THE OPTIMIZATION STUDY OF TRUSS BRIDGE

ABSTRACT

A recent drift in bridge development has been the optimization of the cost-to-performance proportion. Steel truss bridges are employed for long spans. It can carry heavy loads which cannot be economically carried by simple girder bridge. Efficient and economical design of truss bridge usually results in high cost. As urbanization increases and city populaces rapidly increment, transportation of individuals and stock must ended up more proficient. Bridge development plays a crucial part in the urban transportation framework, in order to move forward transport frameworks and foundation. Recently, as bridge construction advances, development proficiency, maintainable support, and development taken a toll optimization are getting to be imperative issues in bridge development. In order to resolve these issues, numerous ponders have been conducted to optimize the cost-to-performance proportion. Among those concerning bridge advances, the superstructure of a bridge was distinguished as a basic viewpoint in maximizing bridge effectiveness.The most compelling way to optimize the cost-to-performance proportion is to maximize the effectiveness of the superstructure. Optimization of truss bridges is a popular topic in civil, and structural engineering due to the complexity of problems and benefits to industry. The topic of optimization, which accumulates the enthusiasm of numerous exploration groups, where the reason for it existing; is to simplify a goal work. Minimum weight shape and measuring ideal plan was executed, with the tallness of the truss and the cross-segment regions of its individuals constituting the outline factors of the issue. The auxiliary investigation and configuration were led as per the details of the

Eurocodes. The impact of both the tallness to-traverse proportion and the deck write on the heaviness of the truss, the aggregate weight and the cost is talked about in view of the outcomes got from the streamlining technique.

Keywords: Finite element method, optimal design, steel truss bridges, frp deck, concrete deck, steel deck, cost analysis of bridges.

TABLE OF CONTENTS

Title	Page No
ABSTRACT	i
TABLE OF CONTENTS..	ii
LIST OF TABLES	iii
LIST OF FIGURES	iv
NOMENCLATURE	v
CHAPTER 1 INTRODUCTION	
1.1 Objectives and scope of work …………………………………....	1
CHAPTER 2 LITERATURE REVIEW	
2.1 Introduction………………………………………………………..	10
2.2 Review of Literatures……………………………………………….	10
2.3 Summary……………………………………………………………….	14
CHAPTER 3 FINITE ELEMENT MODELING AND ANALYSIS	
3.1 Introduction ……………………………………………………...	15
3.2 Finite Element Modeling………………………………………...	15
3.3 Geometric Modeling Cases ……………………………………...	15
3.4 Geometric Modeling for Validation…………………………….	20
3.5 Material Behavior Model………………………………………..	21
3.6 Summary……………………………………………………………….	24
CHAPTER 4 RESULTS AND DISCUSSIONS	

4.1 Result And Discussion………………………………………………………… 27

4.2 Summary………………………………………………………………………. 34

References…………………………………………………………….............. 36

LIST OF FIGURES

Figure No.	Title	Page No.
1	Typical deck, through and semi-through type truss bridges	8
2	Typical Rigid frame bridge	9
3	Typical arch bridge	9
4	Suspension bridges	10
5	Section of Plate girder bridge	10
6	Section of Plate girder bridge	11
7	Box girder bridge	11
8	Truss girder bridge	11
9	Overall Bridge Layout	21
10	Front Elevation of Main Fettuccine	22
11	Longitudinal Elevation of below deck	22
12	Bridge Pier Dimension	23
13	Traffic loads	24
14	Abaqus Parts	25

15	Abaqus Material & Section	26
16	Abaqus Optimization	27
17	Abaqus Assembly	28
18	Abaqus 3D model with pier and abutment	28
19	Abaqus Individual Pier Part	29
20	Abaqus Model-1 Meshing	29
21	Material Property	30

LIST OF TABLES

Table No.	Title	Page No.
1	Optimization parameters used for size optimization of truss	**32**
2	SUMMARY OF RESULT FROM SIZE OPTIMIZATION OF TRUSS	**33**

CHAPTER 1

INTRODUCTION

Steel truss bridges are employed for long spans. It can carry heavy loads which cannot be economically carried by simple girder bridge. Efficient and economical design of truss bridge usually results in high cost. As urbanization increases and city populaces rapidly increment, transportation of individuals and stock must endend up more proficient. Bridge development plays a crucial part in the urban transportation framework, in order to move forward transport frameworks and foundation. Recently, as bridge construction advances, development proficiency, maintainable support, and development taken a toll optimization are getting to be imperative issues in bridge development. In order to resolve these issues, numerous ponders have been conducted to optimize the cost-to-performance proportion. Among those concerning bridge advances, the superstructure of a bridge was distinguished as a basic viewpoint in maximizing bridge effectiveness.

As of late, basic optimization has gotten to be an imperative tool for basic originators, since it permits a superior misuse of fabric, hence diminishing a structure's self-weight and sparing fabric costs. In addition, auxiliary optimization makes a difference the originator to discover inventive plan arrangements and auxiliary shapes that not as it were superior misuse fabric but moreover grant the structure more noteworthy tasteful esteem from an building point of see. In this article, the seismic retrofitting of a bridge initially outlined in strengthened concrete is outlined, appearing how helping the bridge superstructure, or maybe than fortifying the as of now completed establishments and projections, permitted these last mentioned highlights to stand up to more noteworthy seismic activities as required in the later overhaul of the Italian seismic code. Hence, other than utilizing the steel-concrete composite typology, the bridge superstructure was helped through basic optimization.

As we know that structural steel has high ratio of strength to cost in tension and a slightly higher strength to cost ratio in compression as compared to concrete as a building material. Structural steel has been used in construction of bridges since 1890. The first steel bridge Firth of the Forth was the first steel bridge.

The steel bridges have following advantages: -

- For the same load carrying capacity they have lower self-weight, this leads to smaller foundations.
- Steel construction takes less time as compared to that of nominal RCC construction.
- Steel has higher quality control.
- Steel is more efficient that concrete to withstand seismic and blast forces.
- Steel bridges have longer life span from that of concrete bridges.

With the advent of new paints corrosion is no longer a problem for steel bridges.

The steel used for bridges can be classified into three broad categories.

1. Carbon steel – The yield strength of this steel is up to 250N/mm^2. The steel confirming to ASTM A36, Euronorm 25 grades 235 & 275 and British grades 40 belong in this category.
2. High strength steel – The higher strength is due to the addition of alloying element. The steel conforming to ASTM A572, Euronorm 155 grade 360 and British grade 50 belong in this category. Varieties of this steel with resistance to atmospheric corrosion are also available.
3. Heat-treated carbon steel – These are the strongest categories of structural steel available. It derives its strength from various processes involved in the production of steel like quenching, normalisation, tempering etc.

Steel bridges are classified into various categories according to –

- Type of traffic they carry
- Carriageway position with respect to main structural system
- Structural system used to carry the load

Based on the traffic the bridges are further classified as –

- Road Bridges or Highway Bridges
- Rail Bridges
- Rod-cum-rail bridges

Based on Carriageway position with respect to main structural system bridges are classified as –

- Deck type
- Through type
- Semi through type

Deck type bridges – In these types of bridges the carriageway is located on the top of main load carrying structure.

Through type bridges - In these types of bridges the carriageway is located below the main load carrying structure.

Semi through type bridge - The deck lies in the middle of top and the bottom of the main load carrying members

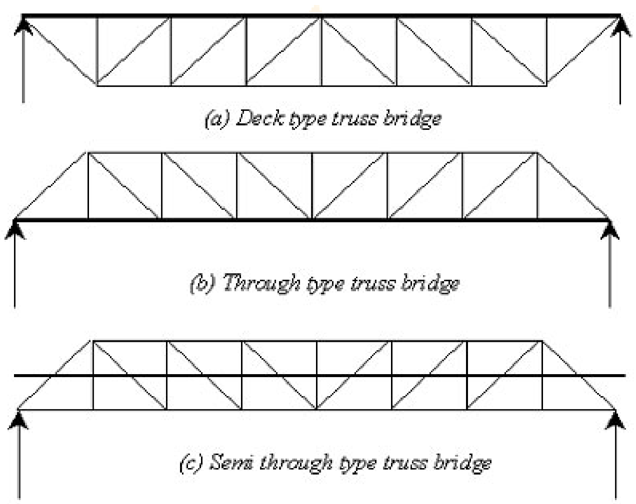

Fig.1: Typical deck, through and semi-through type truss bridges

Based on main structural system the bridges are classified into following main categories–

- Rigid frame bridges
- Arch Bridges
- Cable stayed bridges
- Suspension bridges
- Girder Bridges

Rigid frame bridges – The stability in these bridges is achieved through stability joint between longitudinal girders and vertical or inclined members. The major forces in members are flexure and axial (compressive). These are suitable for spans ranging from 25m to 200m.

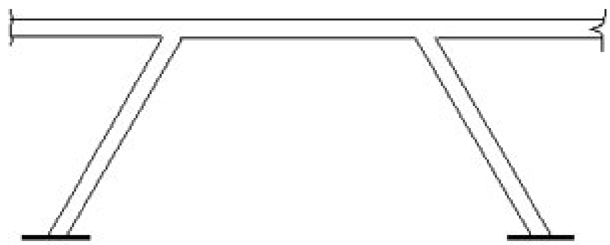

Fig.2: Typical Rigid frame bridge

Arch bridges – As the name implies the major load carrying system in these types of bridges is arch. Arch may be 3-hinged, 2-hinged or fixed. The major force in the arch is compression associated with some bending. Their spans range from 200m to 500m

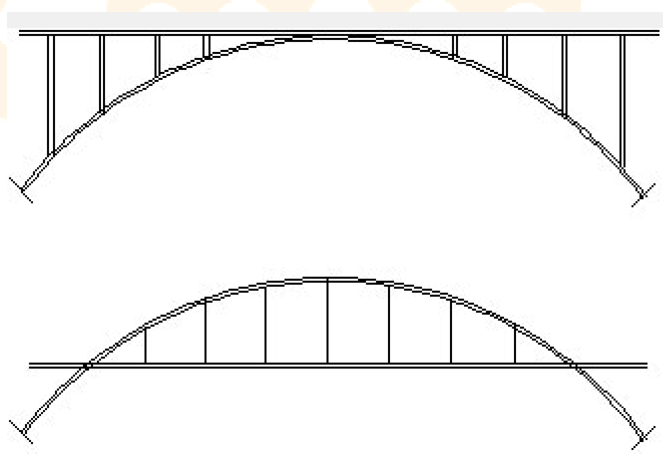

Fig.3: Typical arch bridge

Cable stayed bridges - The main longitudinal girders are supported by cables running in the vertical or near vertical planes. The cables run from towers and are anchors to the deck at bottom. Their spans range from 150m to 700m.

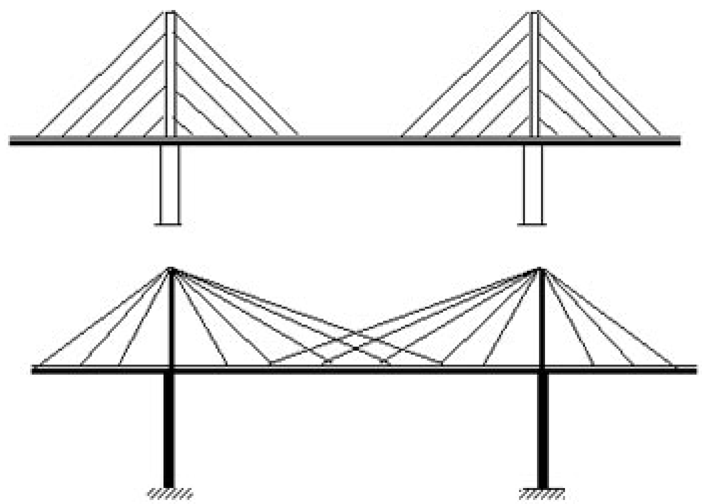

Fig.4: cable stayed bridges

Suspension bridges – For long span bridges the suspension bridge is the best and most economical solution. In this the deck is supported by the cables running between two tall towers and are anchored to the ground in the end.

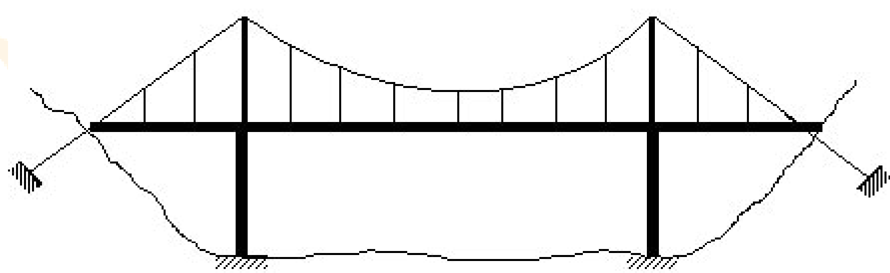

Fig.5: Suspension bridges

Girder bridges – The main forces in these types of bridges bending in vertical members. Girders are of three types – Solid girders, box girders and truss girders. For spans up to 50m plate girders are used, for spans up to 250m box girders are used. The span of Truss bridges can range from 30m to 375m.

Girder bridges can be further classified into simply supported spans, continuous spans and suspended & cantilever spans.

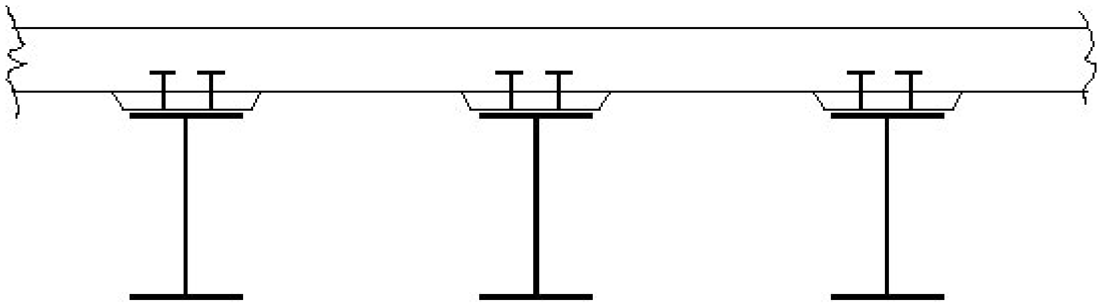

Fig.6: Section of Plate girder bridge

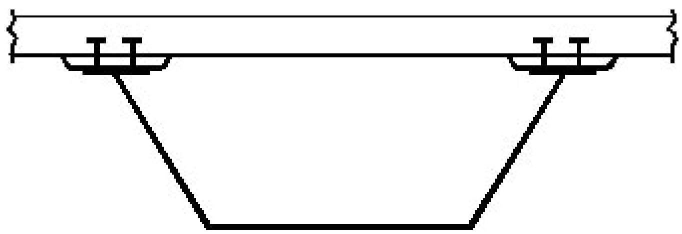

Fig.7 Box girder bridge

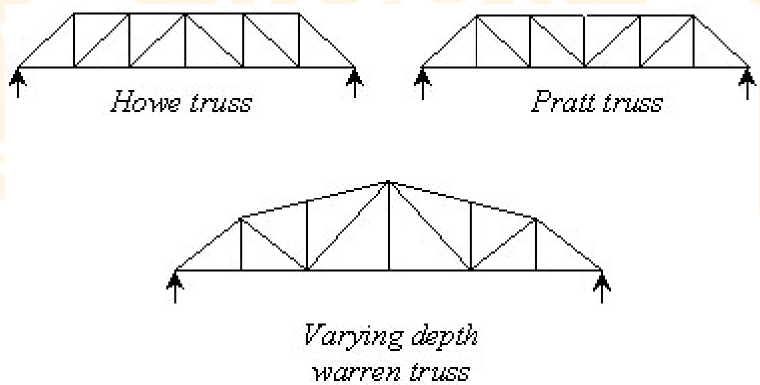

Fig.8 Truss girder bridge

1.1 OBJECTIVES AND SCOPE OF THE WORK

The reason for the examination is to explore the impact of both the tallness to-traverse proportion and the deck type on the heaviness of the truss, the aggregate weight and the cost in view of the outcomes acquired from the improvement technique. Finite element analysis software, ABAQUS is utilized.

The main objective of the study is:

1) The optimization analysis of truss bridge.

2) To investigate various design alternatives including cross-sectional area of members.

3) To compare the different shape and geometries of the truss bridges.

4) To investigate the effect of several influential parameters, with the focus on construction process-related and design-related variables.

1.2 ORGANIZATION OF THE THESIS

The thesis is organized into five chapters

1) The first chapter gives brief description of present thesis work, objectives and scope of the work.

2) The second chapter presents review of literature in this area.

3) The third chapter deals with the steps involved in the finite element modelling and optimization.

4) The fourth chapter deals with validation of the study and numerical investigation carried out for the present study.

5) The fifth chapter presents the conclusions of the presents study.

CHAPTER 2

LITERATURE REVIEW

2.1 BACKGROUND

Various tests and explanatory examinations have been directed to research the conduct of steel truss bridges. This section summarizes some of the previous work done in the development of bridge, and it depicts a brief history of the optimization. Various methods for predicting optimum solution discussed in order to understand the general trend of research and the limitation of their applications. Researches carried out on genetic algorithm is also discussed.

2.2 REVIEW OF LITERATURES

Chang et al. (2003) studied thermal elastoplastic investigation, utilizing finite element analysis (FEA), to examine the thermomechanical conduct and assess the leftover stresses in butt-welded joints. The leftover stresses at the surface of the weldments were estimated by X-ray diffraction method. The consequences of FEA were contrasted with trial residual stress data with an aim of affirming the precision of the technique.

Peric et al. (2006) conducted a numerical and exploratory examination on leftover stresses and twists incited by the T-joint welding of two plates. Inside the structure of numerical examinations, a thermo-mechanical FEA was performed by utilizing a shell/three-dimensional modelling technique to enhance both the computational proficiency and the precision. The impact of the decision of the regional 3D display estimate on the temperature, leftover pressure, and relocation dispersions was examined. An insignificant 3D zone estimate that had both fitting unions of the arrangement and exactness was characterized. To approve the numerical model, a progression of investigations utilizing a completely robotized welding process was led. A thermographic camera and an optical estimation framework were utilized to quantify the temperature and displacement distributions.

Mullins (2009) directed welding pretend programs utilizing isotropic, kinematic and mixed hardening models. The isotropic hardening model gave the best general concurrence with test estimations. The mixed hardening models gave great assertion for forecasts of the hoop stress yet tends to under-predict the greatness of the axial stress. The kinematic hardening model reliably under-anticipated the extent of both the axial and hoop pressure.

Gannon et al. (2010) performed numerical pretend programmes in view of finite element modelling to think about the impact of welding arrangements on the dissemination of leftover pressure and twisting produced when a flat bar stiffener is welded on to a steel plate. The reenactment comprises of consecutively coupled warm and auxiliary investigations utilizing an element birth and death system to display the expansion of weld metal to the workpiece. The temperature field amid welding, leftover stresses and contortion was anticipated, and outcomes were contrasted and test estimations and investigative forecasts.

Andersen (2001) proposed a strategy which consolidates a few strategies to anticipate welding twists in vast scale industrial welding applications. The format was set up in three stages. Initial, a generic model was made enabling the basic welding mechanics to be caught in fillet welding. Thereafter, techniques such as mesh grading or dynamic meshing were utilized to upgrade the computational productivity. At last, sub-structuring, global shell models and dynamically meshed models were consolidated in a layout for an additional increment of the proficiency. A critical increment in computational productivity has been acquired by the systems displayed however found in connection to the welding application, which contains in excess of 8 m of the weld, it is as yet a tedious assignment to register the welding distortions.

Wang et al. (2007) performed FEA to foresee the basic conduct of welded I-segment aluminum parts subjected to four-point bending. In the recreations, the quality of the material was decreased consistently in the HAZ near the welded stiffener. The yield capacity and work solidifying parameters for the warmth influenced zone, weld and construction material was resolved with respect to material tests and trial information accessible in the literature. The numerical recreations contain unequivocal investigations for an essential, moderately coarse mesh and certain examinations for a similar fundamental mesh and a refined mesh. Recreations were performed with impeccable and flawed geometries since a few beams failed while buckling locally. The numerical outcomes are contrasted and existing test information, and, by and large, great concurrence with the exploratory outcomes was acquired. Notwithstanding, the arrangements were observed to be mesh dependent for parts flopping by strain localization and crack in the tension flange.

Deng et al. (2007) built up an elastic finite element technique, in view of characteristic strain hypothesis, to exactly foresee welding contortion amid the gathering procedure considering both neighborhood shrinkage and root gap. To begin with, thermal elastic-plastic finite element method was utilized to evaluate intrinsic distortions for various ordinary welding joints. Second, the proposed versatile FEM was utilized to anticipate welding contortion for vast welded structures in view of the obtained inherent deformations. Impact of the initial gap on welding distortions is additionally examined. At last, tests were done to check the simulated results. The viability of the proposed versatile FEM was affirmed utilizing experimental results.

Nezo et al. (2011) proposed a blended time incorporation strategy to build the proficiency of welding simulation. The welding strategy can be separated into two extremely discernable parts with fundamentally unique qualities. The initial segment, the stage of the actual welding, is a quick paced, quickly changing procedure that includes exceedingly nonlinear material conduct. The second part, the period of cooling off, moderate paced and gradually changing procedure that does not bring about sensational changes in the material conduct. Unequivocal time combination was utilized for the thermo-mechanical investigation of welding and absolute time incorporation was proposed for the ensuing cooling stage.

Wanga et al. (2012) performed examinations to consider the physiognomies for the welding bending on thin plate structures. The test model was a thin plate cemented structure, and a broad contorting was taken note. The welding bending of a comparative structure was analyzed as an enormous distortion issue using a warm versatile plastic FEM and a flexible FEM in perspective of the possibility of normal deformation. The prepared results of the two procedures showed the curving bending, and the degree of this mutilation agreed well with the test estimation.

Chacon et al. (2009) researched a definitive load limit of the steel plate girders subjected to patch loading by methods for three unique techniques. The primary technique was by the presentation of sensible, exact shapes of initial imperfections already estimated for every girder together with a typical idealized residual stress pattern. The second strategy was by the presentation of initial imperfections in light of various eigenmodes (together with auxiliary defects, i.e. an idealized residual stress pattern). The third, by utilizing the identical starting defects proposed in EN1993-1-5, which incorporates both geometric and structural imperfections.

Chacon et al. (2012) directed an appraisal of the impact of auxiliary flaws, on a definitive load limit of steel plate girders when subjected to patch loading. A few residual stress patterns found in the writing were actualized in numerical reenactments of four distinct tests on girders subjected to patch loading. The fundamental resolution of such correlation is that, for every single numerical propagation of the tests, all idealized residual stress patterns tend to lead indistinguishable outcomes for both extreme load potential and structural reaction.

2.3 SUMMARY –

- Several techniques such as fully coupled stress analysis, sequentially coupled stress analysis and element birth and death technique described in the literature can be used for simulation of small or medium structures. But these methods are inapplicable to simulate in large structures because of computational time constraint.

- For large structures, other techniques such as inherent strain interface element, mixed time integration, graded element, dynamic meshing, and iterative substructuring method can significantly reduce computational time with an acceptable accuracy of predicting stresses and distortion.

CHAPTER 3

FINITE ELEMENT MODELLING AND ANALYSIS

3.1 GENERAL

In this chapter, the modelling strategy and features of Finite element (FE) analyses under ABAQUS, a general purpose program is discussed. ABAQUS 6.18 is used for the present study as it facilitates actual simulation of physical problem and provides an excellent alternative to the expensive experimental work. In this study Abaqus optimization computational approaches are employed. The simulation performed optimization based on topology concept to predict the final shape then non-linear analysis for finding ultimate load carrying capacity of bridge. From modelling perspective, it will be exceptionally valuable if the parameters of intrigue which add to the burdens and twists in different kinds of joint and structure application can be mimicked numerically. So, execution as for the different angles could be surveyed and assessed in an effective way.

3.2 GEOMETRY OF BRIDGE MODEL

In this work, basically sustained through-truss steel bridges with a Pratt truss configuration is used as shown in fig. 3. Pratt truss arrangement effects in rigidity of the slanting participants below the perpendicular loading. Bridge length is 150 meters with Truss element. As shown in the fig3. the bridge, with a continuous three span 50m-50m-50m has an overall distance of 150m. The floor has a continuous profundity lengthways and its longitudinal axis is straight and horizontal. The material used is bridge Iron.

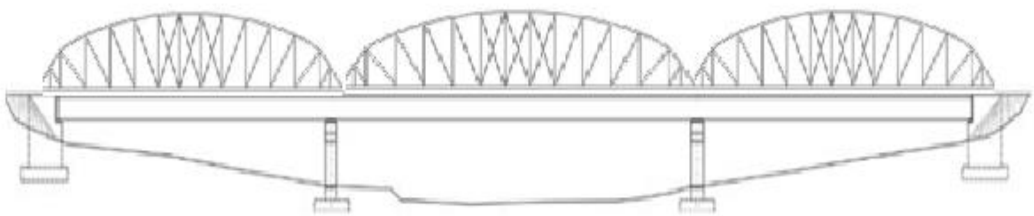

Figure 9 : Overall Bridge Layout

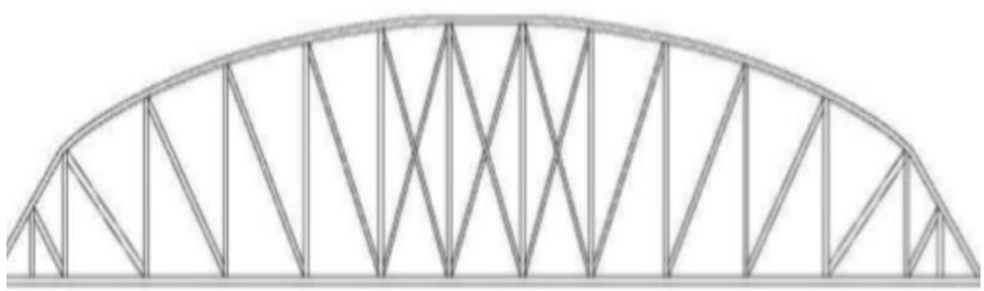

Figure 10 : Front Elevation of Main Fettuccine Bridge

In this work, solely supported through-truss steel bridges with a Pratt truss configuration is used as shown in fig. 3. Pratt truss arrangement brings about strain of the corner to corner individuals under the vertical stacking.

The extent of the vertical truss members are carefully chosen to confine the angle of the diagonals between 35° and 65° approximately to reduce the buckling length of the compressed members.

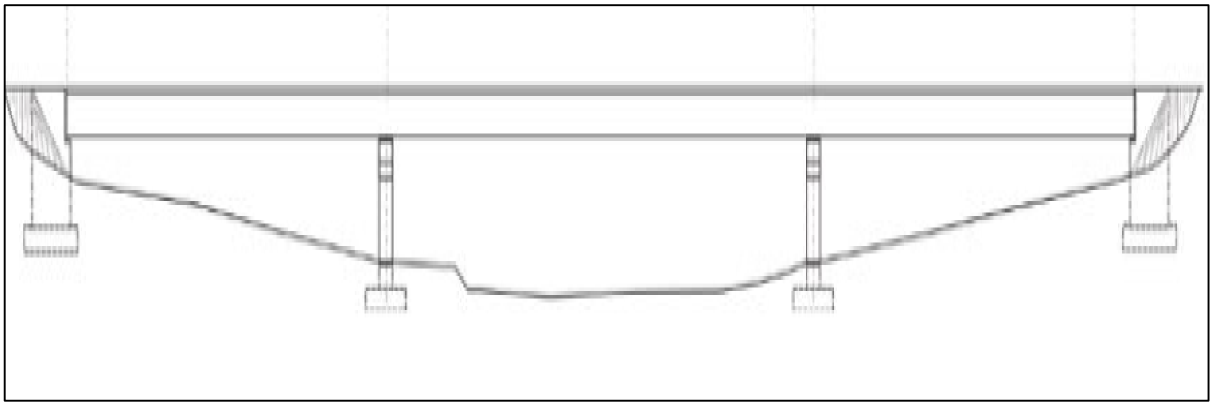

Figure 11 : Longitudinal Elevation of below deck

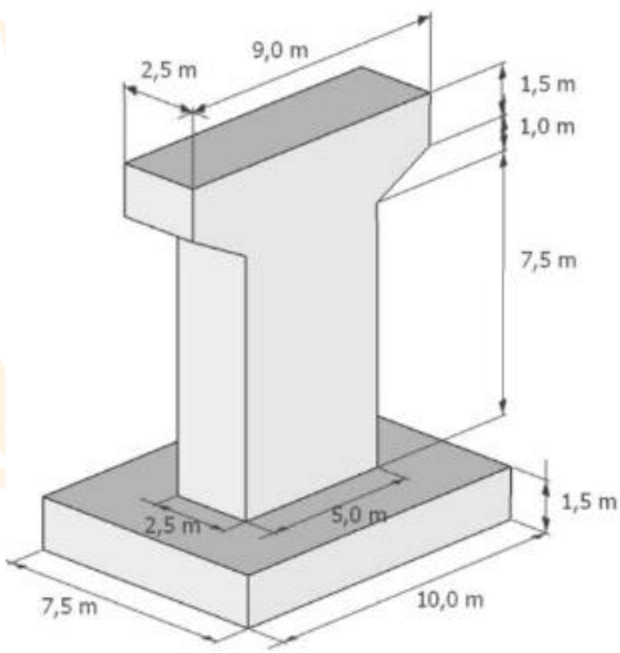

Figure 12 : Bridge Pier Dimension

Bridge structures consists of two parts: substructure and super structure. Substructure includes all component which transfer the loads from bearing to soil. Substructure of bridge consists of pier, abutment piles etc. while the superstructure is the part above bearing includes deck, girders, road way etc.

Bridge Pier transfer the loads to the soil. It is provided intermediately for long bridge it provides supports to the spans of the bridge. Scaffold Piers ought to be solid to take both axial, vertical and horizontal loads. Wharfs fundamental capacity is to exchange the heap from the bearing to connect establishment underneath it. They are subjected to substantial hub loads, bi-pivotal minutes and shear powers in longitudinal and transverse bearing. The wharfs are 10 m high with a strong rectangular cross-segment. The deck is upheld at docks with coordinate contact with bearing. Extension docks are of two kind - Fixed and Free wharfs. Wharfs supporting a settled bearing are called settled docks and those under free orientation are called free docks. Settled wharfs are liable to transverse and longitudinal powers while free docks exchange just pivotal powers from the bearing to the dirt.

The target capacity of the advancement issue was the heaviness of the truss, with the Eurocode's plan prerequisites being the limitations. The objective of the enhancement process was to limit the goal work. Shape advancement of the truss was directed by choosing its tallness at midspan as the plan variable. The cross-sectional zones of the individuals constituted the estimating advancement plan factors. European standard hot-rolled section are chosen for the steel individuals.

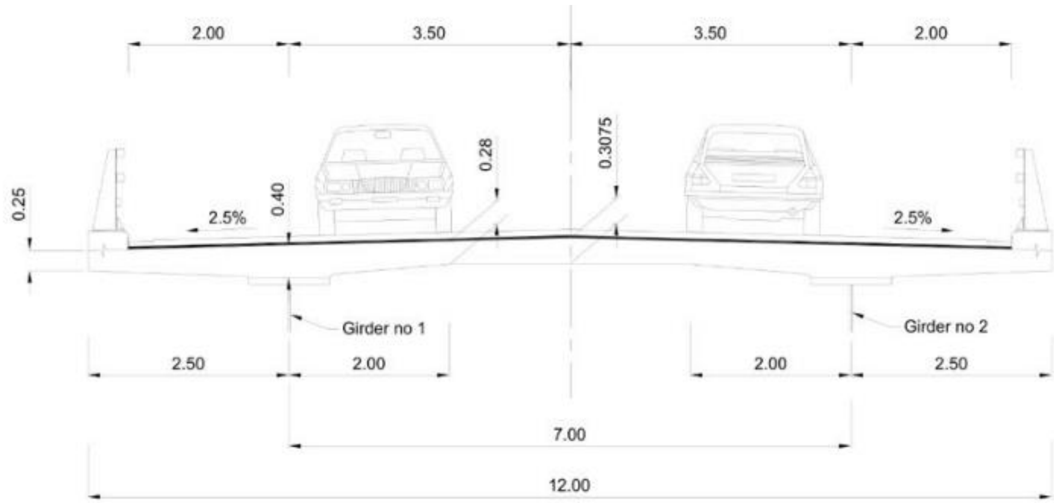

Figure 13 : Traffic loads

The road has two traffic lanes 3.5 m wide, with total width of 11 m which includes 2m wide hard strips see Fig 1. Considering 0.5 m for the vehicle parapet of each side,the total width of the concrete slab equal to 12 m.

The traffic loads specified in EN1991-2 were applied to the studied bridge. Wind action per EN1991-1-4 was also taken into account. All relevant analyses were performed by the commercial software Abaqus, which implements the Finite Element Method (FEM). the modelling strategy and features of Finite element (FE) analyses under ABAQUS, a general purpose program is discussed. ABAQUS 6.18 is used for the present study as it facilitates actual simulation of physical problem and provides an excellent alternative to the expensive experimental work. In this study Abaqus optimization computational approaches are employed. The simulation performed optimization based on topology concept to predict the final shape then non-linear analysis for finding ultimate load carrying capacity of bridge. From displaying perspective, it will be extremely helpful if the parameters of intrigue which add to the anxieties and mutilations in different sorts of joint and structure application can be reenacted numerically. So, execution regarding the different viewpoints could be surveyed and assessed in a productive way.

3.2 FINITE ELEMENT DESCRIPTION OF MODEL

Within the framework of numerical investigation, three models are considered for calculation of inherent deformation and comparison with inherent strain method, second web panel for finding influence of design variable and process related variables, third girder with two imperfection patterns first one with buckling mode and second with inherent deformed shape.

ABAQUS/CAE, a graphical preprocessor program is utilized to define finite element model. Parts are the building blocks of model, each part is created by part module. In this work 6 parts are created which includes deck, truss, girder, pier, abutment, then assembly module is used to assemble instances of this parts.

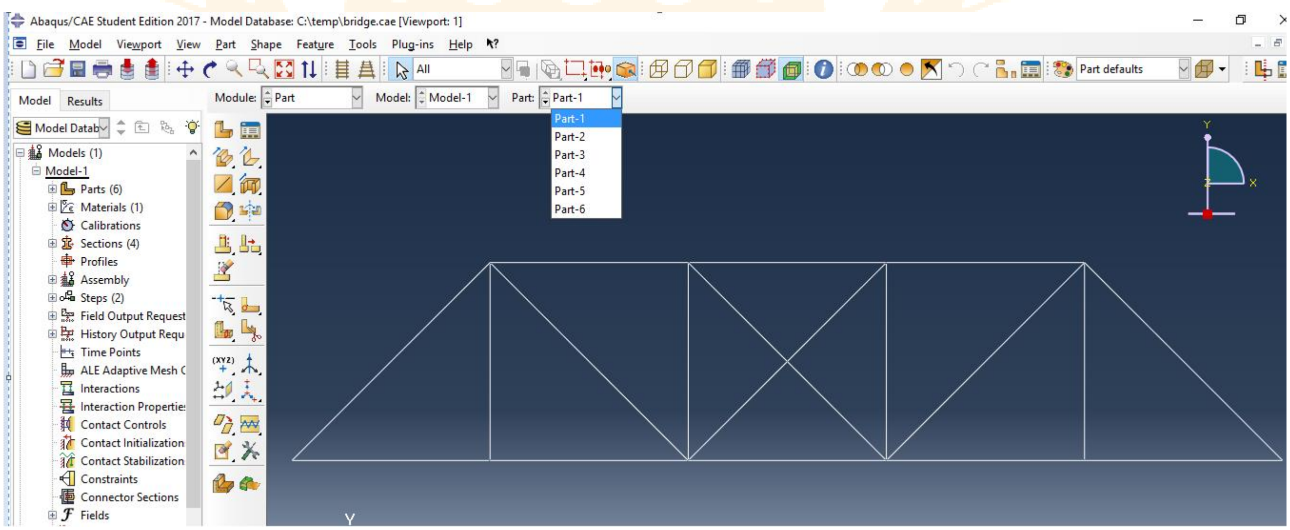

Figure 14 : Abaqus Parts

In this work Model-1 is created using 6 parts, different part of the model have different property some are solid, truss, shell etc. For 3D model, deformable solid feature extrusion type is used and for modeling pier and abutment. They are defined along the length of base then cell portion is used and divided into numbers of chunks.

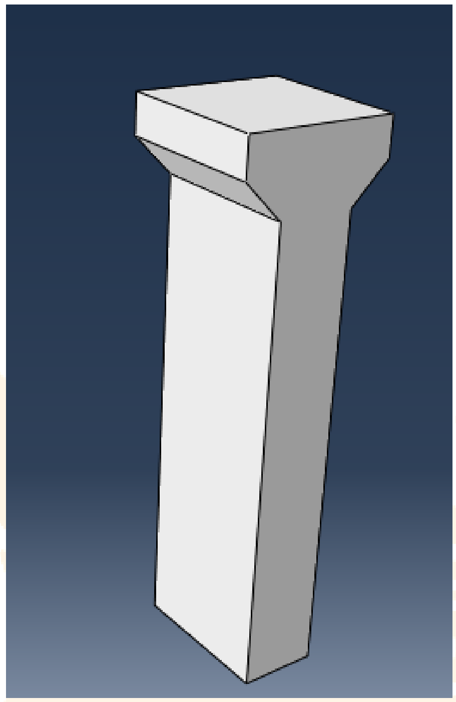

Figure 14 : Abaqus Individual Pier Part

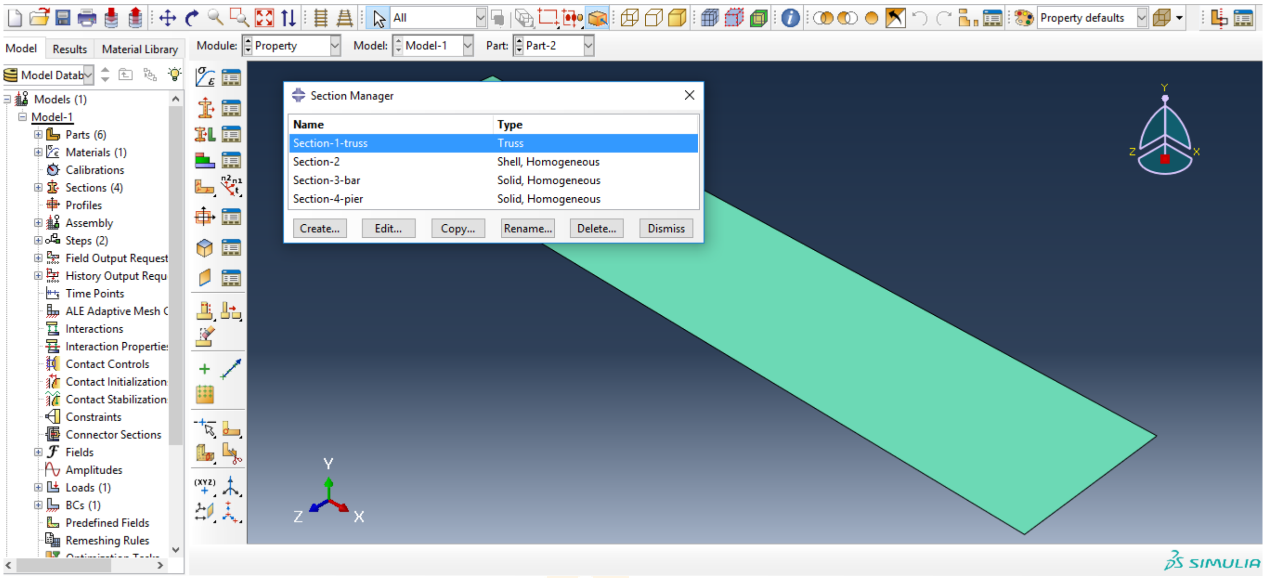

Figure 15 : Abaqus Material & Section

Material Property is defined in material module then different section is assigned to different parts using material defined. Both geometric and material non-linearity is considered in the numerical simulation and the stress-strain relationship for iron is assumed to be elasto-plastic. Non-linear temperature dependent material properties are used and material is assumed to follow the von Mises yield criterion. Linear isotropic hardening is assumed and autogeneous weldment is used.

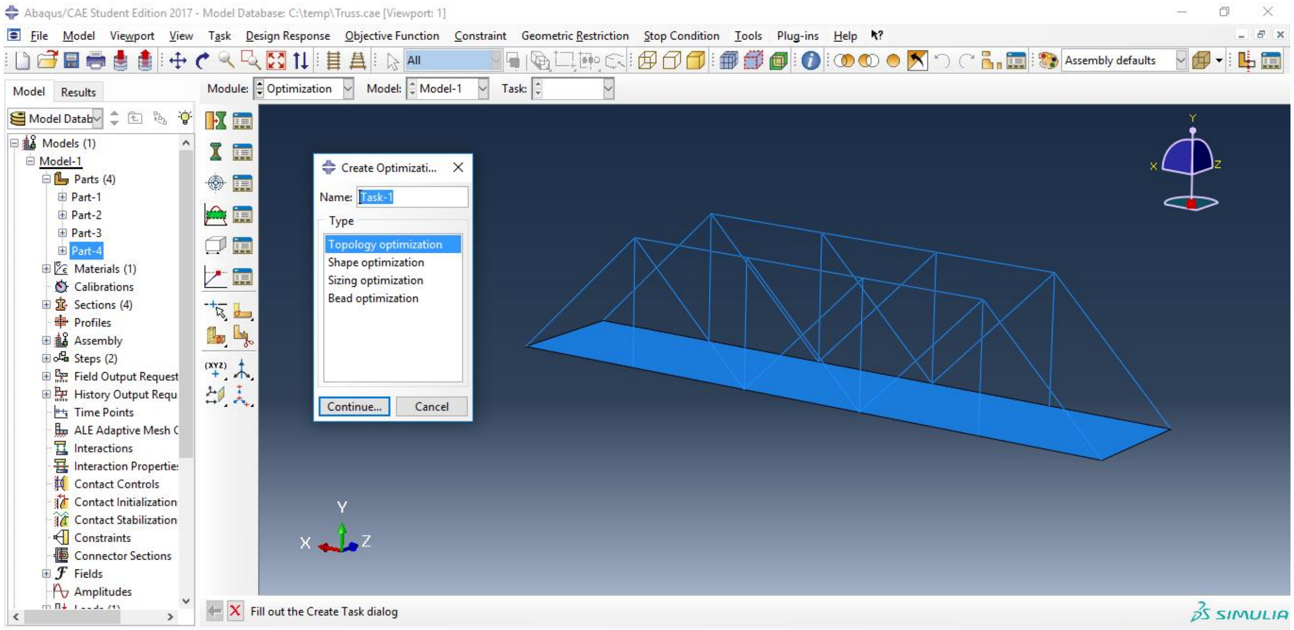

Figure 16 : Abaqus Optimization

Four type of optimization can be performed in Abaqus :Topologic Optimization , Shape Optimization , Size Optimization, Bead Optimization. Streamlining is a subject which assembles the enthusiasm of numerous examination groups, where the reason for existing is to limit a goal work. Slightest weight shape and measuring ideal plan was executed, with the tallness of the truss and the cross-segment zones of its individuals constituting the outline factors of the issue. The basic examination and configuration were directed as per the particulars of the Eurocodes. The impact of both the tallness to-traverse proportion and the deck write on the heaviness of the truss, the aggregate weight and the cost is talked about in light of the outcomes acquired from the improvement system.

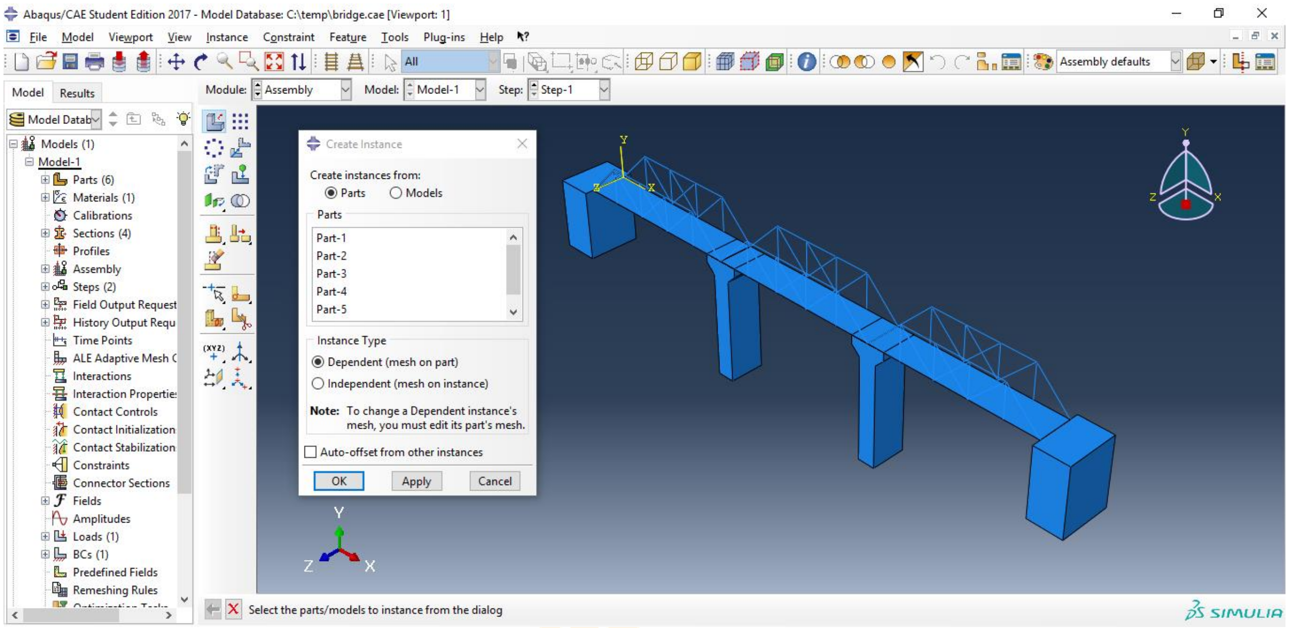

Figure 17 : Abaqus Assembly

INTERACTION MODULE IS USED TO DEFINE MECHANICAL INTERACTIONS BETWEEN REGIONS OF A MODEL AND BETWEEN THE REGION OF A MODEL AND ITS SURROUNDINGS. INTERACTIONS ARE STEP-DEPENDENT OBJECTS, WHICH MEAN THAT THEY ARE ACTIVE FOR SPECIFIC STEPS OF THE ANALYSIS. IN ASSEMBLY MODULE ALL PARTS ARE ASSEMBLED.

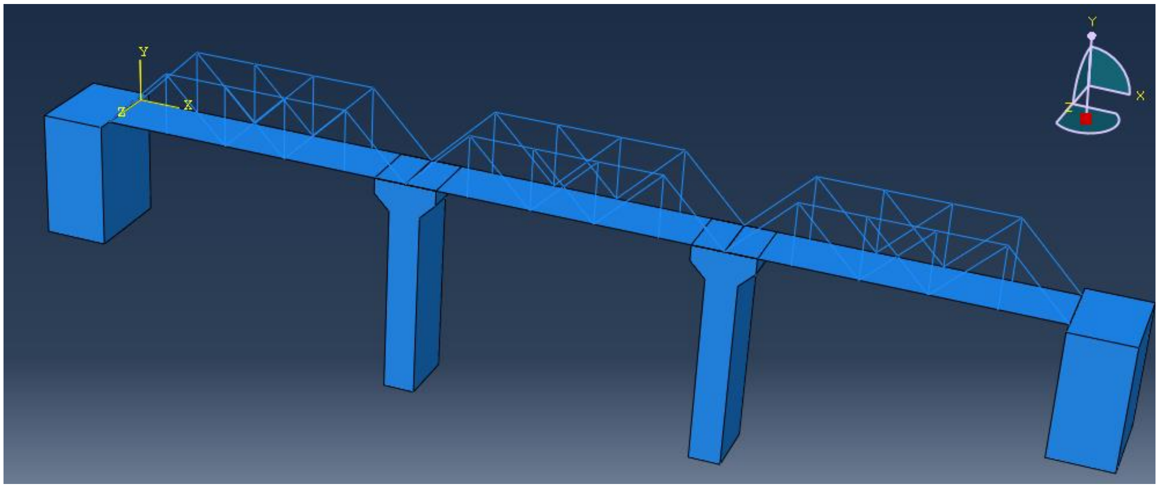

Figure 18 : Abaqus 3D model with Pier and Abutment

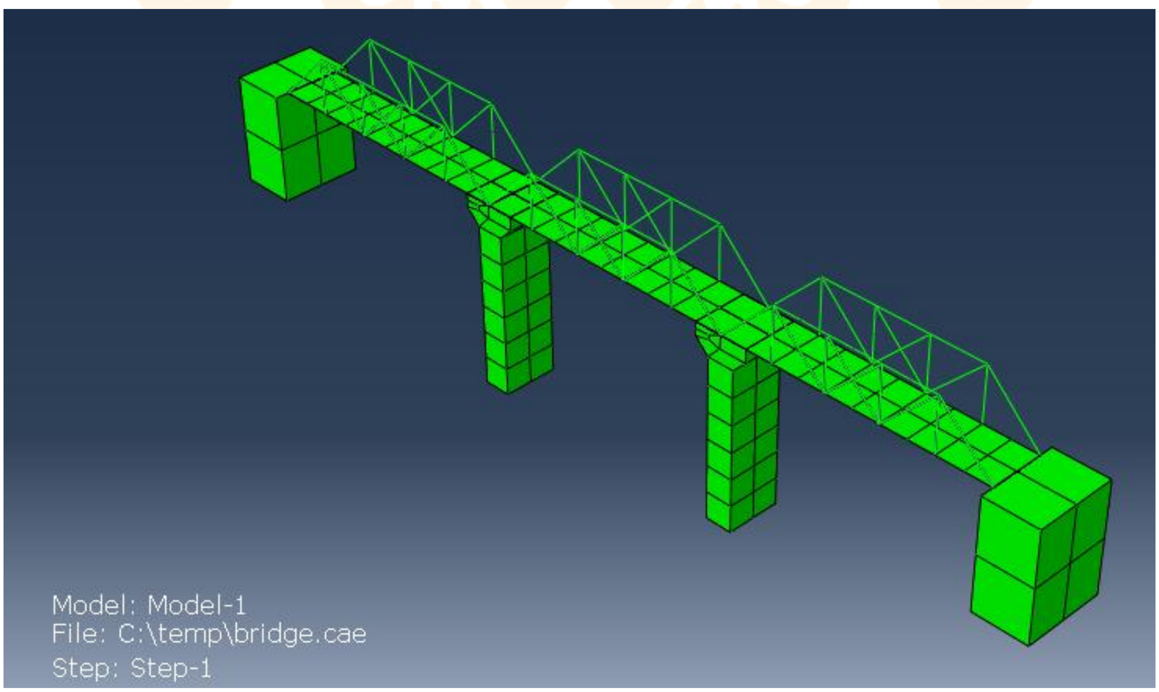

Figure 20 : Abaqus Model-1 Meshing

Model-1 consist of different parts of bridge, it is meshed according to the type of parts. The size of the mesh restricts the result. finer the mesh better the result but too fine mesh result in large computational time so here balance level is selected considering both the factor accuracy and computational time.

Specialists and researchers utilize limited component investigation (FEA) programming to construct prescient computational models of true situations. The utilization of FEA programming starts with a PC supported plan (CAD) show that speaks to the physical parts being reproduced and in addition information of the material properties and the connected burdens and requirements. This data empowers the expectation of genuine conduct, frequently with large amounts of precision.

The accuracy that can be obtained from any FEA show is particularly related to the constrained segment work that is used. The constrained part work is used to subdivide the CAD show into tinier territories called segments, over which a game plan of conditions are handled. These conditions around address the regulating state of interest by methods for a course of action of polynomial limits portrayed over each part. As these segments are made more diminutive and smaller, as the work is refined, the enrolled game plan will approach the veritable course of action.

A decent limited component examiner begins with both a comprehension of the material science of the framework that will be broke down and an entire portrayal of the geometry of the framework. This geometry is spoken to by means of a CAD display. An ordinary CAD model will precisely portray the shape and structure, yet frequently likewise contain corrective highlights or assembling subtle elements that can turn out to be unessential for the motivations behind limited component demonstrating. The investigator should put some designing judgment into analyzing the CAD model and choosing if these highlights and subtle elements can be expelled or disentangled preceding lattice. Beginning with a straightforward model and including intricacy is quite often simpler than beginning with an intricate model and streamlining it. Specialists and researchers utilize limited component investigation (FEA) programming to construct prescient computational models of true situations. The utilization of FEA programming starts with a PC supported plan (CAD) show that speaks to the physical parts being reproduced and in addition information of the material properties and the connected burdens and requirements. This data empowers the expectation of genuine conduct, frequently with large amounts of precision.

The precision that can be acquired from any FEA demonstrate is specifically identified with the limited component work that is utilized. The limited component work is utilized to subdivide the CAD demonstrate into littler areas called components, over which an arrangement of conditions are tackled. These conditions around speak to the overseeing condition of intrigue by means of an arrangement of polynomial capacities characterized over every component. As these components are made littler and littler, as the work is refined, the registered arrangement will approach the genuine arrangement.

The investigator ought to likewise know the greater part of the physical science that are pertinent to the issue, the materials properties, the heaps, the limitations, and any components that can influence the consequences of premium. These information sources may have vulnerabilities in them. For example, the material properties and burdens may not generally be correctly known. It is essential to remember this amid the demonstrating procedure, as there is no advantage in endeavoring to determine a model to more prominent precision than the information concedes. When the greater part of this data is collected into a FEA show, the examiner can start with a preparatory work. Right on time in the examination procedure, it bodes well to begin with a work that is as coarse as conceivable – a work with vast components. A coarse work will require less computational assets to understand and, while it might give an extremely mistaken arrangement, it can in any case be utilized as a harsh confirmation and as a mind the connected burdens and limitations.

Subsequent to figuring the arrangement on the coarse work, the procedure of work refinement starts. In its most straightforward shape, work refinement is the way toward settling the model with progressively better and better networks, looking at the outcomes between these distinctive lattices. This examination should be possible by breaking down the fields at least one focuses in the model or by assessing the basic of a field over a few spaces or limits.

By looking at these scalar amounts, it is conceivable to judge the union of the arrangement regarding network refinement. Subsequent to looking at least three progressive arrangements, an asymptotic conduct of the arrangement begins to develop, and the adjustments in the arrangement between networks wind up littler. In the long run, these progressions will be sufficiently little that the expert can view the model as met. This is dependably a careful decision with respect to the expert, who knows the vulnerabilities in the model data sources and the satisfactory vulnerability in the outcomes.

Examining merging requires picking a proper work refinement metric. This metric can be either neighborhood or worldwide. That is, the metric can be characterized at one area in the model or as the indispensable of the fields over the whole model space. A case of a neighborhood metric is the removal or worry at a point inside an auxiliary investigation. A case of a worldwide metric is the vital of the strain vitality thickness over all spaces. Both the anxieties and the strain are figured in light of the slope of the arrangement and the relocation field. Inclinations of the arrangement are constantly processed to one request bring down polynomial estimation.

While picking a metric, it is critical to recollect that distinctive measurements will have diverse merging conduct. This is delineated in the figure beneath, indicating distinctive cross sections being utilized to illuminate the same FEA display. These cross sections vary as far as the component estimate and are looked at as far as the quantity of degrees of opportunity (DOF) inside the model. The DOF is identified with the quantity of hubs, the computational focuses that characterize the state of each limited component. The computational assets required to illuminate a FEA show are straightforwardly identified with the quantity of DOF.

From the figure underneath, it shows up as though certain measurements focalize speedier than others, yet it is imperative to remember that the rate of work merging for a specific issue proclamation is needy whereupon work refinement method is utilized.

With respect to work refinement, there is a suite of frameworks that are consistently used. An expert customer of FEA programming should be alright with each one of these frameworks and the tradeoffs between them

Reducing the Element Size –

Lessening the Element Size is the most easy work refinement system, with segment sizes decreased all through the showing territories. This approach is engaging owing to a direct result of its ease, however the drawback is that there is no extraordinary work refinement in zones where a locally better work may be required.

Increasing the Element Order -

Expanding the Element Order is significant as in no remeshing is required; a comparable work can be used, however with different part arranges. Remeshing can be dull for complex 3D geometries or the work may begin from an external source and can't be altered. The downside to this system is that the computational requirements increase speedier than with other work refinement methodologies.

Global Adaptive Mesh Refinement-

Worldwide versatile work refinement utilizes a blunder estimation methodology to decide the point in the demonstrating space where the neighborhood mistake is biggest. The FEA programming at that point takes this mistake estimation and utilizations the data to produce a totally new work. Littler components are utilized as a part of areas where the nearby mistake is huge, and the neighborhood blunder all through the model is considered. The preferred standpoint here is that the product will do the greater part of the work refinement. The disadvantage is that the client has no power over the work. In that capacity, over the top work refinement may happen in areas that are of less intrigue, districts where a bigger nearby blunder is satisfactory.

Local Adaptive Mesh Refinement-

Neighborhood versatile work refinement contrasts from worldwide versatile work refinement in that the blunder is assessed just finished some subset of the whole model space, concerning a particular metric. For instance, it is conceivable to refine the work to such an extent that worries at the limit of an opening are all the more precisely settled. This cross section procedure will in any case remesh the whole model with the goal of diminishing the blunder in one district. On the off chance that a consistent and alluring nearby metric exists regarding which work can be refined, the neighborhood versatile approach is better than worldwide versatile work refinement.

Manually Adjusting the Mesh-

The most work escalated approach is for the investigator to physically make a progression of various limited component networks in view of the material science of the specific issue and an instinct as to where better components might be required. For 2D models, a blend of triangular and quadrilateral components can be utilized. On account of 3D models, a blend of tetrahedral, hexahedral (additionally called blocks), triangular kaleidoscopic, and pyramidal components can be utilized. While triangular and tetrahedral components can be used to work any geometry, the quadrilateral, hexahedral, kaleidoscopic, and pyramidal components are useful when the arrangement is known to change continuously along at least one headings. By extending, or contracting, components in specific bearings, the work can be tuned to the variety in the fields.

Time-Domain and Frequency-Domain Meshing-

Alongside the majority of the above methods, extra contemplations ought to be remembered when fitting issues that have time-fluctuating burdens. A model with nonlinear material reactions or subjective time-changing excitations would should be explained in the time space. Then again, if the connected excitation is of a solitary recurrence or a scope of known frequencies and the material properties are straight, at that point it is favored for the demonstrating to happen in the recurrence area. There are extra work refinement procedures for every one of these cases.

Time-Adaptive Mesh Refinement-

Time-versatile work refinement remeshes the model at unmistakable time interims and considers a mistake gauge of the arrangement at every interim as the metric by which to remesh the model. This is valuable when the locales requiring great work determination move after some time.

Wavelength-Adaptive Mesh Refinement-

Alongside the majority of the above methods, extra contemplations ought to be remembered when fitting issues that have time-fluctuating burdens. A model with nonlinear material reactions or subjective time-changing excitations would should be explained in the time space. Then again, if the connected excitation is of a solitary recurrence or a scope of known frequencies and the material properties are straight, at that point it is favored for the demonstrating to happen in the recurrence area. There are extra work refinement procedures for every one of these cases.

Trends in Finite Element Meshing -

The key point to remember with these methodologies is that, regardless of which technique is utilized, they will all meet toward a similar answer for the postured issue. The contrast between these different methodologies is just in the rate at which they unite. This is, be that as it may, a huge handy contrast. Contingent on the issue, one method may join considerably quicker than others, and nobody refinement system is proper in all conditions. Each issue will have its one of a kind cross section challenges, which keeps on posturing troubles for experts.

A few changes are in progress that will facilitate these difficulties. A standout amongst the most essential advancements in the course of the most recent couple of years has been progressively simple access to reasonable distributed computing assets, empowering the running of a few distinct cases in parallel. This enables experts to examine numerous model and work varieties in significantly less time, enabling them to rapidly address the majority of the vulnerabilities.

The calculations used to produce the cross sections themselves are likewise consistently enhancing and taking more noteworthy favorable position of multicore registering. Furthermore, the solvers are ending up more effective, with the capacity to fathom tremendous models on group PCs. These progressions will give more exact arrangements in less time, while quickening the examination and configuration process.

3.5 MATERIAL BAHAVIOUR MODEL

Both geometric and material non-linearity is considered in the numerical simulation and the stress-strain relationship for iron is assumed to be elasto-plastic. Non-linear temperature dependent material properties are used and material is assumed to follow the von Mises yield criterion. Linear isotropic hardening is assumed and autogeneous weldment is used. Different parameters used for analysis-Film property: film coefficient=0.025 mJ/s/mm^2/k, Radiation property: emissivity=0.9

Ambient Air temperature: 21.1 °C

Temperature dependent mechanical and thermal property used for modeling is shown i

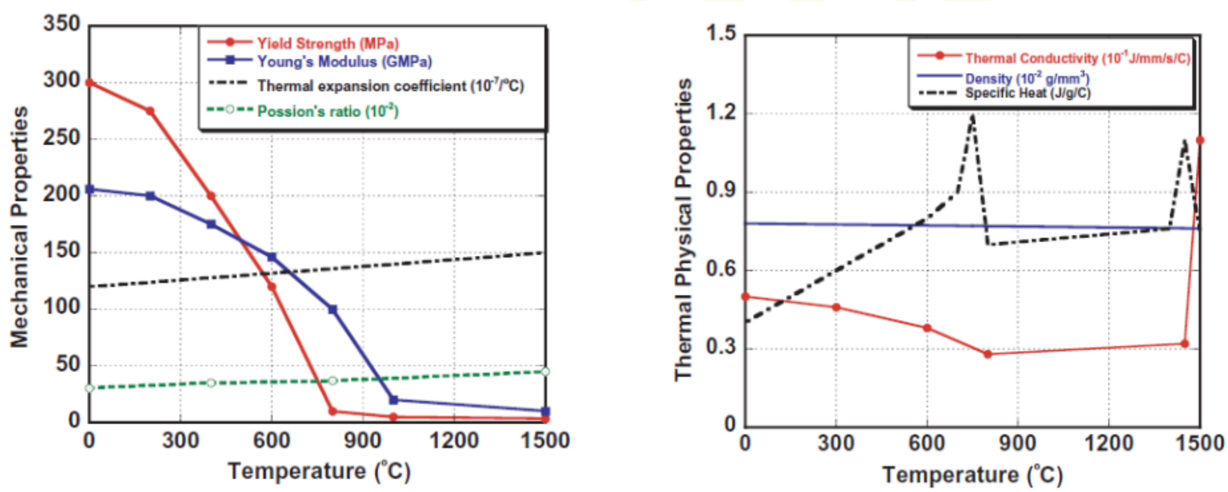

Figure 21 : Material

NONLINEAR ANALYSIS

In structural analysis, nonlinearity is used to describe a problem where the relationship between load and deflection is not constant. This occurs when stiffness of the structure changes over the course of simulation. Two types of nonlinearities are geometric nonlinearity and material nonlinearity. Here, both the nonlinearities are considered.

Static Riks analysis in ABAQUS/Standard is used to study unstable behaviour. It is performed if there is worry about material nonlinearity, geometric nonlinearity before clasping, or temperamental post clasping reaction. Riks technique utilized the heap size as an extra obscure; it clarifies all the while for burdens and removals. In this manner, another amount must be utilized to gauge the advance of arrangement; ABAQUS/Standard utilizes the 'circular segment length' along the static balance way in stack relocation space. This approach gives arrangements paying little respect to whether the reaction is steady or unsteady. ABAQUS/Standard uses Newton's technique to comprehend nonlinear harmony conditions. The Riks strategy utilizes just a 1% extrapolation of the strain increase.

In structural analysis, nonlinearity is used to describe a problem where the relationship between load and deflection is not constant. This occurs when stiffness of the structure changes over the course of simulation. Two types of nonlinearities are geometric nonlinearity and material nonlinearity. Here, both the nonlinearities are considered. Static Riks analysis in ABAQUS/Standard is used to study unstable behavior. It is performed if there is stress over material nonlinearity, geometric nonlinearity before fastening, or touchy post catching response. Riks procedure used the stack measure as an additional unknown factor; it lights up at the same time for weights and expulsions. In this way, another sum must be used to check the progress of game plan; ABAQUS/Standard uses the 'round portion length' along the static adjust route in stack migration space. This approach gives courses of action paying little regard to whether the response is relentless or flimsy. ABAQUS/Standard uses Newton's procedure to fathom nonlinear agreement conditions. The Riks procedure uses only a 1% extrapolation of the strain increment.

In structural analysis, nonlinearity is used to describe a problem where the relationship between load and deflection is not constant. This occurs when stiffness of the structure changes over the course of simulation. Two types of nonlinearities are geometric nonlinearity and material nonlinearity. Here, both the nonlinearities are considered.

Static Riks analysis in ABAQUS/Standard is used to study unstable behaviour. It is performed if there is worry over material nonlinearity, geometric nonlinearity before attaching, or sensitive post getting reaction. Riks methodology utilized the stack measure as an extra unknown; it illuminates in the meantime for weights and removals. Thusly, another entirety must be utilized to check the advance of approach; ABAQUS/Standard uses the 'round part length' along the static modify course in stack movement space. This approach gives strategies paying little respect to whether the reaction is determined or shaky. ABAQUS/Standard uses Newton's strategy to understand nonlinear assentation conditions. The Riks methodology utilizes just a 1% extrapolation of the strain increase.

In structural analysis, nonlinearity is used to describe a problem where the relationship between load and deflection is not constant. This occurs when stiffness of the structure changes over the course of simulation. Two types of nonlinearities are geometric

nonlinearity and material nonlinearity. Here, both the nonlinearities are considered. Static Riks analysis in ABAQUS/Standard is used to study unstable behavior. It is performed if there is worry about material nonlinearity, geometric nonlinearity before clasping, or flimsy post clasping reaction. Riks strategy utilized the heap extent as an extra obscure; it understands at the same time for burdens and relocations. In this way, another amount must be utilized to quantify the advance of arrangement; ABAQUS/Standard uses the 'circular segment length' along the static harmony way in stack removal space. This approach gives arrangements paying little heed to whether the reaction is steady or precarious. ABAQUS/Standard uses Newton's technique to comprehend nonlinear balance conditions. In this chapter, the modelling strategy and features of Finite element (FE) analyses under ABAQUS, a general purpose program is discussed. ABAQUS 6.18 is used for the present study as it facilitates actual simulation of physical problem and provides an excellent alternative to the expensive experimental work. In this study Abaqus optimization computational approaches are employed. The simulation performed optimization based on topology concept to predict the final shape then non-linear analysis for finding ultimate load carrying capacity of bridge. From displaying perspective, it will be extremely helpful if the parameters of intrigue which add to the burdens and contortions in different kinds of joint and structure application can be recreated numerically. So, execution as for the different angles could be surveyed and assessed in a productive way.

The Riks system uses just 1% extrapolation of the strain increase. Static Riks examination in ABAQUS/Standard is utilized to contemplate insecure conduct. It is performed if there is worry about material nonlinearity, geometric nonlinearity preceding clasping, or flimsy post clasping reaction. Riks technique utilized the heap greatness as an extra obscure; it tackles at the same time for burdens and relocations. Thusly, another amount must be utilized to gauge the advance of arrangement; ABAQUS/Standard uses the 'curve length' along the static harmony way in stack uprooting space. This approach gives arrangements paying little respect to whether the reaction is steady or unsteady. ABAQUS/Standard uses Newton's technique to comprehend nonlinear harmony conditions. The system utilizes just a 1% stain increase extrapolation.

3. SUMMARY

The finite element simulation parameters are selected. Geometric, material features of model are discussed. Details concerning the analysis procedure type and sequence of analysis performance are identified.

CHAPTER 4

RESULTS AND SUMMRY

RESULT

The outcomes from the size optimization performed on the truss can be found in Table 3. As found in the table, the volume diminishes with around 38 % after the two stages, contrasted and the estimations of the un-improved truss. It can likewise be seen that the pressure constrains connected in the second step isn't come to. It ought to be noticed that the diversion for all cases is 2.01 mm, which is the same as far as possible.

This article reports and examines the utilization of improvement to streamline the plan of truss spans. This is a testing enhancement issue related with blended plan factors, since it includes recognizable proof of the extension's shape and topology setups notwithstanding the sizing of the auxiliary individuals for least weight. An answer calculation to this issue is created by joining diverse variable-wise adaptations of versatile ESs under a typical streamlining schedule. In such manner, size and shape improvements are actualized utilizing discrete and constant ESs, separately, while topology advancement is accomplished through a discrete adaptation combined with a specific philosophy for producing topological varieties. In the examination, an outline area approach is utilized in conjunction with ESs to look for the ideal shape and topology arrangement of a scaffold in an extensive and adaptable plan space. It is demonstrated that the subsequent calculation performs extremely well and produces enhanced outcomes for the issues of intrigue.

TABLE 1 OPTIMIZATION PARAMETERS USED FOR SIZE OPTIMIZATION OF TRUSS

	Step 1	Step 2
Optimization objective	Minimize volume	Minimize volume
Optimization constraints	Deflections limit (2.01mm)	Deflection limit (2.01 mm) Stress limit (200Mpa)
Design variable limits	0<A<1000 mm2	0<A<1000 mm2

The optimization is done in two stages. In the initial step, the truss is advanced with the goal to limit the aggregate volume of the structure. Diversion imperatives are connected in an indistinguishable point from the heap. As far as possible is been the redirection from a static examination of the un-streamlined truss. By doing this, the diversion is kept steady through the streamlining procedure which makes it conceivable to perceive how a similar execution can be accomplished with less material. After the first advancement is finished, the bars with low zone are expelled from the model. Keeping those bars would give a truss with unreasonably little individuals, and also a truss that is pointlessly muddled.

Stage two is to re-try the optimization on the new truss, from which a few components have been evacuated. The parameters are fundamentally the same, except for an extra imperative on the burdens. The purpose behind not having the pressure limitation in the initial step is that a pressure requirement all things considered would keep the regions of the bars from getting little. Despite the fact that the powers in the bars would be low, a little territory would at present offer ascent to huge anxieties. The arrangement would in view of that not have a unmistakable distinction between the bars with enormous zones and those with little zones. A rundown of the streamlining parameters utilized for these advancements is appeared in Table 2.

TABLE 2 SUMMARY OF RESULT FROM SIZE OPTIMIZATION OF TRUSS

	Initial Values	After 1st step	After 2nd step
Total volume (mm3)	2.43x10^6	1.56x10^6	1.51x10^6
Max stress (MPa)	70	53	100

In the size optimization, a truss is upgraded utilizing size advancement. Since measure improvement just upgrades sizes of discrete factors, and doesn't include add or remove elements, the truss to be upgraded must be characterized preceding the advancement.

SUMMRY

The optimization of a truss in this case indicates how a more powerful truss can be found, by the utilization of size streamlining. By utilizing the two stage technique, as done in this case, a truss is accomplished where the two stresses and deflection are controlled to be inside as far as possible. It is seen that the volume of the structure, and subsequently the amount of steel expected to manufacture it, can be greatly reduced.

The truss improved in this illustration is very basic, and a greater truss with more loads presents a few issues. The significant issue is that its not clear, which bars, and what number of them, to expel after stage 1. This is because of that the zones are more across the board inside the suitable range, so that there is no different distinction between the most supported bars and the less supported ones. This implies even despite the fact that the estimations and advancements principally are finished by the PC, the technique still required a talented architect to guarantee a decent quality outcome.

REFERENCE

1. ABAQUS user's manual, Version 6.11. (2011). Dassault Systemes Simulia Corp., Providence, RI, USA.

2. Dawson, M.I.(2008) "Modeling and Analysis Optimization Processes in Abaqus using the Virtual Fabrication Technology (VFT) Analysis Software developed by Battelle and Caterpillar Inc," 2008 Abaqus Users' Conference Proceedings, pp. 576-589.

3. Dean Deng, Hidekazu Murakawa, Wei Liang, "Numerical simulation Optimization and distortion in large structures", Computer. Methods Appl. Mech. Engrg. 196 (2007) 4613–4627.

4. Goldak, J. A., and Akhlaghi, M.(2005) "Computational Welding Mechanics Optimization," Springer Science, Business Media, Inc., New York, USA.

5. Memari, A. M. Wei Liang, "Numerical simulation Optimization and distortion in large structures", Computer. Methods Appl. Mech. Engrg. 196 (2007) 4613–4627.

6. Sahirman, S.(2005) "Computational Welding Mechanics Optimization," Springer Science, Business Media, Inc., New York, USA.

Lars Fuglsang Andersen, "Numerical simulation of welding distortions in large steel structures", Practical Design of Ships and Other Floating Structures.

7. Liam Gannon, Yi Liu, Neil Pegg, Malcolm Smith (2010) Effect of welding sequence on residual stress and distortion in flat-bar stiffened plate, Computers and Structures 89, 1859–1873.

8. Lindgren, L. E.(2007) "Computational Welding Mechanics – Thermomechanical and Microstructural Simulations", Journal of Materials Processing Technology 212, 1705–1715.

9. R. Chacon, E.Mirambell, E.Real (2009), "Influence of designer-assumed initial conditions on the numerical modeling of steel plate girders subjected to patch loading", Thin-Walled Structures 47, 391–402.

10. R. Chacon n, M.Serrat, E.Real (2012) "The influence of structural imperfections on the resistance of plate girders to patch loading ", Thin-Walled Structures 53, 15–25.

11. Shanmugham N. E. and Baskar K. (2003). "Steel- concrete composite plate girder subjected to shear loading." Journal of Structural Engineering ASCE, 129, 1230-1241.

12. T. Wanga,, O.S. Hopperstada, O.-G. Lademoa,b, P.K. Larsena (2007) "Finite element modeling of welded aluminum members subjected to four-point bending", Thin-Walled Structures 45, 307–320.

13. Memari, A. M. Wei Liang, "Numerical simulation Optimization and distortion in large structures", Computer. Methods Appl. Mech. Engrg. 196 (2007) 4613–4627.

14. Sahirman, S.(2005) "Computational Welding Mechanics Optimization," Springer Science, Business Media, Inc., New York, USA.

Lars Fuglsang Andersen, "Numerical simulation of welding distortions in large steel structures", Practical Design of Ships and Other Floating Structures.

15. Liam Gannon, Yi Liu, Neil Pegg, Malcolm Smith (2010) Effect of welding sequence on residual stress and distortion in flat-bar stiffened plate, Computers and Structures 89, 1859–1873.

16. Lindgren, L. E.(2007) “Computational Welding Mechanics – Thermomechanical and Microstructural Simulations”, Journal of Materials Processing Technology 212, 1705–1715.

17. R. Chacon, E.Mirambell, E.Real (2009), “Influence of designer-assumed initial conditions on the numerical modeling of steel plate girders subjected to patch loading”, Thin-Walled Structures 47, 391–402.

18. R. Chacon n, M.Serrat, E.Real (2012) “The influence of structural imperfections on the resistance of plate girders to patch loading “, Thin-Walled Structures 53, 15–25.

19. Shanmugham N. E. and Baskar K. (2003). “Steel- concrete composite plate girder subjected to shear loading.” Journal of Structural Engineering ASCE, 129, 1230-1241.

20. T. Wanga,, O.S. Hopperstada, O.-G. Lademoa,b, P.K. Larsena (2007) “Finite element modeling of welded aluminum members subjected to four-point bending”, Thin-Walled Structures 45, 307–320.

21. Liam Gannon, Yi Liu, Neil Pegg, Malcolm Smith (2010) Effect of welding sequence on residual stress and distortion in flat-bar stiffened plate, Computers and Structures 89, 1859–1873.