DESIGN OPTIMIZATION & ANALYSIS OF A CONVEYOR FRAME FOR LINEAR STATIC STRUCTURAL ANALYSIS

Shaikh Habib Hameed[1], Prof. R.S. Shivpuje[2]

[1]M.tech student, Department of Mechanical Engineering, M.B.E.S college of Engineering Ambajogai, Maharashtra, India.
[2] Assistant Professor, Department of Mechanical Engineering, M.B.E.S college of Engineering Ambajogai, Maharashtra, India.

Abstract - *The success of companies depends on their ability to produce high quality products at the lowest cost, design optimization has become a practical method of visualizing manufacturing cost and enabling product design within cost targets. The aim of project work is to discover how design optimization includes in engineering design to minimize the cost of product without sacrificing the application.*

Current globalization enables new market's needs and customization of products. Design/Engineering plays a vital role in growth, efficiency & profitability of an organization. As technological innovation changes the approach of design, industry now focusing on parametric modeling and master model technique, where they can automate the design process which reduces the mundane task, for custom based product the overall design same but changes require in overall sizing of the components.

In Material Handling system each machines vary with different geometry and size as per the customer layout, use of traditional design using a combination of judgment, experience, modeling, opinions of others, etc. the engineer makes design decisions which, to an optimal design. Some engineers are very good at this. However, if there are many variables to be adjusted with several conflicting objectives and/or constraints, this type of experience-based optimization can fall short of identifying the optimum design. The interactions are too complex and the variables too numerous to intuitively determine the optimum design.

The proposed solutions solve this problem by computer-based virtual simulation approach to design optimization. With this approach, we use the simulation to search for the best design according to criteria that we specify. The computer's enormous processing power allows us to evaluate many more design combinations than we could do manually. Further, we employ sophisticated algorithms that enable the computer to efficiently search for the optimum. Often we start the algorithms from the best design we have based on experience and intuition. We can then see if any improvement can be made.

Key Words: parametric modeling, design optimization, etc.

1. INTRODUCTION

In the past two decades, modal analysis has become a major technology in the search for determining, improving and optimizing dynamic characteristics of engineering structures. Not only has it been recognized in mechanical engineering, but modal analysis has also discovered profound applications for machine structures, electro-mechanical problems, space structures, acoustical instruments, transportation and nuclear plants.

The Beams are basic components produced using steel material, while extremely proficient as far as the auxiliary quality and firmness to weight proportions can be powerless to greatly complex unsteadiness phenomena. Presently days in industry to conveying high ton limit of burden different types of material handling equipment's like conveyor, gantry, cranes are used to transfer material from one location to other.

In general, applications of modal analysis today cover a broad range of objectives identification and evaluation of deflection, stress, resultant force, factor of safety. The study of mathematical models which involve physical and geometric parameters such as mass density ρ, elastic modulus E, Poisson's ratio v, lengths, and cross-section shape characteristics.

Current project will be defined as optimization of conveyor structure as per the component or industrial requirements, this will also find out the dimensions of different sections as per the end customer component weight which will help to create a custom based conveyor frame with optimized design.

1.1 Product information

One of the most predominant equipment types in bulk material handling is a conveyor. Although many other types of conveyors are available – such as screw, chain and pneumatic the most commonly used is the roller conveyor and belt. Conveyors are the backbone of bulk material handling systems in many different processing plants in design of the mechanical parts, implementation and use of the designed products have developed optimization methods.

Conveyors are able to safely transport materials from one level to another, which when done by human labour would be strenuous and expensive, they can be installed almost anywhere, and are much safer than using a forklift or other machine to move materials, they can move loads of all shapes, sizes and weights. Also, many have advanced safety features that help prevent accidents.

There are a variety of options available for running conveying systems, including the hydraulic, mechanical and fully automated systems, which are equipped to fit individual needs. Conveyor is customized as per the requirements so every time based on the component weight and size designers has to change the different sizes of conveyor, this may lead to over design or under design:

Fig -1: Roller Conveyor with Component

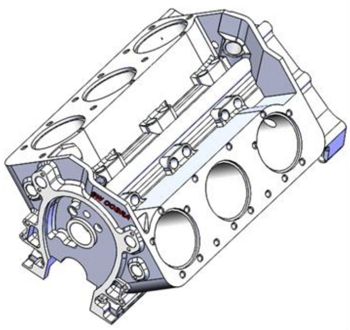


Fig -2: Component to Be Travel

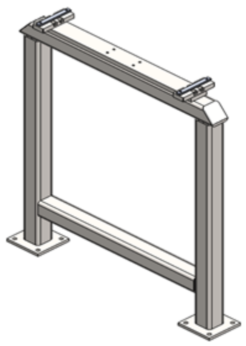


Fig -3: Leg Assembly

2. PROBLEM DEFINATION AND FEA

As discussed in Chapter 2 we have discussed the need of structural optimization of conveyor frame as per the customer requirements, here we will like to design the material handling system and based on the component weight, length of the conveyor and number of component travel, we will like to analyze the frame for correct sizing of component by considering the defined constrained.

We also like to compare the weight, deflection, factor of safety for traditional and new design optimize vertically.

Finite Element Analysis (FEA), also known as the Finite Element Method (FEM), is a numerical technique for describing physical phenomena in terms of partial differential equations. Finite element analysis is widely used in many engineering disciplines for solving structural mechanics, vibration, heat transfer, and other problems.

You use the FEM to predict the behavior of mechanical and thermal systems under their operating conditions, to reduce the design cycle time, and to improve overall system performance.

The basic steps in any FEA process are as follows:

Geometric representation Creates the geometric features of the system to be analyzed stored in a CAD database.

Element formulation Develops the equations that describe the behavior of each element. Material properties for each element are considered in the formulation of the governing Element formulation equations. This involves choosing a displacement function within each element.

Assembly Obtains the set of global equations for the entire model from the equations of individual elements. The loads and support (boundary) conditions are applied to the appropriate nodes of the finite element mesh.

Solution of equations Provides the solution for the unknown nodal degrees of freedom (or generalized displacements).

2.1 Components of a FEA Model

A FEA model is composed of several different components that together describe the physical problem to be analyzed and the results to be obtained. At a minimum the analysis model consists of the following information: discretized geometry, element properties, material data, loads and boundary conditions, analysis type, and output requests.

2.1.1 Discretized geometry

Finite elements and nodes define the basic geometry of the physical structure being modeled.

Each element in the model represents a discrete portion of the physical structure, which is, in turn, represented by many interconnected elements. Elements are connected to one another by shared nodes. The coordinates of the nodes and the connectivity of the elements—that is, which nodes belong to which elements—comprise the model geometry. The collection of all the elements and nodes in a model is called the mesh. Generally, the mesh is only an approximation of the actual geometry of the structure.

The element type and the overall number of elements used in the mesh affect the results obtained from a simulation.

The greater the mesh density (that is, the greater the number of elements in the mesh), the more accurate the results. As the mesh density increases, the analysis results converge to a unique solution, and the computer time.

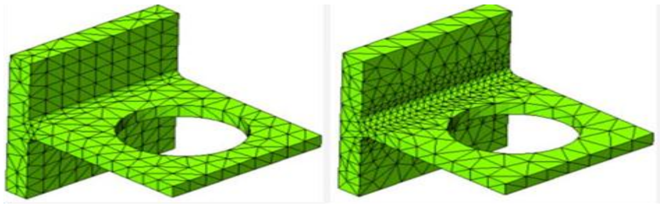


Fig -4: CAD Model divided in Small Pieces (Elements)

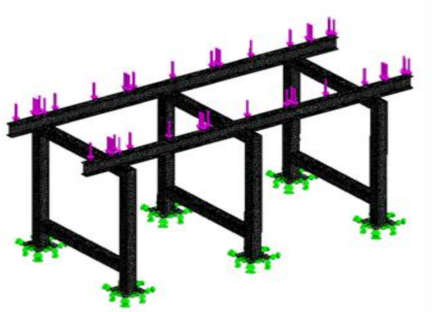


Fig -5: CAD Model divided in Small Pieces-2 (Elements)

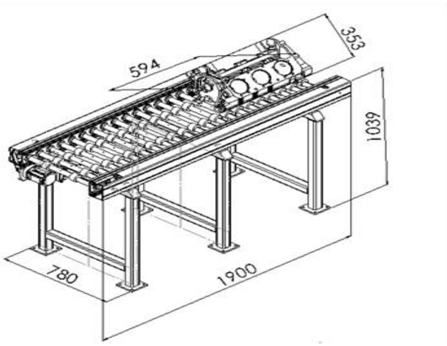


Fig -6: Dimension of Conveyor

2.1.2 Element properties

The choice of the appropriate element is very important for the simulation of the physical problem. Element libraries include elements that are simple geometric shapes with one, two, or three dimensions.

Continuum or solid elements are appropriate for bulky or complex 3D models. Shell elements are suitable for thin parts with thickness significantly smaller than the other dimensions. Beam elements are suitable for structural members where the length is significantly greater than the other two dimensions.

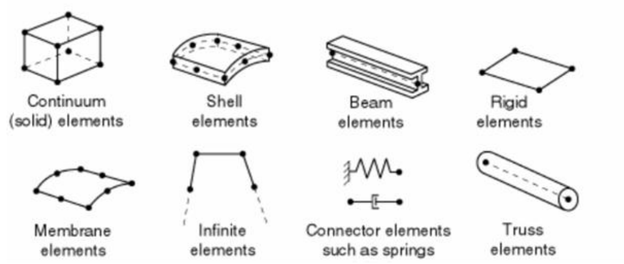


Fig -7: Common Element Families

2.1.3. Material data

Material properties for all elements must be specified. While high-quality material data are often difficult to obtain, particularly for the more complex material models, the validity of the simulation results is limited by the availability of accurate material data.

2.1.4. Loads and boundary conditions

The application of loads distorts the physical structure and, thus, creates stress in it. Boundary conditions are used to constrain portions of the model to remain fixed (zero displacements) or to move by a prescribed amount (nonzero displacements).

The most common forms of loading include:

- Point loads
- Pressure loads on surfaces
- Distributed tractions on surfaces
- Distributed edge loads and moments on shell edges
- Body forces, such as the force of gravity
- Thermal loads
- In a static stress analysis adequate boundary conditions must be used to prevent the model from moving as a
- rigid body in any direction; otherwise, unrestrained rigid body motion causes the simulation to stop prematurely.
- The potential rigid body motions depend on the dimensionality of the model.

2.1.5 Analysis type

Depending on the type of applied load environment, inclusion of inertia effects, and material properties, the appropriate analysis type must be selected for the simulation. Common analysis types include linear static, nonlinear static, dynamic, buckling, heat transfer, fatigue, and optimization.

In a static analysis the long-term response of the structure to the applied loads (which are applied gradually and slowly until they reach their full magnitude) is obtained. In cases where the loads are changing with time or frequency, a dynamic analysis is required. For example, you perform a dynamic analysis to simulate the effect of an impact load on a component or the response of a building during an earthquake.

A nonlinear structural problem is one in which the structure's stiffness changes as it deforms. All physical structures exhibit nonlinear behavior. Linear analysis is a convenient approximation that is often adequate for design purposes. It is obviously inadequate for many structural simulations including manufacturing processes,

Such as forging or stamping; crash analyses; and analyses of rubber components, such as tires or engine mounts.

3.1 software tool could be used: - solidworks and simulation

Dassault Systèmes SOLIDWORKS Corp. offers complete 3D software tools that let you create, simulate, publish, and manage your data. SOLIDWORKS products are easy to learn and use and work together to help you design products better, faster, and more cost-effectively.

The SOLIDWORKS focus on ease-of-use allows more engineers, designers and other technology professionals than ever before to take advantage of 3D in bringing their designs to life.

SOLIDWORKS Simulation Standard is an intuitive virtual testing environment for static linear, time-based motion, and high-cycle fatigue simulation. It delivers a concurrent engineering approach, helping you know if your product will perform properly and how long it will last-lasting the design phase, S OLIDWORKS Simulation Professional enables you tooptimize your design, determine product mechanical resistance, product durability, topology, natural frequencies, and test heat transfer and buckling instabilities, It can also perform sequential multi-physics simulations.

3. FEA ANALYSIS

3.1 Geometry Details: Traditional

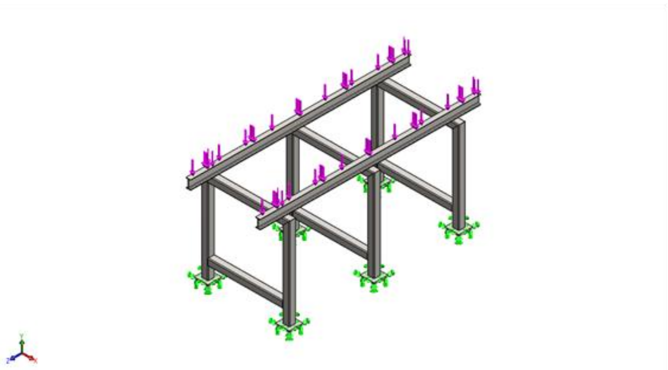


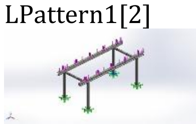
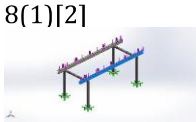
Fig -8: Geometry Details Structure Frame Simplified

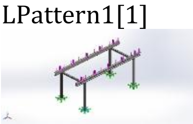
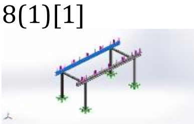
Sub Assembly Details

1) Foundation Assembly
2) Leg Assembly
3) Beam Assembly
4) Drive Assembly
5) Tie Rod Assembly
6) Roller Assembly

3.2 Geometry details: modified design Model Information

Table -1: Geometry Details: Modified design

Document Name and Reference	Treated As	Volumetric Properties
Trim/Extend4[2]	Solid Body	Mass:2.26055 kg Volume:0.00028981 4 m^3 Density:7,800 kg/m^3 Weight:22.1534 N
Trim/Extend6[2]	Solid Body	Mass:2.26055 kg Volume:0.00028981 4 m^3 Density:7,800 kg/m^3 Weight:22.1534 N
Trim/Extend3[2]	Solid Body	Mass:2.26055 kg Volume:0.00028981 4 m^3 Density:7,800 kg/m^3 Weight:22.1534 N
LPattern1[2] 	Solid Body	Mass:1.34784 kg Volume:0.0001728 m^3 Density:7,800 kg/m^3 Weight:13.2088 N
Trim/Extend1[2]	Solid Body	Mass:2.26055 kg Volume:0.00028981 4 m^3 Density:7,800 kg/m^3 Weight:22.1534 N
C channel CH 80 X 8(1)[2] 	Solid Body	Mass:15.5398 kg Volume:0.00199228 m^3 Density:7,800 kg/m^3 Weight:152.29 N

LPattern1[3] 	Solid Body	Mass:1.34784 kg Volume:0.0001728 m^3 Density:7,800 kg/m^3 Weight:13.2088 N
Boss-Extrude1 	Solid Body	Mass:1.34784 kg Volume:0.0001728 m^3 Density:7,800 kg/m^3 Weight:13.2088 N
LPattern1[1] 	Solid Body	Mass:1.34784 kg Volume:0.0001728 m^3 Density:7,800 kg/m^3 Weight:13.2088 N
Trim/Extend4[1]	Solid Body	Mass:2.0287 kg Volume:0.00026009 m^3 Density:7,800 kg/m^3 Weight:19.8813 N
Trim/Extend6[1]	Solid Body	Mass:2.0287 kg Volume:0.00026009 m^3 Density:7,800 kg/m^3 Weight:19.8813 N
C channel CH 80 X 8(1)[1] 	Solid Body	Mass:15.5398 kg Volume:0.00199228 m^3 Density:7,800 kg/m^3 Weight:152.29 N

3.3 Study Properties

Table -2: Study properties

Study name	**Static 1**
Analysis type	Static
Mesh type	Solid Mesh
Thermal Effect:	On
Thermal option	Include temperature loads
Zero strain temperature	298 Kelvin
Include fluid pressure effects from SOLIDWORKS Flow	Off
Simulation	
Solver type	Direct sparse solver
Inplane Effect:	Off
Soft Spring:	Off
Inertial Relief:	Off
Incompatible bonding options	Automatic
Large displacement	Off
Compute free body forces	On
Friction	Off
Use Adaptive Method:	Off
Result folder	SOLIDWORKS document (C:\Users\Admin\Desktop\ME3\conveyor analysis\Conveyor Analysis_Simplified Geometry)

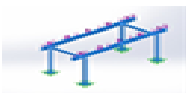
3.4 Units

Table -3: Units

Unit system:	**SI (MKS)**
Length/Displacement	mm
Temperature	Kelvin
Angular velocity	Rad/sec
Pressure/Stress	N/m^2

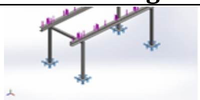
3.5 Material Properties

Table -4: Material Properties

Model Reference	**Properties**	
	Name:	Plain Carbon Steel
	Model type:	Linear Elastic Isotropic
	Default failure criterion:	Max von Mises Stress
	Yield strength:	2.20594e+08 N/m^2
	Tensile strength:	3.99826e+08 N/m^2
	Elastic modulus:	2.1e+11 N/m^2
	Poisson's ratio:	0.28
	Mass density:	7,800 kg/m^3
	Shear modulus:	7.9e+10 N/m^2
	Thermal expansion coefficient:	1.3e-05 /Kelvin

3.6 Loads and Fixtures

Table -5: Loads and Fixture

<table>
<tr><th>Fixture name</th><th colspan="2">Fixture Image</th><th>Fixture Details</th></tr>
<tr><td>Fixed-1</td><td colspan="2"></td><td>Entities: 4 face(s)
Type: Fixed Geometry</td></tr>
<tr><td colspan="4">Resultant Forces</td></tr>
<tr><td>Components</td><td>x</td><td>y</td><td>z</td></tr>
<tr><td>Reaction force(N)</td><td>-9.58554e-05</td><td>10,000</td><td>3.23076e-05</td></tr>
<tr><td>Reaction Moment(N.m)</td><td>0</td><td>0</td><td>0</td></tr>
</table>

3.7 Contact information

Table -6: Contact Information

Contact	Contact Image	Contact Properties
Global Contact		Type: Bonded Components: 1 component(s) Options: Compatible mesh

3.7 Mesh information

Table -7: Mesh Information

Mesh type	Solid Mesh
Mesher Used:	Standard mesh
Automatic Transition:	Off
Include Mesh Auto Loops:	Off
Jacobian points	4 Points
Element Size	17.4328 mm
Tolerance	0.871639 mm
Mesh Quality Plot	High

3.8 Mesh information – details

Table -8: Mesh Information – Details

Total Nodes	73004
Total Elements	36298
Maximum Aspect Ratio	27.798
% of elements with Aspect Ratio < 3	27.3
% of elements with Aspect Ratio > 10	7.1
% of distorted elements(Jacobian)	0
Time to complete mesh(hh;mm;ss):	00:00:09
Computer name:	DESKTOP

Fig -9: Modiied Design

3.9 Resultant forces
Reaction forces

Table -9: Resultant Forces

Selection set	Units	Sum X	Sum Y	Sum Z
Entire Model	N	-9.58554e-05	10,000	3.23076e-05

3.9 Study results

Table -10: Study Result

Name	Type	Min	Max
Stress1	VON: von Mises Stress	6.531e-04 N/mm^2 (MPa) Node: 44251	93.412 N/mm^2 (MPa) Node: 16170

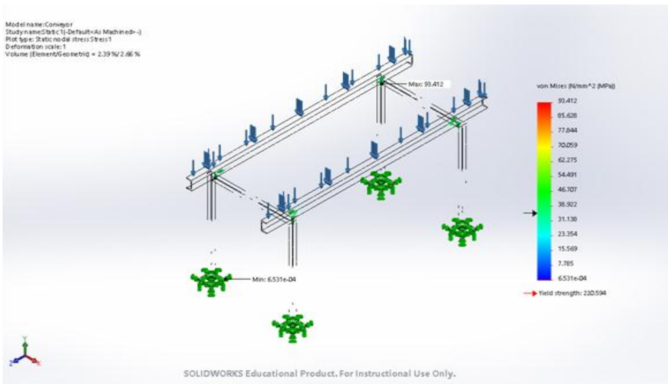


Fig -10: Conveyor-Static 1-Stress-Stress1

Name	Type	Min	Max
Displacement1	URES: Resultant Displacement	0.000e+00 mm Node: 20393	1.258 mm Node: 35596

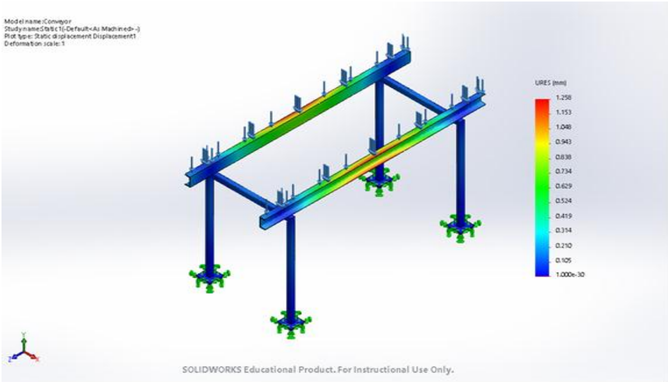


Fig -11: Conveyor-Static 1-Displacement-Displacement1

Name	Type	Min
Strain1	ESTRN: Equivalent Strain	3.250e-09 Element: 22125

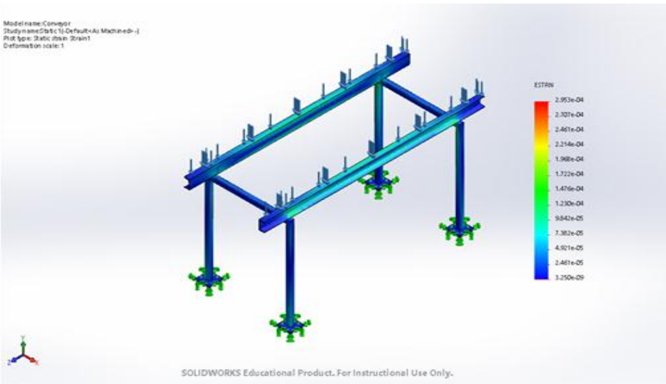


Fig -12: Conveyor-Static 1-Strain-Strain1

Name	Type	Min	Max
Factor of Safety1	Automatic	2.123e+00 Node: 16170	3.378e+05 Node: 44251

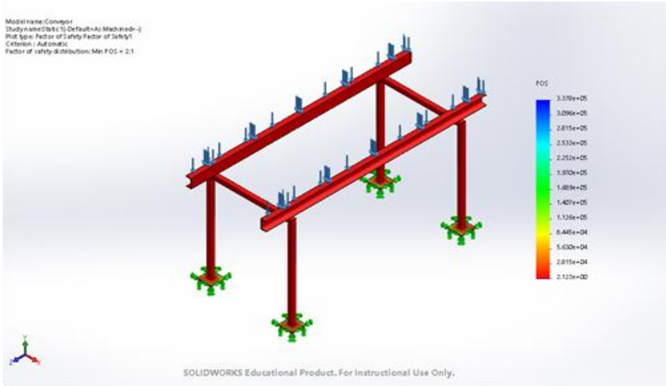


Fig -13: Conveyor-Static 1-Factor of Safety-Factor of Safety1

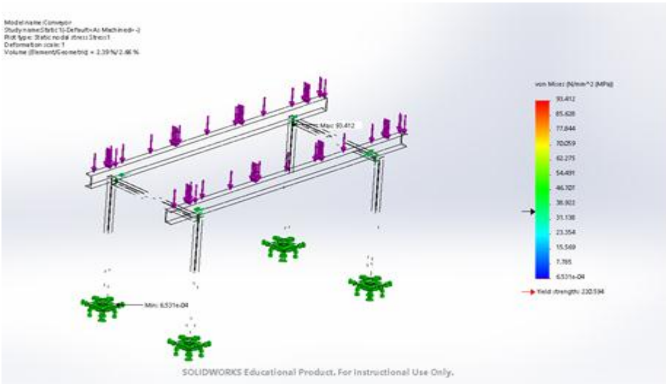


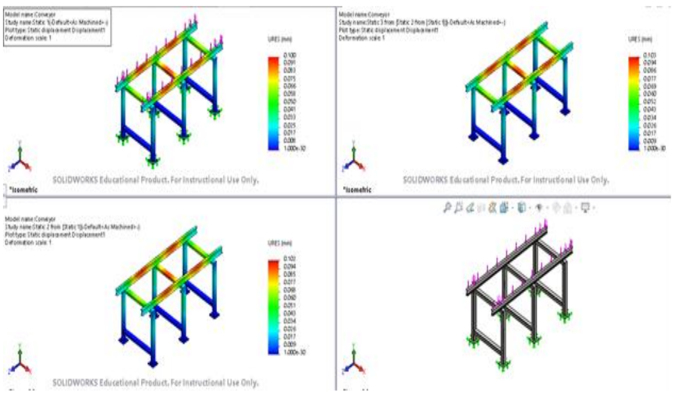
Fig -14: Very Small Portion above 35 MPA

3.14: STUDY RESULTS

- Displacement is 1.258 mm
- Stress 93.41 MPA
- Factor of Safety = 2.1

4. COMPARE RESULT

4.1 Compare result displacement, stress, fos: traditional



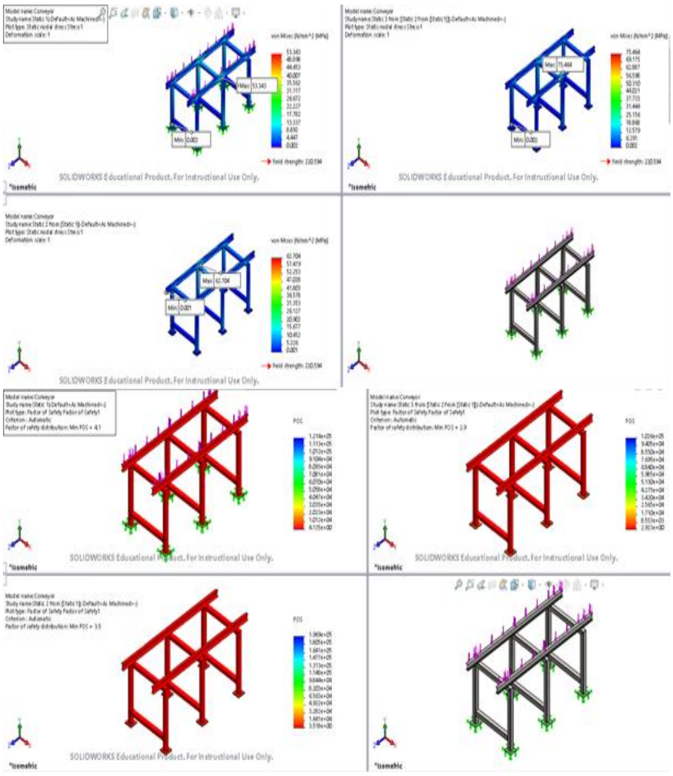
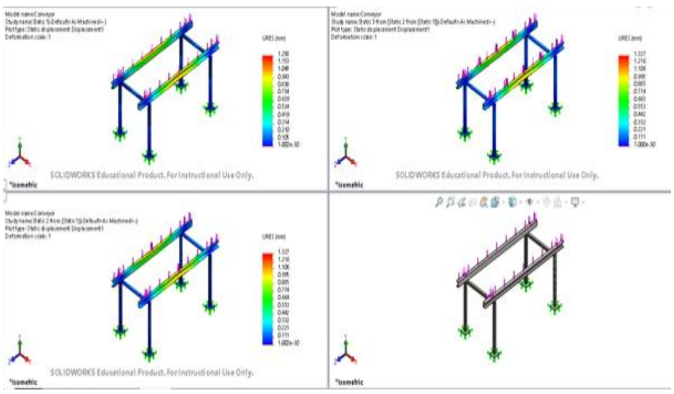
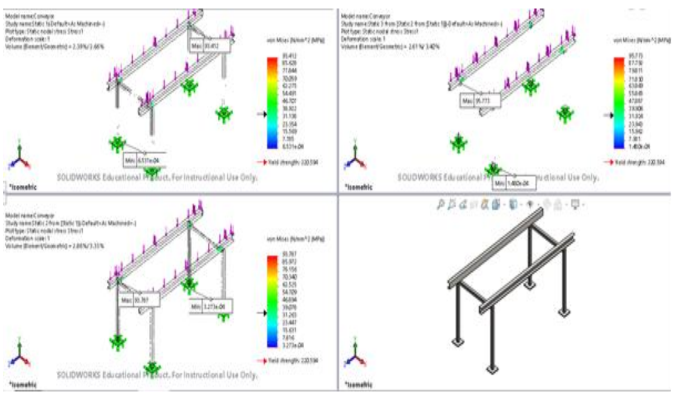


Fig -15: Compare Result Traditional

4.2 Compare result displacement, stress, fos: new modified





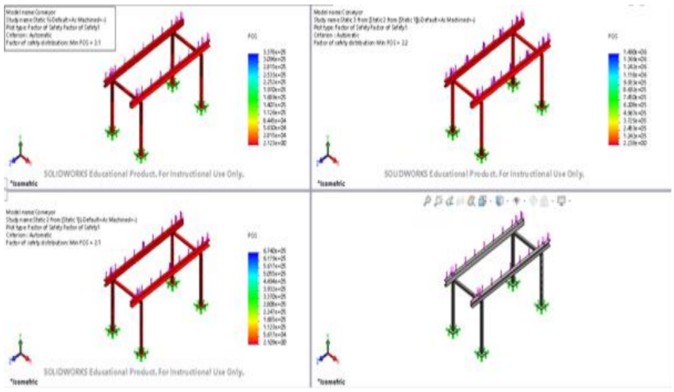


Fig -16: Compare Result Modified

5. CONCLUSION AND FUTURE SCOPE

5.1 Conclusions

This project work gives an idea about structural strength of conveyor frame, based on the above study, we observed

Table -11: Traditional Design

Traditional Design	
Weight :- 78039.78 grams	
Volume:- 10005100.14 cubic millimetres	
Iteration 1	
Stress	53.34 MPA
Displacement	0.1 mm
FOS	4.1
Iteration 2	
Stress	62.7
Displacement	0.102
FOS	3.5
Iteration 3	
Stress	75.464
Displacement	0.103
FOS	2.9

Table -12: Modified Design

Modified Design	
Weight :- 6355.19 grams	
Volume:- 6.355e+06 cubic millimetres	
Iteration 1	
Displacement	1.258 mm
Stress	93.41 MPA

FOS	2.1
Iteration 2	
Displacement	1.327
Stress	93.78 MPA
FOS	2.1
Iteration 3	
Displacement	1.327
Stress	95.77 MPA
FOS	2.2

• 36 % of weight optimization, directly link to material cost.

5.2 Future scope

There future scope of research work may be as follows:

• Further we can analyses for linear dynamics to check the real world scenario

REFERENCES

[1] Hong-Xia Wan, Mahen Mahendran, "Bending and torsion of hollow flange channel beams" Engineering Structures 84 (2015) 300–312, 2015

[2] Jiho Moon a, Jong-WonYi b, ByungH.Choi c, Hak-EunLee, "Lateral– torsional buckling of I-girder with corrugated webs under uniform bending", Thin-Walled Structures 47 (2009) 21– 30,2009

[3] Fatimah Denan, Mohd Hanim Osman & Sariffuddin Saad , "The Study of Lateral Torsional Buckling Behaviour of Beam With Trapezoid Web Steel Section By Experimental And Finite Element Analysis", IJRRAS 2 (3) March 2010

[4] Ismail Gerdemeli, Serpil Kurt, Hasan Onur Alkan , "Main Girder Beam Design And Finite Element Analysis Of 2x160 Ton Gantry Crane", Trends in the Development of Machinery and Associated Technology,2010

[5] M.R.Wakchaure, A.V. Sagade, "Finite Element Analysis of Castellated Steel Beam", International Journal of Engineering and Innovative Technology (IJEIT) 2, Issue 1, July 2012

[6] G.S.G. Beveridge and R.S. Schechter, Optimization: theory and practice, McGraw-Hill, New York , 1970