

# Optimization of a Special Moment Resisting Frame (SMRF) Under the Consideration of Earthquake Loading

Mohd Talha Usmani

M.Tech Student, Department of Civil Engineering, Dr. A.P.J. Abdul Kalam Technical University Lucknow, Lucknow Institute of Technology, India

**Abstract:** *The main challenge faced by Structural Engineers is to go for the most efficient and economical design with accurate results in solution to ensure that the final design of a building must be serviceable for its function purpose over the design life time. This Research presents the detailed knowledge about optimization of special moment resisting frame under the consideration of earthquake loading. Various aspects of designing and its use in constructing important and challenging structures is explained in detail. The observations are being made on STAAD Pro. The various changes in member properties is also mentioned in detail. Civil engineering is the vast branch and this deals with the construction of various structures whether it may be building, bridges, towers etc. This research presents the changes being made on a special moment resisting frame used in a building to make it economical. Some changes are being made in sectional dimensions of the members to resist load on a structure and these changes are made to decrease dead load of structure. Earthquake load is considered to check effect of earthquake loading on the structure. Analysis is done to check various outcomes due to dead load, live load and earthquake load.*

## 1. General Introduction

This research is based on analysis and optimization of a multi-storey residential building. The results and required output are calculated using latest and trending designing tool in civil engineering which is STADD Pro. The location of building is designed such that it comes under seismic Zone III. STAAD Pro is a structural design software used for analysis and design of structures. The structures are easily modified and cross sections can be changed simultaneously, which is then revised to get economical design. This is due to different analytical mechanism and the way this analyse the structure take place. This rise in need to do a comparative study between various structures, to know the actual advantages and disadvantages of this software. In order to get a safe and economical design it is necessary to check which digital technology is good and convenient for analysis and design purpose. So, this gives us the actual data required to know

how to optimize a moment resisting frame to get economical design by use of STAAD Pro.

Most of the cases consists of moment resisting frame as it good to resist lateral loads. If buildings are multi-storeyed then it is utmost responsibility to provide the members with proper ductile detailing. Buildings with five storeys are more than five storeys it is necessary to consider Earthquake loading and taking provision of various IS Codes with IS 1893 as main recommendation code for Earthquake Design. Special moment resisting mostly consists of monolithic beams and columns so that they can transfer moment to other members easily.

This research focus on the main concern of Engineers while designing which depicts the main observation such as quality control, serviceability and economy. These points can be achieved by an Engineer if and only if the design has

gone according to recommendation provided by earthquake design code.

## 2. Objective of the Research

Designing is one of core value considered in engineering. After completion of Engineering Degree, one has technical knowledge to perform design and to deliver projects. But delivering project is not as easy as it looks, it needs knowledge of every aspect considered in a structure from designing codes to planning process. The main objectives of this Research are pointed below:

- Modelling and analysis of G+5 R.C. framed structure by the use of STAAD Pro.
- Designing of regular and irregular building plans as per Indian standard codes IS-456 & IS-1893:2002
- Calculation of shear forces, bending moments and reinforcement details for the structural components of the building particularly consisting of beams and Columns, and finally verifying the results.
- Comparison of results obtained in previously design with revised design result values.
- To observe how much accurate result is given by software and does it come under range given in design codes.
- How easily we can make changes using software instead of making manual changes.
- In addition to above points the main purpose of this research is to check up to what extent changes to be made to make structure economical.

## 3. Overview of the Methodology

As this research is related to a building STAAD Pro is one of the best software which is presently in practice for Analysis and Design of Structures. To design a Structure or to Analyze needs different methodology, if done manually. It is easy to analyze a determinate Structure but to deal with indeterminate structure it needs lot of numerical calculation.

So, I found STAAD Pro is the one which can be very helpful to complete my research.

#### a) STAAD PRO

STAAD Pro is a structural analysis and design software used in civil engineering field. This technology was developed by Research Engineers International Yorba Lynda California in 1997. Then after the deep research on this software by large group of technical teams from civil engineering field and software field, it was bought by Bentley Systems a software company in California based to make this happen to work on this software from 2005. This is the latest tool used by civil engineers in structural analysis and design purpose. This software consists of design codes of more than 70 countries. Every designing firm dealing with structural design works on this software as it gives good result and increase productivity. The main aim of Engineers is to get economical, serviceable and durable design at the end. So, to achieve this structural Engineers with some software engineering professionals developed this tool to make it easy for design and analysis of complex structures.

Multi-storied building need large team to perform design if done manually and there may be some design issues including some complexity in approval of cross sections. It is known that manual process needs time to complete a design. So, to overcome this, it is utmost responsibility for an Engineer to elect such type of tools which can perform design according to IS recommendations. There are many tools in civil engineering field but STAAD Pro is well known for its result and is widely used in industries. The output observed on STAAD Pro gives the total detail for a structural design and then drawings are made to forward those designs for implementation. For the drawing purpose AutoCAD is found to be most important and simple to create reinforcement detailing.

#### 4. Parameters of the Structure

A G+5 Structure is analysed and designed as per IS Codes. Earthquake loading is being considered to know the effects of Earthquake loading on the structure. The building with proper recommendations as per IS codes mentioned below are done by inbuilt codes present in STAAD software. A model of G+5 multi-storey building with framed structure is taken as a problem to get the complete thesis result with proper values. A proper model is being modelled on STAAD Pro and property is assigned which include material type, cross section of members etc. After assigning the material, supports are assigned, which is normally taken as fixed support as the structure is building. The various parameters taken are given below:

Type of Building – Residential Multi-storey building.

Type of Structure – Framed Structure.

Beam Size – 300mm X 200mm

Column Size – 230mm X 230mm

Location of the Building – Lucknow

Soil Type – Soft

Frame Type – RCC

Grade of Concrete Used – M30

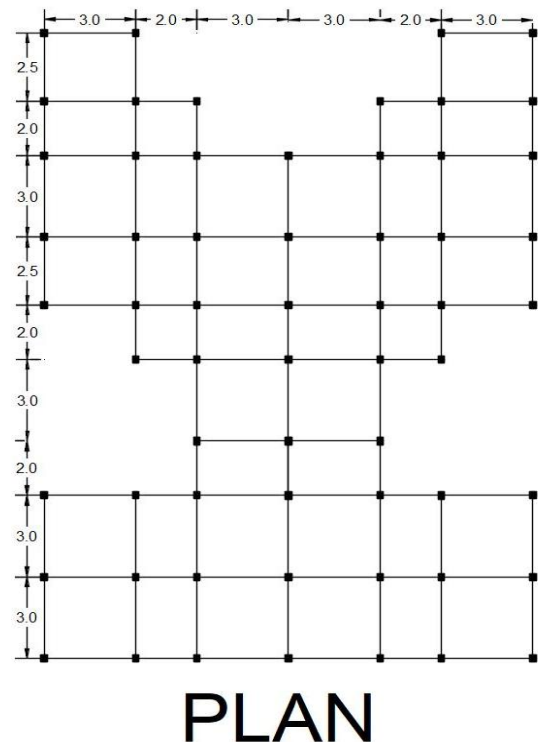
Grade of Steel – Fe 415

Clear cover in case of beams – 25 mm

Clear cover in case of columns – 40 mm

Live load on each floor – 3 KN/M<sup>2</sup>

Live Load on Roof – 1.5 KN/M<sup>2</sup>

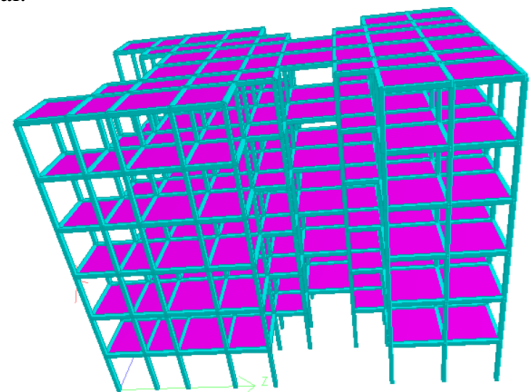


**Figure 1:** PLAN of building with dimensions (STAAD Pro).

Note: All dimensions are in meters.

The fig 1 is the Plan of a building which is to be analysed. Building consists of 6 bays in X-direction and 9 bays in Z-direction as per global axis in STAAD Pro. Dimension of each bay is shown in Plan. The first and foremost step is to select a Plan for this research which must fulfil all requirements.

The column is provided at every intersection. The rectangles at every intersection denotes column and the lines joining these lines are beams in single line diagram. Model consists of 434 nodes which denote joints. They help in load distribution to adjacent members. Moment is distributed to members to stabilize the structure and resist applied loading. First step is to model a structure which shows the beams and columns and these are joined together to make a proper skeletal.



**Figure 2:** 3D Rendered view of Structure

## 5. Earthquake Loading

Earthquake Load is considered as main loading in this project work. Moment resisting frame should be earthquake resistant. So, the loading is taken by proper consideration of location of building, topography factor, soil type, earthquake zone etc. These factors define how much intensity of Earthquake load can be found on a structure. Earthquake load is a lateral load and can be found in two directions i.e. X and Z direction if structure is designed in STAAD Pro as the Y axis show elevation of building. The DOF and some parameters of structure is given below:

Original/final band-width= 49/ 40/ 216 DOF

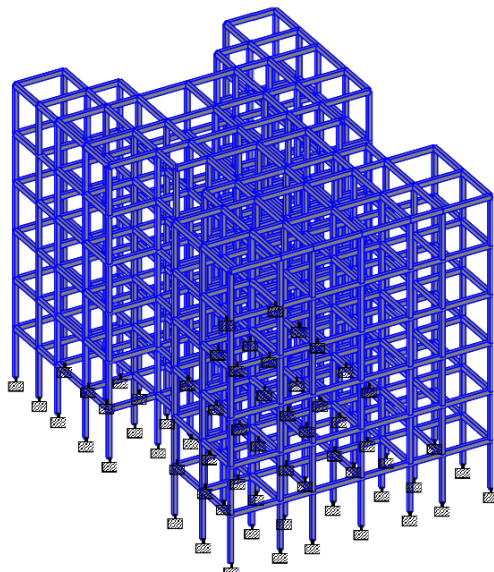
Total primary load cases = 6,

Total degrees of freedom = 2232

Total load combination cases = 27 so far.

Size of stiffness matrix = 483 double kilo-words

Req./avail. Disk space = 18.9/ 76141.2 mb



**Figure 3:** Solid sections showing beams and columns of whole structure.

Solid sections give us the idea about the outline of the members. If there is any need to change the orientation of the member then it can be easily done just before viewing the element in 3D View. Solid sections can be changed by to single line by going to structural diagrams tab.

## 6. Analysis Result

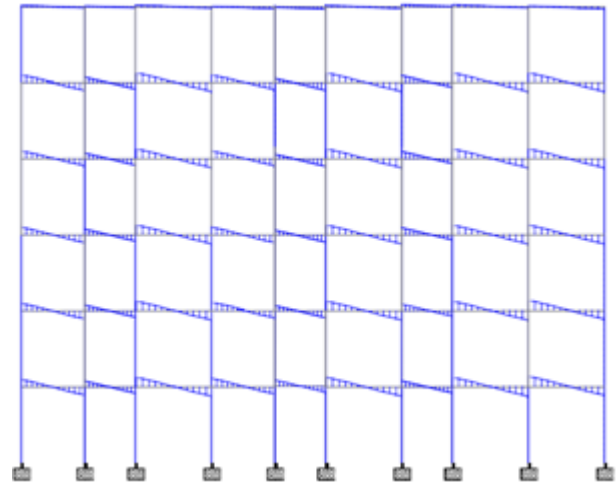
### Introduction

The effect of loading on the structure is known as analysis. Analysis means the generation of Shear Force, Bending Moment, Torsion, Stresses etc due to loading on structure is known as analysis. The various analysis results with suitable diagrams are given section wise:

#### a) Shear Force

The force between two surfaces generated due to external forces is known as shear force. Shear force is found maximum when there are two surfaces in contact. It is found that SF is maximum at ends as there is presence of supports. In case of beams SF gives us the idea about how much reinforcement is needed to counter the external forces.

Stirrups in beams exist as shear force reinforcement whereas main reinforcement is decided based on Bending Moment. In case of columns shear reinforcement is called rings. Stirrups can be two legged, three legged and more based on with of member and number of main reinforcement bars provided.

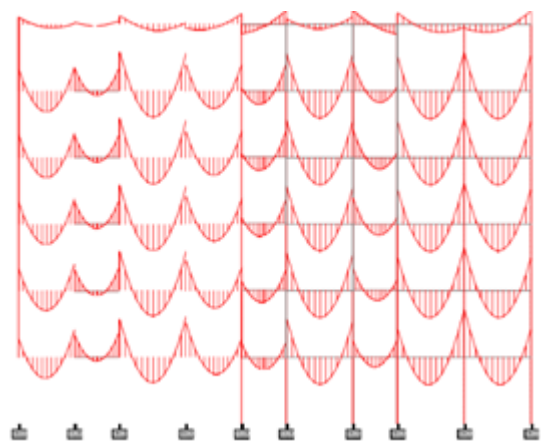


**Figure 4:** Shear Force Diagram of a single frame.

As the loading is always uniformly distributed that is why the above diagram shows shear force changing uniformly as we move beyond support and it will be zero at centre. Same type of diagram is found in every member but there be change in intensity according to loading.

#### b) Bending Moment

The Bending moment defines how much force is responsible for bending at a particular section. Bending will be maximum at the point of application of loading. It is clear from diagram that the bending moment is created due to uniformly distributed loading. In case of point load the Bending, Moment will change linearly whereas in case of uniformly distributed loading bending moment diagram will change parabolically. There will be transfer of moment at joints as the structure is framed type. The distribution of loading takes place due to monolithic nature of structure.

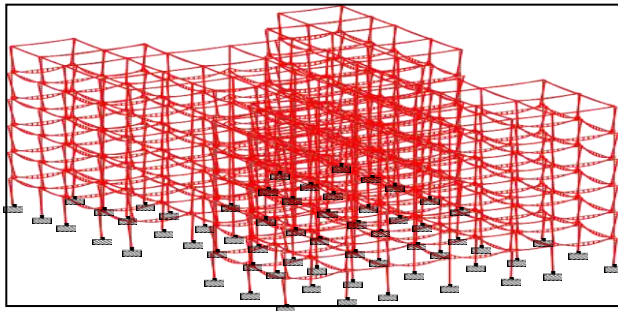


**Figure 5:** Bending moment diagram of a frame

With the help of bending moment values, we can decide how many bars are needed to get embedded in the member so as to resist applied loading. Bending moment also gives us the idea about provision of spacing provided. Ultimate moment at any member is selected for calculation of area of



reinforcement required. The required area of reinforcement is then revised to get area of reinforcement provided. Clear cover for beams is taken as greater or equal to 20mm and in case of columns clear cover is taken as greater or equal to 40mm.

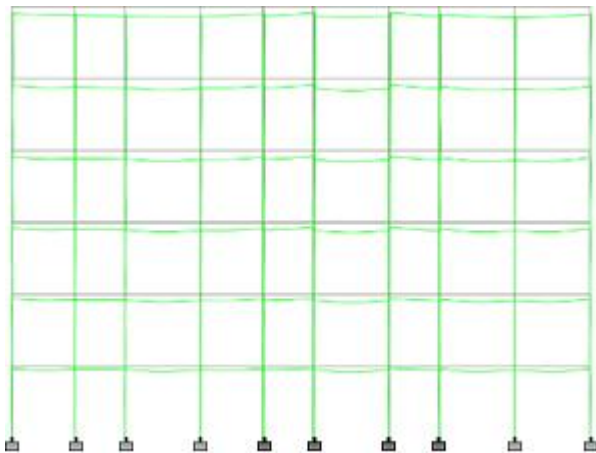


**Figure 6:** Bending Moment Diagram of Whole Structure

Diagrams below show the Shear force and Bending Moment of beam number 37. In fig at the top the bending moment  $M_z$  is maximum at node number 8 with magnitude of 5.56 kN-m. In second figure it is clear that shear force is zero in Y- direction.

### c) Deflection

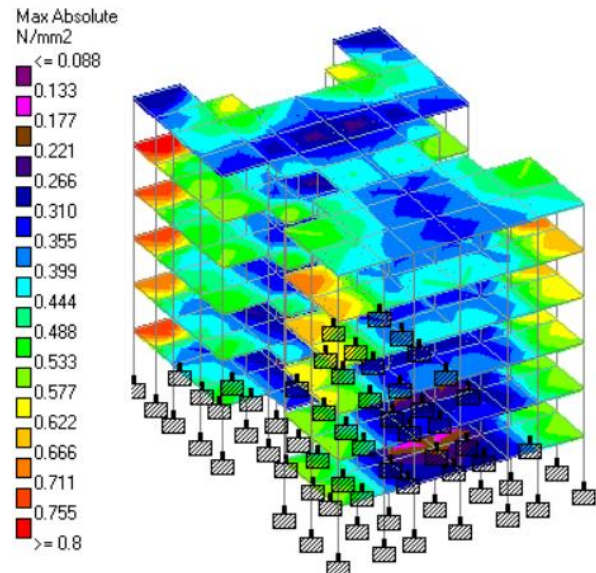
Deflection is the main concern for civil engineers to control. With economical section structural Engineers should kept in mind to minimize deflection as much as possible. According to IS-456 the deflection must not exceed to  $L/250$ . Where "L" denotes the span and is taken in millimetres.



**Figure 7:** Deflection diagram of a frame

It is obvious that deflection maximum deflection is found in long span beams but it can be minimised by placing a column in between or to change section properties. In some cases, sectional changes cannot decrease deflection value. So, it is necessary to place a vertical member to resist that amount of deflection.

Deflection takes place due to loading and loading is of various types. So, Deflection is checked due to Load combination. The deflection is mainly in two directions i.e. vertical deflection and horizontal deflection. Vertical deflection is found due to gravity loads which include self-weight of structure and live loading. Whereas horizontal deflection is found due to lateral loading which in other terms known as Earthquake loading.



**Figure 8:** Stress shown by contours in slab with colour coding.

## 7. Material Take-Off

Reinforcing steel quantity represents reinforcing steel in beams and columns designed above. Below is the quantity take-off showing Volume of concrete and weight of steel. Steel data shows weight of bars used with bar dia.

Total Volume of Concrete =  $153.9 \text{ m}^3$

BAR DIA (mm)	Weight (N)
8	60567
10	30259
12	46776
16	6504
20	3483
Total	147589 N

### Material take-off in Second Iteration

The main aim of structural design engineer is to get the result which will meet all the needs such as economy of the structure, durability and serviceability. The change of cross section leads to decrease volume of concrete but in some cases, there is increase in weight of steel.

The section taken to design the columns were first taken as 230mm X 230mm but to make it possible that structure can withstand at lower cross section also, some changes were made. The size of columns was selected as 230mm X 200mm which made a tremendous decrease in volume of concrete. But after proper analysis process, it was found that two columns at ground floor has failed in design by providing revised cross section. So, I put the same value of cross section as taken before.

In case of beams the section was changed from 300mm X 200mm to "250mm X 200mm" and it was found that all the beams show pass for design. The volume of concrete gets decreased as it is obvious that volume of concrete will decrease with decrease in dimensions of the beam.

Total Volume of Concrete =  $130.5 \text{ m}^3$

BAR DIA (mm)	Weight (N)
8	58214
10	27494
12	46255
16	17653
20	8127
Total	157743 N

After changing grade of steel from Fe 415 to Fe 500, the weight of reinforcement gets decreased from 157743 N to 154954 N.

## 8. Conclusion

Dealing with this Research was really an incredible experience to increase my knowledge in field of designing. While going through this Research I came to know that Degree is incomplete if one is not familiar with such an interesting technology which has made our life as easier as possible. Modelling a virtual model and checking the structural assemblies and adjusting them as your own will gives the most achieving target.

Virtual models nowadays play an important role in checking the suitable locations of each member and their representation. Virtual models help us to check without the loss of resources. I found STAAD Pro as one of the best software to check these properties and to know the behaviour of structure, configuration of building and to check result at each load type. Modelling a G+5 framed structure gives the idea about creating complex models. Modelling a structure in STAAD Pro is one of the major steps which is considered as the main step before applying the loading. Any discrepancy in Modelling leads to failure of structure. While working with this software I came to know that user interface of software is very easy to generate any type of model of the structure. This project helped me to learn IS codes in detail. The main IS Codes considered in this structure are

IS-456:2000 ----- Concrete Design Code.

IS-800:2007 ----- Steel Design Code.

IS-875(Part 1):1987 ----- Dead Loads.

IS-875(Part 2):1987 ----- Live Loads.

IS-875(Part 3):2015 ----- Wind Loads Design Code.

To get the Bending moment and Shear force of an indeterminate structure is very complex to find out by manual process. So, by the software we can find shear force diagram and bending moment diagram of any structure at location.

Structures are of two types; Determinate and Indeterminate. Determinate structures can be solved by three equilibrium conditions while as it is not possible to get Bending and Shear force diagram of indeterminate structures by these equilibrium conditions. To get the results various methods are considered e.g. Slope Deflection Method, Moment Distribution Method, Unit Load Method, Conjugate Beam Method etc. Shear Force Diagram gives the idea to provide Shear Reinforcement (e.g. Stirrups, Rings) while as Bending Moment gives us the idea about provision of main reinforcement. The cross sections were provided on random basis but after getting the results it was found that some

beams are getting failed and finally the section was selected as 200mm X 300mm for beams and 230mm X 230mm for Columns. **“If section provided shows section is ok and can resist the given loading, then section is redesigned to check the least value of cross section which will resist applied loading on a particular member”**

If section gets failed for beam section design then STAAD Pro displays “Section fails while Designing”. In case columns get failed, the message gets displayed in STAAD output as “Section is not adequate”. If the structure is designed for a particular section then there are made some changes which can be made by simply working on property page of STAAD Pro. This Research gave me the idea about the simple methods to make changes in a structure. However, it is observed that if changes are made in sections in case of manual calculations then all values need to be calculated separately which needs lot of efforts. So, to get an economical section we can change section frequently in STAAD Pro by a single command without making the problem complex. So, while optimizing the structure I got two values at the end as given below;

In first iteration the result was found as:

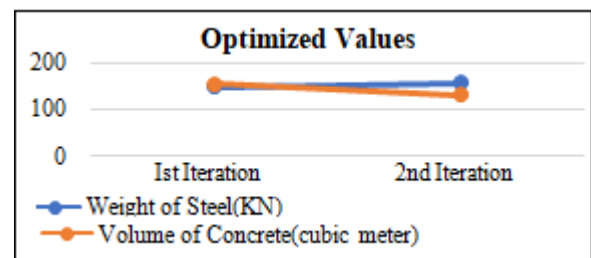
Volume of concrete =  $153.9 \text{ m}^3$

Weight of Steel = 147589 N

In second iteration the result found as below:

Volume of Concrete =  $130.5 \text{ m}^3$

Weight of steel = 157743 N



**Figure 9:** Graph showing change in weight of steel and volume of concrete after optimization of cross sections.

It can be clearly observed that volume of concrete gets decreased but weight of steel gets increased. A designer can change cross section of a member but cannot make any alteration in steel calculation, as steel calculation is given by software. So, by taking this example we found that software made over reinforced section which is the main drawback found in this software. To consider this problem structural Engineers are suggesting ETABS for building design process. STAAD Pro is found good for steel design. But analysis is given accurately by both of the software’.

## References

- [1] Analysis and Design of a multi-storey building by Singhanian and Pares (2011).
- [2] Design by Software to check analysis result by Poonam et al. (2012).
- [3] Dynamic Analysis of a structure using latest technologies by Prasanth’s et al. (2012)
- [4] Analysis and Design of Multi-storey building by Mahesh Suresh Kumawat<sup>1\*</sup> and L G Kalurkar [2014].
- [5] Analysis and Design of G+22 Residential Building by Abhay Guleria [2014].

- [6] Analysis and Design of Structure by Tushar R. Agrawal et al [2015]. Source: IRJET.
- [7] Software approach for analysis and design by Sagar R Padol1 et al [2015].
- [8] Static check of a framed structure by Mahammad sober et al [August 2015].
- [9] Seismic Analysis and Design of Multi-Storied Building by Rinkesh R Bhandarkar et al [2016]..
- [10] Analysis and Design of a Multi Storied Residential Building Of (Ung-2+G+10) By Using Most Economical Column Method by M. Mallikarjun1 [2016].
- [11] Earthquake Analysis of a Multi-storied building by Richa Agarwal et al [2017].
- [12] Structural design; technology and computerization not the problem by Francis SugloLamuo, Sayyed Ahad et al [2017].
- [13] Study on Static and Dynamic Analysis of a multi-storied building by Suchi Nag Choudhary et al [June 2017].