Report

On

Structural Topology Optimization of Steel Section using SIMP Technique

Submitted in partial fulfilment of the requirements of the degree of

Master of Technology

In

CIVIL ENGINEERING

(STRUCTURAL ENGINEERING)

By

ANKIT TOMAR

Department of Civil Engineering
IIT Roorkee
Roorkee-247667
May, 2019

DECLARATION

This is to certify that this submission represents my original work in my own words and where others' ideas or words have been included, I have adequately cited and referenced the original sources. I also declare that I have adhered to all principles of academic honesty and integrity and have not misrepresented or fabricated or falsified any idea in my submission. In case this undertaking is found incorrect, this M.Tech work may be withdrawn unconditionally.

Ankit Tomar
MTech (Structural Engineering)
Enrolment No- 17523004
Date:18/05/2019
Place: Roorkee

CERTIFICATE

This is to certify that the report entitled **Structural Topology Optimization of Steel Section using SIMP Technique** submitted by Ankit Tomar in partial fulfilment of the requirements of the degree of Master of Technology (Structural Engineering) to IIT Roorkee has been carried out under my supervision.

Date:18/05/2019

Place: IIT Roorkee

Dr. Rajib Chowdhury

Associate Professor,

Department of Civil Engineering

Indian Institute of Technology Roorkee

ACKNOWLEDGEMENT

I would like to warmly acknowledge the supervision, guidance and the continuous encouragement offered by my guide **Dr. Rajib Chowdhury**, Department of Civil Engineering, IIT Roorkee, and for helping me in the completion of this project. I sincerely thanks to all my friends and all who helped me in the completion of the project.

Date: 18/05/2019 Ankit Tomar

Place: Roorkee Enrolment No- 17523004

TABLE OF CONTENTS

LIST OF FIGURES AND TABLES

CHAPTER 1 INTRODUCTION

1.1 General

In the twentieth century, architect and designers have utilized inventive and new techniques to create effective and optimum types of structures. These innovative designers employed the techniques that give rise to effective, aesthetic and novel shape but they all have common restriction. Every technique used by these designers needed that within structures number of openings needed to be known prior to the structure obtained by these techniques. These techniques required the use of same type of model. But in Topology optimisation, this restriction does not restrict the formation of new structures which only required the design domain to be defined by the designer. Topology Optimisation also gives the advantage of optimising the openings in the structure and provides an innovative novel section in the study of novel structural shapes.

In 1988, Topology optimisation was introduced. It is a mathematical method to optimise the structures. Elements are selected in a FE mesh which represents the given design domain and then effectively optimises the material to the minimum as well as maximise the strength for the given process. The result obtained from topology optimisation show the similarity to the structures which are found in nature. Structures found in nature are generally structurally efficient and aesthetically pleasing.

1.2 What is Topology Optimisation?

In Structural Optimisation, a given objective is fulfilled utilising fixed quantity of resources to the maximum. In this, three categories exist, these are shape, size and topology optimisation.

In topology optimisation, optimum layout of design is determined by finding out the number of members required and the arrangement of these members in the structure. The location, shape, size and number of openings can also be found out using it. The new design is not restricted by the initial design of the structure in the topology optimisation but in shape and size optimisation it is restricted. The size of the structure varies in the size optimisation and shape of the structure remains the same and in the shape optimisation, the shape of the design space varies and size remains same. Topology optimisation differs from shape and size optimisation in a way that both size and shape vary in it.

Topology Optimisation helps in finding out the optimum distribution of the material for a given optimisation goal and constraints. A solid material of any given shape is removed from the structure to minimise or maximise objective function such as displacement, mass, strain energy and given set of constraints are satisfied like maximum displacement or stress.

In other words, it is a technique which uses mathematics to optimises the shape of material in a given domain for given boundary constraints and loads with an aim of increasing the effectiveness of the system to the maximum. The structure can take any shape and size within the given design space instead of tackling with predefined configurations.

1.3 Need of Topology Optimisation

High rise structures are required to solve the problem of overcrowding in modern cities and topology optimisation provides the solution. The requirement of the material in the high-rise buildings is much more than that of low-rise buildings due to the requirement of the bracing in taller buildings. It also gives structure designer a tool to meet the architectural aspirations in high rise construction.

Topology Optimisation has been successfully utilised in aerospace, automotive and mechanical engineering to achieve weight saving in structures. But in civil engineering, safety is more important than weight saving. But this cannot restrict the engineers using topology optimisation in the era where resilient and sustainable infrastructures are required and also where the redundancy concept plays an important role and every structure needs to be optimised.

1.4 Methods of Topology Optimisation

FEM is used in topology optimisation to evaluate a performance of the structure. The optimisation of the given space is done either using gradient-based mathematical programming techniques like method of moving asymptotes or optimality criteria algorithm or non-gradient based algorithms. There are many methods of Topology Optimisation such as Homogenisation method, Evolutionary Structural Optimisation (ESO), Bi-Directional Evolutionary Structural Optimisation (BESO), Solid Isotropic Material with Penalization algorithm (SIMP) or Power Law algorithm etc.

Earlier Homogenisation approach [1] was used in Topology Optimisation but it has some disadvantages. The evaluation and determination of microstructures and the orientations of these microstructures is difficult if not resolved and also it cannot be built as no length-scale of these microstructures is defined. Despite that homogenisation method is important in topology optimization in a way it bound the theoretical performance of the structure.

SIMP approach is also a micro approach like homogenisation method and is most widely accepted by the researchers in the papers. In this technique, material properties are taken same in every element and it is used to transform the design space in small elements and relative densities of these elements are taken as variables. The property of material is described as relative density of it raised to some power the properties of the solid material. The SIMP technique of topology optimization already applied to problems with more than one material, multiple physics and many constraints.

ESO is developed by Steven and Xie and the difference between ESO and SIMP is that it is macro approach. The removal of material takes place in ESO in the shape of finite element from underutilized design space. ESO is different from SIMP because in this Finite Element mesh is changed during the optimisation.

BESO approach is an improvement and extension of ESO method. It is a combination of additive evolutionary structural optimization (AESO) and ESO. It is an iterative method like others in which underutilised material is eliminated from the space and effective material is placed to it simultaneously.

1.5 Objectives

The objectives of this study are

- To understand the SIMP technique.
- To understand how topology optimization can be used to perforated steel beams.
- To determine the optimised shape of various models using SIMP technique.
- To do a parametric study of the Topology Optimisation

Various studies have been done to understand the concept of topology optimisation and finding out the method to determine the optimised shape of the beam. Some studies are explained below-

O. Sigmund [2] wrote a topology optimisation code in 99 line and performed it in MATLAB. In the code number of simplicities were added to make the MATLAB Code simple. SIMP technique and optimality criteria were used to optimise the beam. Firstly, the problem based on SIMP was introduced where aim was to minimize the strain energy of the beam and written as

$$\min x : c(x) = U^T K U = \sum_{e=1}^{N} (x_e)^p \, u_e^T k_0 u_e$$

$$\text{subject to} : \frac{V_{(x)}}{V_o} = \text{f}$$

$$: \ KU = F$$

$$: \ 0 < \ x_{min} < x < 1 \qquad (1)$$

where, U and F are the global displacement and force,

K is the global stiffness matrix,

and u_e and k_e are the element displacement and stiffness matrix

x is the design variable

x_{min} is minimum density

f is the volume fraction

p is penalisation power

and then it was solved using optimality criteria method. Optimality criteria is a heuristic updating scheme. During the solution, various check type patterns were obtained which were called as checkerboard pattern. These patterns were removed using mesh independency filter which worked by changing the element sensitivities. The optimised shape obtained by Sigmund is shown in fig 2.2.

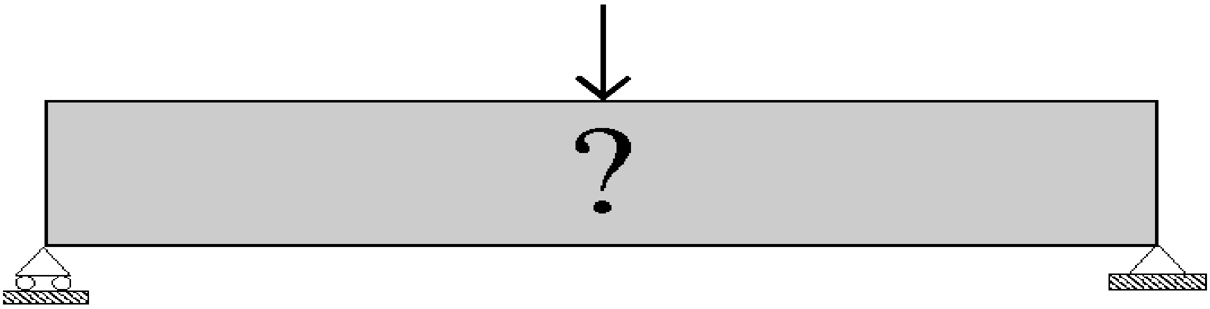


Fig. 2. 1 MBB-beam Full design domain (O. Sigmund, 2001)

Fig. 2. 2 Topology optimisation of the MBB-beam (O. Sigmund, 2001)

Fig. 2. 3 Checkerboard Pattern

A.Aremu et al [3] explained the usage of topology optimisation in Additive manufacturing. The SIMP and BESO algorithms are also explained in the paper. Additive manufacturing is a new approach to manufacturing wherein the formation of the component is done layer by layer which is usually taken from 3D CAD data. It requires less manufacturing constraints than the traditional manufacturing techniques like subtractive or formative (casting) techniques. It has the ability to build the components with intricate complexities and enable the production of optimal parts with better structural performance. Hence, topology optimisation can be adopted for Additive manufacturing by simplifying the constraints within these techniques.

Mello et al [4] carried out an analytical study on perforated steel beams and performed an experiment to verify and understand the behaviour of beams. Seven specimens were taken in which two were cellular beams and rest were perforated beams having new web openings. These tests were done to find out the mode of failure and web strength between two openings of web. The effect of change in space of web opening and in the depth of web-posts were observed to find out how action of strut took place in the buckling of web post. Then it was seen stability of web loaded with vertical load also depends on spacing and depth of web openings. This study tried to give maximum web opening area that could be possible for providing different services in the structure with keeping the minimum weight possible of the structure for different types of loading. These beams with new web openings find its application in long spans where light uniform load is applied.

It was observed in the paper that these novel shapes increase the effectiveness of beams when it was investigated in the failure mode of web-post due to buckling. In addition to this, the manufacturing process of these new openings of web have greater advantage than that of more popular cellular beams. It was also found out that the specimens fail under the action of both moment and shear and high deformation was there when post of webs were under high shear with big malformation in web-opening. The web post buckles under the loads which is slightly more than the deformation were observed first.

K.D Tsavdaridis et al [5] tried to attempt the alternative of the traditional cellular beams by finding out the new web opening I section using the method of Topology Optimisation method. To obtain optimum web opening, several different types of study were done in the paper. These are listed below

- Topology Optimisation was used to find out the optimised shaped of steel I-section.
- To conduct a parametric study to find an effective opening shape for different varieties of cross-sections.
- To understand the behaviour of simply supported beam which is under pressure load using non-linear FE analysis.

To design optimised steel, I-section, UB 305*165*40 beam was taken because they are most commonly used in practise and optimisation study was done. The FE steel model UB 305*165*40 beam was made to find out the response of the full model to use it in algorithm of topology. Shell elements was used to model the beam. The model was linear elastic material with Elasticity constant 200 GPa and Poisson's Ratio value is 0.3 were taken. Pressure which was uniform at all points was applied on top surface of upper flanged of UB. Supports were given at lower surface of flange and then FE analysis was done. In the model, stiffness of structure was subjected to restriction on available material was maximised. The compliance was minimised with volume constraint. The use of SIMP technique to optimise the beam. Then topology optimization was performed for the penalisation factor of 3 with volume fraction constraints of (a) 0.5 (b) 0.4 and (c) 0.3.

*Fig. 2. 4 Topology optimisation of 5m span UB 305*165*40 with different volume fraction constraints (K.D. Tsavdaridis et al, 2015)*

Results of this study in the UB 305*165*40 were represented in element density plots as shown in fig 2.4. The solid material was represented by red and the void or opening was represented by blue. The intermediate density is in transitional zone.

The observation made were that the complicated and irregular truss like shape was formed with more material to areas having high shear that is near support and no web material was present at the mid-section. The beam web elements follow path of principle stresses in the web.

As there was issue of manufacturability due to the complex geometry of the optimised beam, optimisation with symmetry constrained was performed. The symmetry is about the longitudinal axis and centreline that means in both vertical and horizontal direction and an optimised shape was obtained with volume fraction of 0.4. The result showed a rhomboidal opening with truss like design that changed across the length of beam. But the big opening towards the mid-span was still there showing that distribution of material in the web with respect to the shear and bending moment ratio at the section. The symmetry constrained topology optimised beam is shown in figure 2.5.

Fig. 2. 5 Symmetry constrained optimisation study on 5 m span UB 305 165*40 with volume fraction of 0.4 (K.D Tsavdaridis et al ,2015)*

The parametric study on variety of beam cross sections with the help of a short section was done so that every cross-section of beam were analysed and placed at any point across the length. A FE model was developed of a short section and bending moment and shear force was applied as shown in fig 2.6.

Fig. 2. 6 FE model and loading approach (K.D Tsavdaridis et al ,2015)

The parameters that were changed were flange-web thickness ratio and depth of web. In the study, it was seen that flange-web ratio did change resulting shape. The optimal topology changed with the depth of section as shown in fig 2.7. The optimal opening topology was similar in-depth ranges from 270 mm to 750 mm. In a beam of depth more than 750, extra openings were present to maintain balance between weight and stiffness. Based on the result, an optimum opening of web had been proposed as shown in fig 2.8. These were the studies performed in the paper.

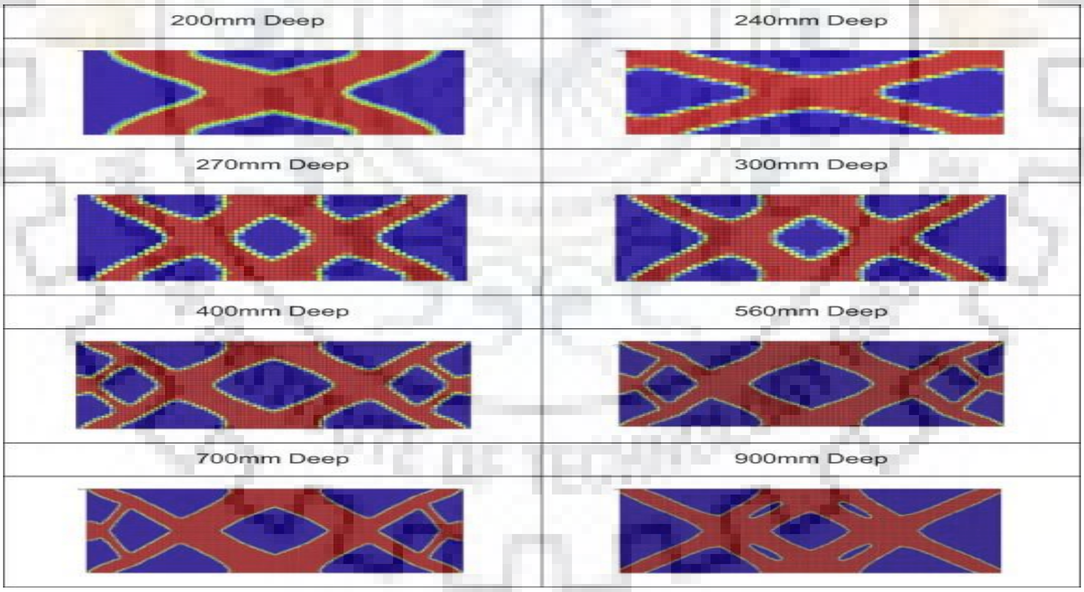


Fig. 2. 7 Results of parametric topology optimisation studies with change in beam depth (K.D Tsavdaridis et al, 2015)

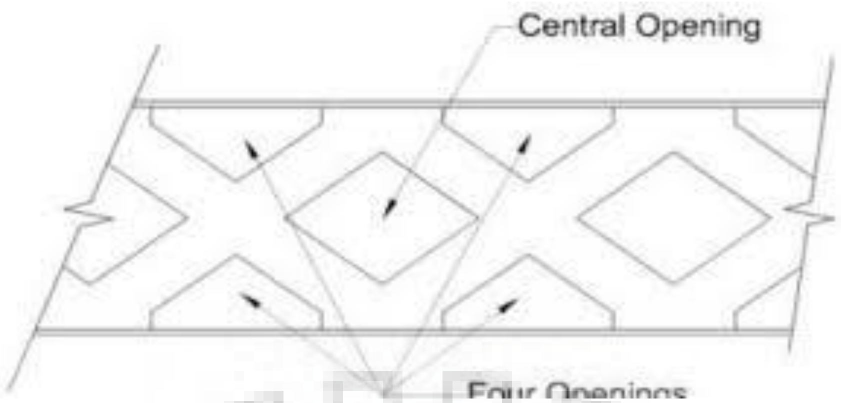


Fig. 2. 8 Suggested topologically optimum web opening (K.D Tsavdaridis et al ,2015)

G. Morkhade and M. Gupta [6] carried out an experimental investigation on the seven models of steel beams with web openings (SBWOs) to identify the maximum load behaviour and deflection of SBWOs in the web. Only vertical loading was considered in the experiment. The experiment was done on ISMB 100 with rectangular and circular openings. The beams were simply supported at ends. The analysis was done by the FEM and the results were verified with those obtained experimentally. One of the seven models shown in fig 2.9. Similarly, there were seven models were used. The work was predominantly experimental oriented and experiments were performed till failure.

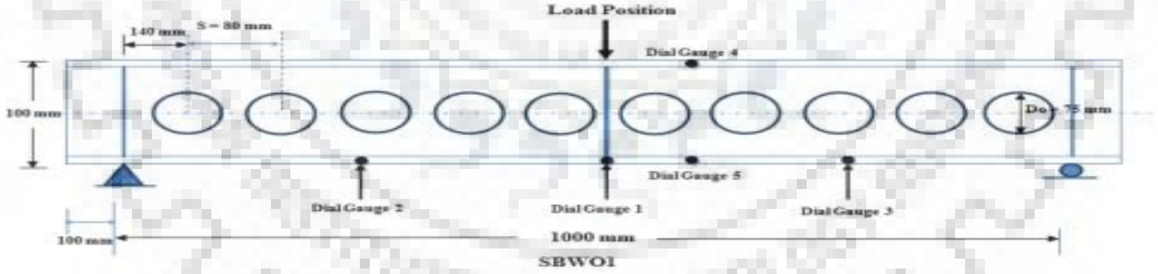


Fig. 2. 9 Beams specimen details (G. Morkhade et al, 2015)

The overall depth of the beams was 100 mm, width of flanges was taken 55 mm and its thickness was 5 mm, the thickness of web was 4.7 mm. The locations where concentrated loads were present, transverse stiffeners were provided, which is at the supports and the mid span. All the beams were initially laterally unrestrained beams. The Dial gauges were installed at L/4, L/2 and 3L/4 to measure vertical displacements and two more dial gauges were placed to measure the lateral deflection of both flanges.

Nonlinear FEA was carried out to find out the ultimate load carrying capacity for comparison. These were used to replicate the experimental work for the verification of the test results. A bilinear stress-strain curve was used.

It was found in the study that rectangular openings were very critical as it showed stress concentration in the corner regions of rectangular openings. Also, it was observed that with increase in opening area, ultimate load and stiffness of the steel beam increased.

Further parametric study was done in the paper to find out the performance of different web openings shapes. The beam of ISMB 400 of 4 m length was simply supported and concentrated load was applied on the top surface of flange.

The load vs mid-span deflection graph of the parametric study was shown in fig 2.10.

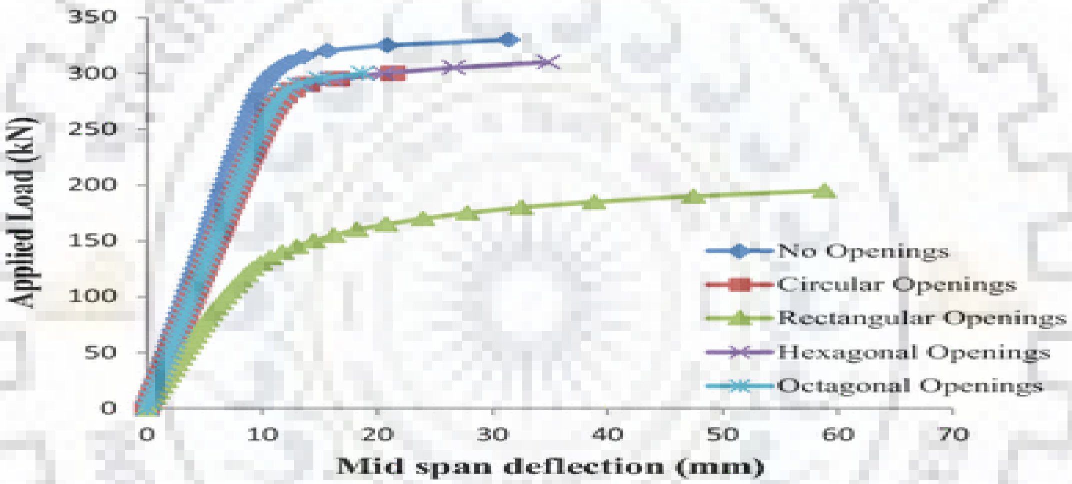


Fig. 2. 10 Load vs mid-span deflection graph (G. Morkhade et al, 2015)

It was found out that most effective opening is circular, it showed least concentration among the web opening while rectangular opening showed highest stress concentration in the corner of web openings.

Some common algorithms used for topology optimization are Homogenization algorithm, Solid Isotropic Material with Penalization algorithm (SIMP) or Power Law algorithm and Bi-directional Evolutionary Structural Optimization (BESO) algorithm. The most common of the methods is SIMP technique and the optimality criteria is used as an iterative method to converge the element densities. These methods are explained in detail below –

3.1 Solid Isotropic Material with Penalization Technique

It is also called as Power Law approach. The SIMP technique vary the density of the material within the given space to find out the optimal shape of the structure. The discretisation of the design domain is generally done and the behaviour of structure is determined by employing FE technique. The design space is converted in number of small pixels and picture of the optimal structure is produced by switching off and on each pixel. In it, Finite Element analysis of given domain is succeeded by optimisation of density of all the elements. Then domain with the new density is re-analysed and the optimisation is done again and again till the convergence is obtained.

It is attractive to build up the supposed 0-1 plan, where distribution of material inside the structure, is involved totally of either void or whole material. The answer of this 0-1 issue has been tried; in any case, it is commonly the situation that the utilization of such procedures is computationally restrictive because of the quantity of limited finite element components important to display the design space. The SIMP method tends to address this issue by characterizing the material inside every one of the limited components, as a nonstop plan variable. By changing over the structure variable into persistent from discrete type it is conceivable to utilize all the more proficient mathematical programming techniques for the answer of the first issue.

Intermediate density materials [7] are those which cannot take the value of fully-solid nor any opening. These are usually not required as it is impossible to make intermediate densities in the real-world structures. To remove these such in the final product, a penalisation is required to reduce advantage obtained from intermediate densities. The penalisation of the densities is done using SIMP, by introducing a relation the material stiffness to its density as

$$E = \rho^{P} \tag{2}$$

Where p is penalization power,

P is material density and

E is stiffness of the material

Two approaches to topology optimisation are there in SIMP techniques. These are explained below-

- Minimum Strain Energy Design – In this approach, the minimum of a particular performance is subjected and restriction on the resources which are available. Generally, strain energy of structure is taken as objective function and restriction is imposed on the volume of the given material.
- Minimize Weight Design – In this, the minimum weight of design space of structure is required along with restriction on particular measures is done.

It was observed that the minimum compliance (strain energy) method has found effective in identifying the conceptual structural design but this approach is criticised by few practitioners for not being able to enable any particular performance parameters like the stress is taken in optimisation process. The minimum weight approach contains specific constraints like maximum stress, buckling load, displacement etc are complicated to solve.

3.2 Optimality Criteria Method

The problem in topology optimisation contains large number of design variables. Hence iterative optimisation techniques are used to solve this problem. Optimality criteria is a heuristic scheme. After every iteration of the Optimality Criteria method, the design variables are updated.

Heuristic updating scheme for the design variable which formulated as

$$x_e^{new} \quad = \quad \max(x_{min}, x_e - m)$$

if $x_e B_e^{\eta} \leq \max(x_{min}, x_e - m)\,, x_e B_e^{\eta}$

if max $(x_{min}, x_e - m) <$ $x_e B_E^{\eta} < \min(1, x_e + m)$

if min $(1,1, x_e + m) \leq$ $x_e B_e^{\eta}$,

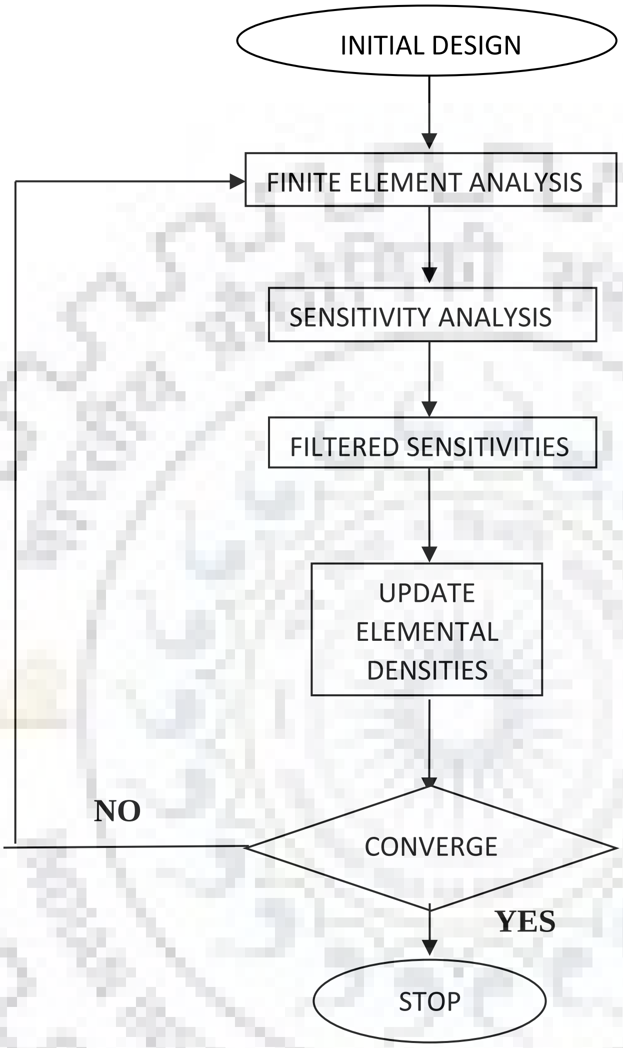
$$B_e = \frac{-\frac{\delta c}{\delta x_e}}{\lambda \frac{\delta V}{\delta x_e}} \qquad (3)$$

where λ is Lagrangian multiplier that can be obtained by a bi-sectioning algorithm. The method of Lagrange multiplier is a strategy for finding the local maxima and minima of a function subject to single constraints.

The sensitivity of the objective function is found by differentiating the compliance as

$$\frac{\delta c}{\delta x_e} = -p x_e^{p-1} u_e^T k_o u_e \qquad (4)$$

The step by step procedure of SIMP approach is shown in Flow Chart as follows –



3.3 FEM MODELLING

3.3.1 ABAQUS

Finite element model software ABAQUS [8] is used to developed the three-dimensional finite element modelling of beams for this study. This software package has five different core software products CAE, Standard, Explicit, CFD, and Electromagnetic. For modelling and obtaining the topology optimised shape of beams ABAQUS CAE is used. Standard version uses the implicit integration scheme to analyse the finite element model. Explicit version uses as per the name suggest explicit techniques of integration for analysis. CFD and Electromagnetic are used for the computational fluid dynamic purpose and computational electromagnetic problems respectively. To find out the topology optimised shape, standard or explicit model is used.

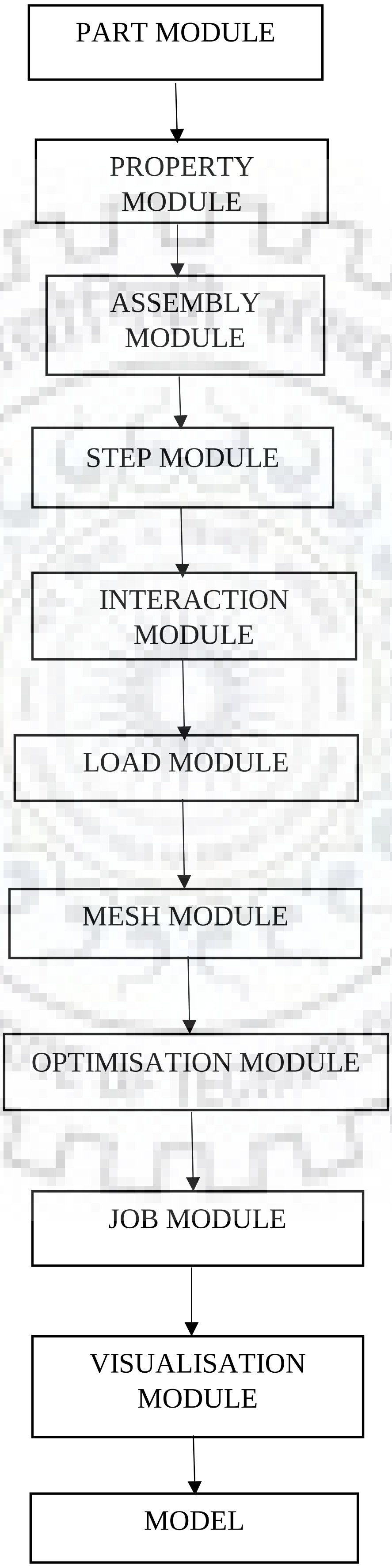
3.3.2 Modelling

A basic topology optimization process in Abaqus CAE involves 10 steps as shown:

1. Create/import part(s) in the Part Module
2. Assign material property to each part or part section(s) in the Property Module
3. Create instance and form an assembly (>= 1 part) in the in the Assembly Module
4. Define analysis type, steps and outputs in the Step Module
5. Add constraints/interactions between part-to-part and/or part-to-datum in the Interaction Module if needed
6. Add loading(s) and boundary condition(s) in the Load Module according to the model requirement
7. Generate FE mesh in the Mesh Module
8. Define optimization type and parameters in the Optimization Module
9. Create/manage optimization job(s) in the Job Module
10. View/save results in the Visualization Module

Hence model can be obtained by following these steps.

The following steps are shown in flow chart below -

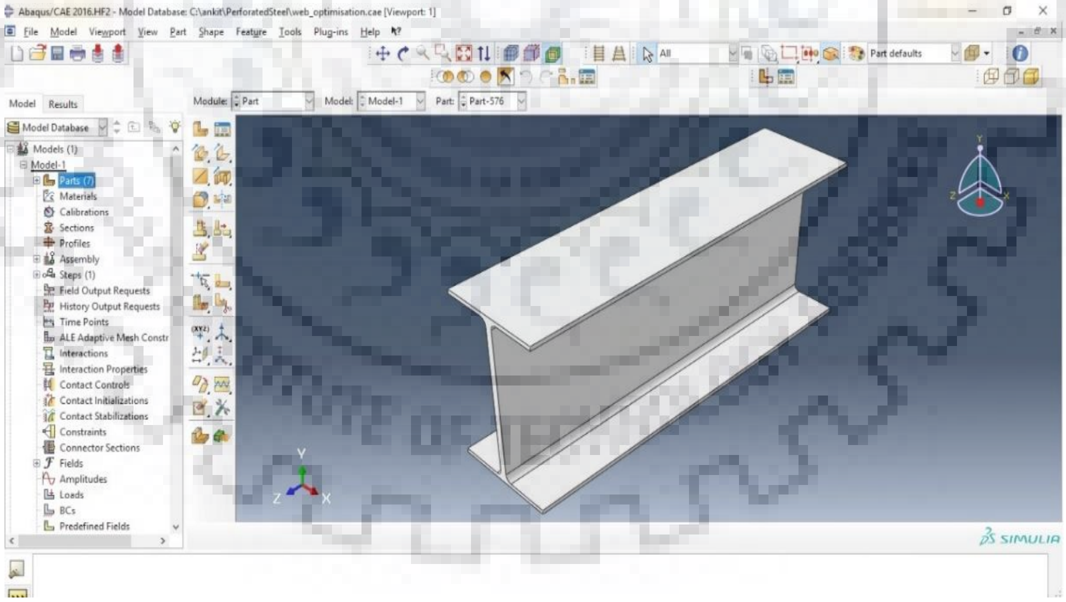


To understand the procedure of getting topology optimised shape of the model, several models are prepared and topology optimisation is done of these models. Three different models are prepared. Steel is the material used and the material properties are defined as given in table.

Table 1 Material Properties of models

Steel density	7850 kg/m^3
Modulus of Elasticity	200 GPa
Poisons Ratio	0.3
Support condition	Fixed

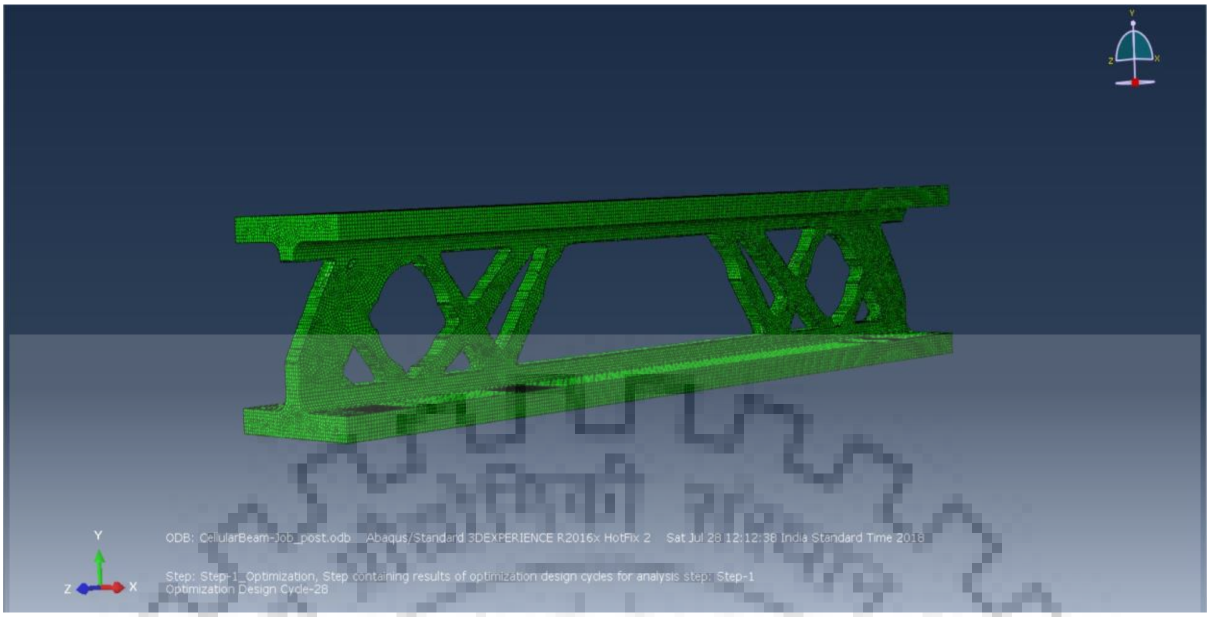
First is I-beam which is fixed at the end at the bottom side of the flange and uniform pressure loading is applied on the beam. Then the meshing of the model is done and optimisation module is defined for the model. Then the analysis is run and the optimised shape is obtained. The model of the I-beam and optimised shape can be seen in figures below.



Abaqus/CAE 2016.HF2 - Model Database: C:\ankit\PerforatedSteel\web_optimisation.cae [Viewport: 1]

Module: Part   Model: Model-1   Part: Part-576

Fig 3. 1 I-Beam model

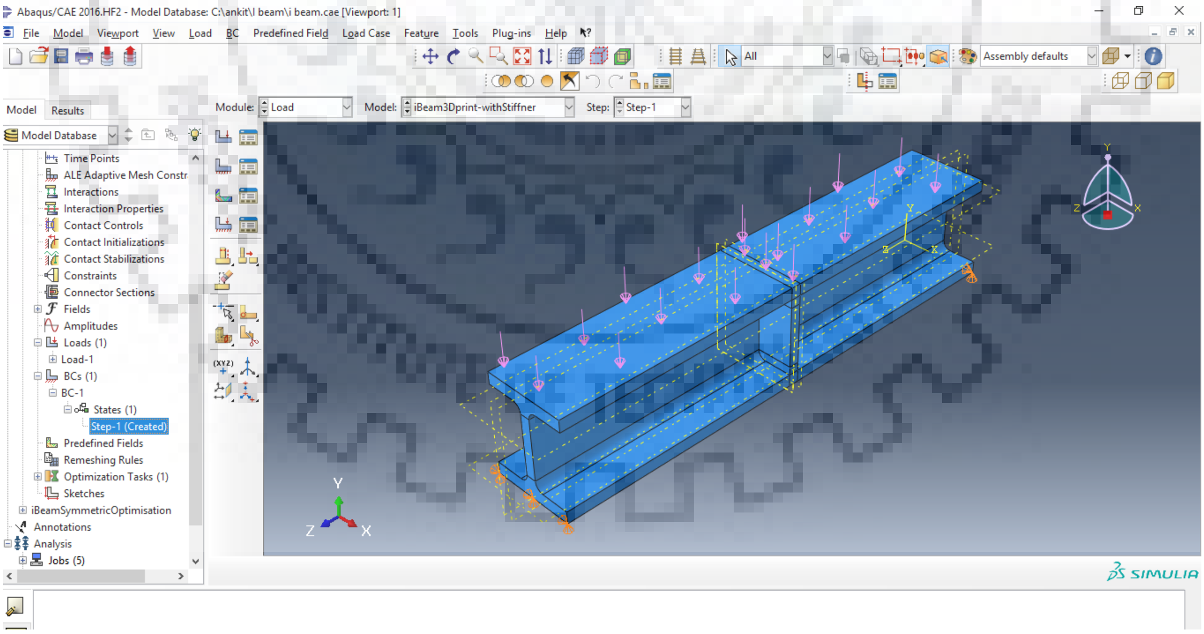


ODB: CellularBeam-Job_post.odb Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2 Sat Jul 28 12:12:38 India Standard Time 2018

Step: Step-1_Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-28

Fig 3. 2 Optimised shape of I-Beam

Second model is steel I-beam with stiffener. The beam is again loaded with uniform pressure and fixed at the end and the bottom of the flange. The same procedure of topology optimisation is performed and the optimised shape is obtained. These can be seen in figures below.



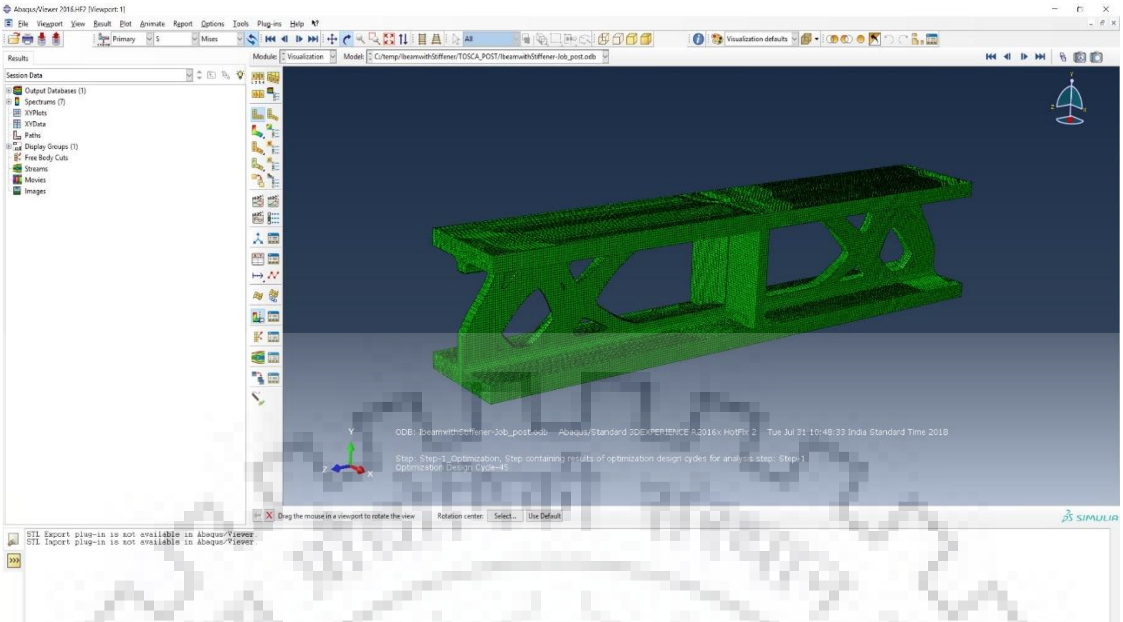
Abaqus/CAE 2016.HF2 - Model Database: C:\ankit\I beam\i beam.cae [Viewport: 1]

Module: Load

Model: iBeam3Dprint-withStiffner

Step: Step-1

Fig 3. 3 I-Beam with Stiffener



ODB: IbeamwithStiffener-Job_post.odb Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2 Tue Jul 31 10:48:33 India Standard Time 2018

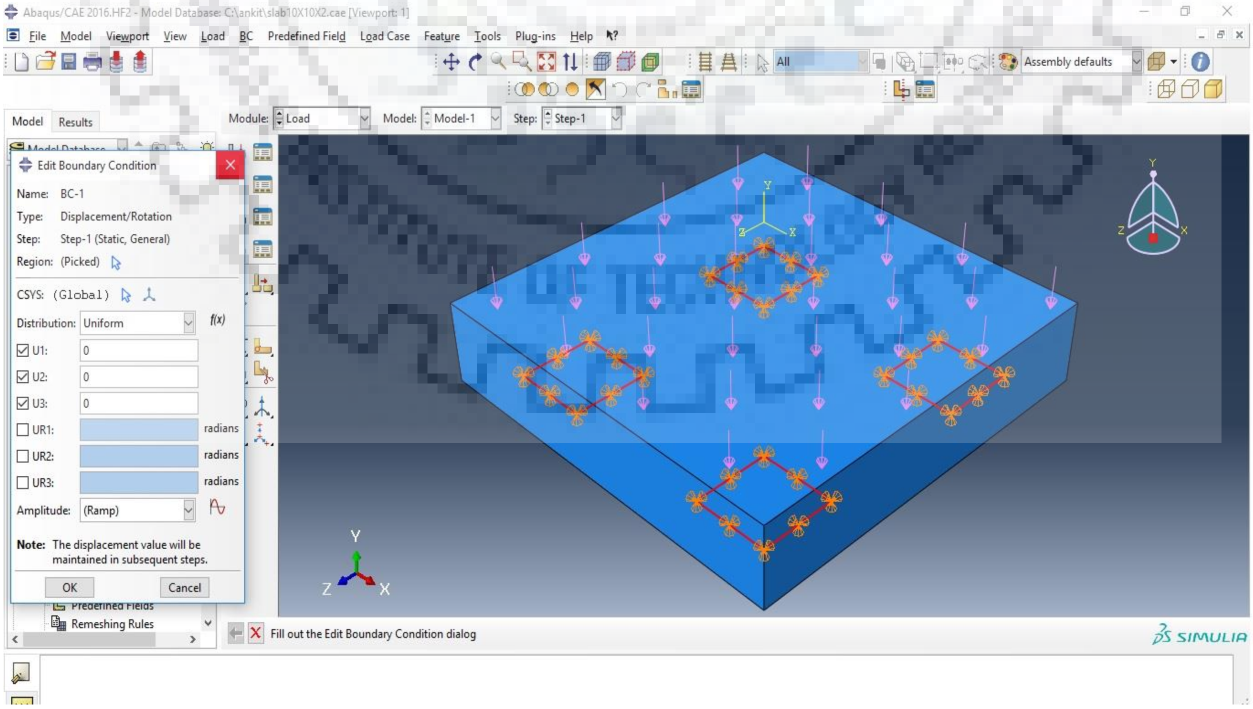
Step: Step-1_Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-45

Drag the mouse in a viewport to rotate the view Rotation center: Select... Use Default

```
STL Export plug-in is not available in Abaqus/Viewer.
STL Import plug-in is not available in Abaqus/Viewer.
```

Fig 3. 4 Optimised shape of I-Beam with stiffeners

The last model amongst the three is the slab. In this case, two volume fraction are used. It means that there are two optimised shape with different volume constraint. These values are 0.2 and 0.3. The slab is made up of steel and fixed at the bottom at certain distance from the corners. Care is taken that all the supports are at equidistant from the centre and symmetric.



Edit Boundary Condition

Name: BC-1
Type: Displacement/Rotation
Step: Step-1 (Static, General)
Region: (Picked)
CSYS: (Global)
Distribution: Uniform
☑ U1: 0
☑ U2: 0
☑ U3: 0
☐ UR1: radians
☐ UR2: radians
☐ UR3: radians
Amplitude: (Ramp)

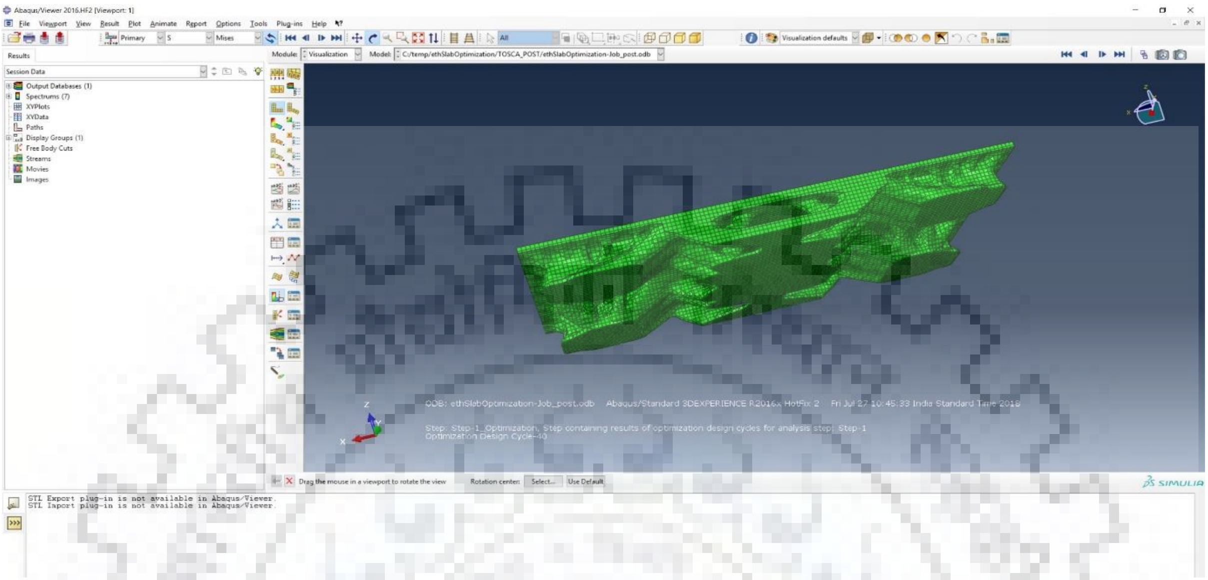
Note: The displacement value will be maintained in subsequent steps.

OK Cancel

Fill out the Edit Boundary Condition dialog

Fig 3. 5 Slab model

The model of slab and its optimised shape with two volume constraint are shown in the fig 3.5 , fig 3.6 and fig 3.7 respectively.



Module: Visualization Model: C:/temp/ethSlabOptimization/TOSCA_POST/ethSlabOptimization-Job_post.odb

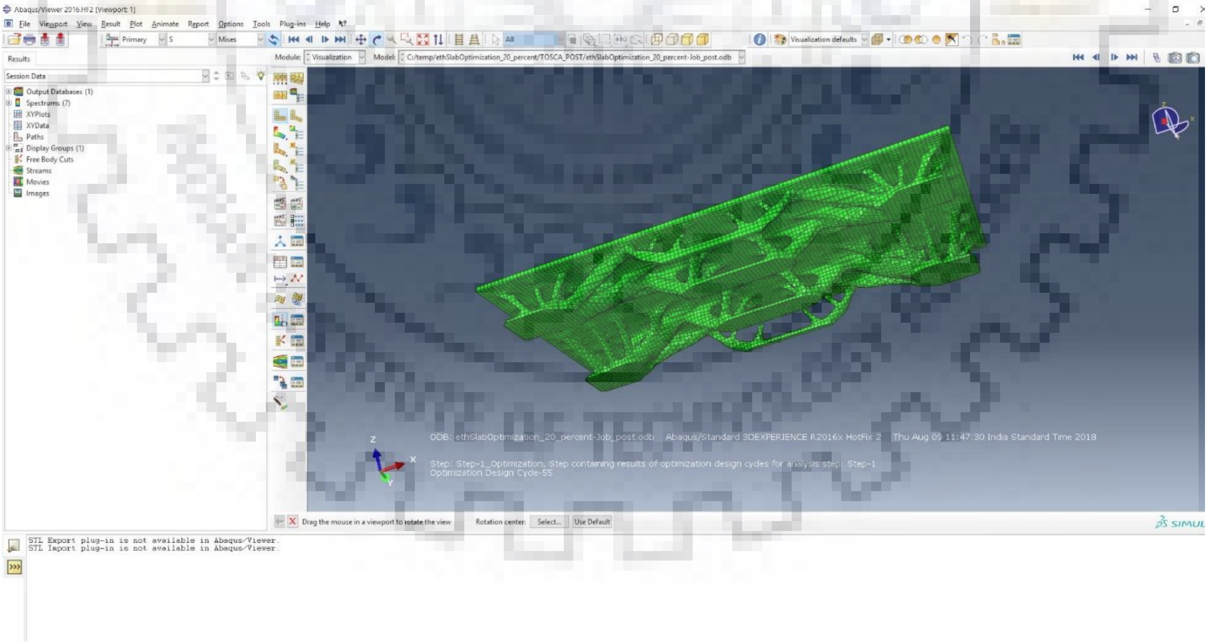
ODB: ethSlabOptimization-Job_post.odb Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2 Fri Jul 27 10:45:33 India Standard Time 2018

Step: Step-1_Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-40

Drag the mouse in a viewport to rotate the view Rotation center: Select... Use Default

```
STL Export plug-in is not available in Abaqus/Viewer.
STL Import plug-in is not available in Abaqus/Viewer.
```

Fig 3. 6 Optimised shape of slab (volume fraction = 0.3)



Module: Visualization Model: C:/temp/ethSlabOptimization_20_percent/TOSCA_POST/ethSlabOptimization_20_percent-Job_post.odb

ODB: ethSlabOptimization_20_percent-Job_post.odb Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2 Thu Aug 09 11:47:30 India Standard Time 2018

Step: Step-1_Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-55

Drag the mouse in a viewport to rotate the view Rotation center: Select... Use Default

```
STL Export plug-in is not available in Abaqus/Viewer.
STL Import plug-in is not available in Abaqus/Viewer.
```

Fig 3. 7 Optimised shape of slab (volume fraction = 0.2)

4.1 General

As the population is increasing day by day, need of high-rise structures are necessary but due to high cost these projects are not increasing as rapidly as needed. To solve this problem, topology optimisation is good tool. But how to optimised any structure. This can be done using ABAQUS CAE software. The optimised shape is light weight for long span and material used is less than ordinary beam. Hence, it is economical. This study is done to get better understanding of Topology Optimised technique. Firstly, the optimised shape of a beam is verified from the research paper. Then several parametric studies are done to understand the parameters on which optimised shape depends. These parameters are depth of beam, support condition, type of loading, web-flange ratio, mesh size and volume fraction. The effect of these parameters is then discussed.

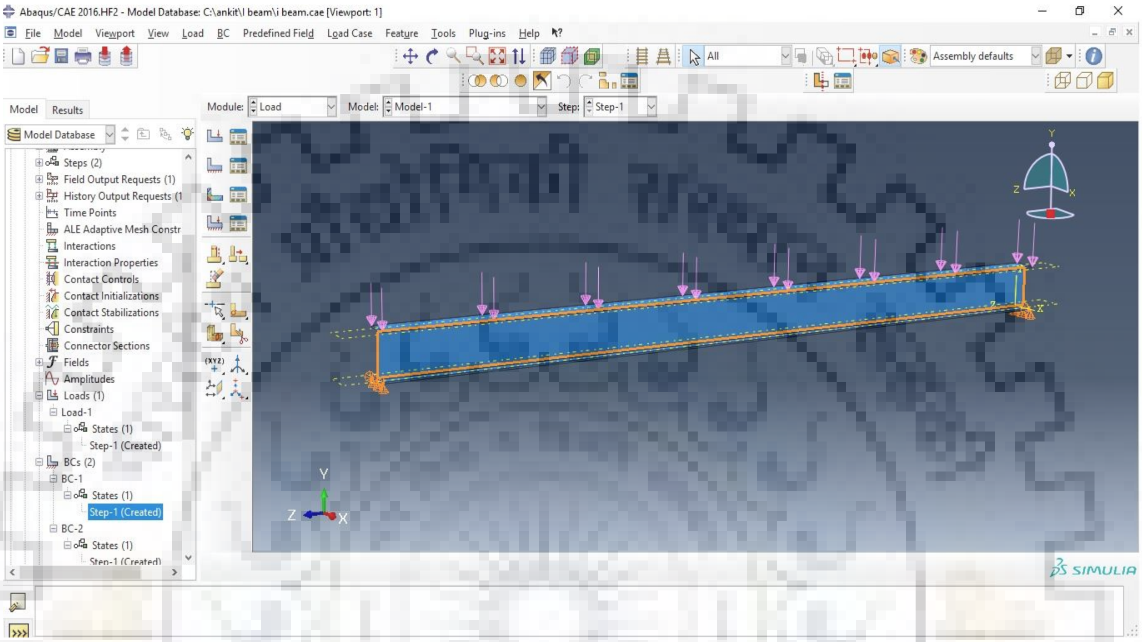
4.2 Validation K.D Tsavdaridis et al (2015)

The optimised shape of the UB 305*165*40 beam of span 5 m is used to verify the topology optimisation in ABAQUS. The sectional properties of the beam are listed in table. The model is prepared in abaqus with the same material properties. Pressure was uniform on the top surface of upper flange. The supports were provided at the surface of lower flange of the beam. The analysis carried out by Tsavdaridis et al was in ANSYS. The optimised shape of beam given in the paper and as obtained by the analysis in abaqus are shown in the fig 4.2 and fig 4.3.

Table 2 Material Property of I-beam of span 5m

Steel density	7850 kg/m^3
Modulus of Elasticity	200 GPa
Poisons Ratio	0.3
Support condition	Fixed
Mesh Size	10mm
Volume Fraction	0.4

It can be seen from the fig that the optimised shape of the beam obtained in ABAQUS is almost the same as obtained by K.D Tsavdaridis (2015) in ANSYS. As suggested in the paper, more material is present at the supports and no material is present at the centre. This implies that material is distributed according to the shear-moment ratio. As we know flanges resisted 80-90 % moment and web resist approximately 85 % shear, this result is the reason that we obtained this type of shapes. The shape obtained is a complicated and irregular truss like structure.



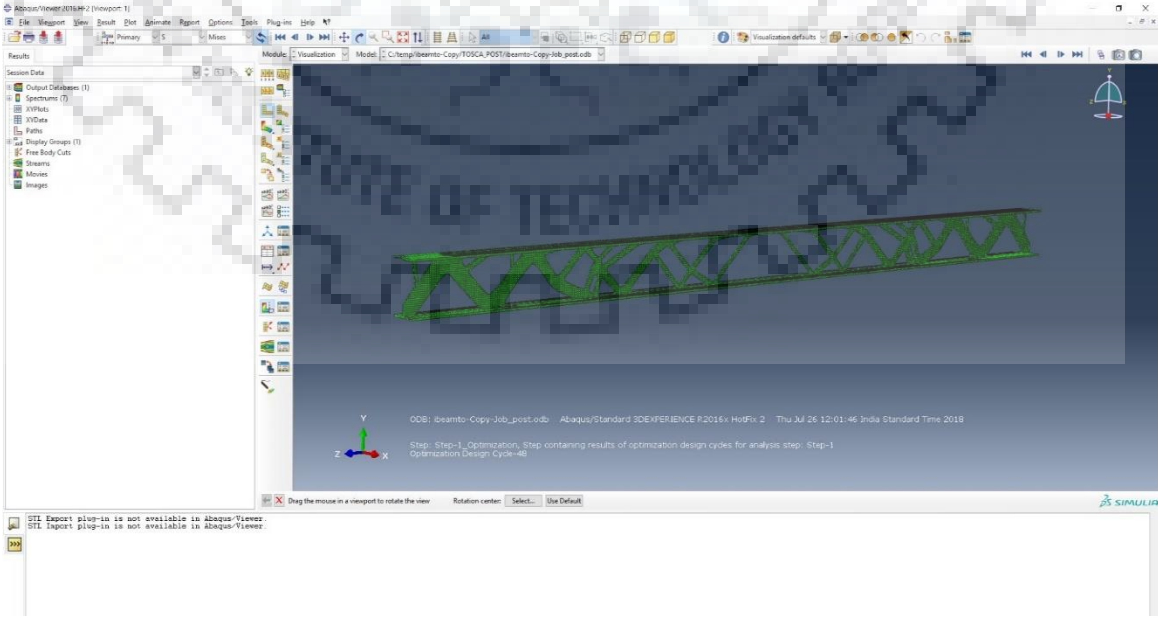
Abaqus/CAE 2016.HF2 - Model Database: C:\ankit\I beam\i beam.cae [Viewport: 1]

Module: Load

Model: Model-1

Step: Step-1

Fig 4. 1 I-Beam model with loading and support condition



Abaqus/Viewer 2016.HF2 [Viewport: 1]

Module: Visualization

Model: C:/temp/ibeamto-Copy/TOSCA_POST/ibeamto-Copy-Job_post.odb

ODB: ibeamto-Copy-Job_post.odb Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2 Thu Jul 26 12:01:46 India Standard Time 2018

Step: Step-1_Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-48

Drag the mouse in a viewport to rotate the view

STL Export plug-in is not available in Abaqus/Viewer
STL Import plug-in is not available in Abaqus/Viewer

Fig 4. 2 Optimised shape of I-Beam having length of 5 m

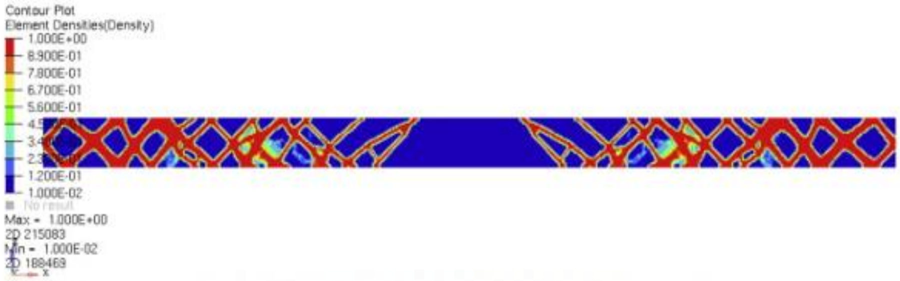


*Fig 4. 3 Topology optimisation studies on 5m span UB 305*165*40 with different volume fraction constraints is 0.4 (K.D. Tsavdaridis et al, 2015)*

Therefore, the optimised shape obtained in the analysis is verified.

4.3 Parametric Study

During the optimisation, there are various parameters that needs to be filled before starting the analysis. These parameters are depth, web-flange ratio, support condition, type of loading, mesh size and volume fraction. The dependency of optimised shape on these parameters is found out in this section. These parameters may or may not affect the optimised shape of I-beam.

4.3.1 Effect of Depth

In this section, the depth of the section is changed while keeping the length-depth, depth-width and length-width ratio same. The other parameters as discussed above also kept constant. Six models are prepared and then optimisation is done. The change in shape and size of the optimised web is observed. During the analysis optimisation of the web is done, flanges are not optimised due to the application of load on upper flange and support constraints on the lower flange. The detail of the models is given in table 3. Uniform pressure is applied on the top surface and the end of the lower flange is simply supported.

Table 3 Section size with variation to section depth

Depth (mm)	Length (mm)	Breadth (mm)	Flange Thickness (mm)	Web Thickness (mm)
300	720	187.5	10	8
350	840	218.75	10	8
450	1080	281.25	10	8
550	1320	343.75	10	8
700	1680	437.5	10	8
900	2160	562.5	10	8

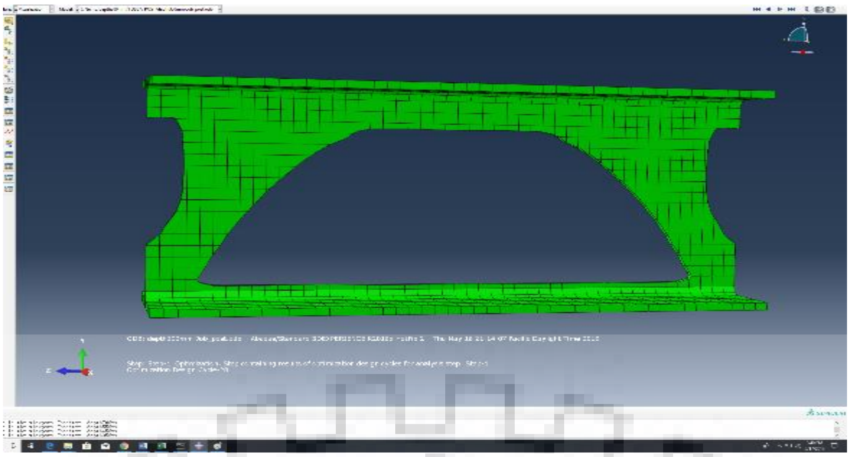


Fig 4. 4 Topology optimised beam of 300 mm depth

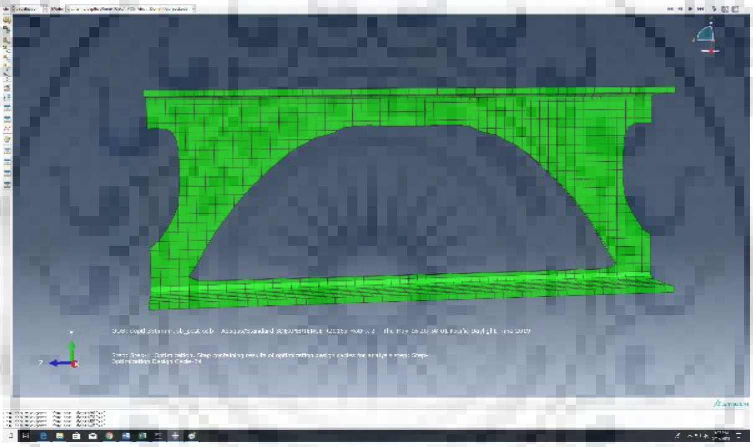


Fig 4. 5 Topology optimised beam of 350 mm depth

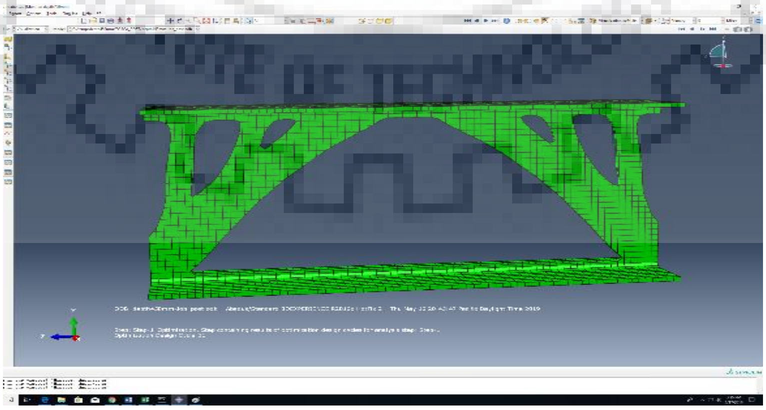


Fig 4. 6 Topology optimised beam of 450 mm depth

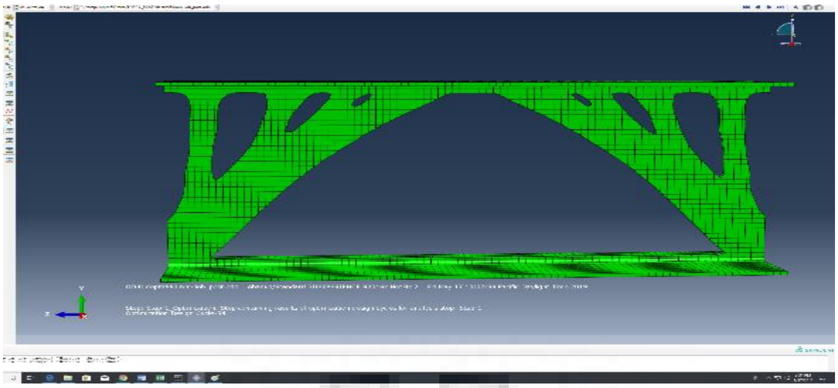


Fig 4. 7 Topology optimised beam of 550 mm depth

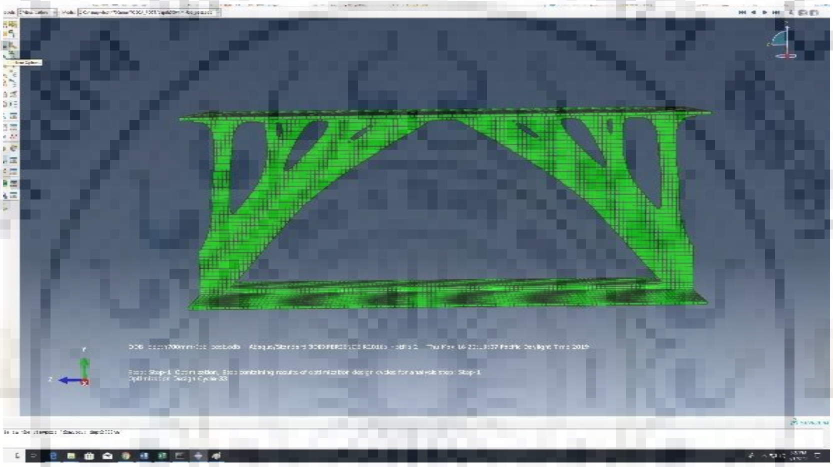


Fig 4. 8 Topology optimised beam of 700 mm depth

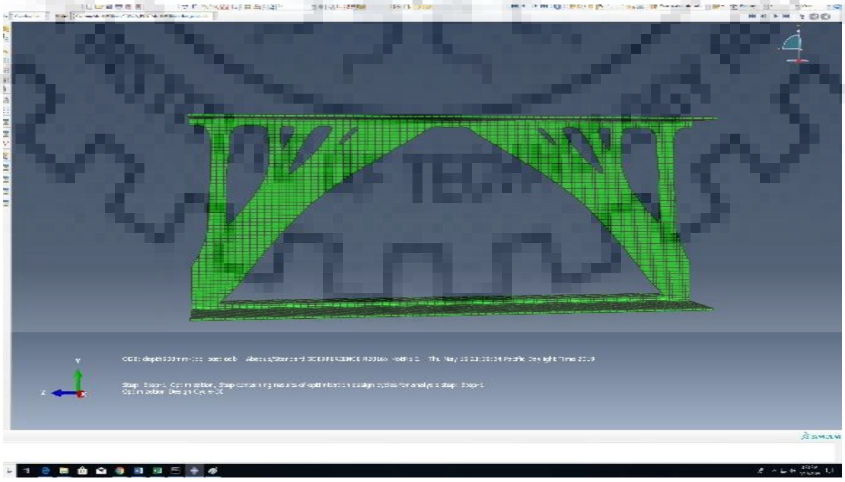


Fig 4. 9 Topology optimised beam of 900 mm depth

As it can be seen from the figures, that the shape of the web at the centre is almost the same in all the section with varying depth. But the number of openings is increasing as the depth is increasing. It is due to the fact that the flange and web thickness is constant in the models and also loading and support condition are same, so to keep the balance between the weight and the stiffness of the section more openings are required. Hence as the depth is increasing, no of openings also increases with same mesh size and volume fraction.

4.3.2 Web-Flange Thickness ratio

Another parametric study was conducted in which the web-thickness ratio was changed while other parameters kept same. The ratio of web and thickness is shown in table. Uniform pressure loading and support condition were applied on the model. The mesh size and the volume fraction also kept same for all the ratio. Five models were made and topology optimisation of the web was performed and the results were obtained in the form of optimised shape of the web. The optimised shapes are shown in the fig 4.10 to fig 4.14.

Table 4 Section size with variation to web-flange ratio

Length (mm)	Depth (mm)	Width (mm)	Web Thickness (mm)	Flange Thickness (mm)	Web-Flange Ratio
1500	350	200	8	5	0.625
1500	350	200	8	8	1
1500	350	200	8	16	2
1500	350	200	8	24	3
1500	350	200	8	32	4

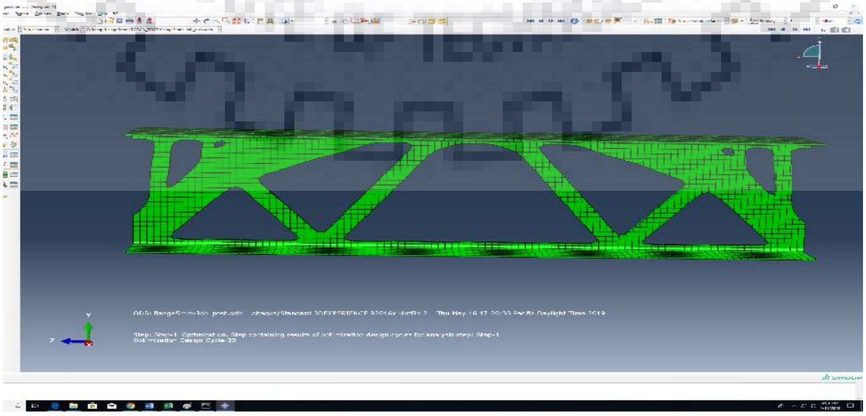


Fig 4. 10 Topology optimised beam of flange-depth ratio is 0.625

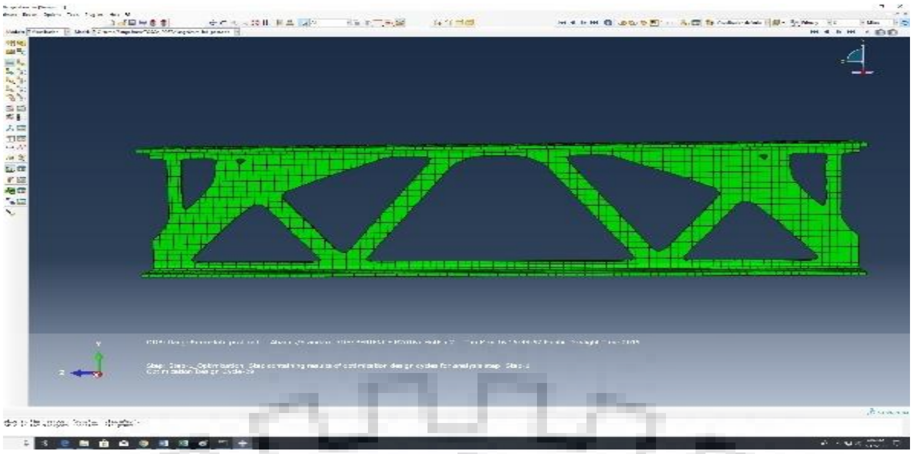


Fig 4. 11 Topology optimised beam of flange-depth ratio is 1

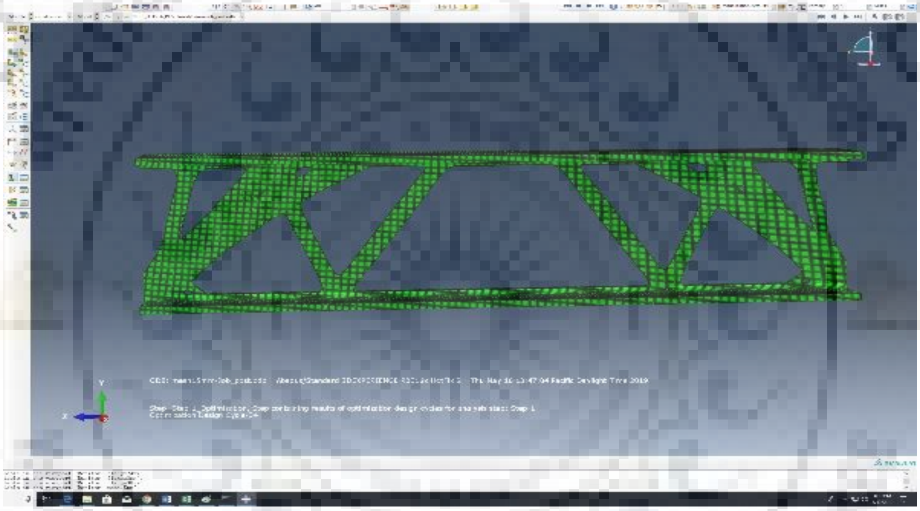


Fig 4. 12 Topology optimised beam of flange-depth ratio is 2

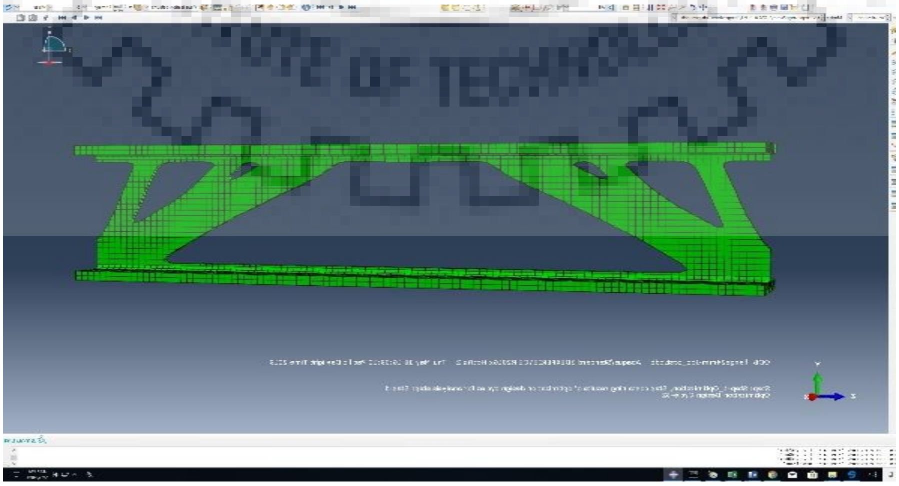


Fig 4. 13 Topology optimised beam of flange-depth ratio is 3

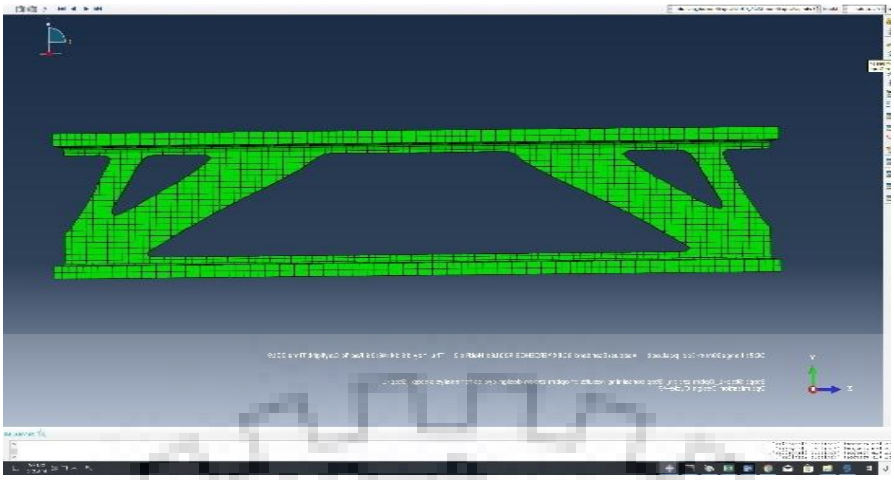


Fig 4. 14 Topology optimised beam of flange-depth ratio is 4

It was concluded from the figures that with increase in web-flange ratio, the shape of the web also changed. Also, the central opening constantly increased. The number of openings also changed; it decreases as the ratio increased. It was also seen that as the ratio was increasing, the truss like structure thickness was decreasing until the ratio becomes 2. Also, the shape of web was almost same till ratio was equal to two. At ratio three and four, shape was very different from the other ratio. This was due to the fact that as the flange thickness increased, the shear needed to resist by the web decreased, so at the centre, size of opening increased. Moreover, number of openings was decreasing to keep the balance between stiffness and weight.

4.3.3 Support Condition

As we know that due to the change in the support, the shear force diagram and the bending moment diagram changes. Thus, the shape of the web opening should change. To verify this prediction, parametric study was performed in which ISWB 350 was loaded with uniform pressure with different support conditions. These boundary conditions were simple support, cantilever and overhang. All the other parameters were same that is mesh size, volume fraction, web-flange ratio and the section size. The topology optimisation was performed on these models with the help of abaqus. The different models and their optimised shape are shown in the fig 4.15 to fig 4.20.

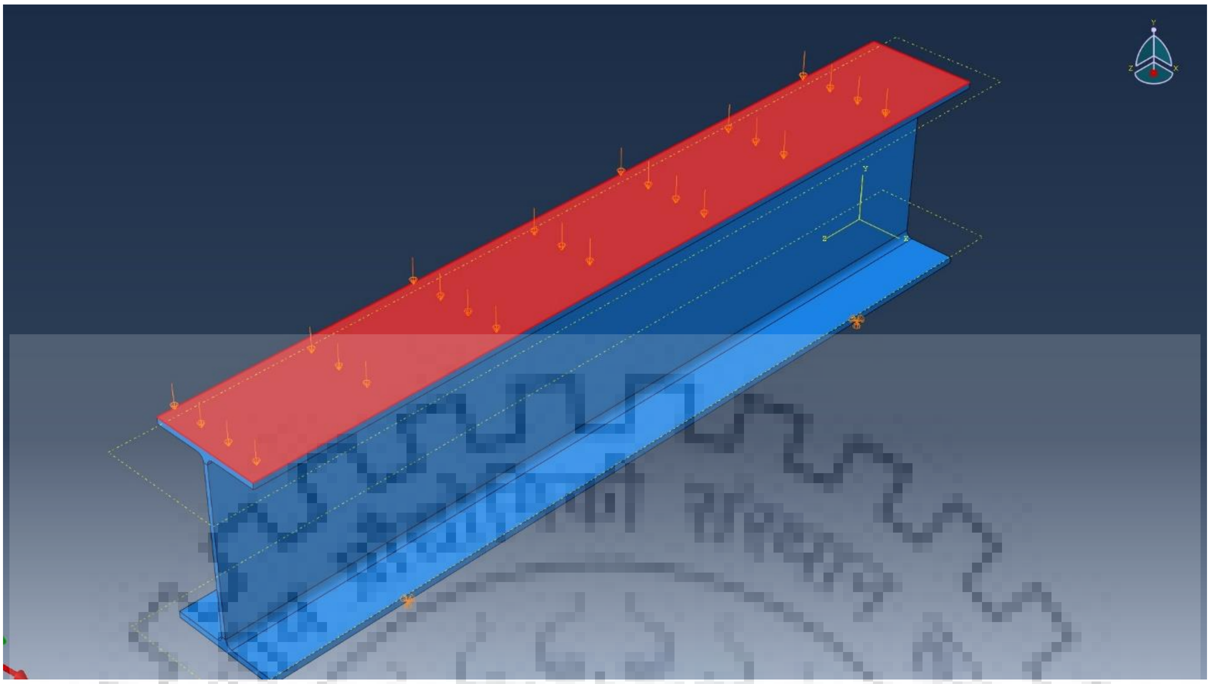


Fig 4. 15 Model of I-beam with overhang support

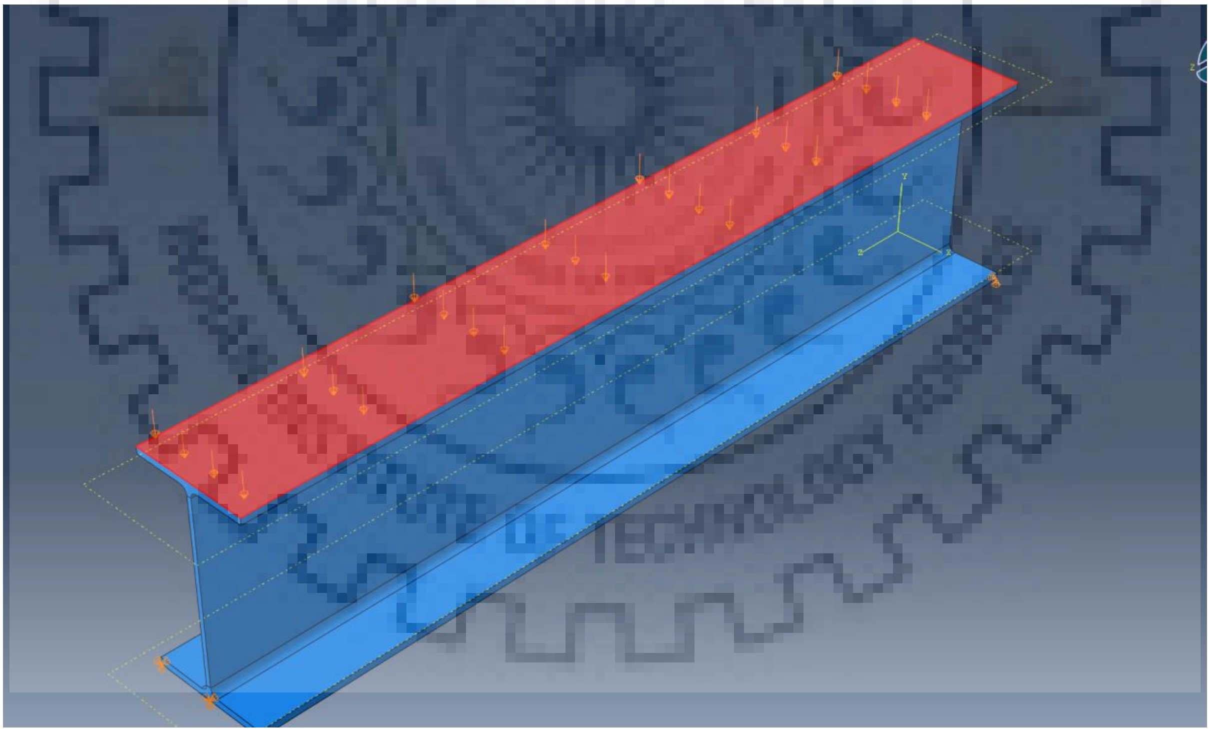


Fig 4. 16 Model of I-beam with simple support

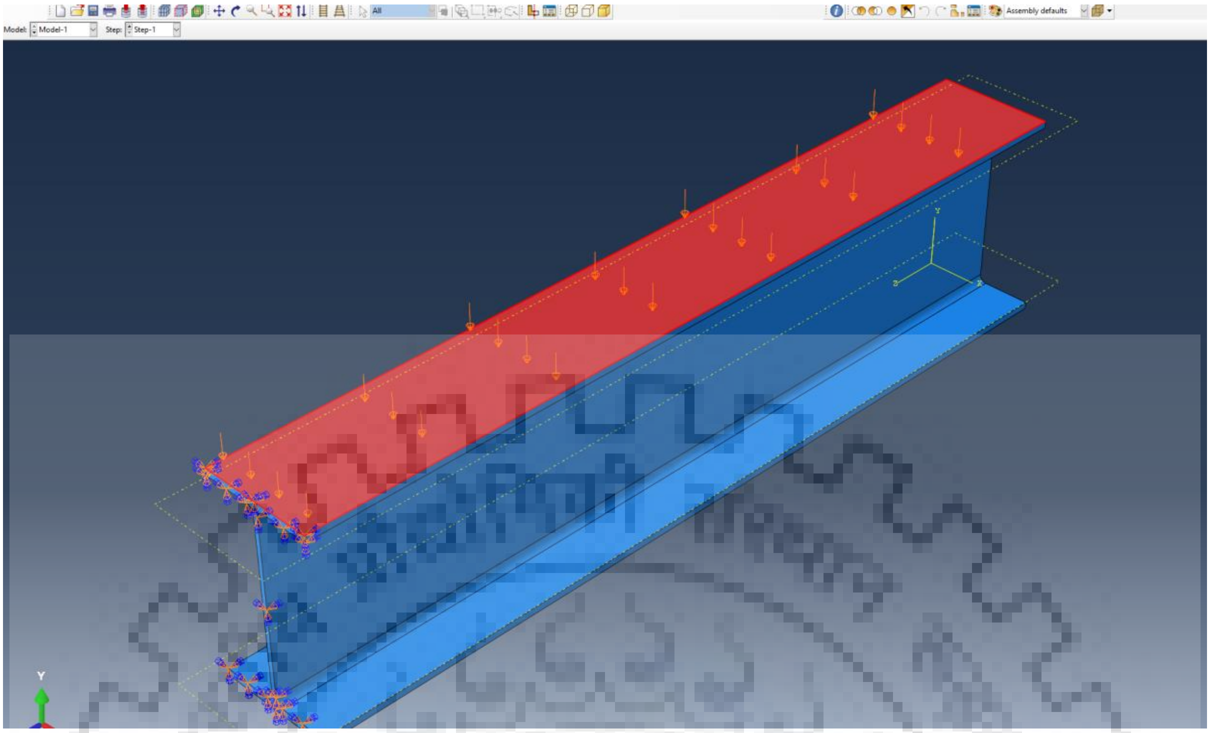
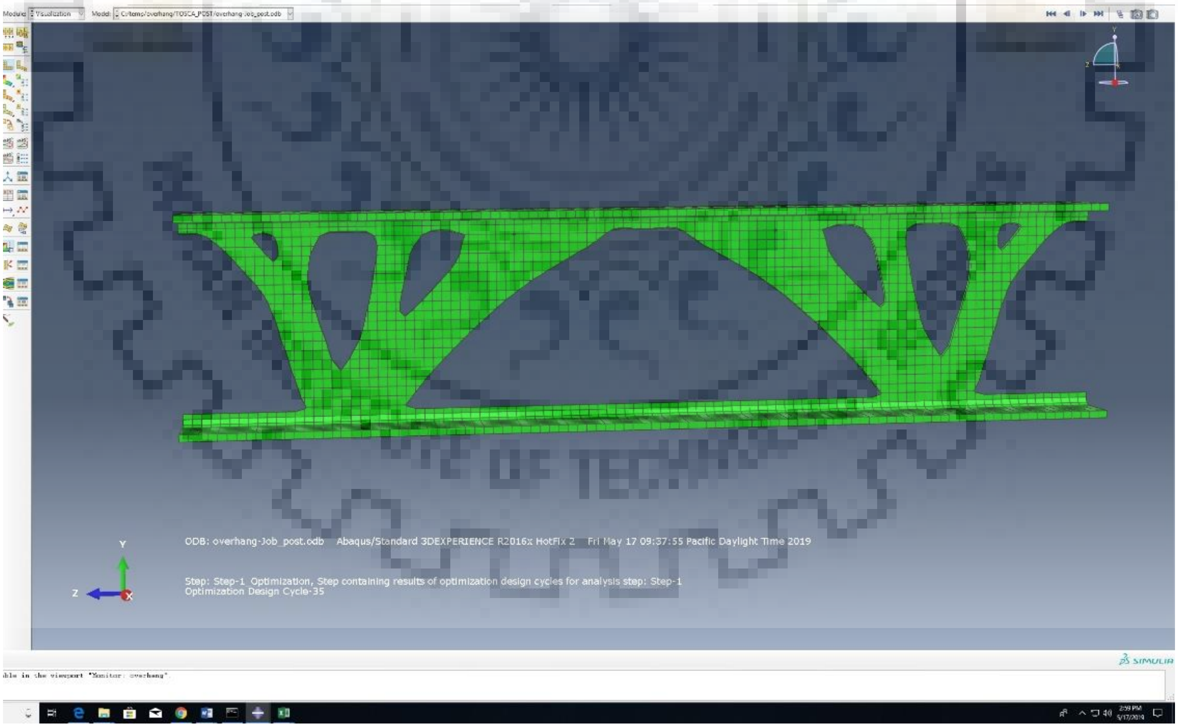


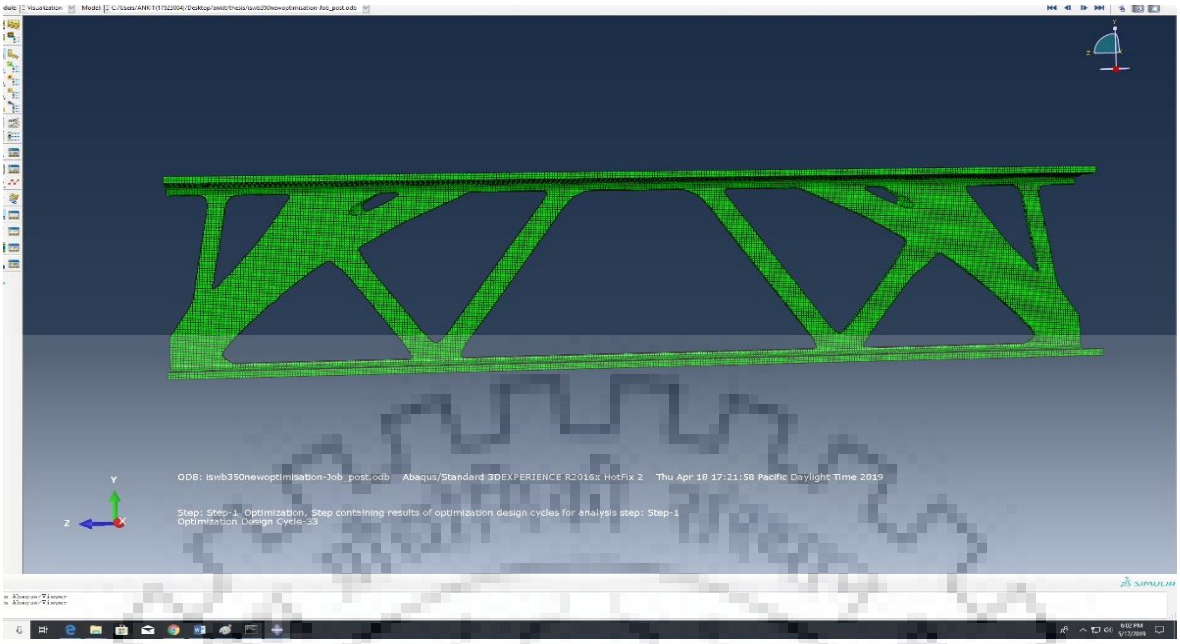
Fig 4. 17 Model of I-beam cantilever support



ODB: overhang-Job_post.odb Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2 Fri May 17 09:37:55 Pacific Daylight Time 2019

Step: Step-1_Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-35

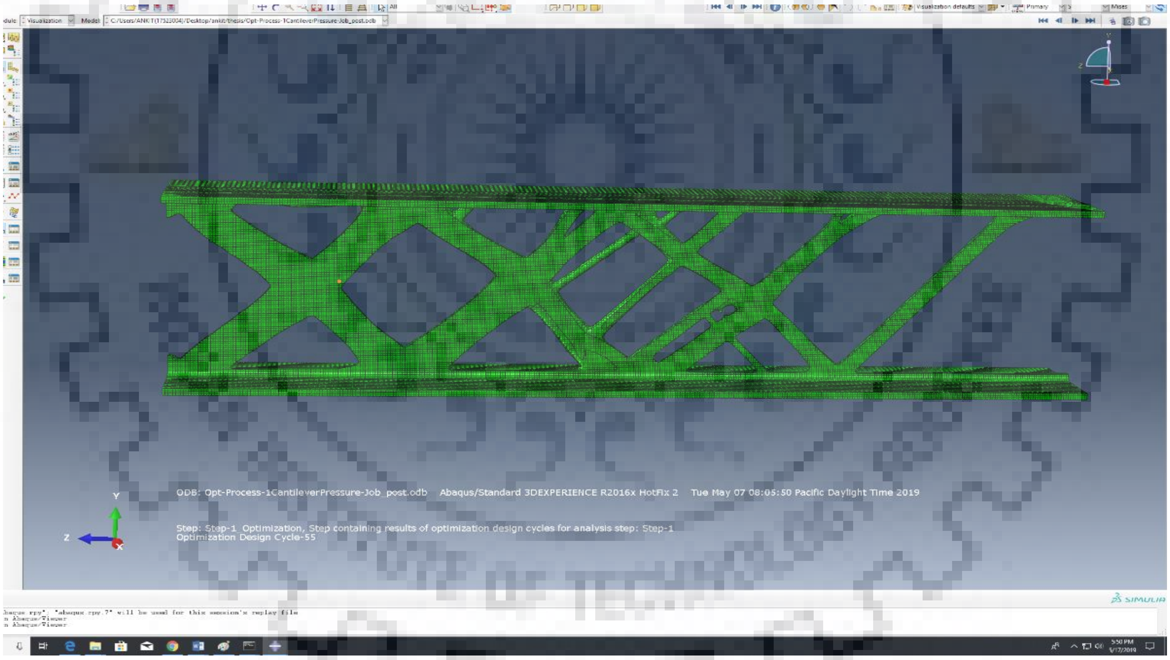
Fig 4. 18 Topology optimised I-beam with overhang support



```
ODB: iswb350newoptimisation-Job_post.odb    Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2    Thu Apr 18 17:21:58 Pacific Daylight Time 2019

Step: Step-1_Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-33
```

Fig 4. 19 Topology optimised I-beam with simple support



```
ODB: Opt-Process-1CantileverPressure-Job_post.odb    Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2    Tue May 07 08:05:50 Pacific Daylight Time 2019

Step: Step-1_Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-55
```

Fig 4. 20 Topology optimised I-beam with cantilever support

It can be seen that all the optimised shapes are different from one another as it was predicted. The pattern was really interesting in all the cases.

In cantilever beam, the material was absent near the free end which can be seen on the right side. As the shear due to pressure loading is almost zero at free end and increases linearly towards the support, the material in the web was also showing this type of behaviour. It meant that material also increasing in the same proportion towards the support.

In case of overhang beam, the thickness of the material is in the region where support is present because at this region shear is maximum. Similarly, at the end and at the mid-section material is absent. The shape of the web between the support is arch type.

The simply-supported model shape is more truss like and material is absent at the centre and maximum at the support due to the high shear-moment ratio at the mid-section and least at the centre. The shape of the web is along the line of principal stresses.

4.3.4 Type of Loading

Another parametric study consists of different type of loading. Two type of loading were applied on simply supported, cantilever and overhang beam. These were pressure load and point load. These loads were applied on the three types of support beam and topology optimisation was done in the abaqus. The optimised shape obtained was analysed. The models and optimised shape are shown in the figures shown below.

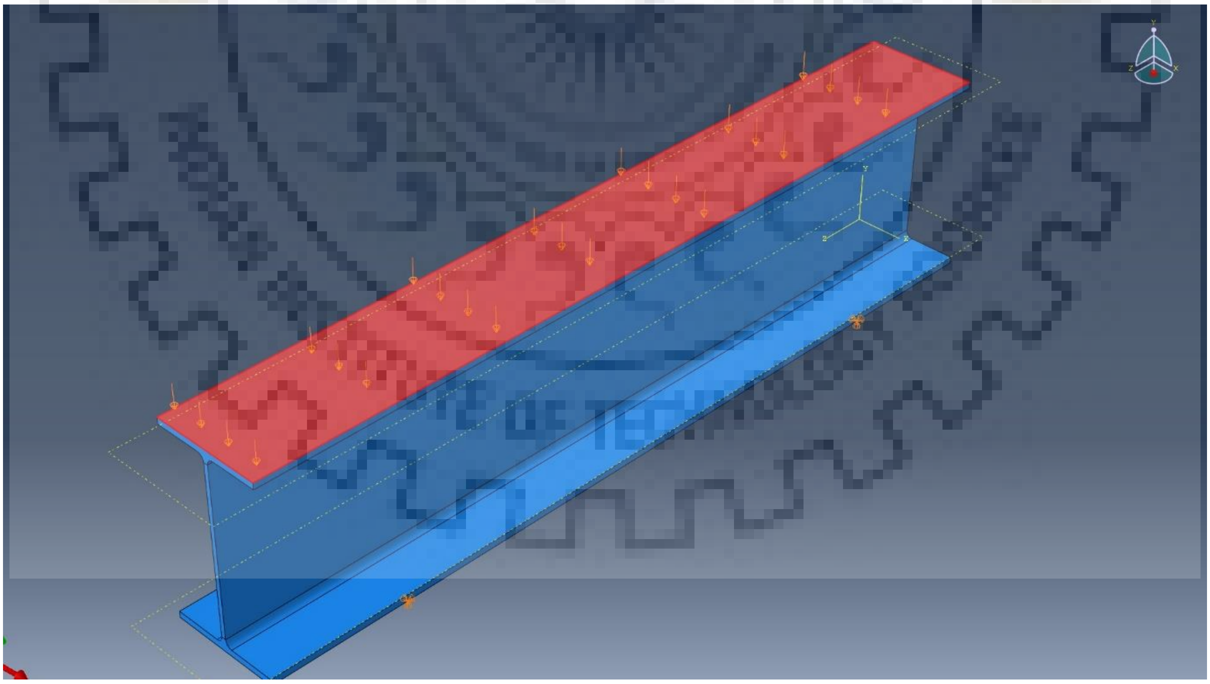


Fig 4. 21 Overhang I-beam with pressure loading

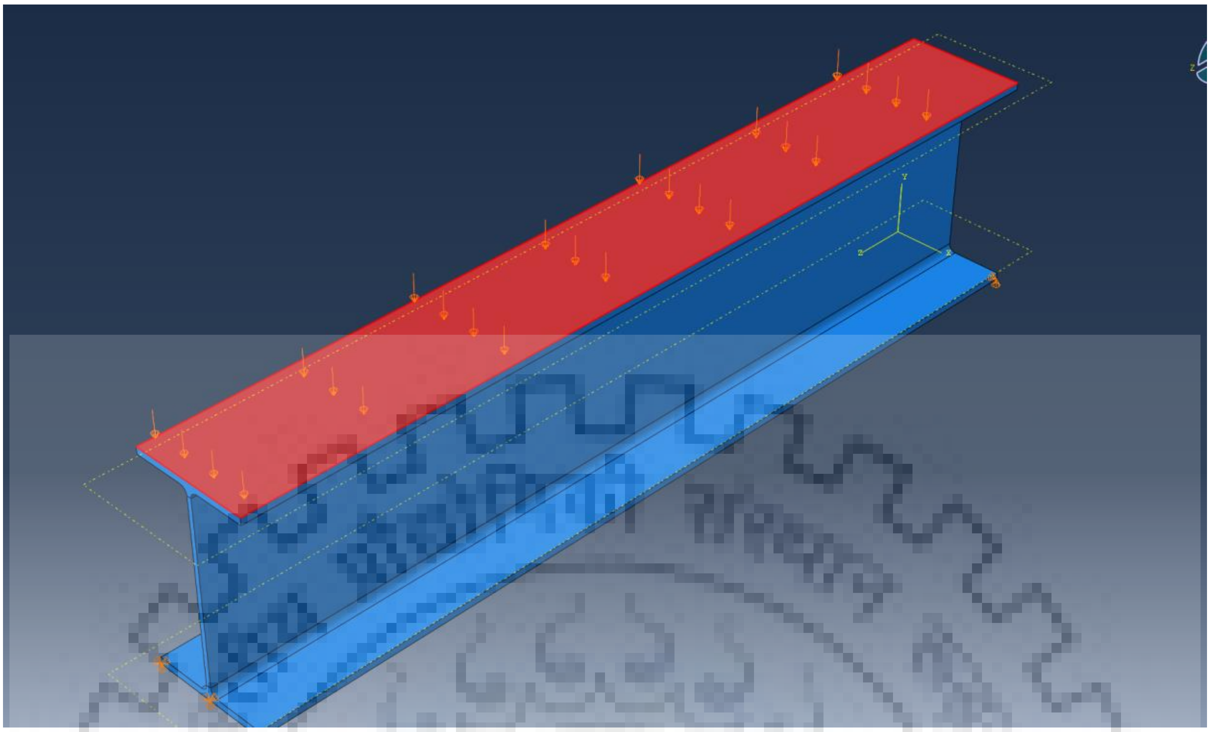


Fig 4. 22 Simply supported I-beam with pressure loading

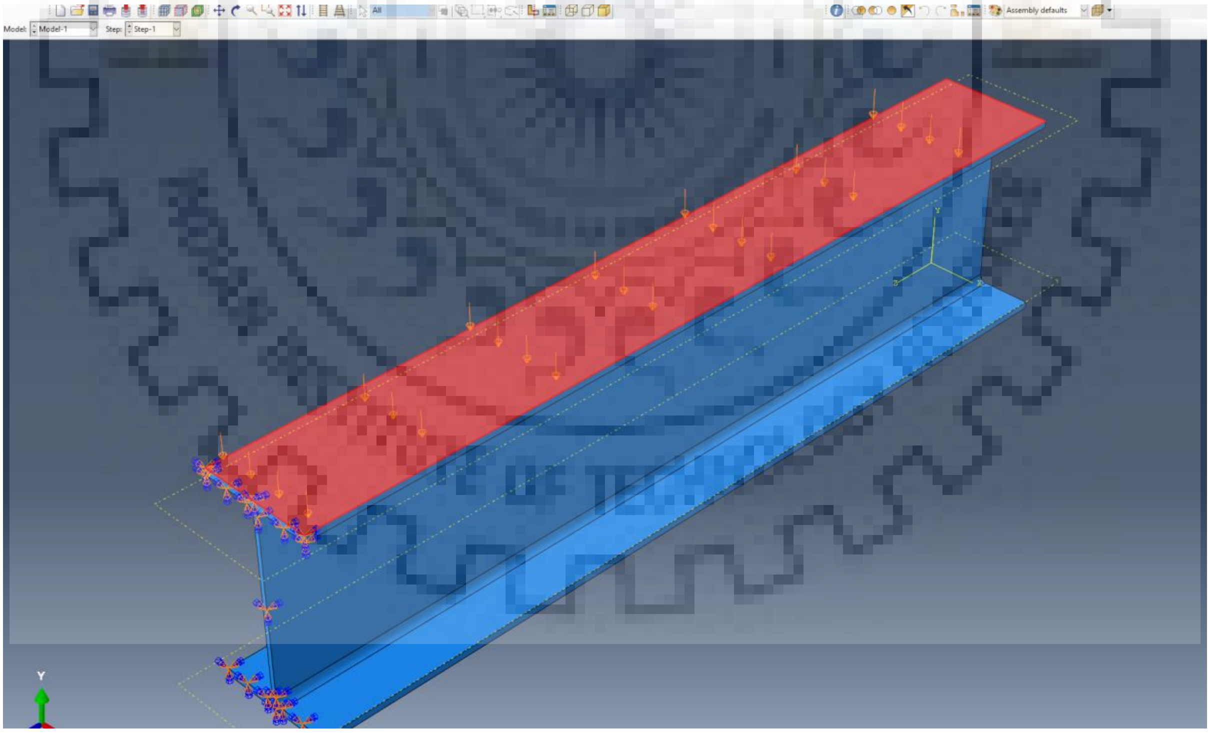


Fig 4. 23 Cantilever I-beam with pressure loading

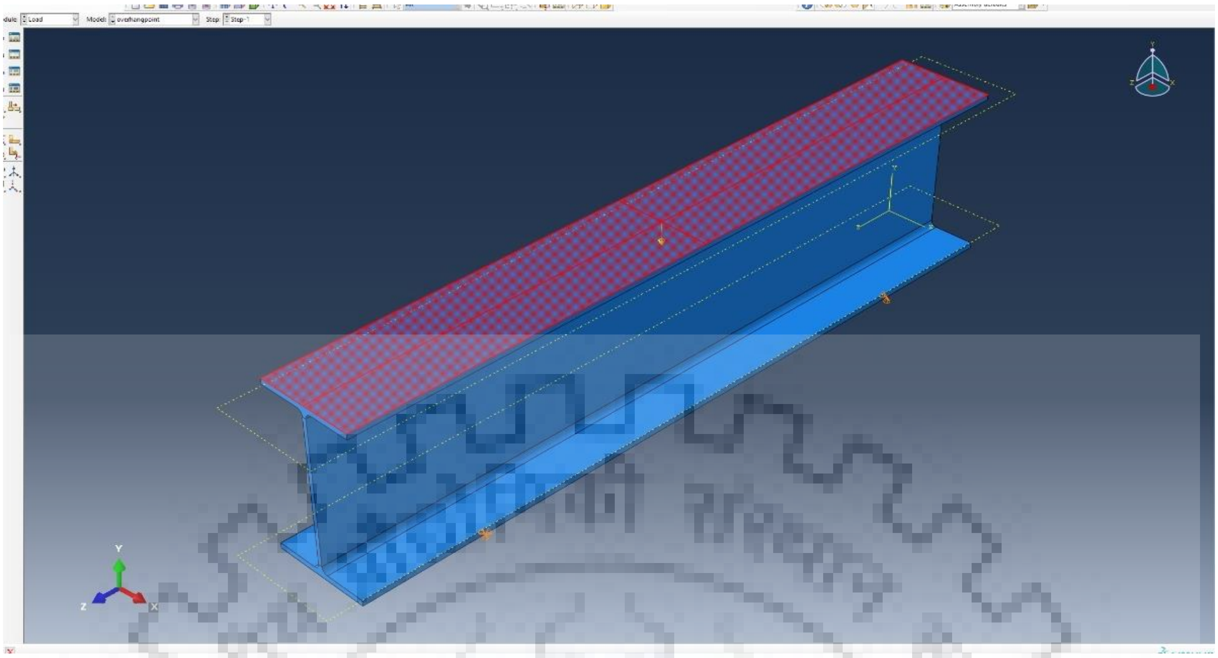


Fig 4. 24 Overhang I-beam with point load

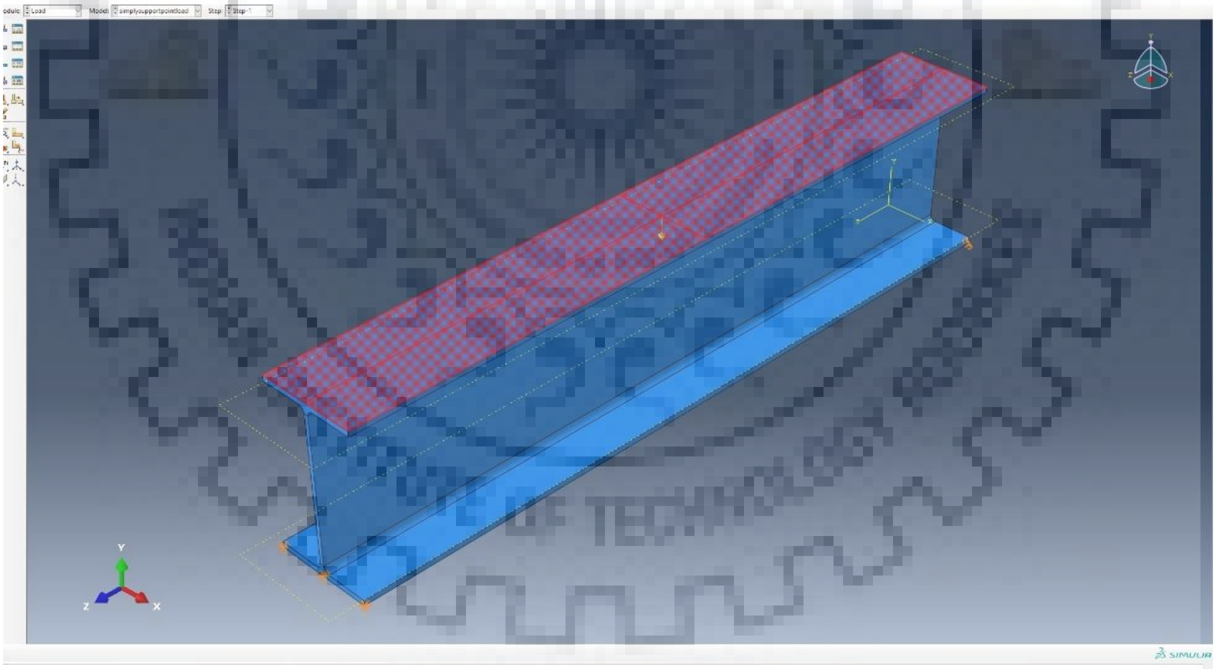


Fig 4. 25 Simply supported I-beam with point load

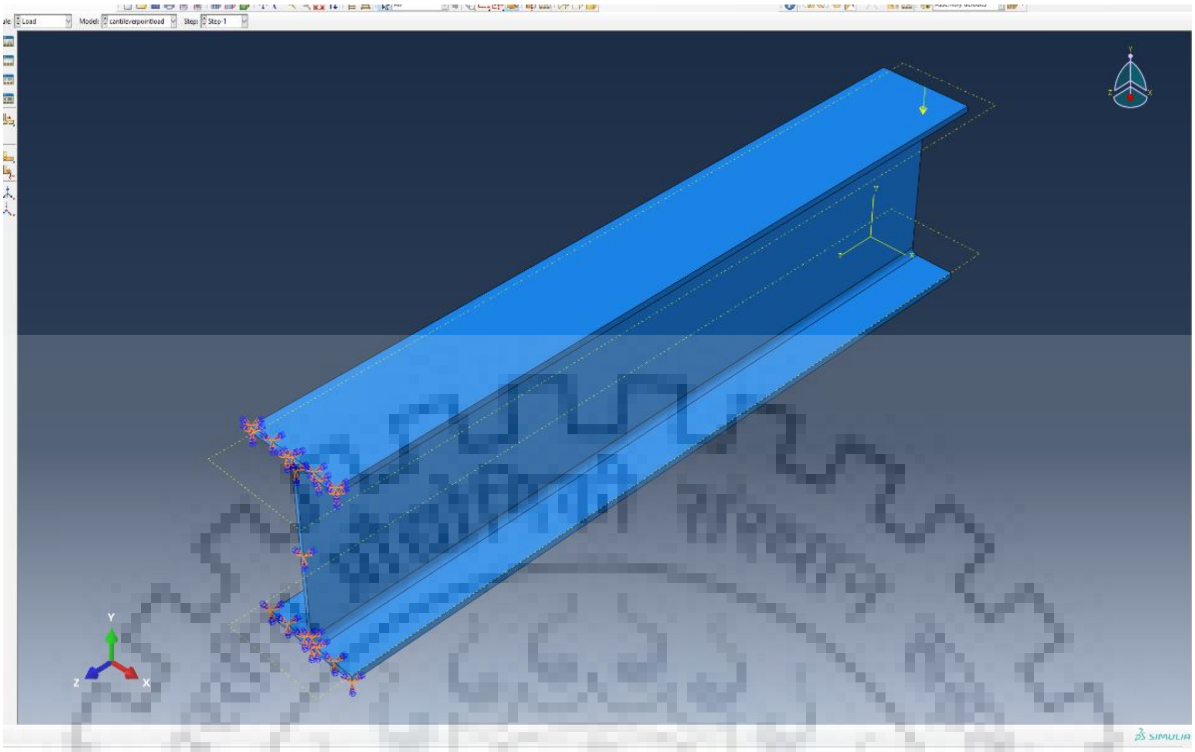
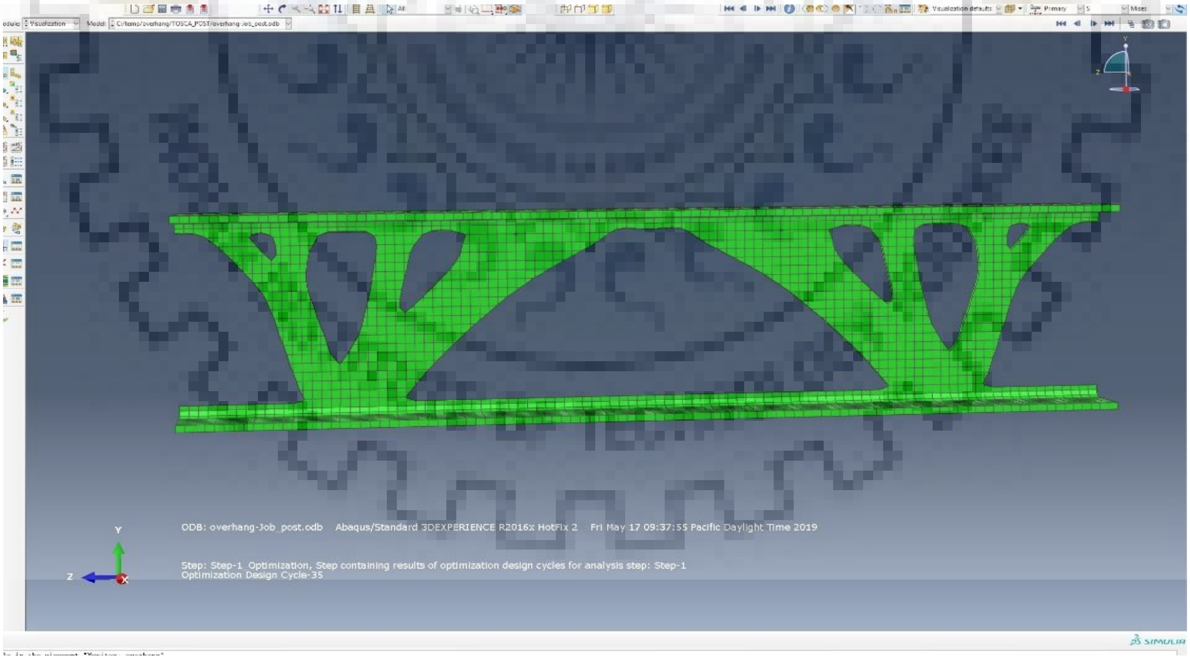


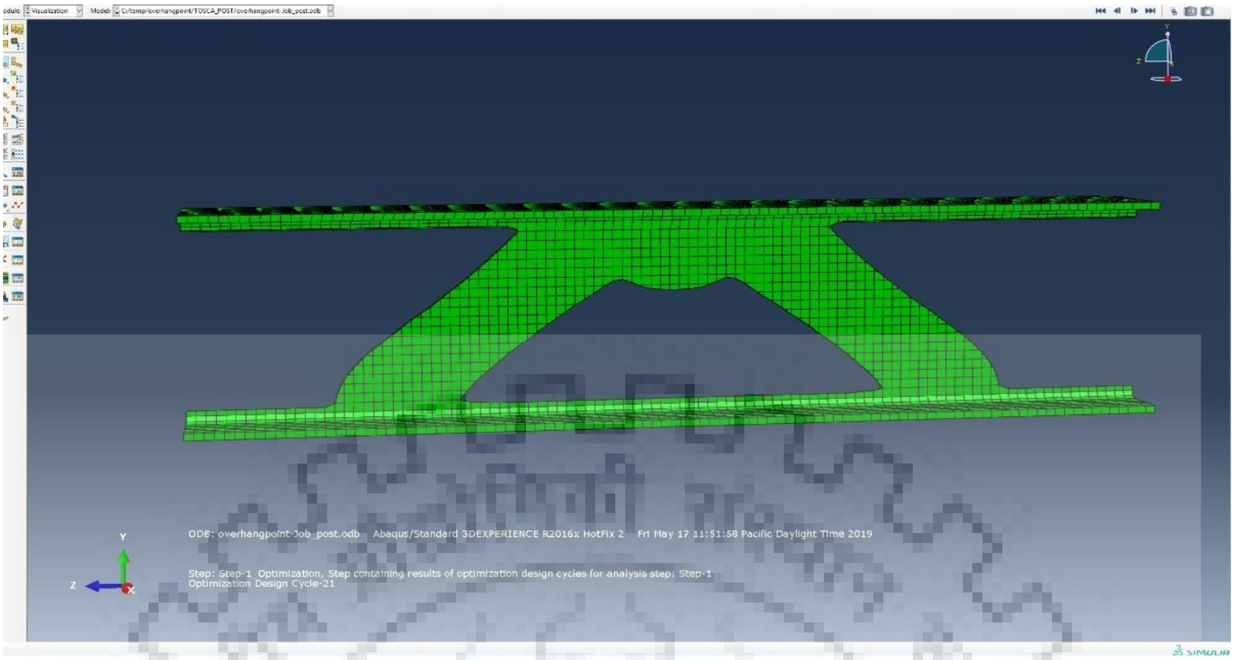
Fig 4. 26 Cantilever support with point load



```
ODB: overhang-Job_post.odb    Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2    Fri May 17 09:37:55 Pacific Daylight Time 2019

Step: Step-1 Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-35
```

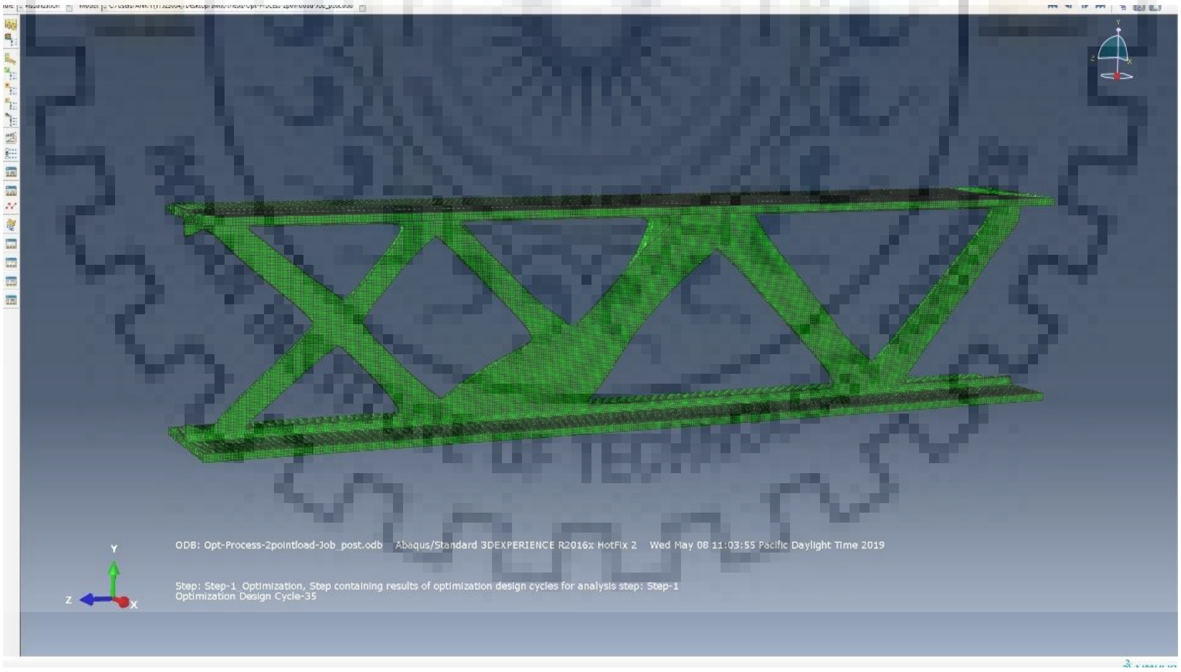
Fig 4. 27 Topology optimised overhang I-beam with pressure loading


odule: Visualization Model: C:/temp/overhangpoint/TOSCA_POST/overhangpoint-Job_post.odb

ODB: overhangpoint-Job_post.odb Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2 Fri May 17 11:51:58 Pacific Daylight Time 2019

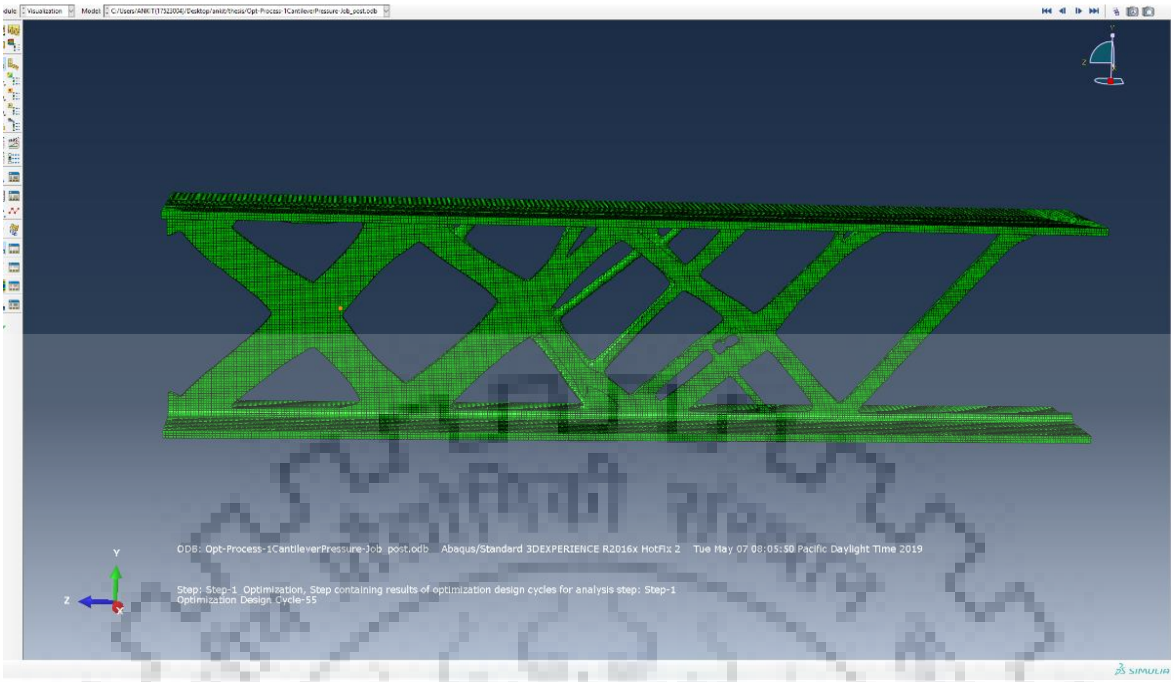
Step: Step-1 Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-21

Fig 4. 28 Topology optimised overhang I-beam with point load


ODB: Opt-Process-2pointload-Job_post.odb Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2 Wed May 08 11:03:55 Pacific Daylight Time 2019

Step: Step-1 Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-35

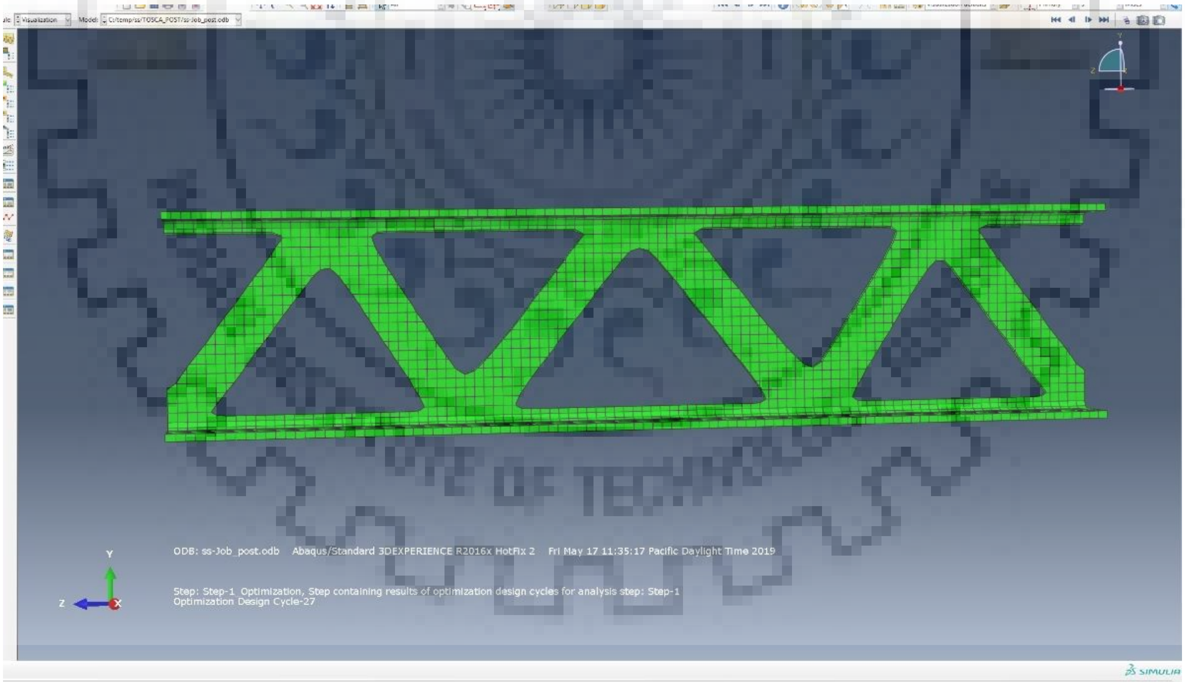
Fig 4. 29 Topology optimised cantilever I-beam with point load



```
ODB: Opt-Process-1CantileverPressure-Job_post.odb    Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2    Tue May 07 08:05:50 Pacific Daylight Time 2019

Step: Step-1  Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-55
```

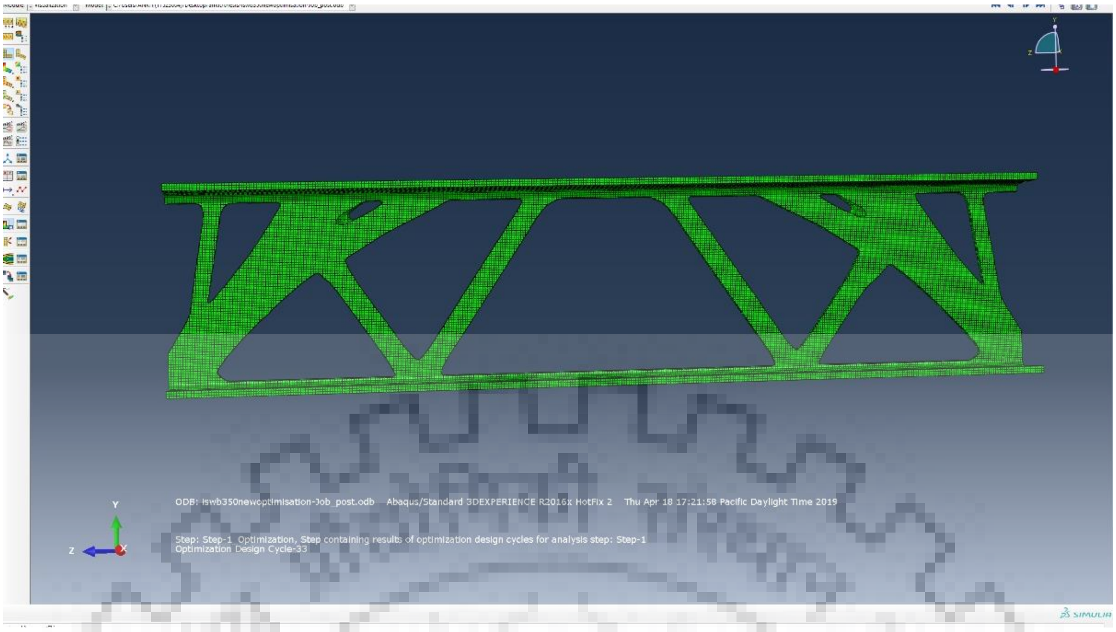
Fig 4. 30 Topology optimised cantilever I-beam with pressure loading



```
ODB: ss-Job_post.odb    Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2    Fri May 17 11:35:17 Pacific Daylight Time 2019

Step: Step-1  Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-27
```

Fig 4. 31 Topology optimised simply supported I-beam with point load


ODB: iswb350newoptimisation-Job_post.odb   Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2   Thu Apr 18 17:21:58 Pacific Daylight Time 2019

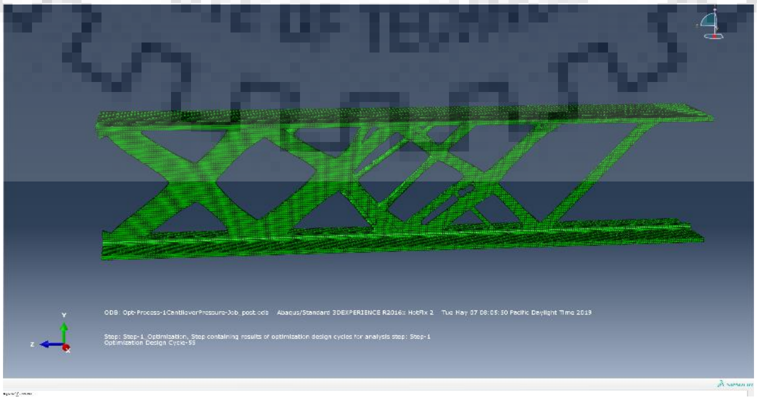
Step: Step-1_Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-33

Fig 4. 32 Topology optimised Simply supported I- beam with pressure loading

It was seen that all the optimised shape of the I-beam were different from each other. It showed that the optimised shape largely depends on the support condition and type of loading. The material was largely distributed towards the support and the section where moment was less, there was no material. This also confirmed that previous observations were correct and the shape of the I-beam depends on shear-moment ratio at the section and it follows the line of principal stresses.

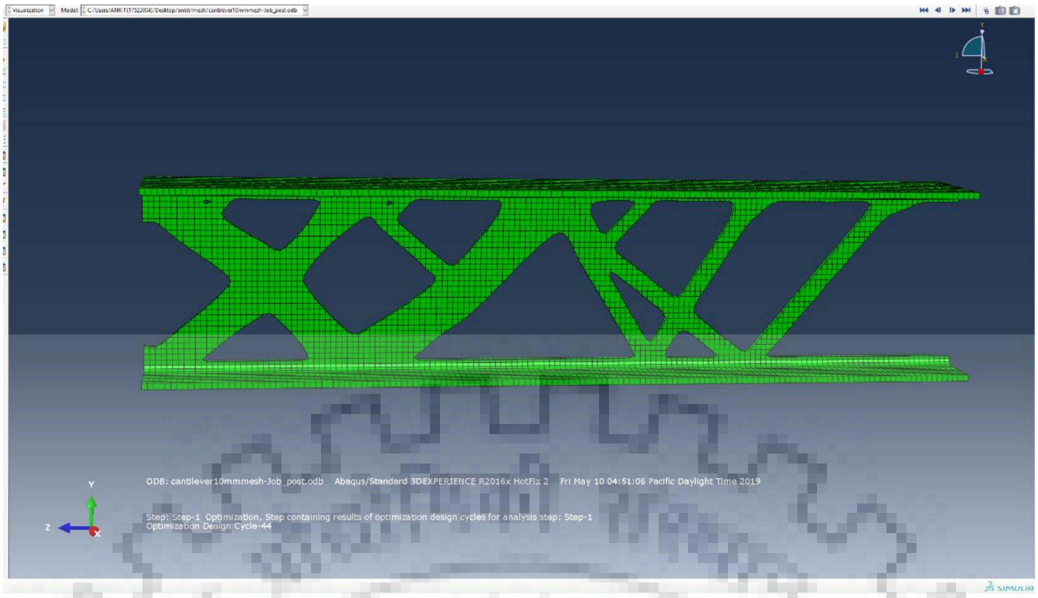
4.3.5 Mesh Size

One more parametricstudy was performed in which cantilever beam with different mesh size was optimised and the shape was obtained. Three cantilever beams with pressure loading were modelled but with different size of mesh. The volume fraction also took the same. The optimised shape are shown in the figures below.


ODB: Opt-Process-1CantileverPressure-Job_post.odb   Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2   Tue May 07 08:05:30 Pacific Daylight Time 2019

Step: Step-1_Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-55

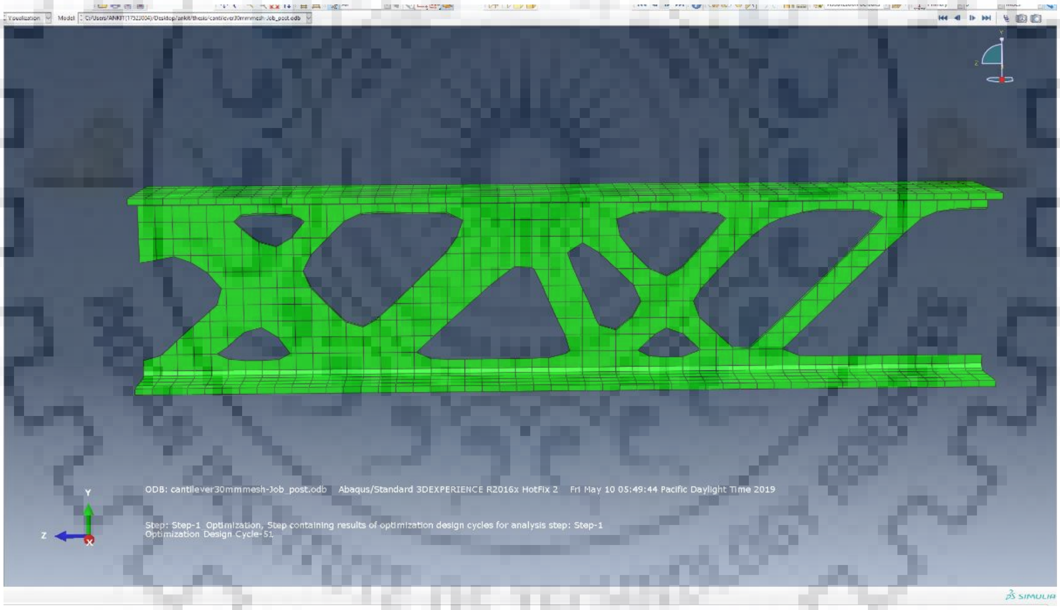
Fig 4. 33 Cantilever beam of mesh size 5mm



ODB: cantilever10mmmesh-Job_post.odb Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2 Fri May 10 04:51:06 Pacific Daylight Time 2019

Step: Step-1 Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-44

Fig 4. 34 Cantilever beam of mesh size 10mm



ODB: cantilever30mmmesh-Job_post.odb Abaqus/Standard 3DEXPERIENCE R2016x HotFix 2 Fri May 10 05:49:44 Pacific Daylight Time 2019

Step: Step-1 Optimization, Step containing results of optimization design cycles for analysis step: Step-1
Optimization Design Cycle-51

Fig 4. 35 Cantilever beam of mesh size 30mm

It was concluded from the above figures that mesh size does not effect the shape of beam as much.Thus, it was concluded that support conditions, load type, depth,flange-web depth changed the shape of the optimised beam more than the mesh size.these were the parametric studies performed.

CHAPTER 5 CONCLUSION AND RECOMMENDATIONS

5.1 Summary

The main purpose of this work was to understand the procedure of finding out the topology optimised shape of different model with the help of abaqus. FEM software ABAQUS was used to analyse and find out the optimised web of different models. Different parametric study were carried out to understand the change in shape and size that took place in the models and the trend followed by optimised shape if it had any trend. There were five parameters that might affect the optimised shape, which were depth of section, web-flanges ratio, support condition, types of loadings and mesh size. One parameter was changed while other were unchanged and study was done.

5.2 Conclusions

The optimised shapes that were obtained by the analysis of different models in abaqus helps in understanding the factors that affect the filling of materials in different areas of the model that affects the shape and size of optimised beam. Some conclusions that can be drawn through the parametric study are listed below

- The change in depth of web section does affect the shape of web opening but the change is very little. As the depth increases, number of openings also increases.
- As shear taken by the web at the section decreases, number of openings increases to keep the stiffness-weight ratio balanced.
- The effect of change in web-flange ratio show drastic change when the ratio changes to three. The shape before this ratio and after this ratio does not change as much.
- It is observed that the matetial is more where shear force is more and it decreases or increases as shear changes. Thus, the shape depends on shear value at the section.
- Due to the change in support condition or type of loading, shear force diagram changes hence optimised shape also changes. The loading type and support condition are the main parameter that have the maximum effect in determining the shape of the optimised beam.
- The volume fraction and the mesh size mainly effects the size of the optimised beam and the change in shape is very little due to these parameter.

5.3 Recommendations

The following recommendations can be drawn on the basis of this dissertation work

- Effect of the parameters on the steel beams with stiffeners can be studied.
- Find out the failure modes of topology optimised beams.
- Find out the load carrying capacities of perforated beams or optimised beams.
- Carry out the material and geometrical non-linear analysis of the optimised beam.

References

[1] M. P. B. a. N. Kikuchi, “Generating Optimal Topology in Structural Design using a Homogenisation Method,” *Computer Methods In Applied Mechanics and Engineering,* 1988.

[2] O. Sigmund, “A 99 line topology optimisation code wriiten in Matlab,” *Structural and Multidisciplinary Optimisation,* 2001.

[3] I. A. R. H. R. W. C. T. A. Aremu, “Suitability if SIMP and BESO Topology Optimisation Algorithms for Additive Manufacturing,” *SOLID FREEFORMFABRICATION PROCEEDINGS,* 2010.

[4] C. D. Konstantinos Daniel Tsavdaridis a, “Web buckling study of the behaviour and strength of perforated steel beams with different novel web opening shapes,” *Journal of Constructional Steel Research,* 2011.

[5] J. J. K. ,. V. V. T. Konstantinos Daniel Tsavdaridis, “Application of structural topology optimisation to perforated steel,” *Computes and Structures,* 2015.

[6] L. M. G. Samadhan G. Morkhade, “An experimental and parametric study on steel beams with web,” *International Journal of Advanced Structural Engineering,* 2015.

[7] J. T. K. a. T. Kingman, “Applications of topology optimization,” in *Civil Engineering for Sustainability and Resilience*, 2014.

[8] A. u. manual, 2008. [Online].