IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 4 Ver. VI (Jul- Aug. 2014), PP 61-69
www.iosrjournals.org

TopologyandFreeSizeOptimizationforFrontFenderofThreeWheelerVehicle.

AjitV.Salunkhe[1*],J.P.Supale[2]

[1]**PGScholarDepartmentofMechanicalEngineering,SKNSinhgadInstituteofTechnology&Science, Lonavala,MaharashtraIndia,*

[2]*AssistantProf.DepartmentofMechanicalEngineering,SKNSinhgadInstituteofTechnology&Science, Lonavala,MaharashtraIndia*

Abstract: *Frontfenderdesignof three wheelervehicleisvery importantwith thefocusonanimprovementaspect inthe automotiveindustry.The goalsare toincreasethe performanceandtofindthesolutiontoreducethecost ofthefenderhenceabletoreducetheproductioncost.Thecostsofthefenderishighbecauseoftheamount of materialused.InthispaperFiniteelementmodelsofthefrontfenderofthreewheeler vehiclewereanalyzed, usinglinearandnonlinearfiniteelementanalysesinCAEsoftware.Experimentalandanalysisstresswere comparedtoevaluatethevalidityof theFEAapproaches.Stress results werefoundto havea7%error whichis wellwithina10%reasonablevalue.Thehandcalculations showeda36%errorwhichwasattributedtothe simplificationsandassumptionsmadetomake thecalculationspossibleaswellasthegeometrycomplexity. Inthisstudy,multi-stagetopology andfree-sizeoptimization offrontfenderstructurehavebeen performedusingOptistructbyAltairengineering, usedtodesignalightweight frontfenderwhensubjectedto serviceloads.Anexistingcommercially availablefrontfendermudguardservesasthereferencedesignspace. The objectivefunctionfor topologyoptimizationis usedduringthedevelopmentstageto determineplacementof supportingribsinfendercomponentandconfirmmaterialreductionofanexistingfeasibledesignspace.Element levelfree-sizeoptimization isusedtodetermine optimal materialdistribution byvariableshellthickness optimization.Functionalrequirementsvalidationis performedusingnonlinearfiniteelementstressandstiffness designchecks.Anoverallreductioninweightof1.04%isachievedoverareferencecommerciallyavailablefront fendercomponent.*

Keywords:*Fender,StrainGauges,StrainIndicator,CADModeling,CAE-ComputerAidedEngineering,HyperMesh,Topology&Free-SizeOptimization,Optistruct.*

I. Introduction

Inrecentyears,withincreaseddemandforlowfuelconsumptionandlowcostvehicles,thelightweight designsarehighestpriorityduringproductdevelopmentstageinautomobileindustry.Thispaperdealswithfinite elementstressanalysisoffrontfenderofthree-wheeler vehicle,experimentalvalidationofthestressdatawith analysisresultsanddesignoptimizationwiththeaimofreducingweightforspecifiedloads,constrainsanddesign space.Fendersprovidesufficienthousingforthewheelsandsuspensionlinkages. Variousmaterialsusedfor fenderdependonthestrength,expectedlifeandsuitability ofmanufacturingmethods.Preferredmaterialsare sheetmetal,plasticandfiberreinforced plastic. Plasticispreferredbecauseofitslightweightcharacteristicbut strengthisaproblem oversheetmetal.Whiledesigning thefenderfollowingfactorsareconsidered.Itshould providesufficientcovertothewheelandsuspensionlinkages,itshouldhavesufficientstrengthtowithstandloads andvibrationunderalloperating conditions.ApartfromnormalloadstheMud-Guardissubjectedtodifferent handlingconditionsduringrepairandmaintenanceofthevehicle. Themanufacturerofthevehicle now cameto knowthat theMud-Guarddesignrequiredtobemodifiedforhandlingduringrepairandmaintenance.RafatAli [1]hasdescribedtheapplicationofthefiniteelementtechniquetothestaticstressanalysisofcompositestructure inwhichfiniteelement[FE]model ofstructureisauthenticatedbyusing strain gauge andstrain indicator.Basil Housari,LianX.Yang[2]explainedtheexperimental stressmeasurementtechniqueusedtomeasurestress concentrationinwhichresultsobtainedfromrosettestraingaugesarecomparedwiththosefromfiniteelement analysis.FEstressresultshavebeenvalidatedthroughcomparisonwithexperimentalstraingaugemeasurements. Mohamad M.Ansari[5]compared straindatafromfiniteelement resultsandtestdatafromstraingaugetest. Quadrilateral shellelementhasbeenusedtogenerateFEmodel.Frontfendershouldbedesignedaslightas possiblewithsufficientstrength,stiffness,andenergyabsorption propertiesintheeventofcrashloads.Thisis significantdesignchallengeastheneedforlightweightmustbebalancedwiththeneedforcomfort,adjustability, andeverincreasingsafety demands. Dieinjectionplasticmoldingallowsforvariablethickness distributionand detailedgeometricfeaturessuchassupporting ribsandothertopographic properties whichhavesignificant potentialfordetailedoptimizationforlightweightdesign.Severaldifferentapproachestooptimization areavailableforstructuraldesign,includingfiniteelement methodsbasedontopology,geometricsizeandshape,andfree-sizeapproaches.Acompleteoptimizationprocess oftenusesacombination ofdifferentoptimization techniques.Finiteelementbasedtopology optimizationhas importantpracticalindustrialapplicationsinmanufacturingincludingautomobileandaerospaceindustries,and islikelytohaveasignificantroleinmicro-andnanotechnologies [6].Topology optimizationusesamaterial distributiontodesignstructuresbyindividuallyassigningmaterialdensitiestofiniteelementsinaspecifieddesign space.Thematerialredistributionisachievedbymathematicaloptimizationalgorithmsusingsensitivityanalysis ofthepreviousdesigninaniterativefashion[7,8].Fortopology optimizationmethodsbasedonacontinuum basedfiniteelementsolution,thedesigndomainis representedasamixtureof materialdensityand“voids”.The resultsobtainedfromthetopologyoptimizationareintheformofmaterialdensitiesofthefiniteelementsinthe designdomain.Anautomated process forinterpretingthree-dimensionaltopologyoptimizationresults based on densitycontoursintoasmoothCADrepresentation isproposedin[9,10].Free-sizeoptimizationisanelement levelmethodwhereindividualshellsurfacethicknessesofeachelementinacomponentarefreedesignvariables [11].Forthin-walledcastcomponentssuchasanautomotivebackrestframe,thin-shellfiniteelementswithindividualthicknesseswhichcanbevariedanddistributedfreelyinatopologically optimizedcastcomponent offeracompellingmethodforachievingsignificantweightsavings.Applicationofmulti-stepengineeringdesign optimizationmethodsandtoolstothedesignofautomotivebody-in-white(BIW)structuralcomponentsmadeof polymermetalhybrid(PMH)materialswasaddressedin[12] wheretopologyoptimizationis usedinidentifying theoptimalinitialdesignsandtheuseofsizeandshapeoptimizationtechniquesinusedtodefinethefinaldesigns subjectedtodifferentin-serviceloadsanddesignedforstiffness,strength,andbucklingresistance.Manufacturing constraintsforinjectionover-moldingwerealsoconsidered.

An objectiveofthe presentworkistoextendthemulti-stepengineeringdesignoptimizationapproachtodie-castcomponentswithtopology optimization underloadcases,andaccountingfordiedirectionmanufacturing constraints inordertodetermineoptimalplacementofsupportribsandthentoutilize“free-size”elementlevel optimizationwhichisidealforoptimizingthethicknessdistributioninthinwalledcastparts.Inthepresentwork, atopologyandfree-sizeoptimizationprocedureisdevelopedbasedonthecommerciallyavailablefiniteelement programOptiStructfromAltair[13],forthe optimalmaterialandsupportingrib distributionforthelightweight designoffender.Theorganizationofthe paperisas follows:a descriptionofthereferencefront fender,material properties,andloadrequirementsarepresented.Anoverview ofthemulti-stagetopologyandfree-size optimizationprocessisthenpresentedanddiscussed.Theresultsobtainedfrom theoptimizationprocessinthe presentworkarepresented anddiscussed,followedbytheinterpretednewdesign andvalidation offunctional requirementsusingnonlinearfiniteelementanalysis.Bothtopologyandfree-sizeoptimization arebasedona linearfiniteelementanalysis, whilethevalidationisperformedusing afiniteelementmodelwhichaccounts for bothmaterialandgeometricnonlinearity. Themainconclusions resultingfromthepresentworkarethen summarized.

II. Objectives OfTheWork

The followingaretheobjectivesofthestudy:

1. Tostudyexistingthreewheelerfrontfender mudguardin Indianmarketforpossibledesignmodifications.
2. Tocarryoutfiniteelementstressanalysisoffront fenderofthree-wheeler.
3. Tocarryoutexperimentalvalidationofthestressdistributionacrossthewhole fenderusingstraingauges.
4. Todomultistageoptimizationthefront fendertoimproveitsstructuralstrengthbyuseofCAEsoftware's.
5. For functionalrequirementvalidationisdonebyperformingnonlinearfiniteelementstressandstiffnessdesign analysis.

III. ProposedWork AndMethodology

Amethodologydescribesdifferentapproachestoanalyzefrontfenderbymeansofthe f i n i t e -elementmethod canbecategorizedintothefollowingsteps:

1. PreparationofCAD modelusingavailabledrawingsoftheexistingfender(Mud-Guard).
2. Pre-processingofthefiniteelement modelofexistingfenderusingHyperMesh.
3. Performa linearstaticanalysisofa FEmodelofexistingfenderusingNastransolver.
4. Post-processingoftheresultsoftheBaselinedesignusinghyper viewaspostprocessor.
5. ComparisonofValuesobtainedfromFEManalysisandthosefromExperimentalvaluesusingstraingauge method.
6. Multi-stageoptimizationprocessareperformedbasedona linearfiniteelementanalysis.

7. Topologyoptimizationisusedtodetermineoptimalplacementofsupportingribsandmaterialdistributionfor fenderofanexistingdesignspaceinAltairOptistruct.
8. Element-levelfree-sizeoptimizationisperformedfor variableshellthicknessinAltairOptistruct.
9. Resultssuggestedfrombothoptimizationoftheshellmodelarecomparedtocreateafinaloptimizeddesigninpro-Esoftwareforthefendercomponentwithreduced weight.
10. Tofunctionalrequirementsvalidation,thefinaloptimizedfendermodelissubjectedtononlinearfiniteelement analysisusingNastransolver.
11. Conclusionsaremadebycomparingtheresultsoftheexistingandfinalmodelsimulations foritsimprovement ofstructuralstrength.

IV. Finite ElementModellingApproach

4.1.CADmodeling

Pro/Engineerisa parametric,feature-basedmodelingarchitectureincorporatedintoa singledatabasephilosophy withadvancedrule-baseddesigncapabilities. Thecapabilities oftheproductcanbesplitintothethreemain headingofEngineeringDesign,AnalysisandManufacturing. Thisdataisthendocumentedinastandard2D productiondrawingorthe3DdrawingstandardASME.Modelingofmudguardisdonewithhelpof Pro-e software.Allgeometricanddimensionsofthemudguardsystemincludingribsareplotted.Themodelwhichwas developedinpro-e softwarerequiredforFEmodellingwasacquiredinInitialGraphicalExchangeSpecification (IGES)format thatissupportedinalltheCAEpre–processing software. ByImporting IGESfileintohyper meshsoftwarefor Cleaningof geometryandpreparingthefileformeshgeneration.

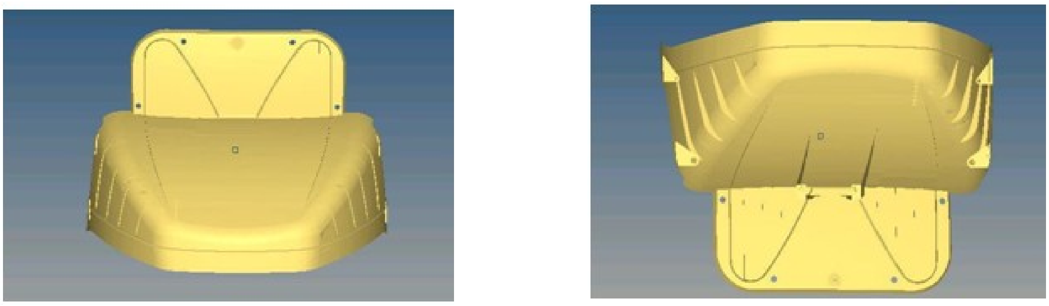

Figure 1. Existingmudguard model

4.2.FEmeshgenerationusinghypermesh

Initially,theimportedmodelasIGES,wasControlled forthecorruptedandtwistedSurfacesandalso fortheboundaries andEdgesbetweensurfaceswherethegapinBetweenisbeyondtheacceptable range. The modifications arefollowedbysuppressingtheboundary patchestoavoidtheUndesirablelayoutofthemesh. Therearetwodifferent2D meshtypes,namelyquadsandtries.ThequadsshowBetterresultsincomparisonwith tries.Hence,eachmodel'ssurfacesweremeshedusing2DMeshwithaCombinationoftriesandquads.However, theResultingmesh comprisedmainlyof quadElements.FinerMesh isrequiredtoshowthenonlinearBehavior ofthematerialandfailure.Inthemeshingapplication,theaimwastokeepthemajorityoftheelementssize around4mmwhichisfiner/smallerthanthemeshSize-around10mm.Butontheotherhand,applyingfinermesh raisesthesimulationtimedurationDuetotheexplicitnatureofthe solverforDynamicanalysis.

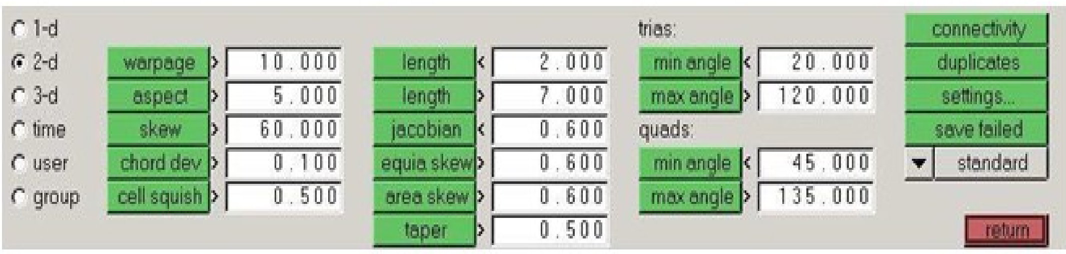

Figure 2. Meshcriteria

4.3.Materialinformation

IntheFEA,propertiesofmaterialplayanimportantrole.Thepropertyof materialisa basicinput forstructural analysis.Materialused forfrontfender mudguardisnormallypolypropylene(PP).

Materialpropertiesforfender	Symbol(Unit)	Value
Young'smodulus	E (MPa)	1100
Poisson'sratio	ν	0.381
Density	ρ(ton/mm³)	1.56e−9
Yieldstress	σ_y(MPa)	30-34
Ultimatestress	σ_t(MPa)	42-50

Table1. Mechanicalproperties ofpolypropylene.

4.4. Linearstaticstructuralanalysis

Staticanalysis determinesthedisplacements,stresses,strains,andforces instructuresorcomponentscausedby lodesthatdonotinducesignificantinertiaanddampingeffects.Steadyloadingandresponseconditionsare assumed;thatis,theloadsandthe structure'sresponseareassumedto varyslowlywithrespecttotime.Duringrepairandmaintenanceofthe vehicleisnormallyhandledbyservicemen,so upwardliftingforcesacting onfenderarecriticalforwhichitisnotdesignedandmanufactured.Inthispaperthestructuralanalysisperformed for variousloadsrangesfrom0-100kgatintervalof10kg,respectivelyresultsareplotted.

Linearstaticanalysisis carriedoutinstagesas follows:

- ImportingPartgeometryinHypermesh
- Meshingthesurface
- Applyingmaterialandproperties
- Applyingloadsandboundaryconditions
- SolvinginNASTRANSolver(sol101)
- ViewingresultsinHyperview

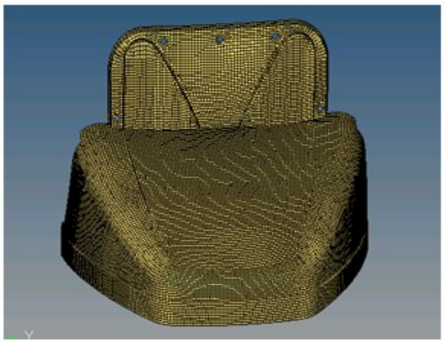

Figure 3. Existingfendermeshed model

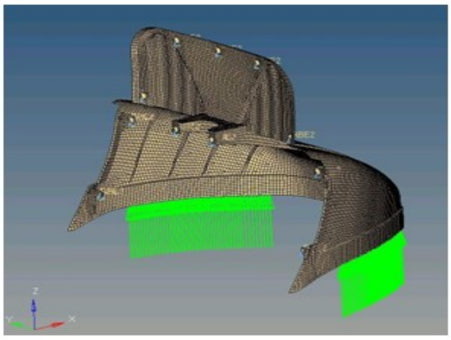

Figure 4. Appliedloads &boundary condition onmodel

Criticalloadcondition

Loadcapacityactingonfenderismaximumat100kgrelativetothat resultsare plotted formaximumdeflection, maximumstress valuesarecheckedforexistingdesign.From the analysis,thefenderisfoundtoexperiencethe largeststresses.Hence,theresultofthemaximumprinciplestressesisusedfor furthertopologyoptimization.

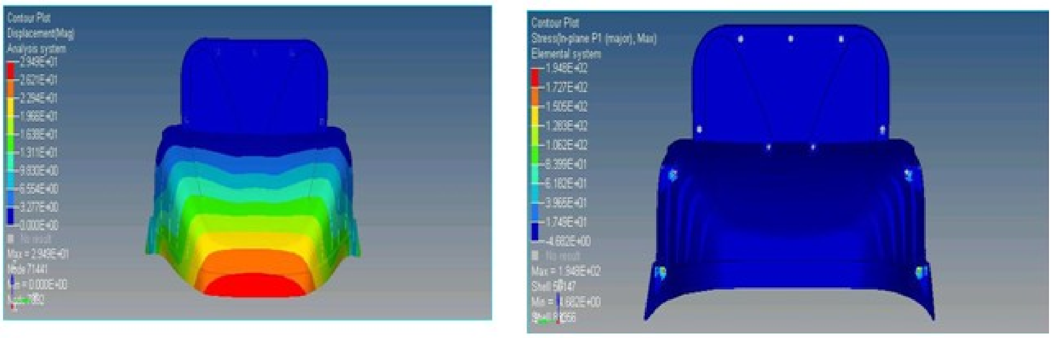

Figure5: Displacement&stressplots ofthe existingmodel.

V. Experimental Validation

Forvalidation oftheresult,theFenderistobeloadedaspertheactualconditionuntilfendergetstotally deformed.Suitablefixtureisfabricatedandmountedonarigidframeorwall.Forthesakeofconvenience the Fenderwasmountedinreversepositioni.e. upsidedown.Loadsappliedonfenderfrom0-100kgatinterval10kg andfurtherincreasingloadthestructuregetstotallydeformed.Henceapplied deadweightswillwork assimilar toactualliftingloadduringserviceandmaintenanceconditionwhilestraininducedwasrecorded onstrain indicator. From experimental readings,ithasbeenseenthattherearenostressactinginzdirection.so2Dplane stressconditionexist.

Figure 6. Experimentalsetup

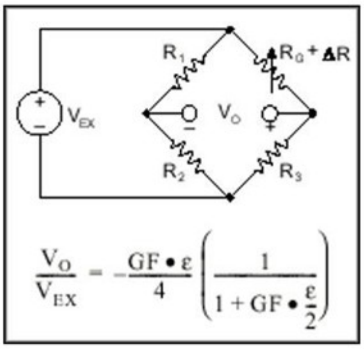

Figure 7. Quarter-bridge strain gauge

Circuitwithtemperature compensation

UsingGauge SensitivitiesandGageFactor,theoreticalstraininducediscalculatedforappliedloads.The Gauge Factorformetallicstraingaugesistypicallyaround2.Thentheoretically experimentalstressiscalculatedby usingstressstrainrelationfor variousloads.

Formula for Experimental stress: $$\sigma = \frac{E}{(1 - \nu)} X \, \varepsilon$$ where ε = Experimental strain

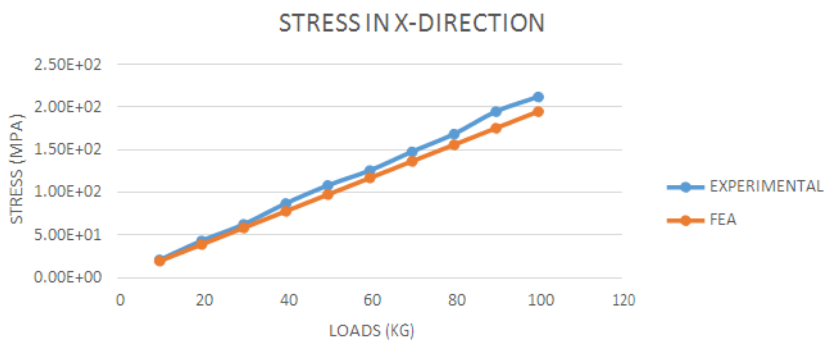

Figure8. Experimental&FEAstress(x-dir.)vs. load.

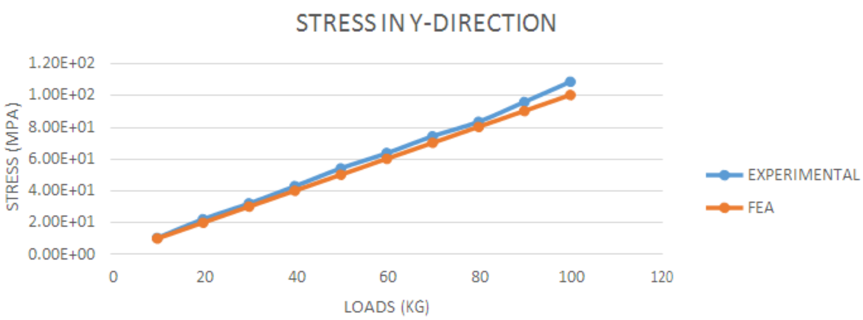

Figure9. Experimental&FEAstress(y-dir.)vs. Loads

Experimentalandanalysisstress werecomparedtoevaluatethevalidityoftheFEAapproaches.Stress resultswerefoundtohavea7%errorwhichiswellwithina10% reasonablevalue.ForMathematical model an assumptionisdonebyconsideringacantileverbeamhavingUDLonpartofspanofbeamasshowninfig.10.So bysolvingmathematicallyforfindingmaximum deflectionandmaximumstressacrossthecantileverbeamfor validationpurpose.Thehandcalculations showeda36%errorwhichwasattributedtothesimplifications and assumptionsmadetomakethecalculationspossibleas well asthegeometrycomplexity.

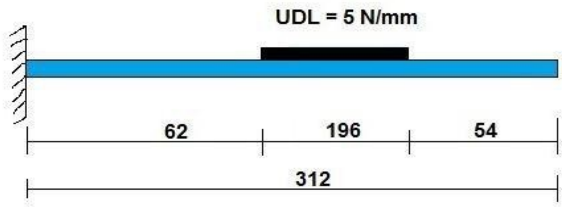

Figure10. Cantileverbeamwith UDL on partofspan.

VI. Multi StageOptimization

Themultistageoptimizationiscarriedoutintwomainstagesasfollows:

Thetotalmassofthereference fenderwasmeasured tobe1.48kgandoverall thickness offenderis3mm.The referenceframeisusedasabasisfordesignoptimizationwiththegoalofreducedmassunderconstraintsofstress.
Inthepresentwork,bothtopology andfree-sizeoptimization arebasedonalinearfiniteelementanalysisin Optistructsolver,whilethevalidationisperformedusinganonlinearfiniteelementmodelwhichaccountsforboth materialandgeometricnonlinearity.

6.1 TopologyOptimization

ItadoptsDensityapproachforoptimalmaterialdistribution.Availabledesignspaceisdefinedwithproperloading and boundaryconditionswithobjectiveto minimizethe globalweightofthe structure,subjectedtostressconstraint.
Thebelowmentionedcriteria'sare used fortopologyoptimization.
DesignVariable:Densityofeachelementwithindesignspace
DesignConstraint:Stresswithspecifiedlimit
DesignObjective:Minimizethemass

Multi-Stage Engineering Optimization Process

6.2 FreeSizeOptimization

Theoutputdesignbytopologyoptimizationwithintroducingcutouts issetforsizeoptimizationtoget theoptimizedthicknessofall structuralmembers.
Thefollowing criteriaaredefinedforsize optimization.
DesignVariable:Thicknessofthecomponents
DesignObjective:Minimizemass
DesignConstraint:Stresswithspecifiedlimit.

6.3 InterpretationFinalDesign

Theresultssuggestedfromthefree-sizeoptimizationoftheshell modelandtopology optimization ofthe fender framemodelare comparedandusedasaguidetocreate afinaloptimizeddesignforthe fender frame component with reduced weight. Since fullyautomatedinterpretationtoolswhichaccount fullyformanufacturingconstraintsarenotcurrentlyavailable, the interpretationstepmustgenerallybedonemanuallyandthepartredra wn.Basedontheobservationsfrom topologyandfree-sizeoptimizationresultsofshellmodel,anewsurfaceshellmodelofth efenderframeis generated.

6.4 ValidationofOptimizedDesign

Toverifystrengthanddeflectionrequirements,thefinaloptimizedfendermodelissubjectedto nonlinearfinite elementanalysis.

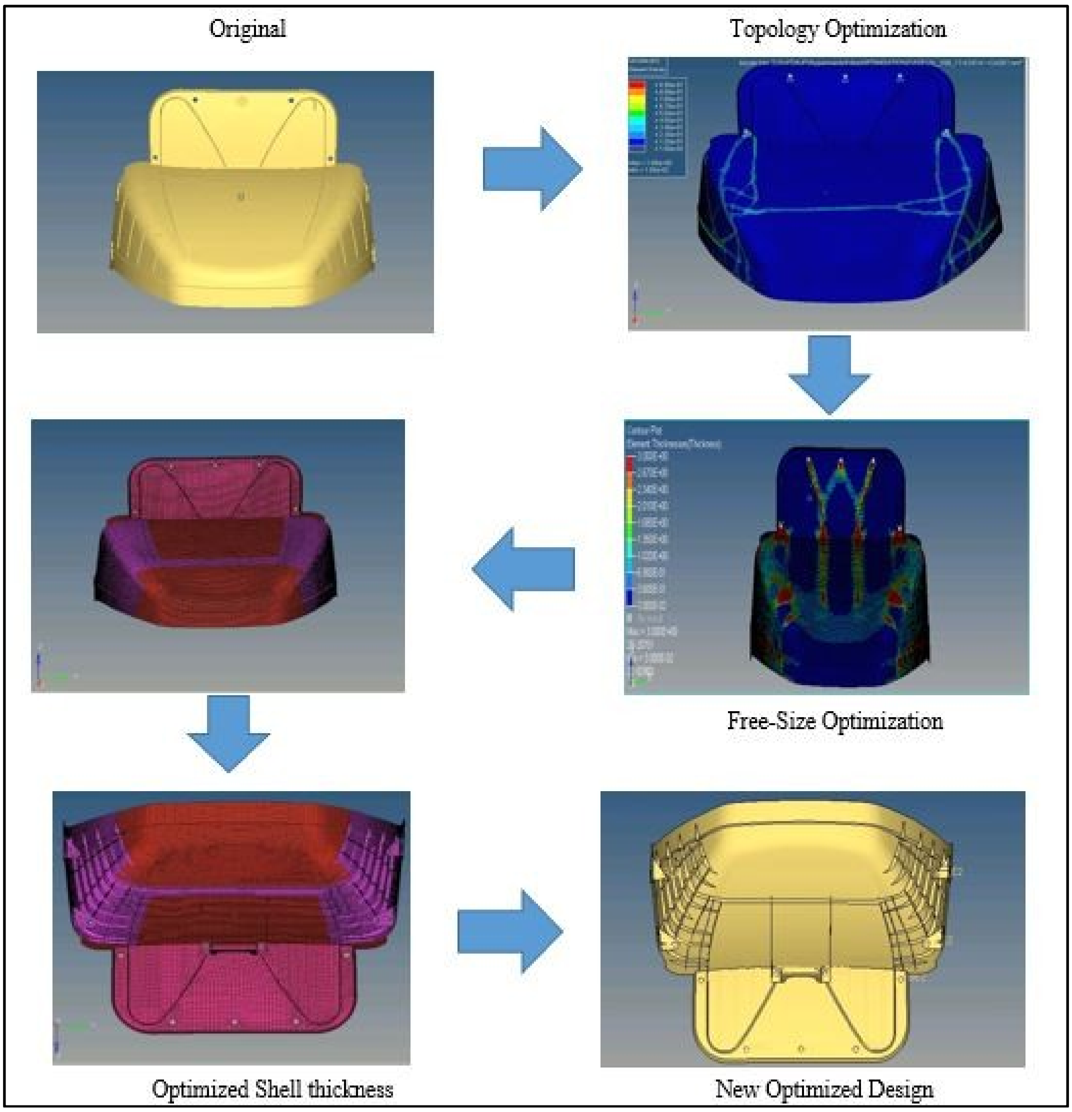

Figure11. Multi-stage engineeringoptimization processfor fenderframe

VII. NonlinearStatic Structural Analysis

As materialused forfenderis polypropylenewhichisnonlinearelasticmaterial.So nonlinearstructuralanalysis isperformedbyconsideringmaterialnonlinearity.Materialsnon-linearstressstraincurveshavebeenobtained experimentallybyconductinguniaxialtests.Theseresultsareenoughtocarryout nonlinearanalysisof homogeneousmaterials.

7.1 TensionTest

Tensiletestingisoneofthemorebasicteststodeterminestress–strainrelationships. Asimpleuniaxialtest consistsofslowlypullingasampleofmaterialintensionuntilitbreaks.Testspecimens fortensiletestingare generally eithercircularorrectangularwithlargerendstofacilitategrippingthesample.Itgeneratesastress-straincurve,whichcharacterizesamaterial'smechanicalperformance,e.g.,yieldstrength,tensilestrength,elastic modulus,elongation,andtoughness.Whenperformedproperly,thetensiletestcanbeaninvaluabletoolfor materialcharacterizationandverification.TheAmericanSocietyofTestingandMaterials(ASTM)havespecific protocolstofollowfortestingawiderangeofmaterials.ASTMD638StandardTestMethodforTensileProperties ofPlasticsmatchedtheplannedexperimentmorecloselythananyotherstandardsreviewed.According to standards prescribethemethodbywhich thetestspecimenwillbeprepared andtested,aswellashowthetest resultswillbeanalyzedandreported.

Figure12. Uniaxialtensile testsetupandtestspecimensbyASTM D638standard

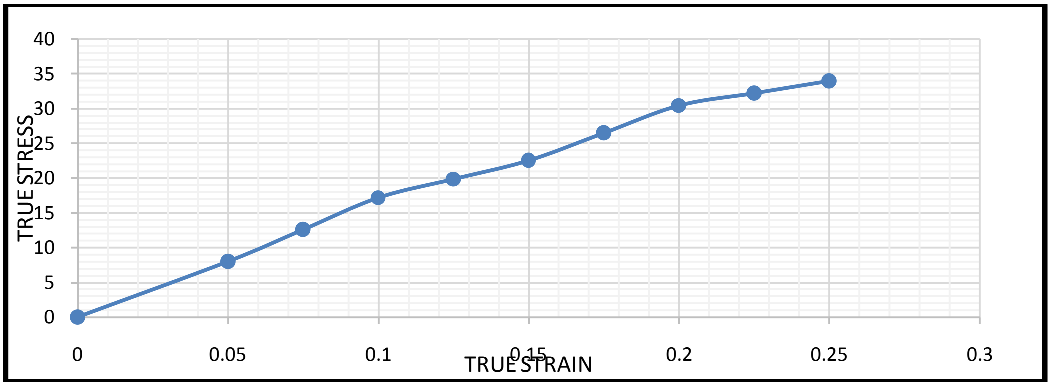

Figure13. Materialtruestressstrain curve forpolypropylene (PP)

7.2 DesignValidationbyNonlinearAnalysisResults

Basedonthematerialnon-linearity,anon-linearstaticanalysisisconducted.MSC.NASTRAN solverisusedtosolve forthestressesandstrains.Resultwereobtainedforallloadcasesupto100kgbutdesign pointviewstressesfor nonlinearelasticmaterialshouldbelessthanultimatestrengthofmaterial.Resultsobtainedupto70kgarewithin designlimitsandabovethatareconsiderasfailure,thusforcomparison between exitingandoptimizemodel resultsplotsare shown for70kg.Thehigheststresslevelsareobservedin localizedregionsontherearsideofthe frameata supportrib neartheboltlocations.

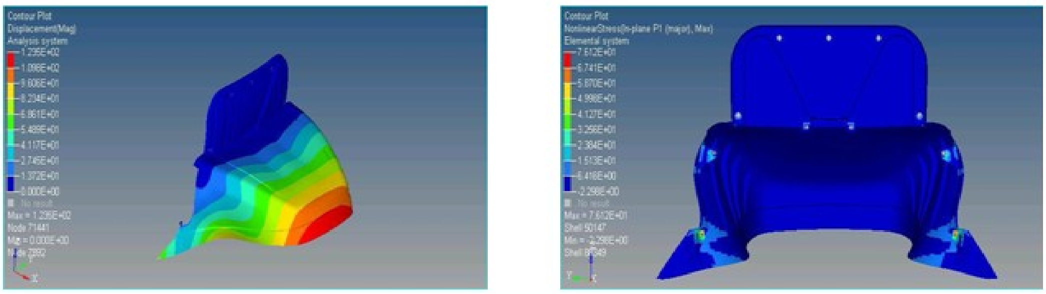

Figure14. Stress&displacementplots of the existingmodel

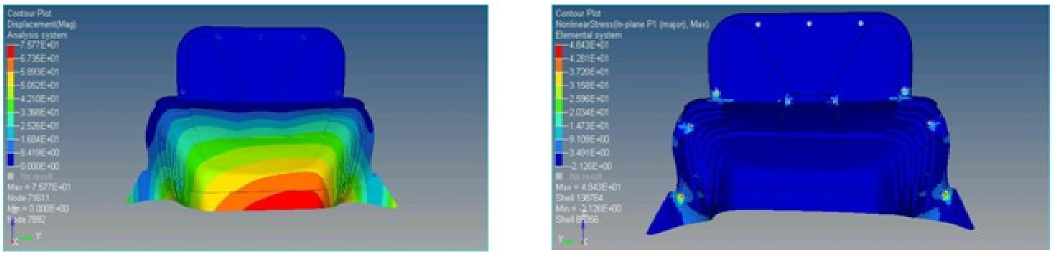

Figure15. Stress&displacement plots of theoptimized topologymodel

Themaximumprinciplestressis48.43MPAwhichisabovetheyieldstressvaluefor(PP),butbelowthe ultimate strengthforthematerialindicatinglocalplasticyieldingbut nofracturehasoccurred.Themaximum displacementfortheoptimizedfenderdesignpredictedfromnonlinearanalysisoccursatthefrontofthefender witha valueof75.8mm whichis38.7%belowthevalueof123.5mm whichwascomputedasthemaximum valuefortheexistingreferencefendercomponent.

VIII. Result Table

Model	Weight(kg)	Deflection(mm)	MaxPrincipalStress (Mpa)
Existingdesign	1.48648	123.5	76.12
Optimizeddesign	1.4710	75.77	48.43

IX. SummaryAnd Conclusions

Basedontheresultsobtainedinthepresentwork,thefollowingsummaryandmainconclusionscanbe made:

1. Thepresentwork demonstrateshowtopologyandfree-size optimizationwithloadrequirements,stress andstiffnessconstraints,togetherwithmanufacturing constraintsondiedrawdirectionandsymmetry,canbe usedtodesignalightweightfenderframecomponent.Tousetopology optimization asatoolformaterial placementismoresystematicthantouse moreorlessguessesbasedonexperience.
2. Thebasedesignresultsformostcriticalloadingconditionareidentifiedandcorrespondingstress&displacement plots. Itisconcludedthattheseregionsarepotential areastoremoveweight.Moreover, theregionswithlower factorofsafetyarealsotargetedforimprovementusingthe optimizationtechnique.
3. Byobservingresultsobtained,thedisplacementandstressvaluesarelessinoptimizedfendercomparedwith existingfendermodel.Basedonresultstheoptimizedfendermodelissafeandbetterforutilization compared withreferenceexistingmodel.
4. Usingthisengineering designoptimizationprocess, anoverallreductioninweightof1.04%isachievedovera referencecommerciallyavailablefenderframecomponent.

References

[1]. RafatAli,"FiniteElementStudyof aComposite Material Sump Panof anI.C.Engine",SAEPaper No.950942, 1995.
[2]. Basil Housari,LianX. Yang,"ExperimentalTechniques for Strain MeasurementandValidationof CAE Model", SAEPaper No.2005-01-0587, 2005.
[3]. Muniyasamy K.,GovindrajanR.,Jayram N.,Ravikharul,"Vibration FatigueAnalysis of MotorcycleFrontFender",SAEPaperNo.2005-32-0030, 2005.
[4]. K. BelKnani,P.Bologna,E.Duni,G.Villari,G.Armando,MTortone,MLeghissa,SBorone,"CAEMethodologiesfor Virtual Prototypingof CastAluminum Suspension Components",SAEPaperno.2002-01-0677, 2002.
[5]. MohammedMAnsari,"Validationof FiniteElement(FE) Model for AllRadiatorEndTank",SAEPaperNo.2002-01-0951, 2002.
[6]. Rozvany,G.,(2008) "Acritical reviewofestablishedmethods of structuraltopologyoptimization"JournalStructural andMultidisciplinaryOptimization,OnlineForum,Publisher SpringerBerlin/Heidelberg,February21,2008.
[7]. Bendsoe,M.P.(1995).Optimization of StructuralTopology,Shape,and Material.Berlin:Springer.
[8]. Eschenauer,H.A.,& Olhoff,N.(2001).Topologyoptimization of continuum structures- areview.AppliedMechanicsReviews, 54(4),331-390.
[9]. A.V.Kumar,D.C.Gossard,(1996) "Synthesisof optimalshapeand topology ofstructures",J.MechDes,118, 68-74.
[10]. S.K.Youn,S-H.Park,(1997)"Astudyon theshapeextractionprocessin thestructural topology optimization usinghomogenized material",ComputStruct, 62(3), 527-538.
[11]. Vanderplaats,G.N.,"Numerical Optimization Techniquesfor EngineeringDesign:With Application,3rdEdition,VanderplaatsResearch&Development,1999.
[12]. Grujicic,M.et.al."Applicationof Topology,SizeandShape OptimizationMethods inPolymer MetalHybridStructuralLightweight Engineering", MultidisciplineModeling inMaterialsand Structures,Volume4,Number4, 2008, pp. 305-330(26).
[13]. Altair EngineeringInc., 2007,"OptiStruct 11.0, User's Guide".