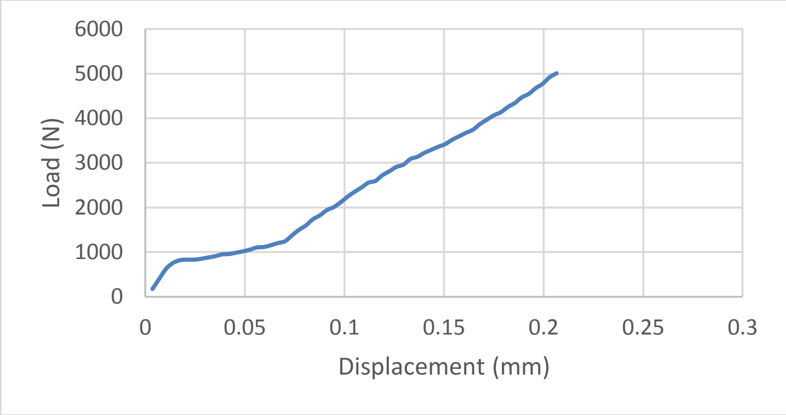
VII. EXPERIMENTATION AND VALIDATION

The component produced for the experimentation is given by the company which is used for the testing. The input conditions are recreated in the lab while the component is being tested for performance. The loading and the boundary conditions will be matching the practical working conditions in which the vehicle is expected to perform. For simplicity, a Universal Testing Machine along with a suitable fixture for the component shall be engaged for testing purposes and strain gauges are being used for recording the strain in the component while loading. The load applied is 5000N which is the maximum load acting along the strut region.

Experimental set up consists of following components

1. UTM (universal testing machine)
2. Steering knuckle
3. Plunger
4. Strain gauges
5. Data logger
6. Fixtures for holding the component.

Figure10 Experimental set up



Graph 2 Load Verses Displacement

Table 4 Result comparison between FEA and experimental

Parameter	Stress determined by FEA	Stress recorded during exp.	Percentage variation in results
Max stress (Mpa)	231.4	233.8	1.02%

VIII. Validation

The Test Report for the component for verifying the results with the Analytical method of analysis are compared it is observed that the results are much closer. Typically, depending on the type of Test and the application, an error margin or about 5 to 10% could be considered close towards validating the proposed design.

The stress determined by FEA is 231.4 Mpa and the stress determined by the test is about 233.8 Mpa. Hence the percentage variation in result is 1.02 %.

IX. Conclusion

The results obtained are quite favourable which was expected. Finite element analysis is effectively utilised for addressing the conceptualisation and formulation for the design stages. The stresses derived during the analysis phase normally indicate the potential solution. The iteration are carried out in the analytical phase which yields the suitable values for design parameter.

As the conclusion following points can be drawn

1. Part is safe under the given loading conditions.
2. The working stresses are less than the yield stress by around 22% to 24% which improves the design life of steering knuckle.
3. To improve performance geometry has been modified using topology and free size optimization which enables to reduce stress levels marginally below yield limit.
4. Less process time will required to melt the material of modified component is reduced.
5. As the cost of component is reduced, market demand and profitability will increase which makes the product competitive in the market.

References

[1]. S. Vijayarangan, N. Rajamanickam, 'Optimization and Finite Element Analysis of Steering Knuckle, Altair technology conference, 2013.
[2]. Viraj Rajendra Kulkarni, Amey Gangaram Tambeb, 'Evaluation of metal matrix composite to replace spheroidal graphite iron for a critical component, steering knuckle,' pp 532–541, 2013.
[3]. Kwang-Seon Yoo, Seog-Young Han, 'A modified ant colony optimization algorithm for dynamic topology Optimization,' pp 68–78, 2013

[4]. J. Paris, F. Navarrina, 'Stress constraints sensitivity analysis in structural topology optimization,' pp 2110–2122, 2010.
[5]. Chang Yong Songa, Jongsoo Lee, 'Reliability-based design optimization of knuckle component using conservative method of moving least squares meta-models,' pp 364–379, 2011.
[6]. Xu Guo, Wei Sheng Zhang, 'Stress-related topology optimization via level set approach,' pp 3439–3452, 2011.
[7]. Hong-Seok Park, Xuan-Phuong Dang, 'Structural optimization based on CAD_CAE integration and metamodeling techniques,' pp 889-902, 2010.
[8]. Wan Mansor Wan Muhamad, Endra Sujatmika, 'Design Improvement of Steering Knuckle Component Using Shape Optimization,' International journal of advanced computer science, Vol. 2, No. 2, pp. 65-69, Feb.2012.
[9]. S.S.Rao. "Finite Element Method in engineering" Fourth edition Elsevier (2006).
[10]. J.N.Reddy "Finite Element Method" McGraw Hill international edition.
[11]. S.Ramamruthum "Strength of Materials" Dhanpat Rai publication Fourteenth Edition (2003).

AUTHOR'S BIOGRAPHY

Kamlesh Lalasaheb Chavan ME (Design engg), D.Y. Patil College of Engg., Akurdi, Pune.

S. R .Deodas Assistance Professor, BE (Mechanical),ME (Heat Power), P G Coordinator (Heat Power Department) D Y Patil college of Engineering, Akurdi , Pune,

Swapnil S.Kulkarni Director, Able Technologies India Pvt. Ltd., Pune. The Company offers Engineering Services and Manufacturing Solutions to Automotive OEM's and Tier I and Tier II Companies. He is a Graduate in Industrial Engineering with PG in Operations Management. With around 20 years of working experience in the domain of R&D, Product Design and Tool Engineering, he has executed projects in the Automotive, Medical and Lighting Industry. His area of interest is Research and Development in the Engineering Industry as well as the emerging sector of Renewable Energy.