International Engineering Research Journal Page No 1977-1990

Topology optimization of steering knuckle arm using finite element method

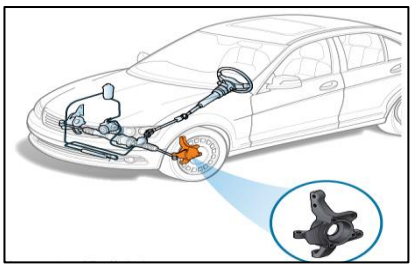
Ms. Kalyani Bhimrao Bhagwat[1], Prof. S. B. Belkar[2], Mr. Swapnil S. Kulkarni[3]
[1]Department of mechanical engineering, Pravara rural engineering college, Loni, India bhagwatkalyani66@gmail.com [2]Assistant professor, Department of mechanical engineering, Pravara rural engineering college, Loni, India belkarsanjay@yahoo.com [3]Director- Able Technologies India Pvt. Ltd., Pune

Abstract *– Mass reduction for the vehicle calls for a comprehensive evaluation of each component in the sub-assembly and the sub-assembly as a whole within the main assembly. For the initial phase of the mass reduction exercise the parts which have higher mass are normally identified as 'eligible' candidates. So steering knuckle arm has been chosen for reducing weight while addressing safety and manufacturability of the part. The chosen part is / about- Kg in weight. Main objective is to reduce it by at least 3~4 % of its existing mass without affecting strength and factor of safety. Mathematical treatment might be limited as steering knuckle arm has irregular geometry and uneven physical features. Topology Optimization is one of tool of weight reduction. Using FEA as computation technique to validate the solution using test setup with static loads.*

Keywords*: Topology Optimization, weight reduction, steering knuckle arm, Optistruct*

I. Introduction

In automotive suspension, a steering knuckle contains the wheel hub or spindle, and attaches to the suspension components. The wheel and tire assembly attach to the hub or spindle of the knuckle where wheel rotates while being held in a stable plane of motion by the suspension assembly. Forces acting on assembly are cyclic in nature. Because of additional luxurious and safety features weight of the vehicle is increasing so it affects fuel efficiency as well overall performance [2-4]. So nowadays it is need to reduce vehicle weight.



"Fig".1 location of steering knuckle arm

Steering knuckle links suspension system through shock absorber, steering system, wheel hub through lower arm and disc brake assembly. Weight reduction of steering knuckle is the objective of this exercise for optimization [1].The finite element software like OptiStruct is used for weight reduction or for optimization as well Nastran/Ansys/Abaqus can also be utilized. The targeted weight reduction is up to 5% without changing structural strength.

II. Literature review

Rajkumar Roy etc. al. [1] (2008) focuses on recent approaches to automating the manual optimization process and the challenges that it presents to the engineering community. **S. Vijayaranganet**.al. [2] (2013) uses the different material than regular material for optimization. Mass reduction upto 55% by using MMC(metal matrix composite) as compared with

SG(ductile) iron, 12% weight reduction by geometry optimization.

Prof. R. L. Jhala etc. al. [3] (2009) assesses fatigue life and compares fatigue performance of steering knuckles made from three materials (forged steel, cast aluminum, and cast iron)of different manufacturing processes. **K. h. Chang and P.S. Tang** [4] (2001) discuss an integrated design and manufacturing approach that supports the shape optimization. The main contribution of the work is incorporating manufacturing in the design process, where manufacturing cost is considered for design. **Jie Zhou** etc. Multi Objective optimization design of a new closed extrusion forging technology for a steering knuckle with long rod and fork To eliminate the internal defects in the existing steering knuckle forming process Orthogonal test method with finite element simulation Influence of each factor as friction condition on forming process & their priorities. **Hong-Seok Park** Structural optimization based on CAD_CAE integration and meta modelling Techniques Improve performance and save material Reduce cost compare to other CAD and CAE tools Meta modelling technique Time saving benefits using meta modelling technique To accelerate product development and to reduce cost.

Summary from literature

i. Use of MMC (metal matrix composite) gives weight saving up to 55% but it is costly as well complex manufacturing process, requires skilled operator.

ii. Review of literature shows that many authors have reported the design to study effect of weight reduction in steering knuckle; they also study stresses developed and material properties.

iii. Analysis is to done to check the effect of variation in the significant design parameter with suitable boundary conditions.

iv. Computational method (CAE software) is identified as the prospective methodology to find effect of weight reduction in steering knuckle with appropriate changes in steering knuckle geometry.

v. Mathematical calculation used to find the stresses developed in steering knuckle geometry and results to be compared with computational method used.

III. Objective

i. Topology optimization of steering knuckle arm of marutisuzuki Estilo.

ii. Analytical and experimental stress analysis (static) of steering knuckle arm.

IV. Methodology

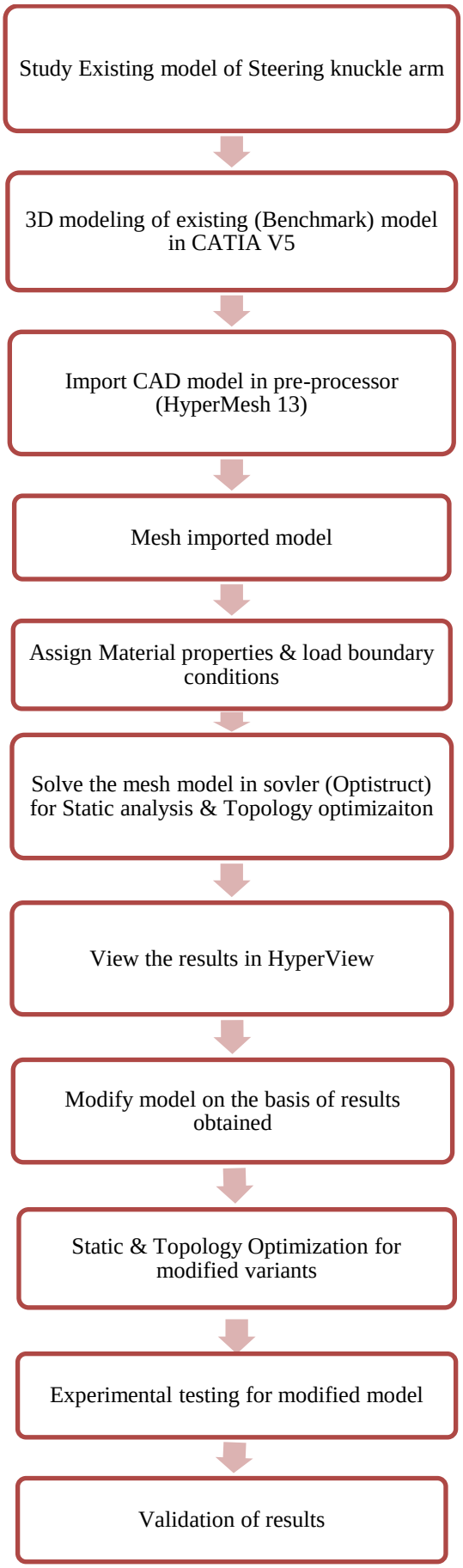
1. Computational Method-

- CAD model of the steering knuckle of automobile.
- Pre-processing the CAD model in Pre-processor
- Applying Material properties, loads, Boundary conditions.
- Exporting to solver to find out the stress developed.
- Viewing results through post-processor
- Revision in geometry and Analysis for improvement
- Find out the best suitable alternative

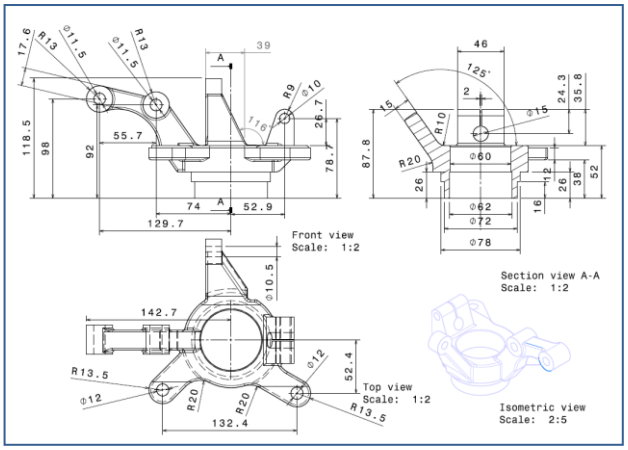
Software Used & to be Used:

- 3D modeling:CATIA V5 (Dassault Systems)
- Pre-processor: HyperMesh (Altair Engineering)
- Solver: Radioss or Optisruct (Altair Engineering)
- Post-processor: Hyperform (Altair Engineering)

.



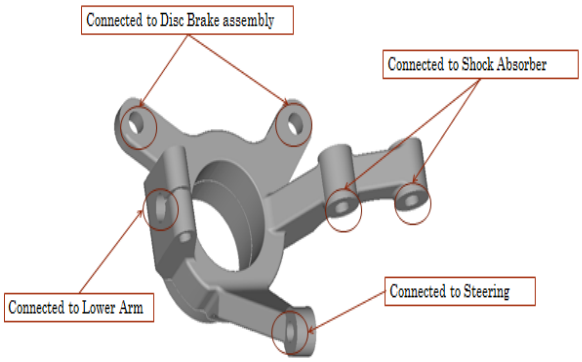
i. CAD model



"Fig" 2 CAD Model

Isometric view of steering knuckle arm having scale 2:5 is shown (see fig. 2). CAD model of steering knuckle was developed in 3D modeling software CATIA. it consists of stub hole, brake caliper mounting points, steering tie-rod mounting points, suspension upper and lower A-arm mounting points. Knuckle design mainly depends on suspension geometry and steering geometry. The 3D model of existing or benchmark steering knuckle arm is created with the help of CATIA V5. Then this CAD model is imported in pre-processor which is Hypermesh.

ii. Connection to matting part



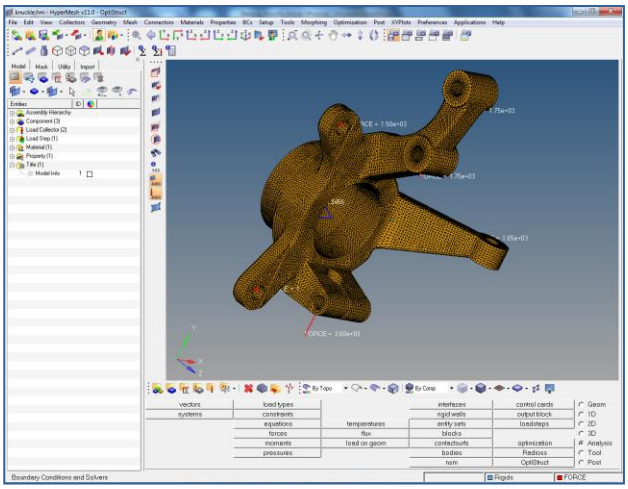
"Fig.3" Connection to matting part

Various connections to matting parts are shown (see fig. 3). As steering knuckle arm links various system, it is connected to lower arm, disc break assembly, steering and shock absorber. CAD model of steering knuckle was developed in 3D modeling software CATIA. it consists of stub hole, brake caliper mounting points, steering tie-rod mounting points, suspension upper and lower A-arm

.

mounting points. Knuckle design mainly depends on suspension geometry and steering geometry. The 3D model of existing or benchmark steering knuckle arm is created with the help of CATIA V5. Then this CAD model is imported in pre-processor which is Hypermesh.

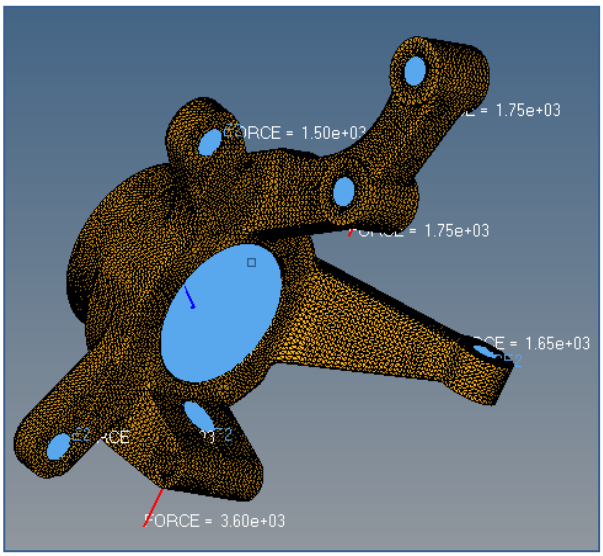
iii. Hypermesh interface



"Fig".4 hypermesh interface

How model looks in hypermesh is shown. CAD model of knuckle converted into STEP file. This model is imported into Abaqus Workbench simulation. Geometry clean-up was performed prior to meshing of model. Optistruct is used as solver. For better quality of mesh fine element size is selected.

iv. Mesh model



"Fig." 5 Mesh model

Mesh Model have :

No of elements = 153627

No of Nodes = 245075

Pre Processing = HyperMesh 12

Solver = Optistruct

Post Processing = HyperView

Analysis = Linear Static

Meshing is fine mesh for highly stressed area and area of interest whereas meshing is coarse in low stressed area. Meshing is done in Hypermesh. Proper meshing is done to the imported model of existing steering knuckle arm.

v. Quality cheh for meshing

warpage	5.000	length	7.500	min angle	20.000
aspect	5.000	length	20.000	max angle	120.000
skew	60.000	jacobian	0.700	quad faces:	
tet collapse	0.100	equia skew	0.600	min angle	45.000
cell squish	0.500	vol skew	0.600	max angle	135.000
		vol AR	5.000		

"Fig." 6 Quality cheh for meshing

Quality check for meshing is done which includes warpage angle (Angle between the normal of warp element), aspect ratio (Ratio of minimum length to maximum length of element), Skrewness, minimum and maximum angle, minimum and maximum angle.

vi. Material selection

	ID	[E]	[G]	[NU]	[RHO]
MAT1	1	1.7e+05		0.275	7.1e-09
	[ST]	[SC]	[SS]		

There are several materials used for manufacturing of steering knuckle such as S.G. iron (ductile iron), white cast iron and grey cast iron. But grey cast iron mostly used. Forged steel are most demanding material for this application. For this Ferrite ductile iron is used.

Material- ASTM 65-45-12 (ISO 450-10) Ductile Cast Iron

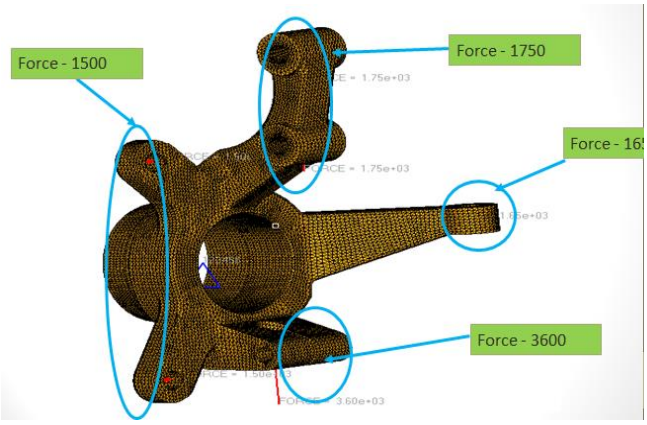
E=1.7x10^5N/mm^2

Poisson Ratio=0.275

Density=7.1x10^-9 ton/mm^3

.

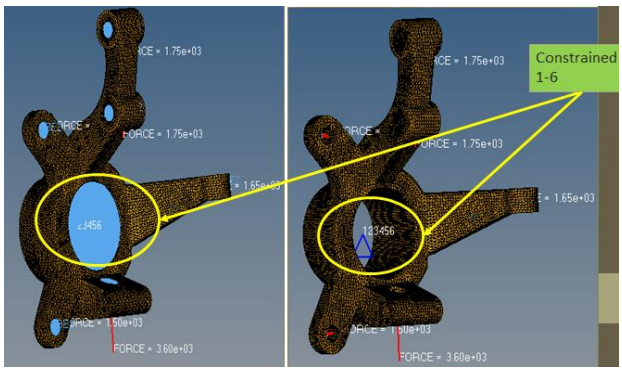
vii. Loading conditions



"Fig".7 loading conditions

The material properties and loading conditions are assigned to imported model of benchmark steering knuckle arm.

viii. Boundary condition

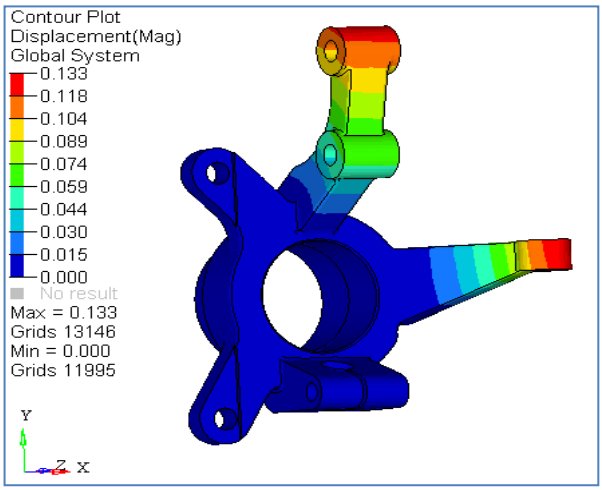


"Fig".8 Boundary condition

Loads are applied and boundary conditions are given to solve the mesh model for static analysis and topology optimization. Fig. shows that all six degree of freedom are constrained i.e. three degree of freedom for translational and three degree of freedom for rotation.

ix. Displacement contour

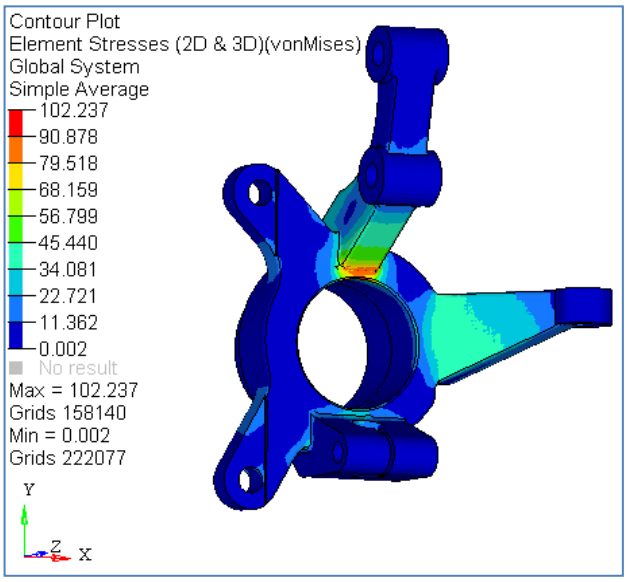
Meshed model is solved in solver which is Optistruct for static analysis and topology optimization. Optistruct solver directly gives the brief idea about highly stressed area and low stressed area and also availability for weight reduction. Post processor used is Hyperview i.e. results are viewed in Hyperview to modify the model on basis of results obtained.



"Fig".9 displacement contour

Displacement of steering knuckle arm is shown when loading conditions and boundary conditions are applied. It shows maximum displacement is 0.133 mm and red color shows it has occurred at shock absorber i.e. at suspension system.

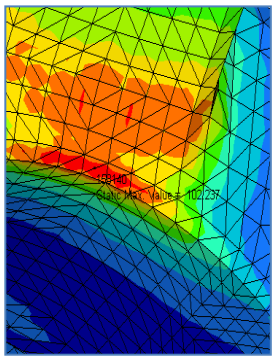
x. (a) Stress contour for von Mises stress



"Fig".10 stress contour for von mises stresses

Von mises stresses developed in steering knuckle arm are shown. It shows that maximum von mises stresses are 102.237 MPa.

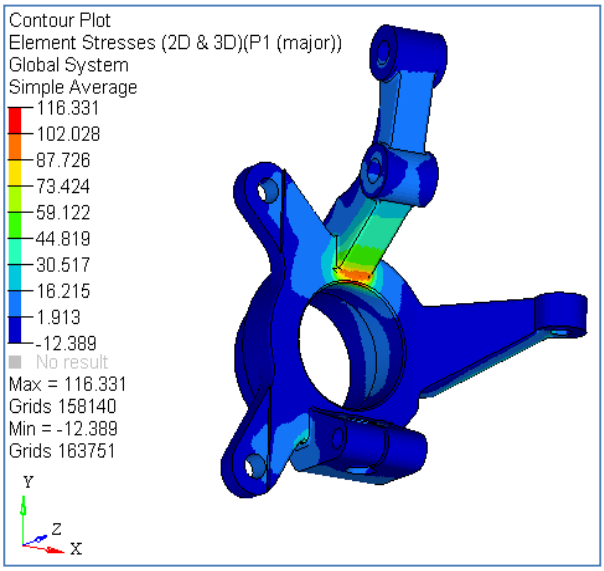
.



"Fig."11 Enlarge view of maximum stressed area

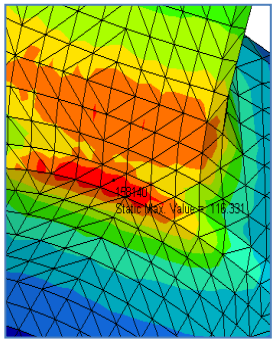
Enlarge view of maximum stressed area is shown. Maximum von mises stresses are developed at end of matting part of shock absorber.

(b) Stress contour for principal stresses



"Fig."12 Stress contour for principal stress

The stress contour for principal stress are shown. It shows that maximum principal stresses are 116.331 MPa.

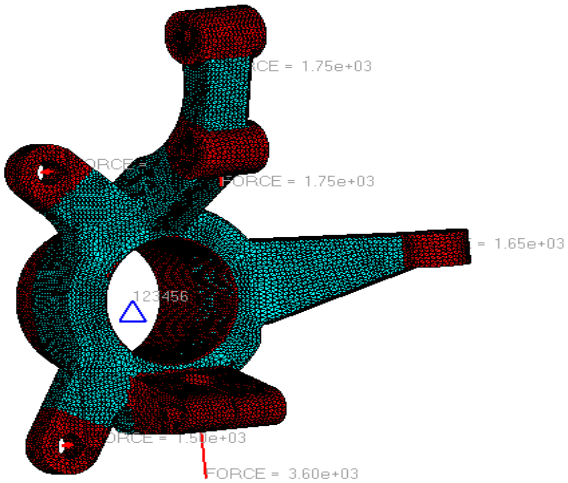


"Fig."13 Enlarge view of maximum stressed area

Enlarge view of maximum stressed area are shown. Maximum principal stresses are developed at end of matting part of shock absorber.

V. TOPOLOGY OPTIMIZATION

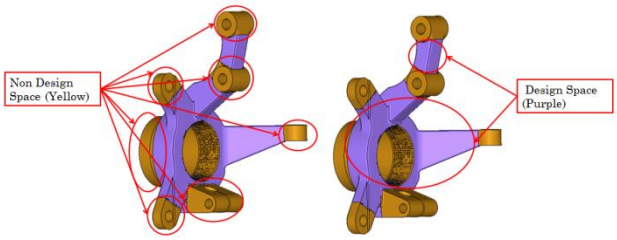
Topology optimization is a mathematical approach which optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets. The model from static analysis step was imported in OptiStruct for optimization. Topology optimization is one of tool of weight reduction.



"fig"13 model in topology optimization

i. Topology design variables

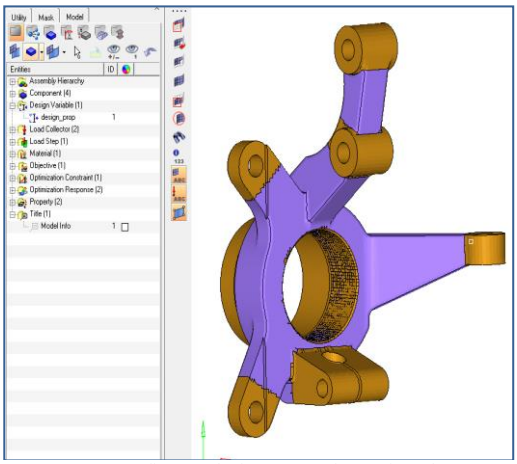
It can be defined as the set of possible designs and design parameters that meet a specific product requirement.



"fig"14 design & non-design variables

Some area can be identified as non-design space and these regions will not be modified during the optimization. In given figure, purple colour is design space and yellow colour is non-design space.

.

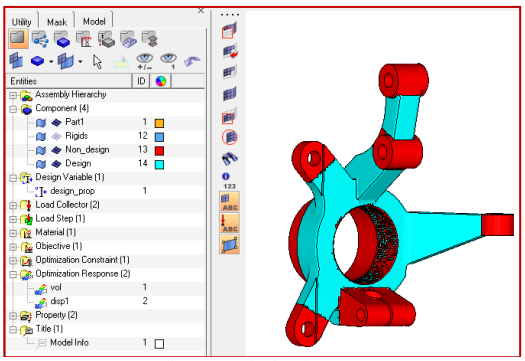


"fig."15 design properties assigned

Design properties are assigned to design variables which are design space and non-design space which can be element density, linear static, linear continuous or linear non-continuous.

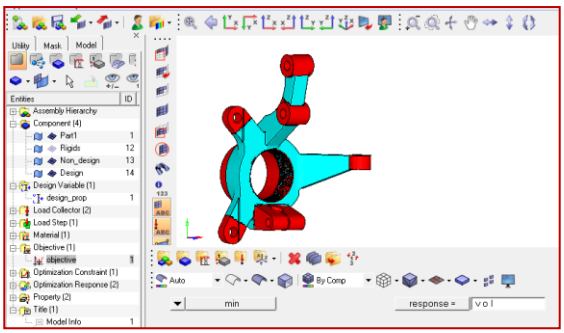
ii. Optimization responses

Response optimization helps to identify the combination of variable settings that jointly optimize a single response or a set of responses.



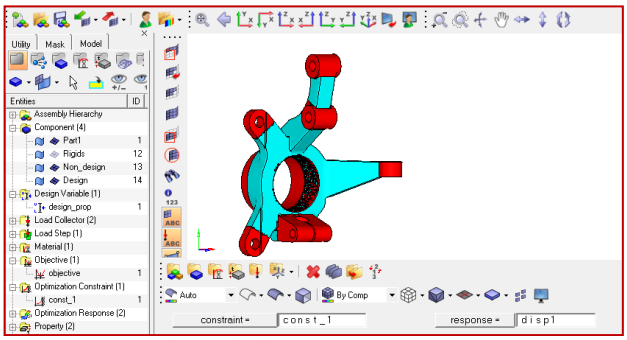
"fig"16 optimization responses

For this optimization, responses are volume and displacement.



"fig"17 volume response

As to minimize volume of steering knuckle arm

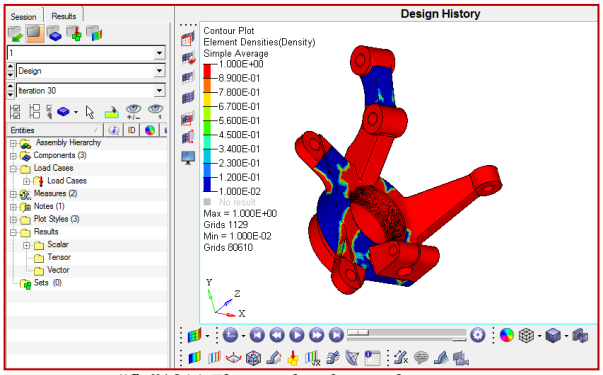


"fig"18 displacement response

constraint displacement of steering knuckle arm.

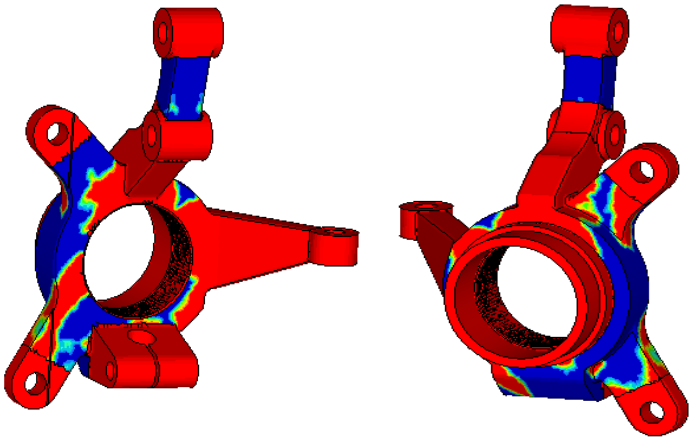
iii. Element distribution density

OptiStruct solves topological optimization problems using either the homogenization or density method.



"fig"19(a) Element distribution density

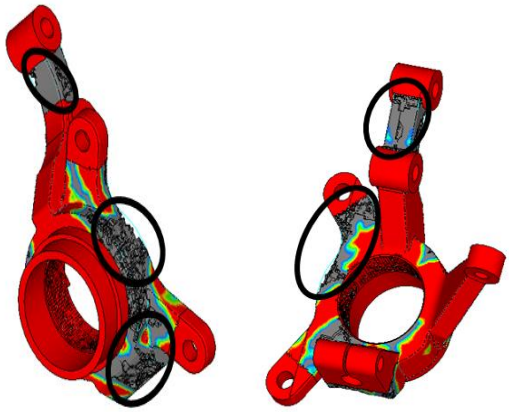
These results helped in changing the CAD model so as to reduce the overall weight of the model.



"fig"19(b) Element distribution density

Blue colour region describes dead zone or elements from same region does not contribute to share the external

load. There is a scope of removing material from the same region.

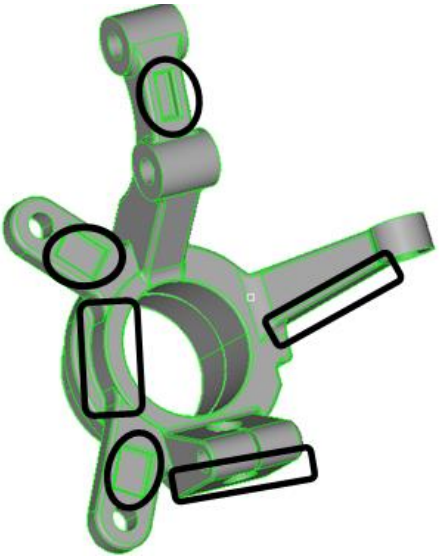


"fig"20 material removed model

Material is removed in encircled region as shown in figure.

iv. Modified geometry

Software has directly given the area where material removal is possible as shown in fig. 20 i.e. material removal from that area does not lead to any failure of component.



"fig"21 modified geometry

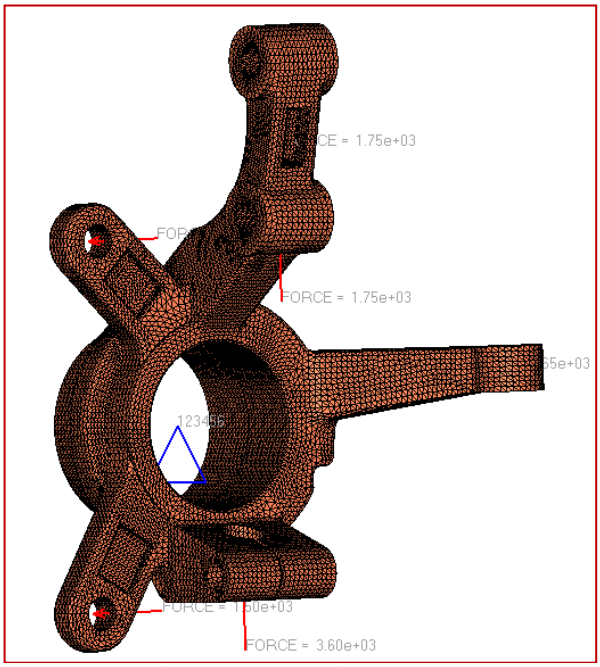
But it is not possible to remove material directed by software as practically it's not possible. So, for convenience of manufacturing process as casting material is removed in encircled region as shown in figure 21.

Geometry can be modified precisely using dimensioning panel. Dimensions are changed in an intuitive and direct way using dimension manipulator, where the required dimension values are entered on the screen and effects are immediate. Geometry can be modified, scaled, extended at required locations exclusively where other areas remains as it is.

VI. Analytical static stress analysis of modified model

i. Mesh model of modified geometry

The results from Topology Optimization changes were made in CAD model to reduce the weight of the model as well as maintaining the strength and rigidity of the model.



"fig"22 meshed modified model

Meshing is done to modified geometry. Meshing is started with critical area first as hole, fillet area. Meshing is fine at hole and fillet area. To avoid localised stress or to avoid triangular edge washer plate is added near hole. Diameter of washer plate is about 1.5 times diameter of hole.

No of elements = 158059

No of Nodes = 252329

Pre Processing = HyperMesh 11

Solver = Optistruct

.

Post Processing = Hyperview

Analysis = Linear Static

ii. Quality criteria for meshing

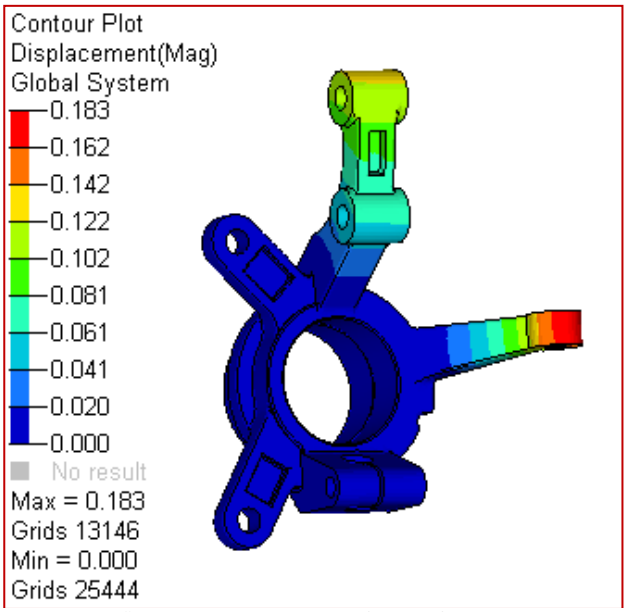
Quality check for meshing is done which includes warpage angle (Angle between the normal of warp element), aspect ratio (Ratio of minimum length to maximum length of element), Skrewness, minimum and maximum angle, minimum and maximum angle.

warpage	>	5.000	length	<	7.500	min angle	< 20.000
aspect	>	5.000	length	>	20.000	max angle	> 120.000
skew	>	60.000	jacobian	<	0.700	quad faces:	
tet collapse	<	0.100	equia skew	>	0.600	min angle	< 45.000
cell squish	>	0.500	vol skew	>	0.600	max angle	> 135.000
			vol AR	>	5.000		

Quality check for meshing is decided by company manual or complexity of geometry.

iii. Displacement contour

Displacement contour shows how much elements are displaced. Displacement contour for modified geometry is as shown.

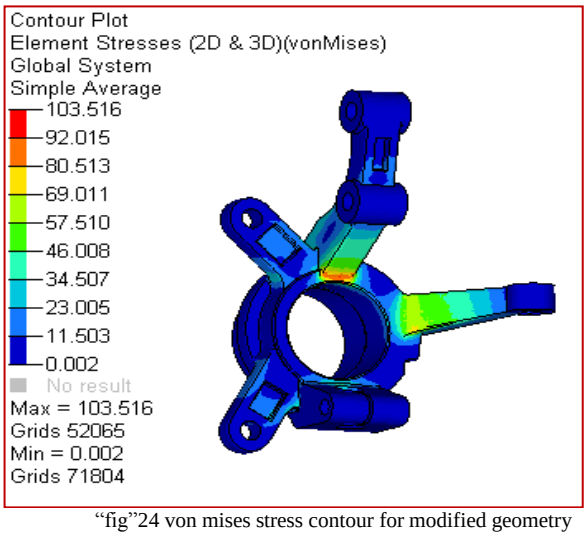


"fig"23displacement contour for modified geometry

Maximum displacement for modified geometry is 0.183 mm whereas for benchmark geometry, it is 0.133mm. Increase in displacement is due to removal of material.

iv. Stress contour

Stress contour gives the stresses developed in model. For modified geometry, maximum stress developed is 103.516 N/mm^2. It is developed at end of suspension system.

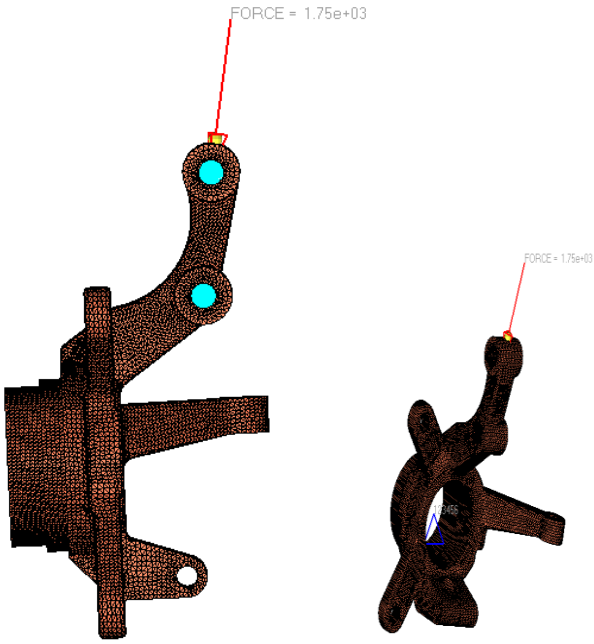


"fig"24 von mises stress contour for modified geometry

As material used for steering knuckle arm is ductile cast iron, only von mises stress theory can be used.

VII. Testing for validation

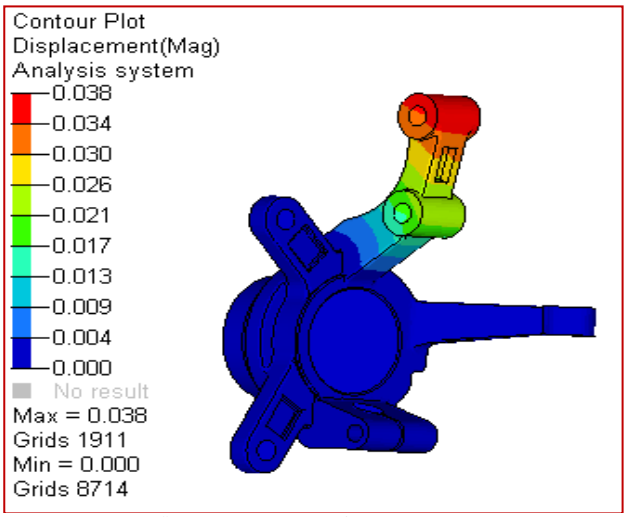
Experimentation results are validated with analytical results. Maximum load acting on steering knuckle arm is 1750 N (9). It acts at where steering knuckle arm mates to suspension system. So analytically, load of 1750 N is applied.



"fig"25 testing for validation

Load of 1750 N is applied over the FE Model. The same shall be applied over the Prototype using the experimental setup. Corresponding displacement shall be

recorded and compared with the results for physical experimentation.



"fig"25 stress contour for testing validation

Displacement contour shows that maximum displacement when load of 1750 is applied at shock absorber is 0.038 mm which should be validated with experimental results.

VIII. EXPERIMENTATION

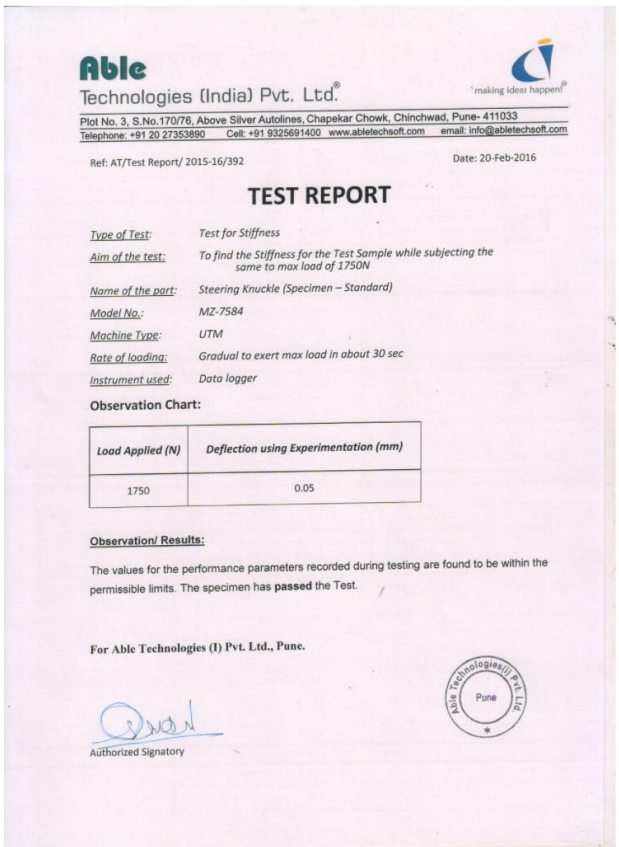
The typical test setup for determination the stiffness of the steering knuckle is shown (see fig 26). The gradually increasing load will be applied and corresponding deformation will be determined.

"Fig".26 Typical Test setup for Physical Experimentation for determining Stiffness

The typical test setup for determination the stiffness of the steering knuckle is shown (see fig 14). The gradually

"fig"27 testing at lab

The load from the load cells present on the UTM machine is applied gradually. Display attached to the machine will give a corresponding plot graph for load vs displacement i.e. stiffness of the component.



Able
Technologies (India) Pvt. Ltd.®
'making ideas happen'®
Plot No. 3, S.No.170/76, Above Silver Autolines, Chapekar Chowk, Chinchwad, Pune- 411033
Telephone: +91 20 27353890 Cell: +91 9325691400 www.abletechsoft.com email: info@abletechsoft.com

Ref: AT/Test Report/ 2015-16/392 Date: 20-Feb-2016

TEST REPORT

Type of Test: Test for Stiffness
Aim of the test: To find the Stiffness for the Test Sample while subjecting the same to max load of 1750N
Name of the part: Steering Knuckle (Specimen – Standard)
Model No.: MZ-7584
Machine Type: UTM
Rate of loading: Gradual to exert max load in about 30 sec
Instrument used: Data logger

Observation Chart:

Load Applied (N)	Deflection using Experimentation (mm)
1750	0.05

Observation/ Results:

The values for the performance parameters recorded during testing are found to be within the permissible limits. The specimen has **passed** the Test.

For Able Technologies (I) Pvt. Ltd., Pune.

Authorized Signatory

Able Technologies(I) Pvt. Ltd. Pune

"fig"28 test result

The results recorded over the physical experimentation concur with the computational results offered by FE Analysis. An error margin of about **8 percent** is noted

.

while comparing the results offered by the two methodologies.

IX. Result

Parameters	Benchmark model	Modified model
Maximum Displacement(mm)	0.133	0.183
Von mises stresses (MPa)	102.24	103.516

	From software	From experiment
Displacement (mm)	0.038	0.05

X. Conclusion

- Steering knuckle is analysed under various loads from connected parts.
- Max Displacement for benchmark model are Observed 0.133 mm at shock absorber mating point.
- Von Mises stresses for benchmark model are observed 102.24 MPa at shock absorber mating point.
- Max Displacement for modified model are Observed 0.183 mm at steering mating part.
- Von Mises stresses for modified model are observed 103.516 MPa at shock absorber mating point.
- Stresses are below yield limit i.e. 310 MPa
- An error margin of about **8 percent** is noted while comparing the results offered by the two methodologies

References

[1] Rao, S.S., "Engineering Optimization Theory and Practice", John Wiley & Sons, Inc., 2009, 4th edition.

[2]Hee-Jin Shim, and Jung-Kyu Kim "Durability analysis of an automotive component in consideration of inhomogeneous hardness", springer, 2010

[3]Rick Bornsand Don Whitacre "Optimizing Designs of Aluminum Suspension Components Using An Integrated Approach" SAE Paper 05M-2

[4] US department of energy Vehicle Technology Office "WORKSHOP REPORT: Light-Duty Vehicles Technical Requirements and Gaps for Lightweight and Propulsion Materials" February 2013.

[5] RajkumarRoy, Srichand Hinduja and Roberto Teti "Recent advances in engineering design optimization: Challenges and future trends" CIRP Annals – Manufacturing Technology 57 (2008) 697–715

[6] S.Vijayarangan, N. Rajamanickam and V. Sivananth "Evaluation of metal matrix composite to replace spheroidal graphite iron for a critical component, steering knuckle" Materials and Design 43 (2013) 532–541

[7] Prof R. L. Jhala, K. D. Kothari and Dr. S.S. Khandare "Component Fatigue Behaviors And Life Predictions Of A Steering Knuckle Using Finite Element Analysis" International Multi Conference of Engineers and Computer Scientists 2009 Vol II

[8] R. d'Ippolito, M.Hack, S.Donders, L.Hermans, N.Tzannetakis ,and D.Vandepitte "Improving the fatigue life of a vehicle knuckle with a reliability-based design optimization approach" Journal of Statistical Planning and Inference 139 (2009)1619 – 1632.

[9]Practical aspects of Finite element simulation-HyperWorks Student guide

[10] Nitin S Gokhale, Sanjay S Despande, Sanjeev V Bedekar and Anand N Thite, book on "practical finite element analyasis", 2012 edition

[11] The ANSYS 13.0 HyperMesh 11.0 Simulation finite element analysis by china press

Acknowledgement

I take this opportunity with honor to acknowledge with great sense of gratitude towards Prof .S. B. Belkar my project guide whose valuable guidance and encouragement during our project made project work possible. His ideas and suggestions provided solid backbone in the development of project. I would like to thank Prof. M. S. Mhaske, my project coordinator for his valuable guideline.

We would like to thank Prof. Dr. R. R. Kharde, Head of Mechanical Engineering Department and Prof. Jahagirdar, Principal of our college for their consistent supervision and motivation. I wish to endow sincere thanks to all technical staff that extended their co-operation to me when I needed throughout the project.

.